iMOD

DRAFT

User Manual

Deltares

Enabling Delta Life

iMOD

User Manual

4.3

P.T.M. Vermeulen
L.M.T. Burgering
F.J. Roelofsen
B. Minnema
J. Verkaik

Version: 4.3
SVN Revision: 56254

June 21, 2018

iMOD, User Manual

Published and printed by:
Deltares
Boussinesqweg 1
2629 HV Delft
P.O. 177
2600 MH Delft
The Netherlands

telephone: +31 88 335 82 73
fax: +31 88 335 85 82
e-mail: info@deltares.nl
www: https://www.deltares.nl

For sales contact:
telephone: +31 88 335 81 88
fax: +31 88 335 81 11
e-mail: sales@deltares.nl
www: http://oss.deltares.nl

For support contact:
telephone: +31 88 335 81 00
fax: +31 88 335 81 11
e-mail: imod.support@deltares.nl
www: http://oss.deltares.nl

Contents

List of Figures

List of Tables

1 Introduction

Welcome to iMOD. This chapter gives a brief introduction to:

- ◇ section 1.1: our motivation to develop iMOD,
- ◇ section 1.2: the iMOD approach of building groundwater models,
- ◇ section 1.3: iMOD's main functionalities,
- ◇ section 1.4: the minimal system requirements,
- ◇ section 1.5: info on where to get help.
- ◇ section 1.6: some general info on Deltares.

1.1 Motivation

Stakeholders (e.g. water companies, water boards, industrial users) and decision makers (e.g. municipalities, provincial governments) are increasingly participating in jointly developing numerical groundwater flow models that cover land areas of common interest. The reason for this is twofold:

1. minimize the undesired high costs of repeatedly developing individual - partly overlapping - models, and
2. facilitate stakeholder engagement participation in the model building process.

In an effort to facilitate this the concepts of MODFLOW were used by Deltares to develop iMOD (interactive MODeling) to;

1. provide the necessary functionalities to manage very large groundwater flow models, including interactive generation of sub-models with a user-defined (higher or lower) resolution embedded in- and consistent with the underlying set of model data, and
2. facilitate stakeholder participation during the process of model building.

A major difference, compared to other conventional modeling packages, is the generic georeferenced data structure that for spatial data may contain files with unequal resolutions and can be used to generate sub-models at different scales and resolutions applying up- and down-scaling concepts. This is done internally without creating sub-sets of the original model data. For modelers and stakeholders, this offers high performance, flexibility and transparency.

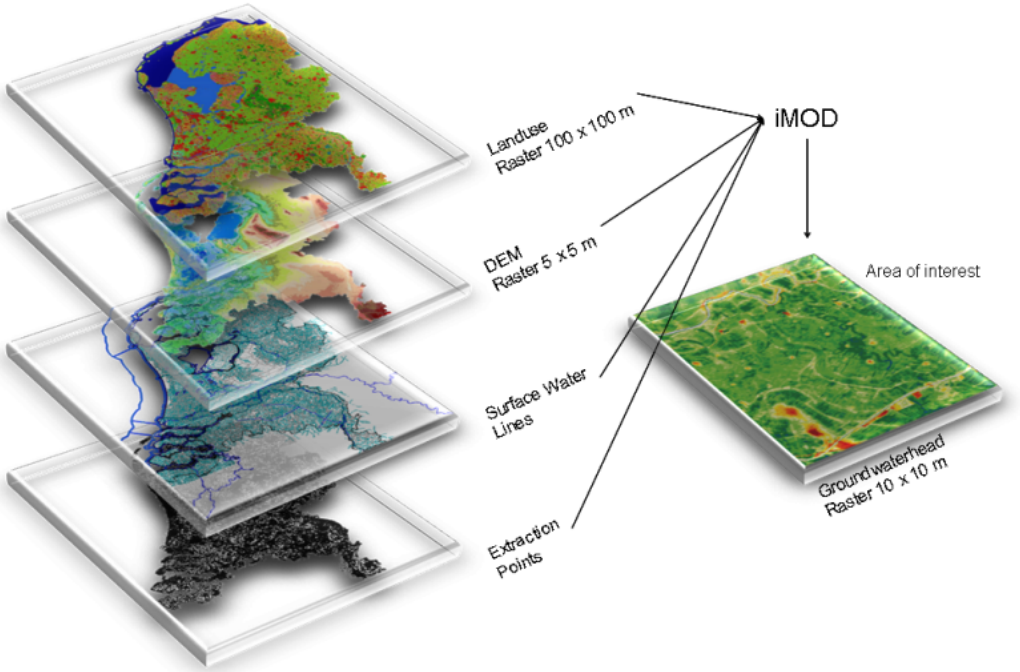
1.2 The iMOD approach

High resolution groundwater flow modeling, necessary to evaluate effects on a local scale, has traditionally been restricted to small regions given the computational limitations of the CPU memory to handle large numerical MODFLOW-grids. Although CPU-memory size doubles every two years ('Moore's law') the restriction still holds from a hardware point of view. This restriction has traditionally forced a model builder to always choose between (1) building a model for a large area with a coarse grid resolution or (2) building a model for a small area with a fine grid resolution. For some time it appeared that finite element models could fill the gap by refining the grid only where hydrological gradients were anticipated. However, unanticipated stress may also occur in parts of the model area where the grid is not yet refined resulting in a possible undesired underestimation of these effects. Theoretically the modeler could choose to design a finite element network with a high resolution everywhere, but then it becomes more economic to use finite differences. This is why Deltares has based its innovative modeling techniques on MODFLOW considering it is largely seen world-wide as the standard finite difference source code. Still, modelers ideally need an approach that allows: (1) flexibility to generate high resolution model grids everywhere when needed, (2) flexibility to use or start

with a coarser model grid, (3) reasonable runtimes / high performance computing and (4) conceptual consistency over time for any part of the area within their administrative boundary. Deltares has invested in understanding all of these requirements and has developed the iMOD software package to advance the methods and approach used by modelers and regulators.

The development of the iMOD approach took off in The Netherlands in 2005 when Deltares and a group of 17 stakeholders decided to jointly build a numerical groundwater model for their common area of interest (Berendrecht *et al.*, 2007, Vermeulen, 2013). The groundwater model encompasses the entire north of the Netherlands at a resolution of 25 x 25 $m^2$and was constructed together via an internet accessible user-interface. This makes it possible for the modelers to easily access the model data, intermediate results and participate in the model construction. The iMOD approach allows gathering the available input data to be stored at its finest available resolution; these data don't have to be clipped to any pre-defined area of interest or pre-processed to any model grid resolution.

The iMOD approach: one input data set:



Resolutions of parameters can differ and the distribution of the resolution of one parameter can also be heterogeneous. In addition, the spatial extents of the input parameters don't have to be the same. iMOD will perform up- and down scaling (Vermeulen, 2006) whenever the resolution of the simulation is lower or higher than that of the available data. This approach allows the modeler to interactively generate models of any sub-domain within the area covered by the data set. When priorities change in time (e.g. due to changing political agenda's) the modeler can simply move to that new area of interest and apply any desired grid resolution. In addition the modeler can edit the existing data set and / or add new data types to the data set. Utilizing the internal up- and down-scaling techniques ensures that sub-domain models remain consistent with the bigger regional model or that the regional model can locally be updated with the details added in the sub-domain model.

Suppose the modeler needs to simulate groundwater flow for the total area covered by the

data set, but the theoretical size of the model is far too big to fit in any CPU-memory. iMOD facilitates generating sub models for parts of the whole area of interest with a user-defined resolution depending on how large the available CPU-memory is and how long the modeler permits her/himself to wait for the model calculations to last. To generate a high resolution result for the whole model domain a number of partly overlapping but adjacent sub models are invoked and the result of the non-overlapping parts of the models are assembled to generate the whole picture. The modeler should of course be cautious that the overlap is large enough to avoid edge effects, but this overlap is easily adjustable in iMOD. A big advantage of this approach is that running a number of small models instead of running one large model (if it would fit in memory, which it often will not) takes much less computation time; computation time (T) depends on the number of model cells (n) exponentially: T = f($n^{1,5-2,0}$). The approach also allows the utilization of parallel computing, but this is not obligatory. Using this approach means that the modeling workflow is very flexible and not limited anymore by hardware when utilizing iMOD.

1.3 Main functionalities

The capability of iMOD to rapidly view and edit model inputs is essential to build effective models in reasonable timeframes. The rapid and integrated views of the geologic / hydrostratigraphic models as well as dynamic model output is critical for the public, stakeholders and regulators to understand and trust the model as a valid decision support tool. iMOD is fast even when working from very large data files because it uses a random accessible data format for 2D grids which facilitates instant visualization or editing subsets of such a large grid file. Also iMOD contains very economic zoom-extent-dependent visualization techniques that allow subsets of grids being visualized instantaneously both in 2D and 3D. Another feature is that iMOD generates MODFLOW input direct in memory, skipping the time-consuming production of standard MODFLOW input files (generating standard MODFLOW input files in ASCII format for large transient models may take hours to a full working day); this efficiency is especially useful during the model building phase when checking newly processed or imported data.

iMOD includes the MetaSWAP-module developed by Wageningen Environmental Research (Alterra); for references to the separate MetaSWAP-documentation see section A.1.8.

1.4 Minimal System Requirements

iMOD works on IBM-compatible personal computers equipped with at least:

- ◇ a Pentium or compatible processor;
- ◇ 512 MB internal memory (2,045MB recommended);
- ◇ 100 MB available on the hard disk (10GB is recommended in case large model simulations need to be carried out);
- ◇ A graphics adapter with 32 MB video memory and screen resolution of 800-600 (256MB video memory and a screen resolution of 1024x768 is recommended). Moreover, a graphical card that supports OpenGL (OpenGL is a trademark of Silicon Graphics Inc.), such as an ATI Radeon HD or NVIDIA graphical card is necessary to use the 3D rendering.

Please note: it is permitted to install the Model System on a different Hardware Platform as long as it is a computer with similar minimum features as listed above. The transfer of the Model System to a dissimilar computer may endanger the working of the Model System and require adjustments in the Configuration.

iMOD can run on 64-bits systems, but iMOD itself is 32-bits. iMOD supports 32- and 64-bit machines working under the following platforms: Windows XP / Server 2003 / Vista Business

/ Vista Ultimate / Server 2008 / 7 .

1.5 Getting Help

Take a look at http://oss.deltares.nl/web/imod. Any questions? Contact the help-desk imod.support@deltares.nl.

1.6 Deltares

Since January 1^{st} 2008, GeoDelft together with parts of Rijkswaterstaat-DWW, -RIKZ and -RIZA, WL | Delft Hydraulics and a part of TNO Built Environment and Geosciences are forming the Deltares Institute, a new and independent institute for applied research and specialist advice. For more information on Deltares, visit the Deltares website: www.deltares.nl.

1.7 Acknowledgements

The development and enhancement of iMOD functionality is project-based. This section lists these project-based developments and specifies its funding and organisations Deltares has collaborated with during the implementation.

Functionality	Funding	Implementation
iMOD Maintenance & Support & iMOD-Helpdesk In 2013 a group of five iMOD-consortia started the project "iMOD Beheer en Onderhoud, Helpdesk en Website" initiating and (co-)financing a coordinated further development of iMOD and enhanced maintenance and support. The current iMOD-consortia are AMIGO, AZURE, IBRAHYM, MIPWA and MORIA; info on the members of each iMOD-consortium can be found here.	**iMOD-CGO consortia**	Deltares
MetaSWAP iMOD includes the unsaturated zone MetaSWAP-module which covers the plant-atmosphere interactions and soil water. MetaSWAP is based on a quasi steady-state solution of the Richards equation. MetaSWAP is developed by Wageningen Environmental Research (Alterra) and was (among others) financed by The Netherlands Hydrological Instrument. For references to the MetaSWAP-documentation see section A.1.8.	NHI	WAGENINGEN UNIVERSITY & RESEARCH
MODFLOW-MetaSWAP coupling The coupling of MODFLOW and MetaSWAP was created in a collaboration between Deltares and Wageningen Environmental Research (Alterra) and was (among others) financed by The Netherlands Hydrological Instrument.	NHI	Deltares WAGENINGEN UNIVERSITY & RESEARCH

Functionality	Funding	Implementation
Quick Scan Tool The MIPWA consortium initiated and funded the QuickScan Tool which is an instrument to efficiently compute effects on groundwater levels and seepage fluxes to- and from drainage systems using a so-called Impulse-Response Database. This database stores pre-computed effects of several measures which can be combined in the QuickScan Tool using the principles of superposition.	**MIPWA consortium**	Deltares
Perched Water Table package The MIPWA consortium initiated and funded the development of the Purged Water Table (PWT) Package. With this package purged water table conditions can be simulated occurring on shallow (clayey) aquitards with a significant vertical resistance. The initial concept was developed in collaboration with Wageningen Environmental Research (Alterra).	**MIPWA consortium**	Deltares WAGENINGEN UNIVERSITY & RESEARCH
3D Tool Waternet funded the development of the first version of the 3D Tool allowing an interactive 3D visualization of the subsurface in combination with the (litho-stratigraphy) of boreholes.	waternet waterschap amstel gooi en vecht gemeente amsterdam	Deltares
GeoConnect Tool The GeoConnect Tool allows the modeller to define and utilize permanent links between 1) the (unassembled) geologic layers (incl. its properties) and 2) the aggregated model layers. With the GeoConnect Tool you can re-calculate the hydraulic conductivities of a model layer after adapting the individual weights of each contributing geological layer. The development of was funded by the IBRAHYM concortium (Waterschap Limburg, Provincie Limburg, Waterleiding Maatschappij Limburg).	**IBRAHYM consortium**	Deltares
ISG The IBRAHYM consortium initiated the development of the concept of line elements (vector format) as a basis to discretize streams as an alternative for grid based parameterization. This yielded considerable data handling efficiency and much more flexibility when parameterization for different model grids sizes. It also facilitated more user-friendliness regarding inspecting and editing the stream data.	**IBRAHYM consortium**	Deltares
Runfile Editor & Plug-In Tool The project manager was extended to support editing of a runfile and project files. Also the iMOD-GUI was extended with Plug-In functionality allowing to invoke external programs in the iMOD-GUI using iMOD files. These functionalities were funded by the iMOD-CGO group.	**iMOD-CGO consortia**	Deltares

Functionality	Funding	Implementation
Parallel Krylov Solver Package Deltares, USGS and Wageningen Environmental Research (Alterra) together with Utrecht University and Technical University Delft have developed the new parallel solver package for iMOD called PKS (Parallel Krylov Solver). It is based on overlapping domain decomposition combining both the techniques of MPI and OpenMP.	Deltares USGS science for a changing world WAGENINGEN UNIVERSITY & RESEARCH	Deltares USGS science for a changing world WAGENINGEN UNIVERSITY & RESEARCH
3D-assignment of fault-lines Extension and improvement of the 3D-parameterization of faults in the Horizontal Flow Barrier package. This extension was funded by the IBRAHYM-consortium led by the Province of Limburg.	**IBRAHYM consortium**	Deltares
Extension of the water balance tool The extension of the water balance tool (released in iMOD 4.2) was created in a collaboration between Deltares and Tauw and financed by the Dutch iMOD-CGO group; it allows visualization (interactive stackbars time series plots and schematic vertical cross-sectional overviews) of water balances for sub-regions and contains several time-aggregation possibilities (e.g. averages per year, month and season).	**iMOD-CGO consortia**	Deltares Tauw
Fence diagrams & Deviated wells The 3D Tool was extended with an option to interactively create so-called fence-diagrams by using clipping planes along the major Euclidian axes. The 3D Tool was also extended with the option to visualize deviated wells. These developments were funded by the Alberta Energy Regulator | Alberta Geological Survey and came available starting from the iMOD 4.2 release.	Alberta Energy Regulator AGS ALBERTA GEOLOGICAL SURVEY	Deltares

2 Getting Started

This chapter describes:

- ◇ section 2.1: How to obtain the Deltares-software executables of iMOD.
- ◇ section 2.2: How to install iMOD.
- ◇ section 2.3: How to get and install third-party MPI software.

iMOD 4.3 includes the new Parallel Krylov Solver (PKS) Package. This package facilitates running (large) iMOD models on Windows-based multi-core computers potentially resulting in a drastic reduction of runtimes. The PKS-package is a new alternative next to the current single-core PCG-solver. The modeller of course is not required to apply the new PKS-package, however, in iMOD 4.3 it is easy to switch between the single- and multi-core solver.

In iMOD 4.3 the LAK-, MNW-, PST-, SFR- and UZF-packages are not supported by the PKS-package; when an iMOD-model contains one or more of these package the single-core PCG-solver package has to be used.

The PKS-package can be used on Windows-based 64-bits systems only.

Please note that the PKS package can be used for iMOD models that contain the MetaSWAP-concept. This potentially gives another boost to the speed of model simulations containing an iterative state-of-the-art coupling between the saturated and unsaturated zone.

The PKS package uses Message Passing Interface (MPI) software. Hence, prior to using the PKS-package MPI-software should be installed on your computer too; section 2.3 describes how to get and install the MPI-software. However, if you are not going to use the PKS-package you are not required installing the MPI software and you can skip section 2.3 all together.

The other sections focus on how to operate iMOD and how new users can familiarize themselves with iMOD:

- ◇ section 2.4: a 3D-appetizer.
- ◇ section 2.5: how to start iMOD after installation.
- ◇ section 2.6: the Main Menu options.
- ◇ section 2.7: how to specify your preferences.
- ◇ section 2.8: how to specify your colours.
- ◇ section 2.9: some Tips and Tricks.

Additionally, the Tutorials in chapter 11 provide a selection of iMOD case studies to introduce the program's functions. New iMOD users are advised to use the Tutorials to familiarize themselves with iMOD.

2.1 Get the Deltares-software executables of iMOD

To get the latest release of the Deltares-software executables of iMOD, please submit the form '*Request form for the Deltares-software executables of iMOD*', see http://oss.deltares.nl/web/imod/get-started.

By submitting this form you are requesting for the use of the Deltares-software executables of iMOD. iMOD is Deltares-software; the source code of iMOD is also available as free open source software at oss.deltares.nl. You may use the Deltares-software executables of iMOD without any remuneration to be paid to Deltares if you accept the iMOD Software License Agreement (iMOD License) which is offered to you as a PDF-file, see http://oss.deltares.nl/web/iMOD/iMOD_Software_License_Agreement. Please go to the PDF-file of the iMOD License, read it and decide whether you want or do not want to accept the iMOD License. Without your acceptance of the iMOD License the use of the Deltares-executables of the iMOD-software is prohibited and illegal.

The iMOD software is distributed in the hope that it will be useful, but WITHOUT ANY GUARANTEE OR (IMPLIED) WARRANTY. Any use of the Deltares-executables of the iMOD-software is for your own risk. See the iMOD License for more details.

For more info, please contact: Stichting Deltares, P.O. Box 177, 2600 MH Delft, The Netherlands. Email: imod.support@deltares.nl.

After your submitted request (section 2.1) was processed successfully you received an email with details on how to download the Deltares-executables of iMOD; this email contains the required download-password.

Please perform the following steps:

1. Browse to https://download.deltares.nl.
2. Click the iMOD-icon.
3. Download the file '**iMOD 4.3 Installation Instructions.pdf**': enter the download-password sent to you in the above mentioned email and click the 'Download'-button.
4. Download the file '**iMOD 4.3.zip**': enter the download-password sent to you in the above mentioned email and click the 'Download'-button.
5. Unzip the file '**iMOD 4.3.zip**'.

After unzipping the file '**iMOD 4.3.zip** you will have the following 3 files:

1. **iMOD 4.3 Installation Instructions.pdf:** the iMOD installation instructions.
2. **iMOD_setup_V4_3.exe:** an installation program that:
   - ◇ determines whether you are installing iMOD on a 32-bit or on a 64-bit system (by processing the output of the DOS-command 'systeminfo')
   - ◇ determines your default PDF-viewer, if available.
   - ◇ calls the self-extracting archive iMOD_zipped_V4_3.000,
   - ◇ creates an initial IMOD_INIT.PRF preference file,
   - ◇ installs the Tutorial Data Set.
3. **iMOD_zipped_V4_3.000:** a self-extracting archive containing the following files and directory:
   - ◇ iMOD_V4_3_X32R.exe: *the iMOD-GUI for 32-bit systems*.
   - ◇ iMOD_V4_3_X64R.exe: *the iMOD-GUI for 64-bit systems*.
   - ◇ iMODFLOW_V4_3_X32R.exe: *iMODFLOW excl. MetaSWAP, for 32-bit systems*.
   - ◇ iMODFLOW_V4_3_METASWAP_SVN1233_X64R.exe: *iMODFLOW including MetaSWAP for 64-bit systems. The MetaSWAP-module is de-*

veloped by Wageningen Environmental Research (Alterra); for references to the separate MetaSWAP-documentation see section A.1.8.

- ◇ netcdf.dll: *a NetCDF-library*.
- ◇ fmpich2.dll, mpich2mpi.dll and mpich2nemesis.dll: *MPICH 1.4.1p1 libraries necessary for PKS*.
- ◇ run_test_mpi_installation.bat, test_mpi_installation.exe: *a test to check a MPI-installation.*
- ◇ USGS Software User Rights Notice.txt: *a copy of the USGS Software User Rights Notice*.
- ◇ iMOD_User_Manual_V4_3.pdf: *the iMOD User Manual, in PDF-format*.
- ◇ iMOD_Software_License_Agreement_V4_3.pdf: *the PDF-file of the iMOD License*.
- ◇ the files and folders of the iMOD Tutorial Data Set.

2.2 Installation of iMOD

To install iMOD please perform the following steps:

1 Make sure you have **read-, write and execute rights** in the directory you want to install iMOD in. You also need **read- and write rights** to create sub-directories and to create, read and write files.

Note: Starting from iMOD version 3.2 you only need **execute-rights** for *{installfolder}* after iMOD has been installed and the iMOD-GUI has been invoked once allowing iMOD to generate the file 'I_accepted_V4_3.txt' in *{installfolder}*; the file 'I_accepted_V4_3.txt' is generated when the user selects the option 'I Accept' during first time use in the iMOD-GUI.
Until iMOD version 3.01 the iMODFLOW-executable was always copied to the actual modelrun-folder and the iMOD-GUI invoked the iMODFLOW-executable copied to that modelrun-folder; starting from version iMOD 3.2 the iMODFLOW-executable is still copied to the modelrun-folder for archiving purposes, however, the iMOD-GUI invokes the iMODFLOW-executable as defined by the keyword 'MODFLOW' in the preference file.

Note: Administrators who wish to assign **execute-rights** only to *{installfolder}*, please invoke the iMOD-GUI once and follow the 'I Accept'-procedure to allow iMOD to write the file 'I_accepted_V4_3.txt' in *{installfolder}*; once the file 'I_accepted_V4_3.txt' is present in *{installfolder}* iMOD only needs **execute-rights** for *{installfolder}*.

During installation no additional access rights are required for other directories; after installation when you start using iMOD you of course need access rights for your to be created iMOD project-files and -directories.

Note: The installation of iMOD does NOT require write access to the Windows Registry, in other words, the installation of iMOD does not make any changes in the Windows Registry.

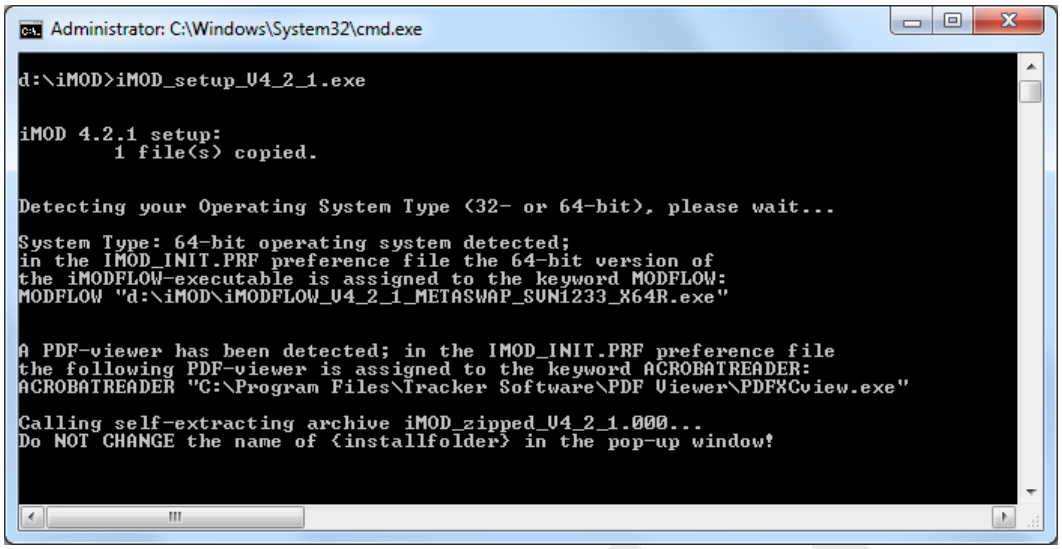
2 Create and go to a new (sub-)directory or go to an existing EMPTY directory where you want to install iMOD.

Note: In this user manual *{installfolder}* refers to the full path of the directory you installed iMOD in (e.g. D:\iMOD).

3 Move the files **'iMOD_setup_V4_3.exe'** and **'iMOD_zipped_V4_3.000'** to *{installfolder}*, e.g. to D:\iMOD.

4 In *{installfolder}* double-click the file **'iMOD_setup_V4_3.exe'** from the Windows Explorer, or start a 'Windows Command Processor'-box in *{installfolder}*,
type 'iMOD_setup_V4_3' and press Enter.
If the archive **'iMOD_zipped_V4_3.000'** is not present, the setup will stop.

5 The setup will first determine 1) whether you are installing iMOD on a 32-bit or 64-bit system (using the DOS-command 'systeminfo') and 2) which default pdf-viewer is available, if any. After that, the Windows Command Processor box looks similar as below (and a pop-up window appears, see step 6):

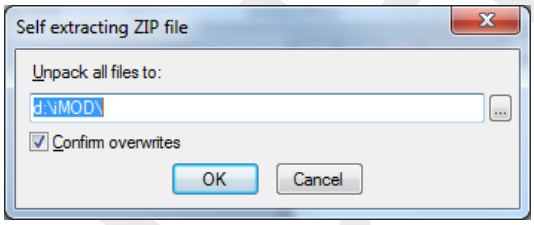


In the preference file IMOD_INIT.PRF (see step 7 below) the keyword 'MODFLOW' will be assigned the 64-bit or the 32-bit version of the iMODFLOW-executable respectively *(64-bit versions of Windows became available starting from Windows XP).*

If during setup your Windows Operating System Type could not be determined the setup uses 64-bit as a default.

If during setup no default PDF-viewer is detected the keyword ACROBATREADER can be specified in the file IMOD_INIT.PRF manually, see step 7 below.

6 In this step the self-extracting the archive **'iMOD_zipped_V4_3.000'** is invoked and the following window appears; do NOT CHANGE the name of the *{installfolder}* and click the 'OK'-button; the archive will unzip.



After the archive **'iMOD_zipped_V4_3.000'** has finished self-extracting (it may take a while to extract more than 3300 files...) also a new sub-folder TUTORIALS has been created in *{installfolder}*, containing a sub-folder for each individual tutorial:

- .\TUT_Map_Display
- .\TUT_Data_Map_Oper
- .\TUT_Map_Analyse
- .\TUT_Initial_Modeling
- .\TUT_Solid_Building
- .\TUT_Model_Simulation
- .\TUT_IPS
- .\TUT_SFR
- .\TUT_LAK
- .\TUT_MNW
- .\TUT_UZF

7 An initial iMOD preference file IMOD_INIT.PRF has been generated, including the keywords 'USER', 'HELPFILE', 'ACROBATREADER', 'MODFLOW' and 'DBASE'. For more keywords please have a look in Section 9.1.

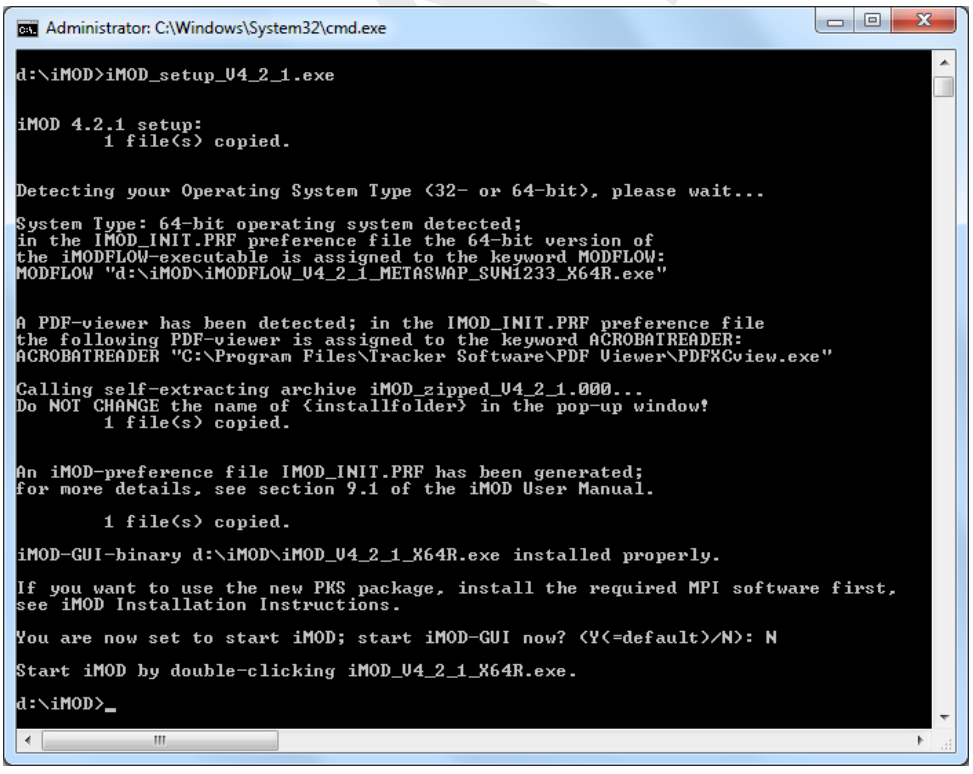
   For the examples used above the content of the IMOD_INIT.PRF-file looks like this:

```
USER "d:\iMOD\IMOD_USER"
HELPFILE "d:\iMOD\iMOD_User_Manual_V4_3.pdf"
ACROBATREADER "c:\Program Files (x86)\Adobe\Reader 11.0\
Reader\AcroRd32.exe"
MODFLOW "d:\iMOD\iMODFLOW_V4_3_MetaSWAP_SVN1233_X64R.exe"
DBASE "d:\iMOD"
```

8 You have now completed the installation of iMOD:

   ◇ When you press Enter, **the iMOD-GUI will start**.
   ◇ Or when you type 'N' and press Enter:

      □ and if in step 2 you invoked the setup by a double-click, the Windows Command Processor-box will now be closed.
      or
      □ and if in step 2 you invoked the setup by typing 'iMOD_setup_V4_3' from a manually opened 'Windows Command Processor-box', after pressing Enter the box will look similar (depending on your {installfolder}) to this:



The installation of the iMOD software is now completed.

Note: To be able to use the Parallel Krylov Solver (PKS) package additional third party MPI software needs to be installed first, see the next section.

2.3 Installation of MPI software

iMOD 4.3 includes the Parallel Krylov Solver (PKS) package. The Parallel Krylov Solver (PKS) package facilitates running (large) iMOD models on Windows-based **multi-core** computers potentially resulting in a drastic reduction of runtimes. This package uses Message Passing Interface (MPI) software. Hence, prior to using the PKS-package MPI-software should be installed on your computer too.

2.3.1 Limitations

The PKS package was implemented for the 64-bit version of iMODFLOW (see Step 5), so the PKS package can be used on Windows-based multi-core 64-bit computer systems, for example on Windows-based laptops and desktops containing Intel CORE i5 or i7 processors or on Windows-based multi-core 64-bit supercomputers.

In iMOD 4.3 the LAK-, MNW-, PST-, SFR- and UZF-packages are not supported by the PKS-package; when an iMOD-model contains one or more of these package the single-core PCG-solver has to be used. So, the PKS-package can be used in combination with the remaining MODFLOW-packages implemented in iMOD; please note that the PKS package can also be used for iMOD models containing the MetaSWAP-concept; this potentially gives another boost to the speed of model simulations containing an iterative state-of-the-art coupling between the saturated and unsaturated zone.

2.3.2 Installation steps for the MPI software

The 64-bit iMODFLOW-executable uses the 64-bit MPICH (1.4.1p1) implementation for MPI, hence the following MPI software `http://www.mpich.org/static/downloads/1.4.1p1/mpich2-1.4.1p1-win-x86-64.msi` should be installed prior to using the PKS package.

Note: In order to install the MPI software correctly, you should do this as Administrator:

- ◇ Download the appropriate MPI software.
- ◇ Open the Command Prompt as administrator by: *Start* → *(Search programs and files)* cmd → *right mouse click* → Run as administrator
- ◇ In the MS-DOS-box named *Administrator: Command Prompt*:
  `msiexec /i mpich2-1.4.1p1-win-x86-64.msi`
- ◇ Follow the instructions of the MPI-installer.

2.3.3 Checking your MPI-installation

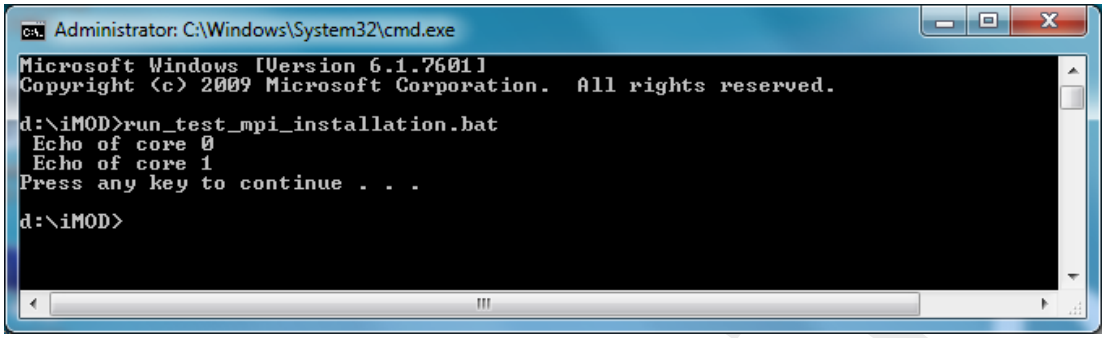
We included a general test-program that allows you to check whether the installation of the MPI-software was successful. To perform this check please do the following:

- ◇ Go to your *{installfolder}*, e.g. to D:\iMOD.
- ◇ Open the batch-file 'Run_test_mpi_installation.bat' in an ascii-editor.
- ◇ In line 2, adjust the reference to the folder and name of the installed MPI-executable, e.g.
  `set mpi "C:\Program Files\MPICH2\bin\mpiexec.exe"`
- ◇ Save the batch-file and close the ascii-editor.
- ◇ Open a 'Windows Command Processor'-box.
- ◇ Type 'Run_test_mpi_installation.bat' and press Enter.

A test is performed using 2 cores; when the test is successful the following message appears:

```
echo of core 0
echo of core 1
```

and the 'Window Command Processer'-box should look similar to this:



Please contact your system administrator for help on installing MPI.

2.3.4 Info on how to use the PKS-package

It is very easy to adapt existing (pre-iMOD 4.0)-runfiles (*.RUN) for the use of the PKS-package; for more info, see section 'Updating a runfile from iMOD 3.6 to iMOD 4.0' in section 10.20.6 of the iMOD User Manual.

The PKS-package can be configured in the 'Solver Settings'-tab of the 'Start Model Simulation'-window of the main menu option 'Tools'; for more info see the iMOD User Manual, section 7.9.

Once the PKS-package has been configured, there are two ways to start a multi-core model simulation:

1 Inside the iMOD-GUI: see section 7.9 ('Model Simulation') of the iMOD User Manual.
2 Outside the iMOD-GUI by typing the appropriate command at the DOS-prompt in a 'Windows Command Processor'-box; here's an example of how to start a multi-core model simulation from outside the iMOD-GUI by entering the following command in a 'Windows Command Processer'-box:

```
mpiexec.exe -localonly 2 iMODFLOW.exe iMODFLOW.run
```

In this example MPI launches two processes of iMODFLOW.exe instances on two computational cores, meaning that the model runs using two subdomains. The `-localonly` option ensures that you should not necessarily have to be connected to your network for running with MPI.
For more detailed info, see section 10.21 ('Start Model Simulation') of the iMOD User Manual.

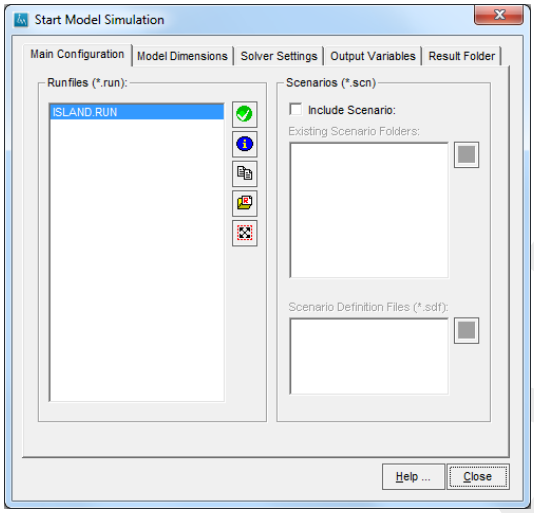
Tutorial 6 (section 11.6) contains an excercise on how to run your model using the PKS package.

You are now set to start iMOD; after completing this chapter new iMOD-users are encouraged to proceed with the tutorials (chapter 11 of the iMOD User Manual). As a 3D-appetizer consider trying to run and visualize one of the pre-defined tutorial models: follow the steps as described in the next section.

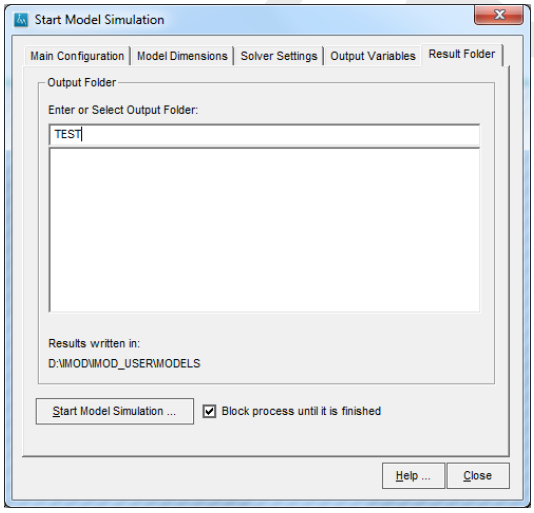
2.4 A 3D-appetizer...

As an appetizer let's try to run and visualize one of the pre-defined tutorial models in 3-D right away by performing the following steps:

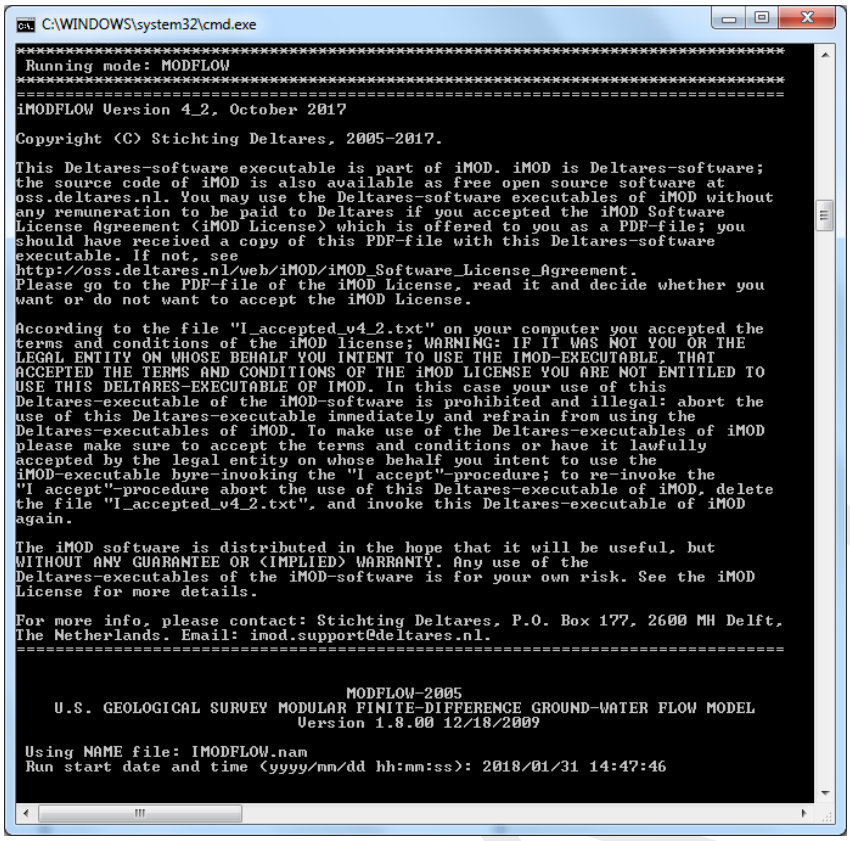
1 If you chose 'N' in the previous step, start iMOD as described in section 2.5.
2 Read and decide whether or not you accept the term and conditions of the iMOD License Agreement.
3 Assuming you accept, click 'Yes, I Accept' and click 'OK'
4 When the 'iMOD Start' window appears, click the 'Start' button.
5 In the main menu click 'Toolbox', and click 'Start Model Simulation ...', the following window appears, displaying the presence of the runfile 'ISLAND.RUN':



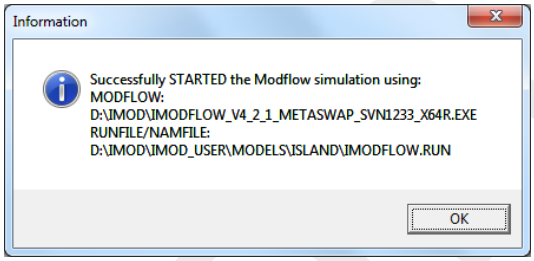
6 Select the tab 'Result Folder' and type a to be created output folder name, e.g. 'TEST':



7 Click the button 'Start Model Simulation' and click 'Yes' in the confirmation pop-up window to start the MODFLOW run; a Windows Command Processor box appears echoing the MODFLOW in- and output process:

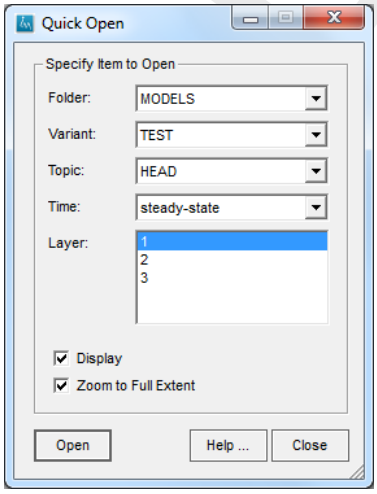


followed by an information window:

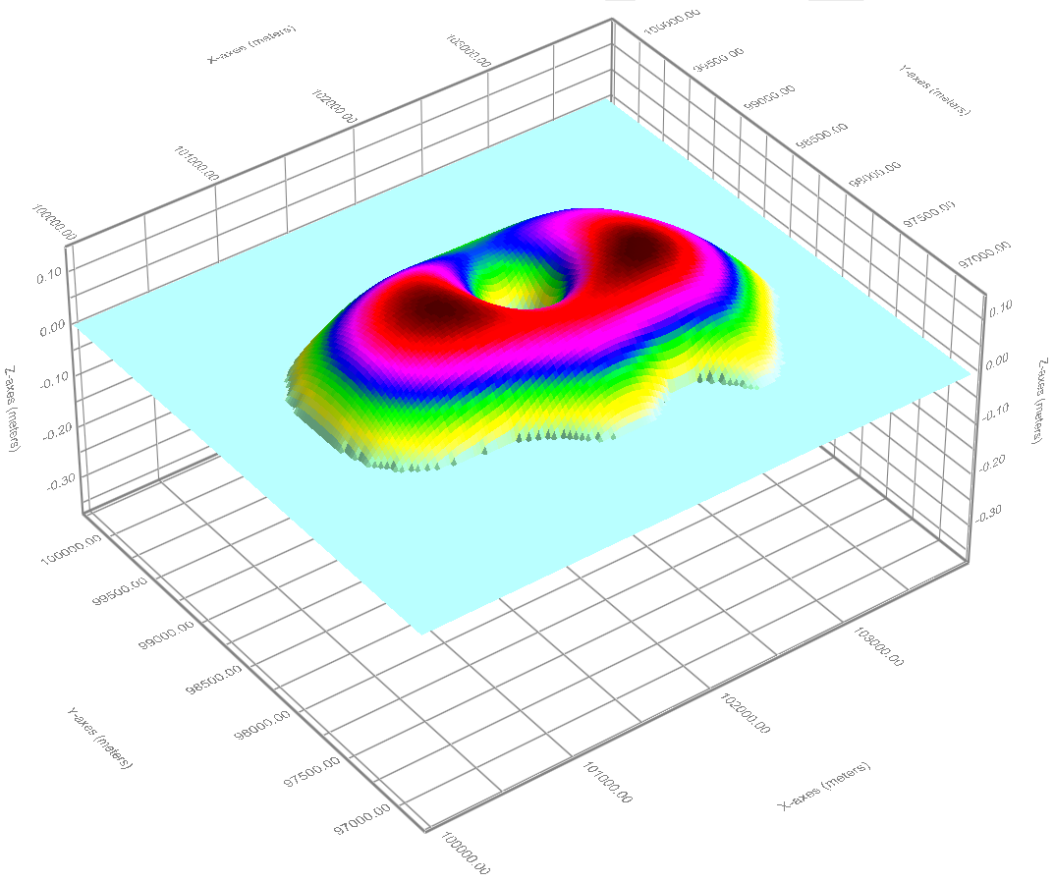


8 Click 'Ok' in this information window, and 'Close' in the 'Start Simulation Model'-window.

9 To visualize the calculated groundwater head in 2D, in the main menu click 'Map' and 'Quick Open ...'; the following window appears:



10 In the 'Topic'-list, choose 'HEAD'.
11 In the 'Layer'-window, click at least one layer, e.g. layer 1.
12 Select 'Zoom to full extent'.
13 Click the 'Open'-button to visualize the calculated heads of layer 1 of tutorial 4; click the 'Close' button.
14 Optionally, you can redraw the map using different colours by choosing 'Map' in the main menu, followed by selecting 'Entire extent' and clicking on 'percentiles'.
15 To visualize these groundwater levels in 3D right away, select 'Toolbox' in the main menu and choose '3D Tool ...'; click 'Apply' to use the default settings.
16 In the '3D Plot Settings'-window, in the 'Colouring' sub-window choose the 'Use Colouring defined in Legend for:'-option to re-use the legend-colours of the 2D-map.
17 Change the 3D-view by dragging over the 3D-map while you keep pressing the left-, middle- or right mouse button. The 3D-map could look something like this:

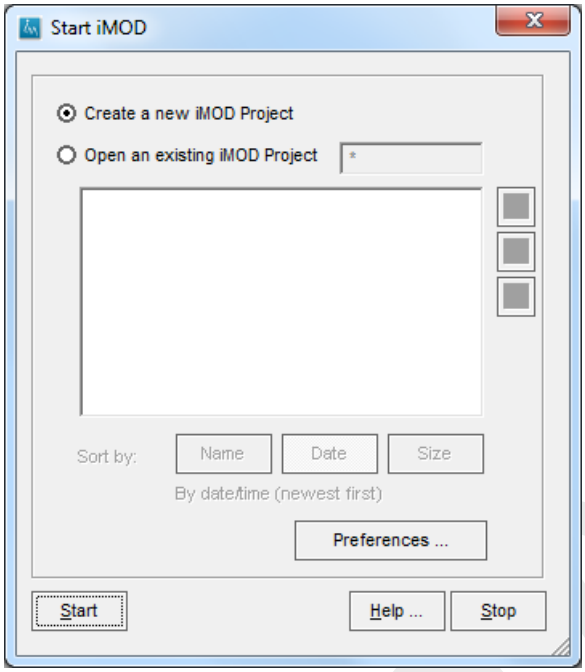


18 Optionally, in the 3D Tool change the Horizontal / Vertical ratio by selecting in the 3D Tool menu bar the option 'View' and select 'Horizontal / Vertical Ratio' followed by the choosing your preferred ratio.
19 Congratulations, you just visualized your first iMOD-model! You are now set to continue exploring iMOD; after completing this chapter new iMOD-users are encouraged to proceed with the tutorials (chapter 11).

2.5 Starting iMOD

To start iMOD, click *Start* on the Windows menu bar and fill in the location and name of the executable: {installfolder}\iMOD_V4_3_X64R.EXE, e.g. D:\iMOD\iMOD_V4_3_X64R.EXE or C:\program files\iMOD_V4_3_X64R.EXE, or double-click on the executable from the *Windows Explorer*. The *Start iMOD* window will appear.

Start iMOD window:



Create a New iMOD Project	Select this option to refresh the iMOD session and release all memory and maps from previous sessions and start iMOD with an empty drawing list.
Open an existing iMOD Project	Select this option to start iMOD with an iMOD configuration saved by a previous iMOD session. Those configurations are stored in *.IMF files and those listed are found in the folder {USER}\IMFILES. Use the wildcard to select a part of existing *.IMF files in the menu field. Use wildcards as "*" (any sign) and "?¿' (any two characters) for specific selections, e.g. *A_??*. The search will be case insensitive and the extension IMF will be added to it automatically. Leaving out any wildcard will act as "*.IMF".
	*Open an IMF-file* Select and search an *.IMF-file from a different location than those presented in the menu.
	*Information of an IMF-file* Click this button to open the selected *.IMF-file in a regular text-editor (*Notepad*) for inspection or adjustments.
	*Delete an IMF-file* Click this button to delete the selected *.IMF file from disk. After that, no recovery is possible.
Sort by:	Select one of the following to sort the list of IMF files: ◇ **Name** Select this to sort the IMF file by Name (case insensitive); ◇ **Date** Select this to sort the IMF file by Date/Time, youngest will be appearing on top of the list; ◇ **Size** Select this to sort the IMF file by Size (largest first).
Preferences . . .	Click this button to open the *Preferences* Window.
Start	Click this button to start iMOD with the selected *.IMF file or with an empty drawing list.

Stop	Click this button to stop iMOD
Help. . .	Click this button to start the iMOD Help Functionality.

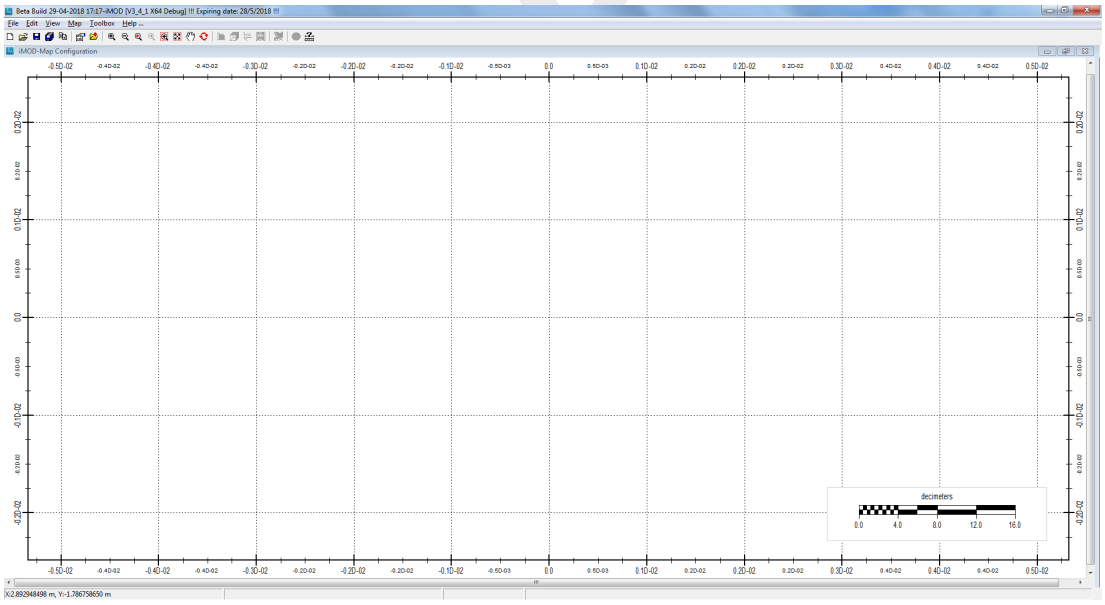
Note: iMOD can be started in different ways, alternatively:

- ◇ **{installfolder}\iMOD_V4_3_X64R.exe**
  will start a regular iMOD session
- ◇ **{installfolder}\iMOD_V4_3_X64R.exe *.IMF**
  will start an iMOD session and read the supplied *.IMF directly;
- ◇ **{installfolder}\iMOD_V4_3_X64R.exe *.IDF**
  will start an iMOD session and read the supplied *.IDF-file directly. This works for *.MDF, *.ASC, *.GEN, *.IFF, *.IPF, and *.ISG-files;
- ◇ **{installfolder}\iMOD_V4_3_X64R.exe *.INI**
  will read the supplied *.INI file. These *.INI files contain specific functionalities of iMOD that can be executed without starting the graphical interface, see chapter 8 for a list and description of all these available functionalities.

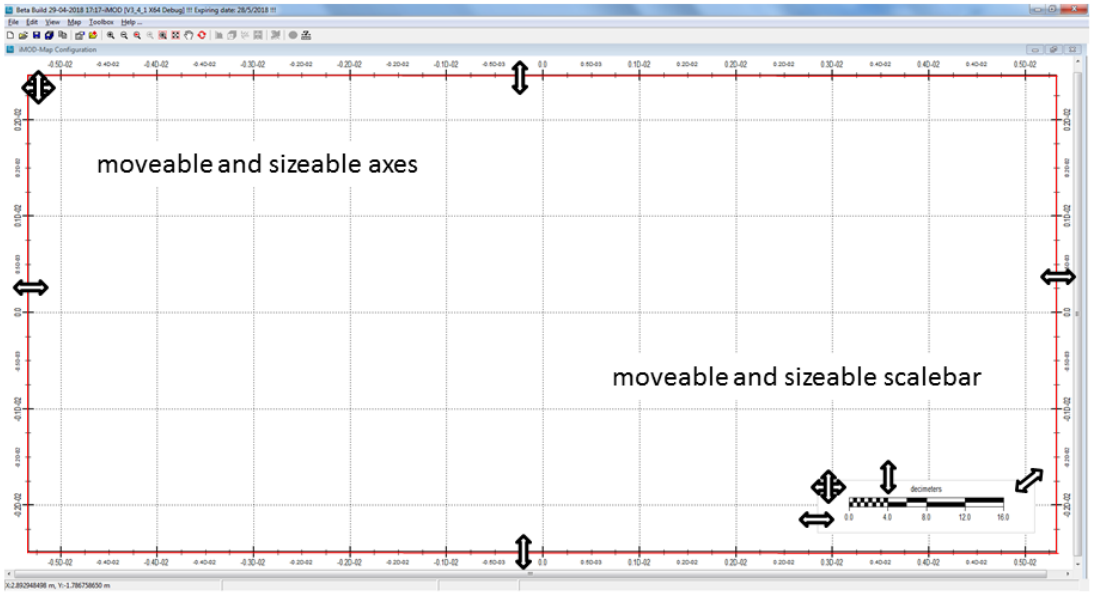
In the examples above the iMOD executable iMOD_V4_3_X64R.exe was used; alternatively the 32-bit version of the iMOD executable can invoked, see section 2.2.

2.6 Main Window

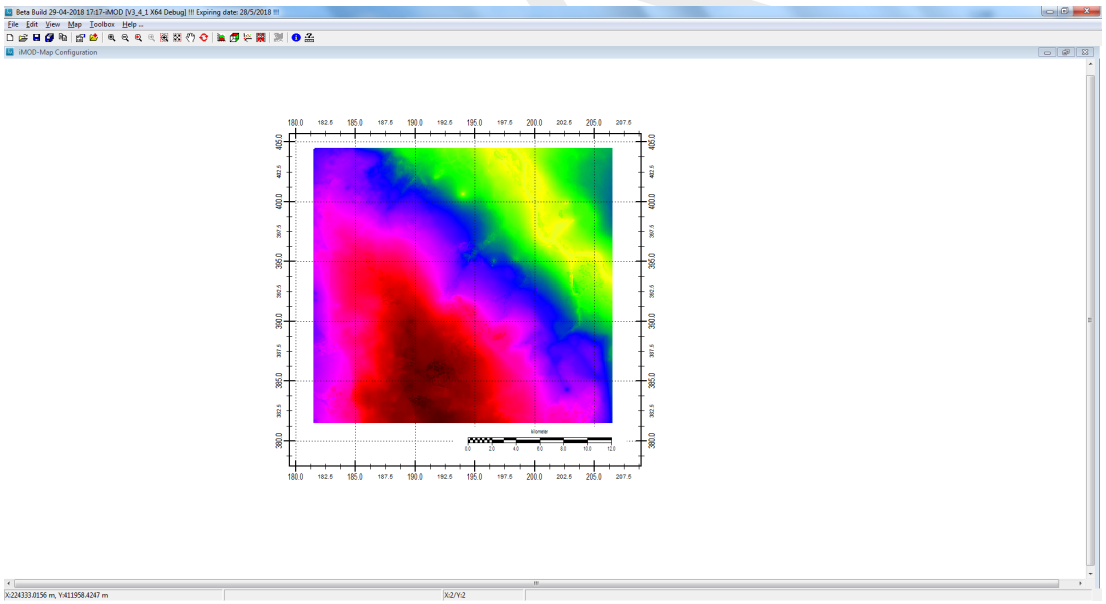
When iMOD is started, the *iMOD Main* window is displayed:



Important to notice that the graphical canvas can be modified, the location of the axes can be changed, as well as the position and size of the scale bar. Whenever the position of the mouse is near the current axes, the axes highlights in red and the cursor changes. At that moment, selecting the left-mouse button, it is possible to drag the axes. This is similar to the resizing and positioning of the scale bar.



The settings of the modified axes and scale bar are saved in the IMF file. Reopening an IMF file will show the modified axes and scale bar. The advantage of repositioning the axes is the ability to closely match the size of the current visible map, sush as a square IDF file.

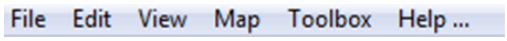


Furthermore, this window contains a menu bar, an icon bar and information displayed on the window status bar.

2.6.1 Menu Bar

To access the iMOD menus, click the menu names on the menu bar, or alternatively use Alt+<*first letter of the menu name*>.

Menu bar:



File	Standard Windows options for saving and opening iMOD MetaFile (*.IMF), export the content of the graphical area.
Edit	This contains a limited set of features to create iMOD Files, such as IDFs out of IPF's, IFF's and GEN-files.
View	This contains functionalities to copy the content of the graphical area onto the *Clipboard* of Windows and a variety of manners to display data.
Map	This menu option offers the ability to open iMOD maps and configure their appearance.
Toolbox	A variety of tools are available, e.g. *Cross-Section Tool*, *WaterbalancingTool*, *ModelingTool*, and more, but also an *ImportTool* for MODFLOW and SOBEK model configurations.
Help . . .	Starts the Help-file (if available in the selected *.PRF file).

Detailed descriptions of these menu options can be found in the Reference section.

2.6.2 Icon Bar

Use the buttons on the *iMOD Icon* bar to quickly access frequently used functions.

Icon Bar:

	*New:* Start a new iMOD Project (*.IMF-file)
	*Open* Open an existing iMOD Project (*.IMF-file)
	*Save* Save the current configurations (maps) in the last saved *.IMF file
	*SaveAs* Save the current configurations (maps) in a new *.IMF file
	*Copy* Click this icon to copy the entire content of the graphical area onto the *Clipboard* of Windows.
	*iMOD-Manager* Click this icon (checkbox) to start or hide (if shown) the *iMOD-Manager* window.
	*OpenMap* Click this icon to open an existing iMOD Map, such as *.IDF, *.IPF, *.ISG, *.IFF, *.GEN, *.NC, *.MAP
	*ZoomIn* Click this icon to zoom IN on the centre of the current graphical dimensions.
	*ZoomOut* Click this icon to zoom OUT on the centre of the current graphical dimensions.

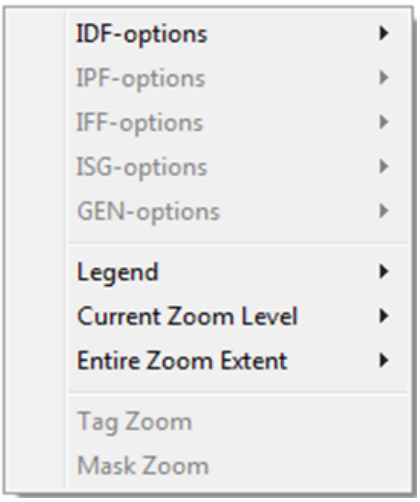
	*Go Back to Previous Extent* Click this icon and the map will return to the previous map extent and view. This view becomes the last view automatically whenever any other zoom button will be used.
	*Go to Next Extent* Click this icon and the map will go to the next extent viewed after the current view. This option becomes available whenever the *Zoom to Previous Extent* button has been selected priorly.
	*ZoomRectangle* Click this icon to zoom in for a rectangle to be drawn. Use your left-mouse button to determine the lower-left corner of the rectangle, click again for the upper-right corner (or vice-versa).
	*ZoomFull* Click this icon to zoom in on the entire extent of the selected maps on the tab *Maps* on the *iMOD Manager* or on the selected overlay Maps in the tab *Overlay* on the *iMOD Manager*.
	*Move* Click this icon to move the current display. Click the left-mouse button on that location where you want to move from, repeat this after the display has been refreshed (automatically). Use the right mouse button to stop the moving process.
	*Cross-Section Tool* Click this icon to start the *Cross-Section Tool* for all the maps selected on the tab *Maps* from the *iMOD Manager* Window.
	*3DTool* Click this icon to start the *3DTool* for all the maps selected on the tab *Maps* from the *iMOD Manager* Window and those selected on the tab *Overlays* from the *iMOD Manager*.
	*TimeSerie Tool* Click this icon to start the *TimeSerie Tool* for all the IDFs (timevariant) and IPFs (with associated files assigned to them) selected on the tab *Maps* from the *iMOD Manager* Window.
	*Topographical Overlay* Click this icon to display the default topographical overlay as defined by the KeyWord TOP25 in the selected *.PRF-file or display the overlays (*.BMP, *.png) as defined by the menu option *Add Background Image*.
	*MapInfo* Click this icon to start the *MapInfo* window to analyse the dimensions of IDFs, IPFs, IFFs, and GENs. For IDFs additional statistics and meta-information can be viewed too.
	*DistanceTool* Click this icon to start the distance tool where you can specify the location where to measure from, by clicking your left-mouse button. Intermediate points can be added by clicking your left-mouse button repeatedly. To stop the process, click your right-mouse button.

Detailed descriptions of these menu options can be found in the Reference section.

2.6.3 Popup Menu

Right-click anywhere in the canvas of the graphical window to open the popup menu. This menu presents several options. The options might be unavailable because no correct file(s) are selected in the *iMOD Manager*.

Popup menu:



◇ **IDF-options**

- *IDF Analyse*, Click this option to get an overview of the analyse options (see section 6.7.1).
  - *Analyse . . .*, Click this option to start *Map Value*.
  - *Plot No Locations*, Check this item whenever no rastercells of the selected IDF-file need to be displayed.
  - *Plot All locations*, Check this item whenever the rastercells of all selected IDF-files need to be displayed. Bear in mind that the performance will slow down whenever many IDF-files are included, and if IDF-files with non-equidistant rasters are included. Whenever this option is checked, all values in Map Value will be coloured differently.
  - *Plot First Location Only*, Check this item whenever the raster cells of the first IDF-file listed in the *Map Value* table, need to be displayed. This is the default.
  - *Points*, Check this item whenever the values for the current location of the mouse need to be listed. This is the default.
  - *Rectangle*, Check this item whenever the values need to be summed within a rectangle that you can draw. Use the left mouse button to locate the first position of the rectangle and the left/right mouse button to stop and close the *Map Value* window.
  - *Polygon*, Check this item whenever the values need to be summed within a polygon that you can draw. Use the left mouse button to locate the first position of the polygon and continue to add more points (as desired) to complete the polygon. Use the right mouse button to stop and close the Map Value window.
  - *Circle (not available in current release)*, Check this item whenever the values need to be summed within a circle that you can draw. Use the left mouse button to locate the first position of the circle and expand the size of the circle while moving the mouse pointer away from the first position (center of the circle). Click again on your left mouse button to stop and close the Map Value window.
- *IDF Calculate,* Click this option to start the *Map Calculator* (see section 6.7.3).
- *IDF Edit,* Click this option to start *Map Edit* (see section 6.7.4).

- *IDF Group,* Click this option to group selected IDF-files (see section 6.5).
- *IDF Ungroup,* Click this option to ungroup selected MDF-file (see section 6.5).
- *IDF Export,* Click this option to export the selected IDF-files (see section 6.7.2) to:
  - *ESRII ASCII Format*
  - *NetCDF Format*

◇ **IPF-options**

- *IPF Analyse*, Click this option to start*IPF Analyse* (see section 6.8.3).
- *IPF Extract,* Click this option to start *IPF Extract* (see section 6.8.4).
- *IPF Configure,* Click this option to start *IPF Configure* (see section 6.8.1).

◇ **IFF-options**

- *IFF Configure,* Click this option to start *IFF Configure* (see section 6.9.1).

◇ **ISG-options**

- *ISG Configure,* Click this option to start *ISG Configure* (see section 6.10.1).
- *ISG Edit*, Click this option to start*ISG Edit* (see section 6.10.3).
- *ISG Show,* Click this option to define the shown attributes (see section 6.10.2):
  - *Nodes*, Click this option to display nodes of ISG-segments;
  - *Segments Nodes*, Click this option to display begin- and end-nodes of the ISG-segments;
  - *Cross-sections*, Click this option to display the position of cross-sections on ISG-segments;
  - *Calculation nodes*, Click this option to display the location of calculation nodes of ISG-segments;
  - *Structures*, Click this option to display the location of structures of ISG-segments;
  - *QH-relationships*, Click this option to display the location of QH-relations on ISG-segments.

◇ **GEN-options**

- *GEN Configure,* Click this option to start *GEN Configure* (see section 6.11.2).
- *GEN Extract,* Click this option to start *GEN Extract* (function not implemented).

◇ **Legend**

- *Plot Legend on Map*, Click this option to display the legend on the graphical window (see section 6.6.4)
- *Legend Columns*
  1. Click this option to display the legend in a single column;
  2. Click this option to display the legend in two columns;
  3. Click this option to display the legend in three columns;
  4. Click this option to display the legend in four columns;
  5. Click this option to display the legend in five columns;
- *Adjust Legend*, Click this option to open a window to adjust the legend (see section 6.6.1)
- *Synchronize Legend*, Click this option to synchronize legends, see section 6.6.3.

◇ **Current Zoom Level**
Click this option to create a legend based on the values for the current zoom level (see section 6.6.2):

- *Percentiles*, Click this option to create a legend with non-linear values,
- *Linear*, Click this option to create a legend with linear values,
- *Unique Values*, Click this option to create a legend with unique values,

◇ **Entire Zoom Extent**
Click this option to create a legend based on the values of the current zoom level (see section 6.6.2):

- *Percentiles*, Click this option to create a legend with non-linear values,
- *Linear*, Click this option to create a legend with linear values,
- *Unique Values*, Click this option to create a legend with unique values,

◇ **Tag Zoom**
Click this option to zoom to the selected Tags (Comments)

◇ **Mask Zoom**
Click this option to zoom onto the last *.MSK file loaded.

2.6.4 Window Status Bar

The status bar of the iMOD main window contains several elements to be considered.

Window Status bar:

X: . . . . m *Y: . . . .. m*	*Current Mouse Coordinates* The coordinates of cursor in the map are displayed in the lower left of the window. The units for the values on the X and Y axis are given in the coordinates of the system in which one is currently working. In the case of Dutch data, this is the National Triangulation System. In principle it is possible to read files in another system (if the system is projected, such as the National Triangulation System, for flat surfaces).
X:6/Y:6	*RasterDisplayResolution* This element on the status bar shows the accuracy of the displayed IDF-file. Since these IDF-files can be enormous in size, iMOD will decline the number of data read as the zoom level is increased. In such a manner, iMOD can display these enormous IDFs raster files quickly. For IDF-files that have non-equidistant cellsizes, iMOD needs to read all data. Consequently, these type of IDFs will be slower in presentation than the equidistant ones. The accuracy can be altered by selecting the menu option *View* and then the option *Accuracy*.
mv_25.IDF	*Current IDF* This element on the status bar shows the current IDF at the current mouse position.
	*WindowExtent* Click on your left-mouse button whenever it is positioned on this lower-right corner of the iMOD main window. By dragging your mouse (while the left-mouse button is pressed), the size of the iMOD main window can increase/decrease.

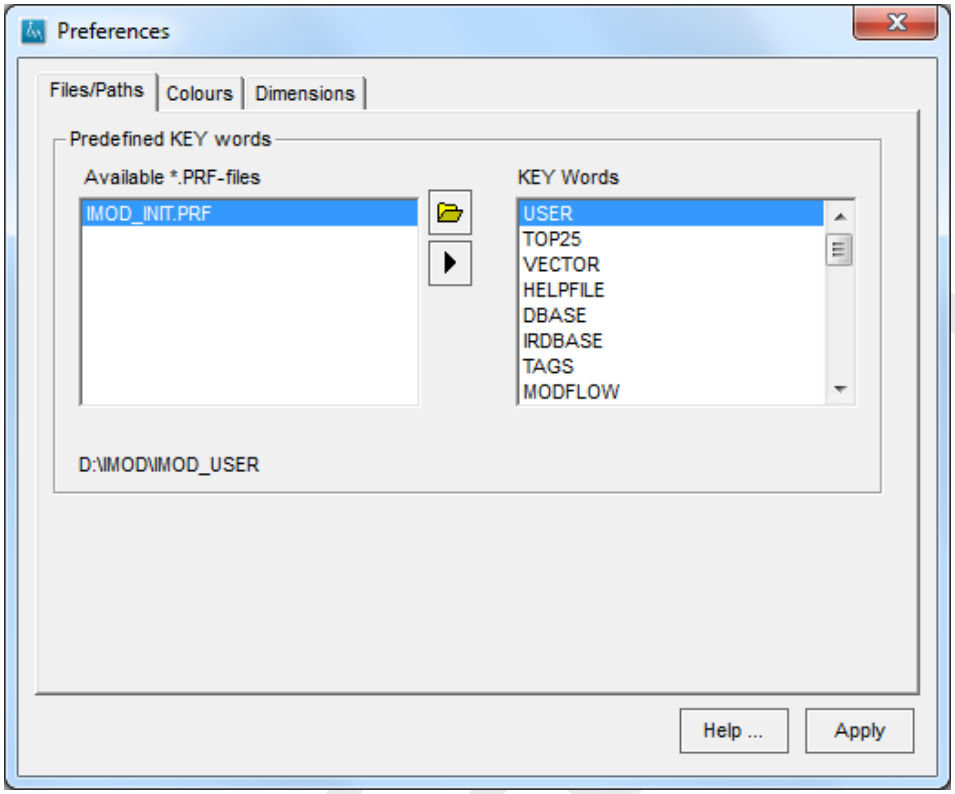
2.6.5 Title Panel

This panel situated at the top of the main window displays the iMOD version and the type of iMOD license. The top of the graphical window displays the name of the *.IMF used last to save the iMOD Project.

2.7 Preferences

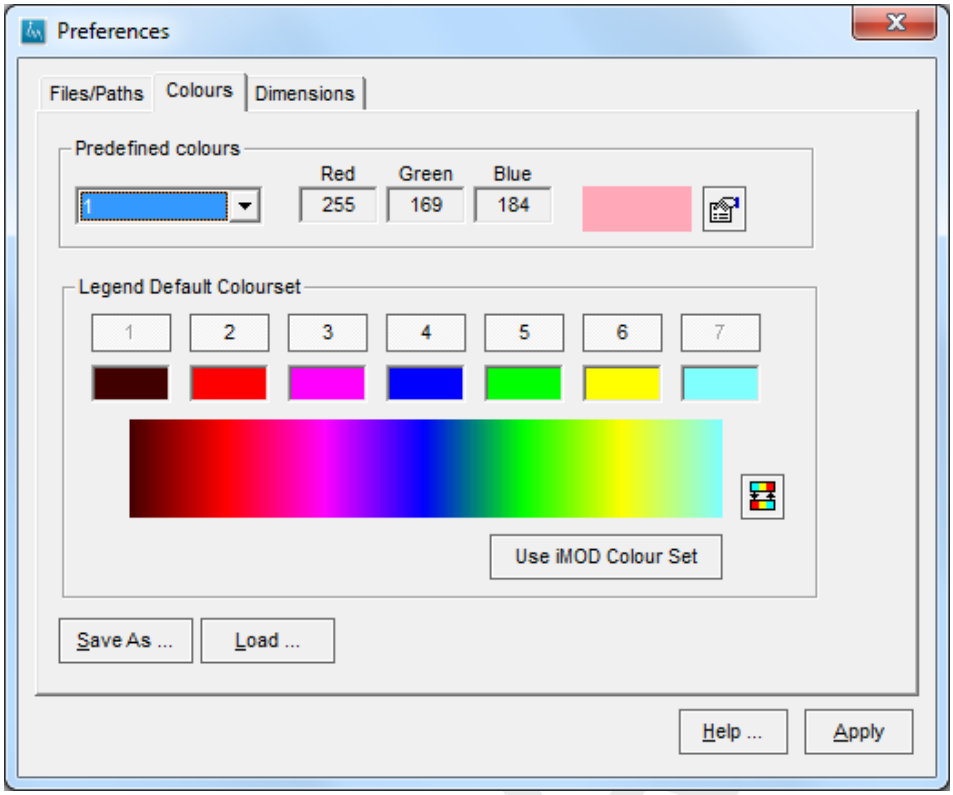
Several settings can be initiated to configure the current iMOD session. These can be defined in a *.PRF file (see section 9.1 for more information). On the menu bar, click the option *File* and then choose *Prefences* to open the corresponding *Preferences* Window.

Preferences window, Files/Paths tab:



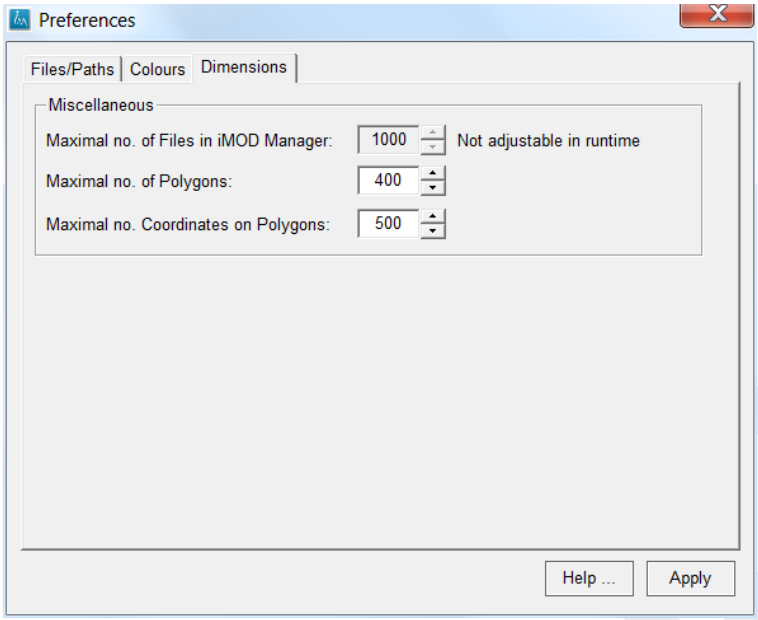
*Available *.PRF files*	Display of all available *.PRF files in the folder in which iMOD was executed. Select one of them to load an iMOD configuration. Different *.PRF file can be stored to switch between different iMOD configurations quickly.
Keywords	Select one of the keywords to inspect the value assigned to it underneath the tab. In this example above the keyword {user} has the value c:\users\peter\work\imodproject\user. To change any keyword, you should open the *.PRF in any third-party software, e.g. *Notepad*.
	*Open *.PRF-file* Click this button to search for a *.PRF-file on disk.
	*Use the selected *.PRF-file* Click this button to read the selected *.PRF-file and use its settings.
Close	Click this button to close the *Preferences* Window.
Help...	Click this button to start the iMOD Help Functionality.

Preferences window, Colours tab:



Predefined Colours	The dropdown menu presents the current colour number. iMOD supports 50 predefined colours to be used as default in a variety of iMOD functionalities, e.g. plotting of *Cross-Sections*, *TimeSeries*.
Red	Value of the red-component of the current default colour (0-255)
Green	Value of the green-component of the current default colour (0-255)
Blue	Value of the blue-component of the current default colour (0-255)
Legend Default Colourset	Change and save the preferred default colour-settings for the 7 basic legend colours in the *.CLR-file. Changing the colours works as similar as described in section 6.6.1.
	Colour Selection Click this button to open a default *Colour* window from Windows
Use iMOD Colour Set	Use the iMOD default legend colours, rainbow colouring: .
	Flip Colours. Click this option to “flip” the colour sequence, e.g. red becomes blue and blue becomes red.
Save As . . .	*Save a *.CLR-file* Click this option to save the current default (legend) colours in a given *.CLR file.
Load . . .	*Open *.CLR-file* Click this button to search for a *.CLR-file on disk. This type of file defines the default (legend) colours used by iMOD.

Preferences window, Dimensions tab:

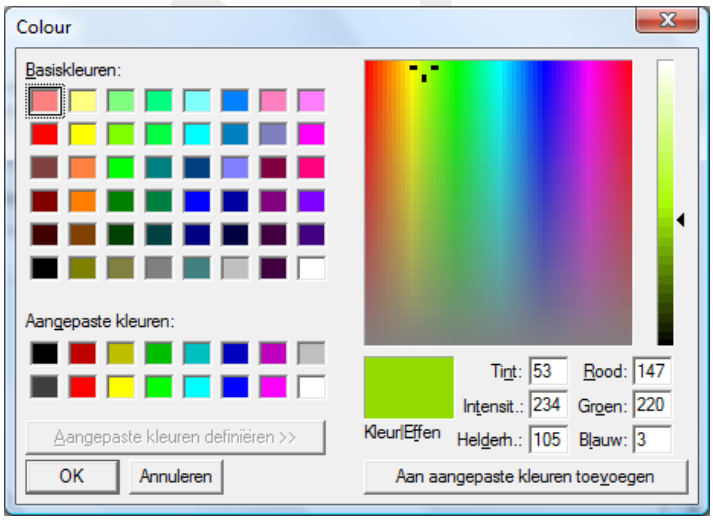


Maximal no. Files in the iMOD Manager:	Maximum number of maps to be loaded in the *iMOD Manager*. This value can not be altered, a change can be applied in the source-code.
Maximal no. of Polygons:	Maximum number of polygons in a single *.SHP and/or *.GEN file.
Maximal no. of Coordinates on Polygons:	Maximum number of coordinates upon each polygon within a *.SHP and/or *.GEN file.

2.8 Colour Picking

On various dialogs in iMOD, you can specify a colour. In all these cases the default *Colour* window is used, as shown below.

Colour window:



OK	Select this button to accept the selected/created colour. The *Colour* window will close.
Cancel	Select this button to cancel any change in colour. The *Colour* window will close.

2.9 Tips and Tricks

2.9.1 Keyboard shortcuts

Use the keyboard shortcuts to directly open a window without selecting the option from the menu bar.

Navigation	
Centre mouse button	Move Map (Pan).
Shift-right mouse button	Zoom Out.
Shift-left mouse button	Zoom In.
iMOD Manager	
Ctrl-M	Open *iMOD Manager* Window.
Ctrl-N	New iMOD Project.
Ctrl-O	Open iMOD Project.
Ctrl-S	Save iMOD Project.
Ctrl-P	Open *iMOD Project Manager* Window.
Ctrl-C	Copy current presentation to Windows *Clipboard.*
F1	Open iMOD Help-file (if available in the selected *.PRF file).
F2	Add Map to the *iMOD Manager.*
F3	Set map information (point, polygon, rectangle).

2.9.2 Exporting Figures

The content of the graphical window can be exported in PostScript (*.PS), Bitmap (*.BMP), ZSoft PC Paintbrush (*.PCX) , Portable Network Graphic Image (*.png) and WMF (Windows Meta Files) format. In the *File* menu, select the option *Export* and select the appropriate export type finally. These files can be later imported in a Word document, for example or added as annex in a report. The option *Copy to Clipboard* from the *View* menu can also be used to copy directly the display in a Word document.

2.9.3 Saving iMOD Projects

The content of the *iMOD Manager* can be save into a *.IMF file. Select the option *Save* or *Save As* from the *File* menu. On default, iMOD will save the content of the *iMOD Manager* each minute whenever the option *Autosave On (1 minute)* from the *File* menu is checked. This file will be called AUTOSAVE-IMOD.IMF and will be located in the directory {USER}\imffiles, where {USER} will be the directory assigned to the keyword USER in the used *.PRF file.

2.9.4 Copying part of a Table

It is possible to copy part of a table in another document, an Excel sheet for example.

If the cursor is placed on a cell of the table, select a specific area by using the dragging the mouse while the left-mouse button is pressed. Then, using the shortcut *Ctrl+C*, this area can be copied and pasted into any other (commercial) Windows oriented software.

3 File Menu options

This chapter contains a detailed description of the menu options for iMOD for general use. The examples in the tutorial section provide a convenient starting point for familiarization with the program.

Besides the familiar Windows options for opening and saving files, the *File* menu contains a number of options specific to iMOD:

- **Autosave On (1 minute)**
  On default, iMOD saves the current content of the iMOD Manager each minute. It yields an AUTOSAVE-IMOD.IMF that will be overwritten each time. The file is located in the directory {USER}\IMFILES, where the variable {USER} directs to the value of the keyword USER in the selected *.PRF file.
- **Print . . .**
  Prints the current content of the graphical window to an installed external printer. iMOD uses the default Windows Print Manager.
- **Export**
  The content of the graphical window can be exported to
  - PostScript (*.PS);
  - Bitmap (*.BMP);
  - ZSoft PC Paintbrush (*.PCX);
  - Portable Network Graphic Image (*.PNG);
  - JPEG/JFIF image (*.JPG; *.JPEG).

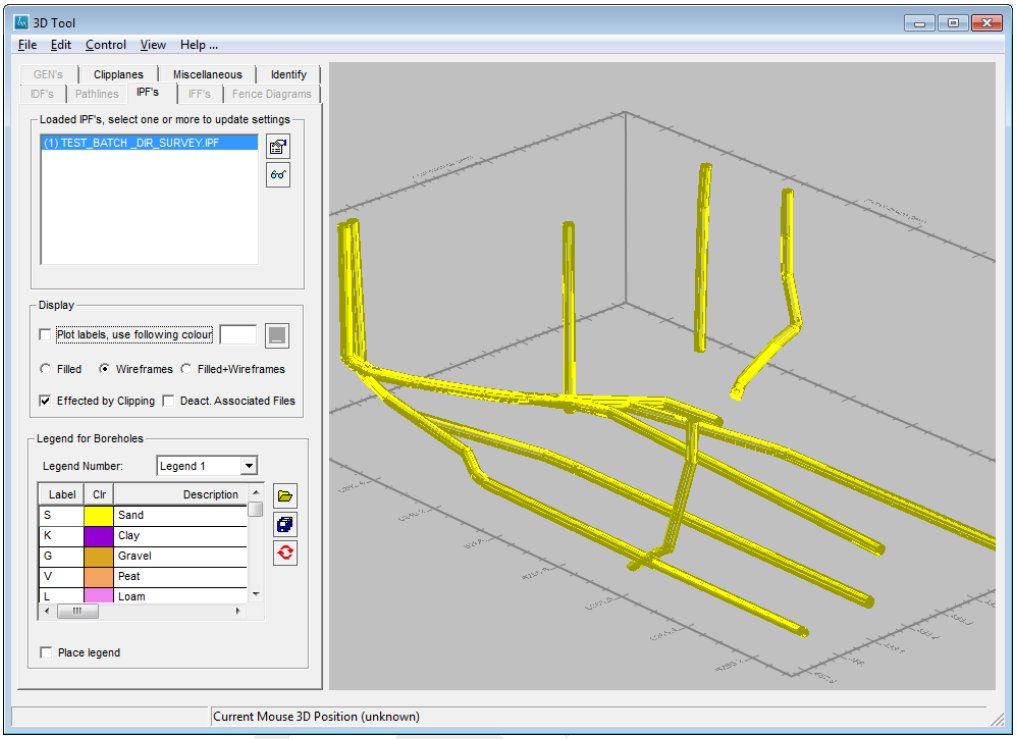
  In the *File* menu, select the option *Export* and select the appropriate export type finally. These files can be later imported in a Word document, for example or added as annex in a report.

◇ **Import . . .**

iMOD offers the limited ability to import a few formats from third-party software packages.

- Import Deviated Wells
  Select this option to import deviated wells into an IPF format (see section section 9.7.

  Example of a yielding deviated well in the 3D of iMOD:

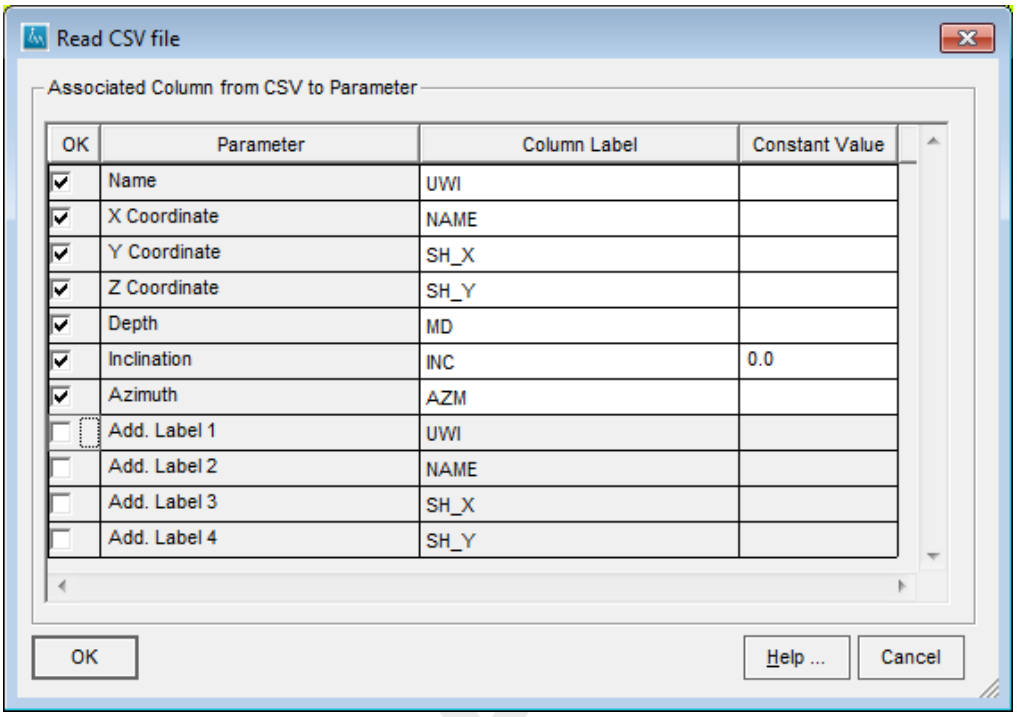


After selecting this item, a CSV file need to be selected from which the appropriate columns within the CSV need to be assigned, or alternatively a constant value can be entered. The following parameters are obliged:

- **Name**
  Select a column that represents the name of the well;
- **X Coordinate**
  Select a column from the drop down menu that represents x-coordinate of the well;
- **Y Coordinate**
  Select a column from the drop down menu that represents y-coordinate of the well;
- **Z Coordinate**
  Select a column from the drop down menu that represents z-coordinate of the well;
- **Depth**
  Select a column from the drop down menu that represents the depth of the well. This is the depth measured as net distance (meter) through the borehole;
- **Inclination**
  Select a column from the drop down menu that represents inclination of the well. The inclination is defined as the angle from the surface (xy-plane) downwards by a positive angle whereby 90.0 degrees is perpendicular downwards;
- **Azimuth**
  Select a column from the drop down menu that represents the azimuth of the well. The azimuth is defined as the angle with the z-axes measured clockwise with a zero angle pointing to the north and 90.0 degrees to the east;

- **Add. Label 1,2,3,4**
  Select a column from the drop down menu that represents an additional label of the well. Up to four additional labels can be assigned.

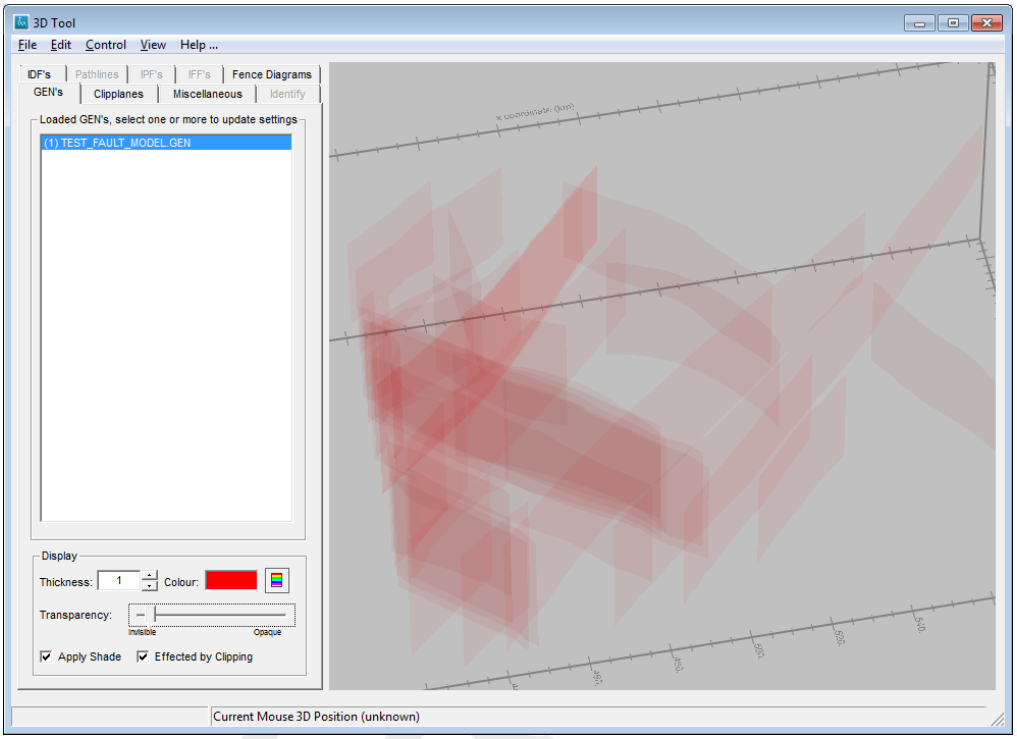
Example of the Read CSV File window:



There is also an iMOD Batch function **DEVWELLTOIPF** available, see section section 8.10.1.

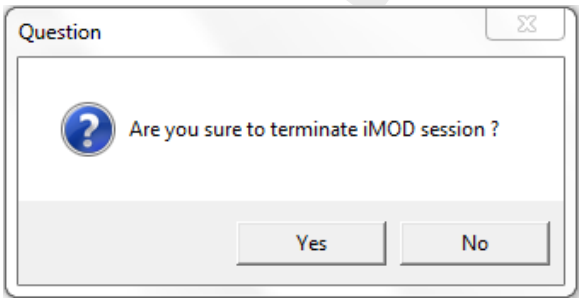
- Import Keypillars Petrel ASCII format
  Use this option to import a *Keypillars Petrel ASCII* format file. This file is exported from Petrel. It assumes a version 4 file format describing the x,y and z-coordinates of so-called Keypillars of faults. After selecting an ASC file, iMOD convert this to a 3D GEN (see section section 9.10). It can then be inspected and visualized in 2D and 3D.

  Example of a converted Keypillars Petrel ACII file into a 3D GEN and visualized in the 3D Tool of iMOD:



- ◇ **Preferences . . .**
  Click this option to open the *Preferences* Window.
- ◇ **Quit . . .**
  Click this option to quit iMOD. Before leaving iMOD you will be asked whether you are sure to leave iMOD, in that case you'll be offered to opportunity to save your work first before leaving iMOD.

  Question window:



4 Edit Menu options

The *Edit* menu contains the following options for the creation of:

- section 4.1: creating new IDF files.
- section 4.2: creating new IPF files.
- section 4.3: creating new GEN files.
- section 4.4: creating new ISG-files.
- section 4.5: creating new Polylines.
- section 4.6: creating new iMOD Batch files.

4.1 Create an IDF-file

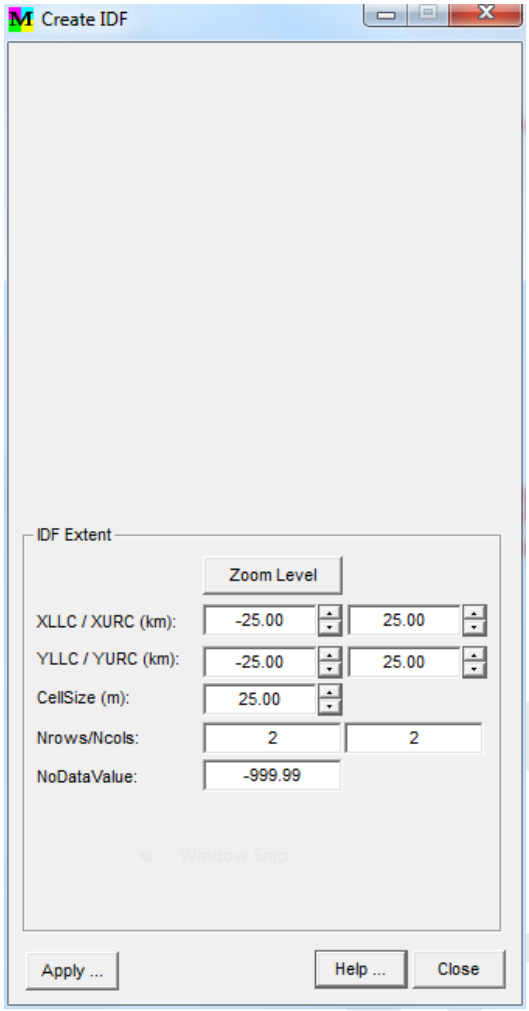
IDF-files can be created from scratch or by conversion from different formats. The available options are:

- **Scratch**
  Click this item to create a new IDF.
- **Points (*.ipf)**
  Click this item to create an IDF from point data stored in an IPF-file.
- **Polygons/Lines (*.gen; *.shp)**
  Click this item to create an IDF out of a (set of) polygon(s).
- **Flowlines (*.iff)**
  Click this item to create an IDF from line data stored in an IFF-file.

To create a new IDF select the main option *Edit*, choose *Create Feature*, then *IDFs from* and then one of the options shown above.

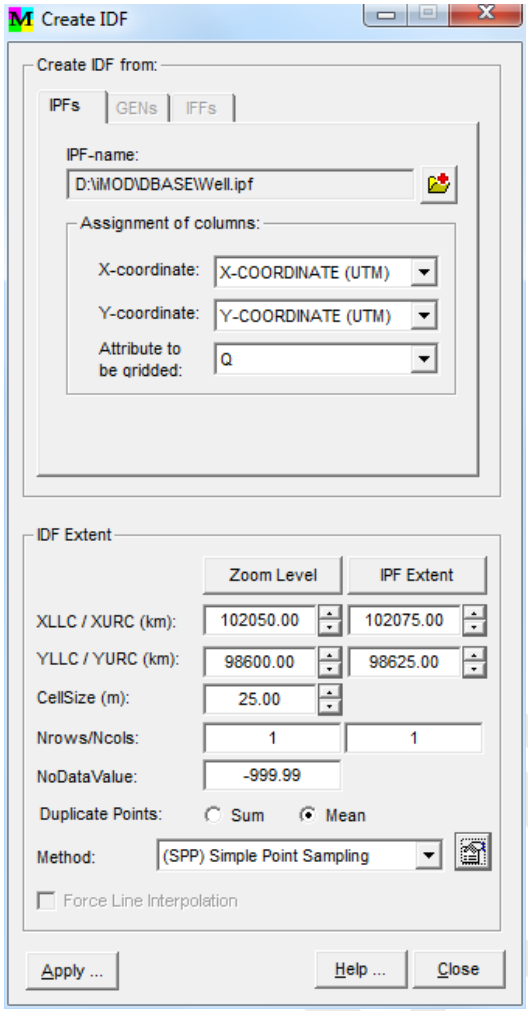
When creating a new IDF from scratch then the IDF is created with NoData values. When creating an IDF from the other formats then the IDF cells are assigned values derived from these files.

Option Scratch, Create IDF window:



Zoom Level	Click this button to adjust the IDF extent to the current zoom level in the graphical display.
XLLC / XURC (m) :	Enter the X coordinate for the lower-left-corner *(XLLC)* and upper-right-corner *(XURC)* of the IDF extent.
YLLC / YURC (m) :	Enter the Y coordinate for the lower-left-corner *(YLLC)* and upper-right-corner *(YURC)* of the IDF extent.
CellSize (m) :	Enter the cellsize of the IDF in meters. **NOTE:** The values for *XLLC, YLLC, XURC* and *YURC* will be trimmed automatically to the *CellSize* value.
Nrows/Ncols:	Displays the number of rows and the number of columns for the current IDF extent. These values are computed automatically and can not be changed directly.
NoDataValue:	Enter the *NoDataValue* for the IDF
Apply	Click this button to start the creation of the IDF
Close	Close the *Create IDF* window. The new IDF is added to the *iMOD Manager* window.

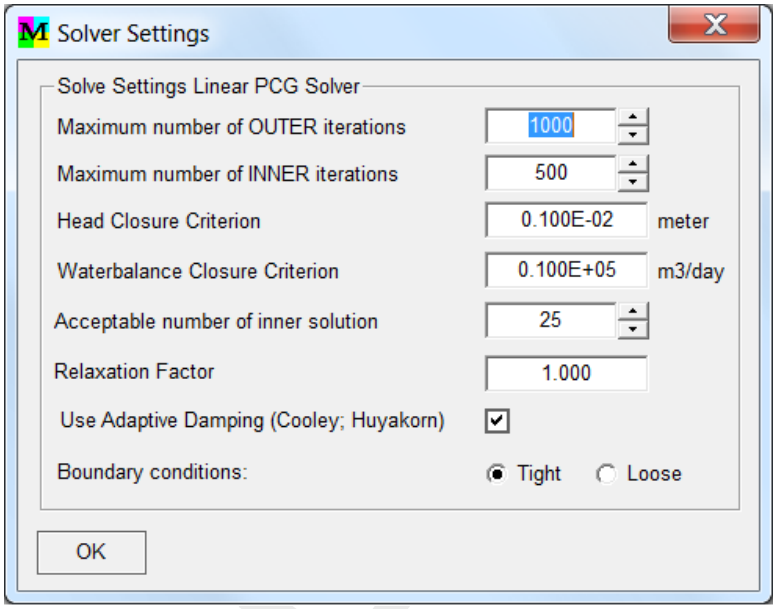
Create IDF window, IPFs tab:



	*Open Map* Click this button to open an IPF-file.
IPF-name:	Displays the name of the IPF-file.
X-coordinate:	Specify a column in the IPF-file that represents the X coordinate
Y-coordinate:	Specify a column in the IPF-file that represents the Y coordinate
Attribute to be gridded:	Specify a column in the IPF-file that represents the values to be gridded. Only numeric values can be gridded.
Zoom Level	*Zoom Level* Click this button to adjust the IDF extent to the current zoom level in the graphical display.
IPF Extent	*IPF Extent* Click this button to adjust the IDF extent to the entire extent of the selected IPF-file.
XLLC / XURC (m) :	Enter the X coordinate for the lower-left-corner *(XLLC)* and upper-right-corner *(XURC)* of the IDF extent.
YLLC / YURC (m) :	Enter the Y coordinate for the lower-left-corner *(YLLC)* and upper-right-corner *(YURC)* of the IDF extent.
CellSize (m) :	Enter the cellsize of the IDF in meters. **NOTE:** The values for *XLLC, YLLC, XURC* and *YURC* will be trimmed automatically to the *CellSize* value.

Nrows/Ncols:	Displays the number of rows and the number of columns for the current IDF extent. These values are computed automatically and can not be changed directly.
NoDataValue:	Enter the *NoDataValue* for the IDF.
Duplicate Points	Select one of the options for points with identical coordinates: Sum: use the sum of the values to be gridded Mean: use the mean of the values to be gridded.
Method:	Select one of the interpolation methods (see for batch creation of IDF's section 8.2.10): *(SPP) Simple Point Sampling:* Click this option to determine grid values on those points that are inside the current grid cell only. As a result, it might be that many grid cells get*NoDataValues.* *(BI) Bivariate Interpolation:* Click this option to determine grid values from a smooth interpolation function Z(x,y), which agrees with the given data (Hiroshi Akima, A Method of Bivariate Interpolation and Smooth Surface Fitting for Values Given at Irregularly Distributed Points, ACM Transactions on Mathematical Software, Volume 4, Number 2, June 1978). *PCG (Preconditioned Conjugate Gradient):* Click this option to apply the Preconditioned Conjugate Gradient method (this is the same as the solver used in MODFLOW) *VG (Variogram):* Click this option to create a semivariogram; this yields no interpolation of the data, it generates a table filled in with a variogram. The results will be written in the VARIOGRAM.TXT file. *(SKI) Simple Kriging Interpolation:* Click this option to apply a Kriging interpolation assuming a constant mean over the entire domain. *(OKI) Ordinary Kriging Interpolation:* Click this option to apply a Kriging interpolation assuming a constant mean in the neighborhood of each estimation point.
	*Open settings window* This function is active for the interpolation methods PCG, SKI and OKI.

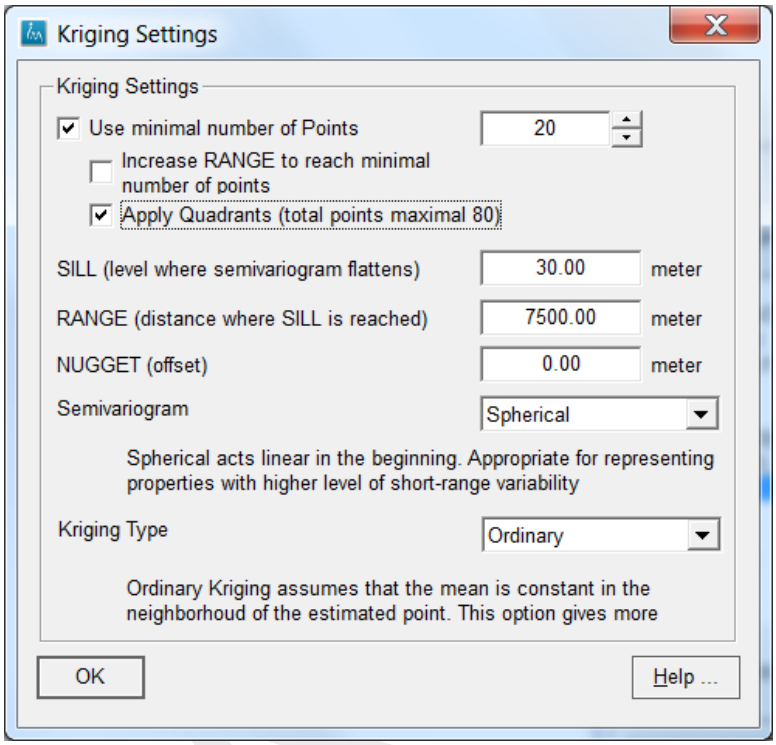
Solver Settings window for PCG interpolation:



Outer Iterations	Specify the maximum number of outer iterations used by the PCG solver;
Inner Iterations	Specify the maximum number of inner iterations used by the PCG solver. The more inner iterations used for a linear problem, the faster a PCG solution will be achieved;
Head Closure Criterion	Specify the closure criterion (e.g. Heads) for the problem to be solved. This value related to the units of the problem to be solved, choose a value at least two order of magnitude less than the desired accuracy;
Waterbalance Closure Criterion	Specify the closure criterion for the water balance for the problem to be solved, e.g. the lumped error of accuracy in the head. This value related to the units of the problem to be solved, choose a high value whenever the usage of the *Head Closure Criterion* is sufficient;
No. Inner Solutions	Specify an acceptable value, e.g. 25, whenever the problem to be solved shows high non-linearities that avoid any convergence of the solver. Solving a Solid might introduce these non-linearities that can be tackled in this manner;
Relaxation Factor	This factor *damps* the subsequent solutions of the solver. Use a high value (1.0) for linear problems and a lower value for non-linear problems. Use the *Use Adaptive Damping* option for non-linear problems instead;
Adaptive Damping	Apply this for non-linear problems as it will adapt the *Relaxation Factor* during the iteration process to yield a more robust solution;
Boundary Conditions	Select the *Tight* option to fixate the known location during the solution, use *Loose* instead to use a different approach in which the known areas are simulated by a boundary condition that allows more change on the known areas;
OK	Select this button to agree with the entered values.

The PCG solver is available in an iMOD Batch functionality as well, see for more information section 8.2.10. **NOTE:** Consult scientific literature regarding PCG solver settings as described above.

Kriging Settings window for Simple and Ordinary Kriging interpolation:



Use minimal number of Points — Specify the minimum number of points (per quadrant).

Increase Range to reach minimal number of Points — Select this option to allow the search increase the Range whenever the number of points is less than the entered minimal number of points.

Apply Quadrant — Select this option to devide the number of points in quadrant to met the minimal number of points individually.

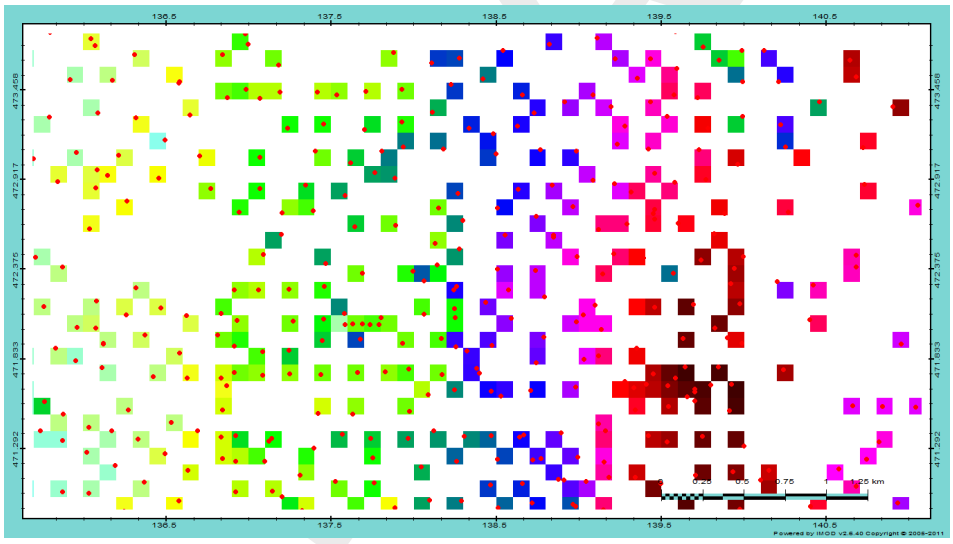
SILL — Specify the SILL value. The value that the semivariogram model attains at the range (the value on the y-axis) is called the sill. The partial sill is the SILL minus the NUGGET.

RANGE — Specify the RANGE value. When you look at the model of a semivariogram, you'll notice that at a certain distance, the model levels out. The distance where the model first flattens out is known as the range. Sample locations separated by distances closer than the range are spatially autocorrelated, whereas locations farther apart than the range are not.

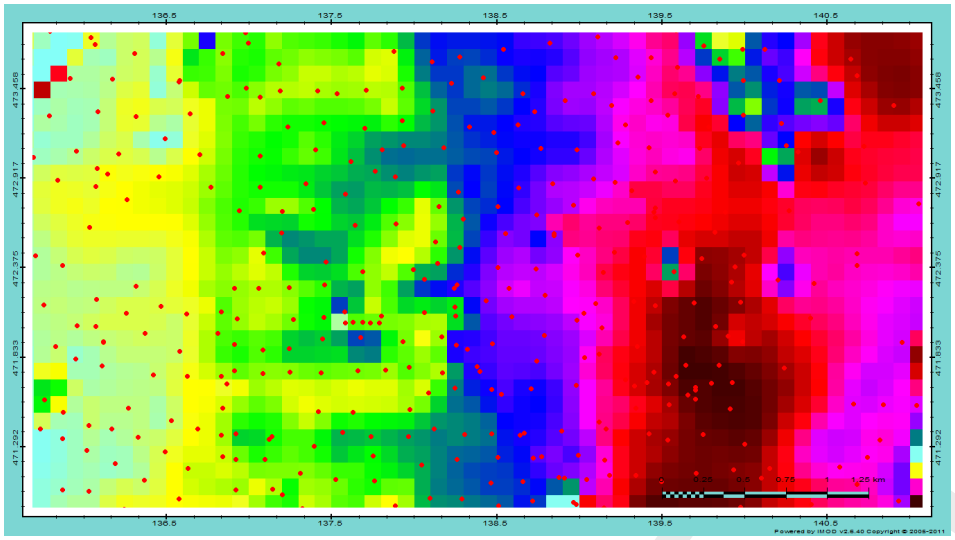
NUGGET — Specify the NUGGET value. The nugget effect can be attributed to measurement errors or spatial sources of variation at distances smaller than the sampling interval or both. Measurement error occurs because of the error inherent in measuring devices. Natural phenomena can vary spatially over a range of scales. Variation at microscales smaller than the sampling distances will appear as part of the nugget effect. Before collecting data, it is important to gain some understanding of the scales of spatial variation. Increased smoothness is applied whenever the NUGGET value is increased.

	Semivariogram Specify the type of Semivariogram, select from: ◇ **Linear Model:** $g(h) = c_0 + c_1 * \frac{h}{a}$ ◇ **Spherical Model:** $g(h) = c_0 + c_1 * 1.5\frac{h}{a} - 0.5(\frac{h}{a})^3$ ◇ **Exponential Model:** $g(h) = c_0 + c_1 * (1 - \exp(-3\frac{h}{a}))$ ◇ **Gaussian Model:** $g(h) = c_0 + c_1 * (1 - \exp(-3\frac{h^2}{a^2}))$ ◇ **Power Model:** $g(h) = c_0 + c_1 * h^{0.5}$ h represents the lag distance, $c_0 + c_1$ is the SILL value, c_0 is the NUGGET value and a is the range. Kriging is also available in an iMOD Batch functionality, see for more information section 8.2.10. **NOTE:** Consult scientific literature regarding Kriging Settings as described above.
Force Line Interpolation	Not active for IPFs.
Apply	Click this button to start the creation of the IDF.

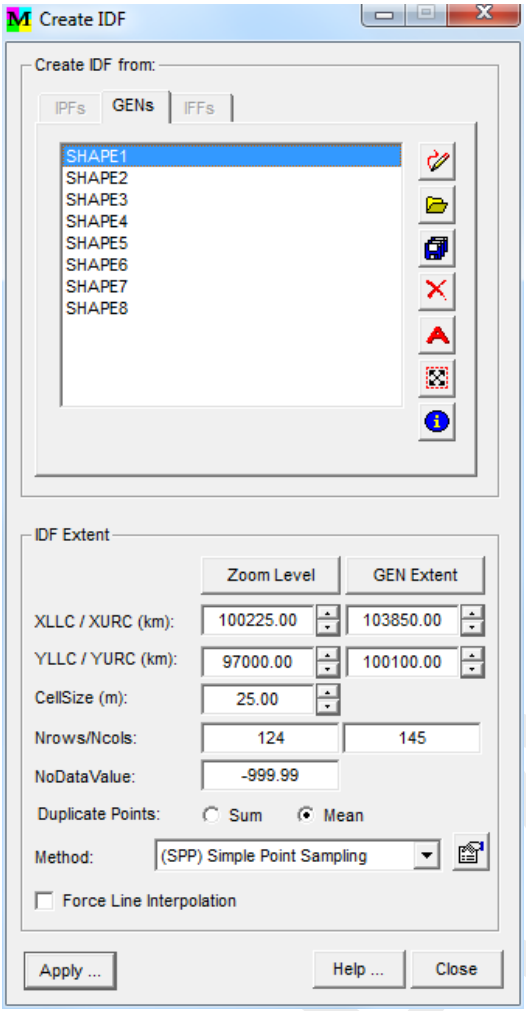
Example of a Simple Point Sampling interpolation:



Example of a Bivariate interpolation:



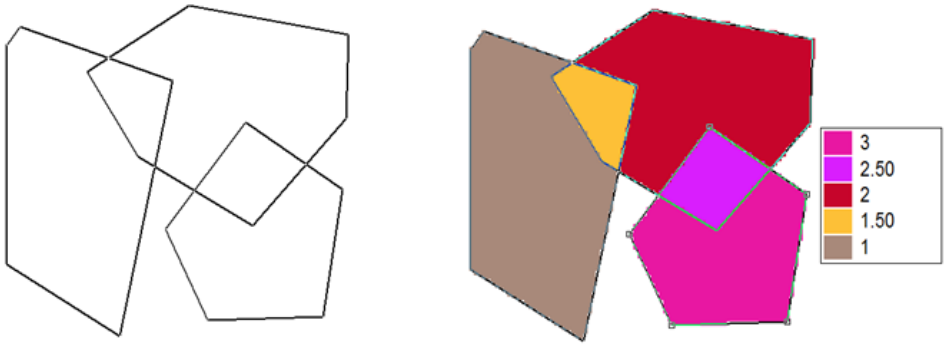
Create IDF window, GENs tab:



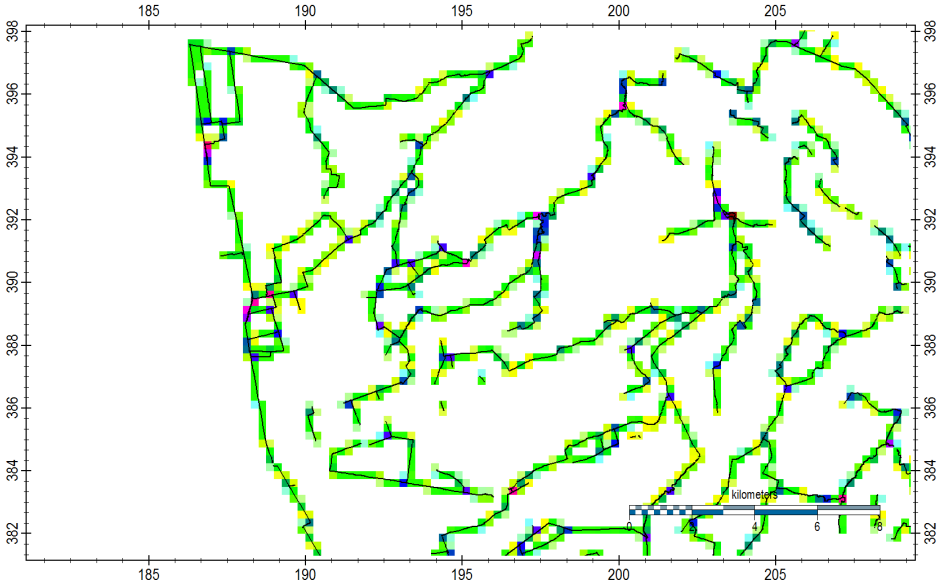
	Click these buttons to draw, open, save, delete or rename a shape. More detailed information can be found in Table 4.1.
IDF Extent	Specify the extent and dimensions of the IDF and the interpolation method. See the description for the IPFs tab for an explanation.
Force Line Interpolation	Create interpolated raster cells only along the lines as specified in the GEN
Apply	Click this button to start the creation of the IDF

Note: The value for a raster cell will be determined by the polygon number. The value of raster cells that are part of overlapping polygons will be equal to the mean value of the polygon numbers. NoDataValues are assigned to raster cells outside any polygon.

Example of a polygon GEN translated into an IDF-file:

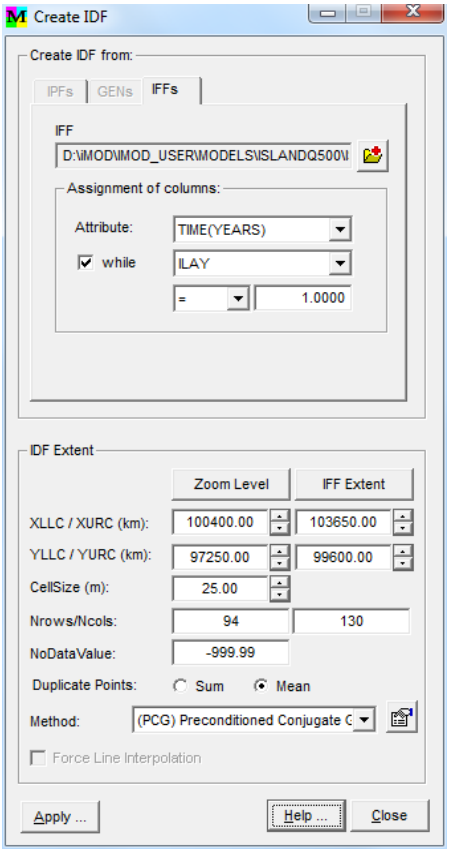


Example of a line GEN translated into an IDF-file:



The above example shows the rasterizing of lines into an IDF-file. The result is an IDF-file with the number of the different lines and another IDF showing the length of the line in each rastercell crossed by it.

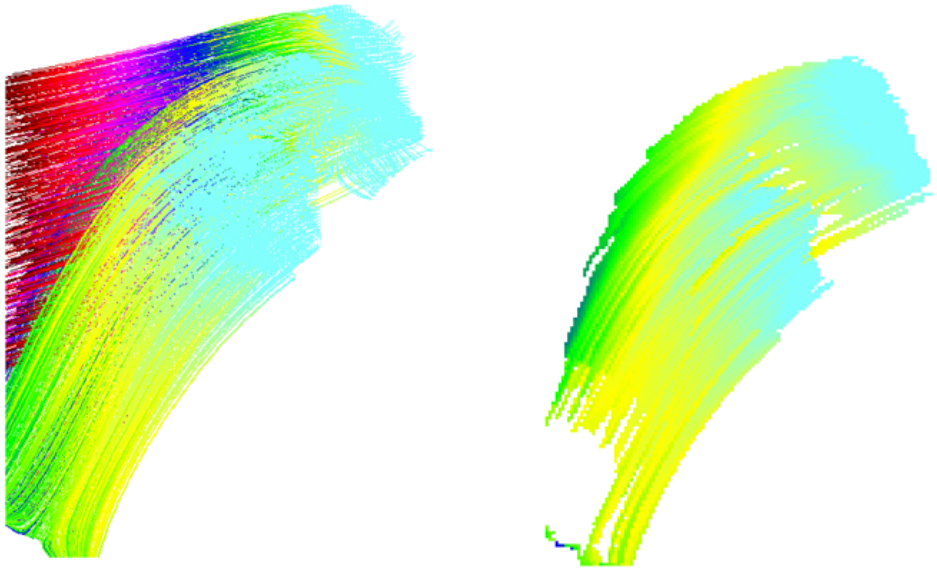
Create IDF window, IFFs tab:



	*Open Map* Click this button to open an IFF-file.
Attribute	Select one of the options to grid: PARTICLE_NUMBER – number of the particle ILAY – modellayer number XCRD – x coordinate YCRD – y coordinate ZCRD – z coordinate TIME(YEARS) – elapsed time in years VELOCITY – velocity (m/day)
while	Click this checkbox to use an extra logical expression. Choose one of the options (see under *Attribute*) and specify a logical operator (“=”;”<>”;”<”;”<=”,”>”;”>=”) and numeric value.
IDF Extent	Specify the extent and dimensions of the IDF and the interpolation method. See the description for the IPFs tab for an explanation.
Force Line Interpolation	Not active for IFFs
Apply	Click this button to start the creation of the IDF

Note: The value for a raster cell will be determined by the particle that passes through. Whenever more particles pass through the same rastercell, a mean value for the chosen attribute will be computed.

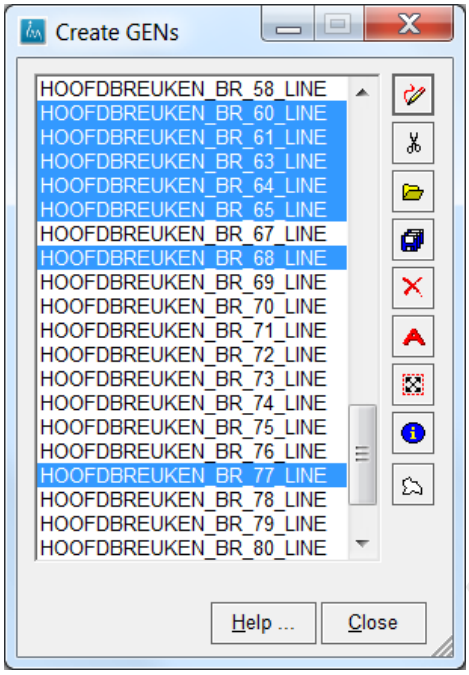
Example of a result of a particle simulation (left) gridded into a single IDF-file (right) for those parts that are within modellayer 3 only:



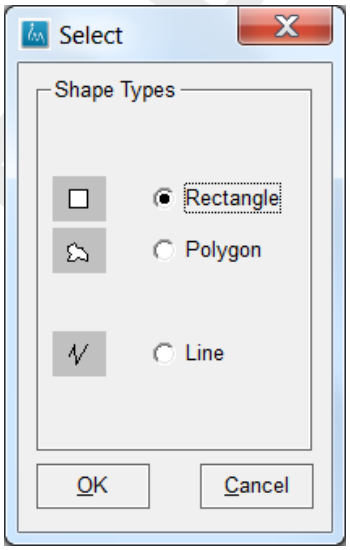
4.2 Create a GEN-file

Select the main option *Edit* and then choose the option *Create Feature* and then the option *GENs* to display the *Create GENs* window.

Create GENs window:

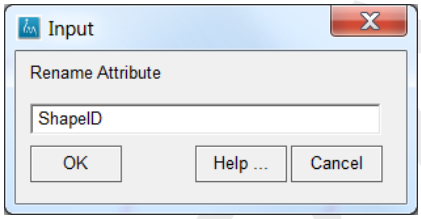


Draw Polygon
Click this button to start drawing a polygon on the canvas, see section 4.4 for more details on drawing polygons. First you need to select the type of shape you want to draw from the *Select* window:

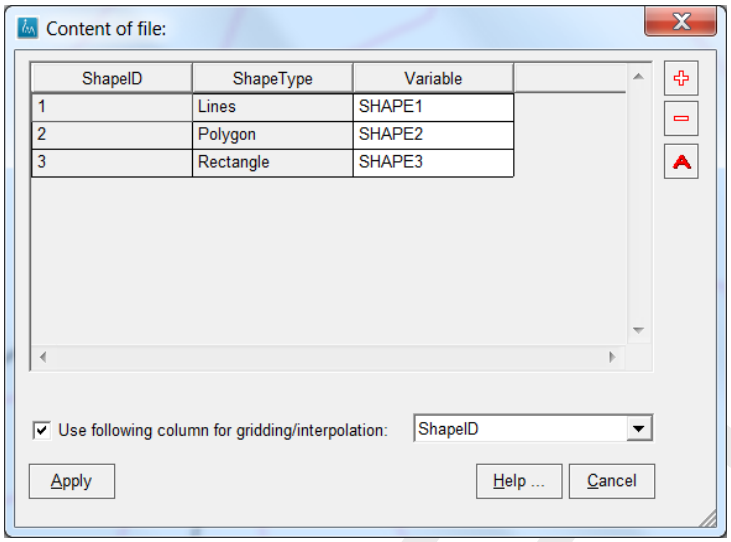


This window shows the available type of topology to supported by the GEN file format.

Rectangle
Click this option to define a rectangle.

	*Polygon* Click this option to define a polygon.
	*Line* Click this option to define a line.
	*Cut* Click this button to start drawing a line that intersects current lines/ polygons or rectangles to split the intersected segments accordingly into separate lines.
	*Open File* Click this button to open a *.GEN or *.SHP file. iMOD will create a name for each shape (polygon, line, point) inside the *.GEN and/or *.SHP file. If no name is specified for a shape (next to the ID identification), iMOD will create a name as follows: {name_of_the_gen}_{shapenumber}_{shapetype}, e.g. AREA_1_POLYGON or AERA_9_POINT.
	*Save File* Click this button to save the polygons to a *.GEN-file (see section 9.9). iMOD will save the names for the individual features in the *.GEN too. Those will be read whenever those files are read in iMOD again whenever the button *Open File* is selected.
	*Delete Polygon* Click this button to delete the selected polygons from the list. This action can not be undone, however, you will be asked first whether you are sure to delete the polygons.
	*Rename Shape* Click this button to rename the shape (line, polygon, rectangle), the *Input* window will appear:  You can enter a different name for the current selected polygon. Click the button *OK* to accept your entry, or click *Cancel* to leave the name unchanged. In both cases you will return to the *Create GENs* window again.
	*ZoomSelect* Click this button to adjust the zoomlevel to the selected shapes.

Information
Click this button to open the *Content of file* window:



This window shows the properties of the shapes which are used to create the GEN-file.

	Insert attribute Add an attribute to the shapes.
	Remove attribute Remove the selected attribute.
	Rename Change the name of the selected attribute, see the *Rename Shape* item previously described here.
Use following column for gridding /interpolation:	Check the box and select the attribute name from the pull down list in case interpolation is to be done for an attribute different from the SHAPEID.
Apply	Make the changes in the shapes.

Select in Polygon
Click this button to create a polygon for which all polygons, lines and/or poins will be selected that are within the polygon. Start drawing the polygon by clicking your left mouse button at the first point of the polygon, click left for each additional polygon point and stop drawing by clicking your right mouse button. After that iMOD will select all topologies within the polygon.

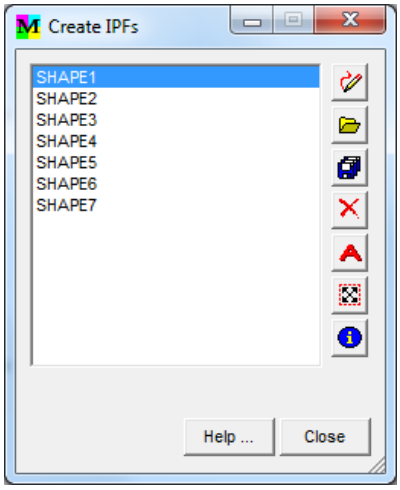
Help ...	Click this button to start the iMOD Help Functionality (if available in the selected *.PRF file).
Close ...	Close the *Create GENs* window, you will be asked to save your shapes first.

Note: The functionalities mentioned above appear throughout iMOD in different windows. The behavior for each of those is similar as explained above.

4.3 Create an IPF-file

Select the main option *Edit* and then choose the option *Create* and then the option *IPFs* to display the *Create IPFs* window.

Create IPFs window:



The options in this window are similar to the options described in the previous section on *Create GENs* window.

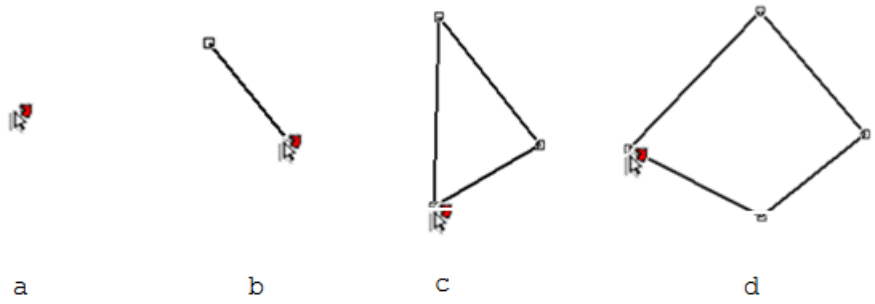
4.4 Create an ISG-file

Select the main option *Edit* and then choose the option *Create Feature* and then the option *ISGs* and then the option *RIV Applicable...* or *SFR Applicable...*. Thereafter you need to enter a name for the ISG to be created. Once a valid file name has been entered, iMOD will create the necessary files that relate to an ISG file, see section 9.9. After that, the *ISG Edit* window will start in which it is possible to add and/or modify the outline of the content of the ISG file, see section 6.10.3.

Note: The content of the files depends on the chosen applicability of the ISG-file. The SFR type of ISG has more attributes in the ISG than the RIV type of ISG file. The latter is used for the conventional RIV/DRN package, as the SFR type of ISG file is specially development to support the SFR package. Moreover, this SFR type of ISG file cannot be used in conjunction with a runfile

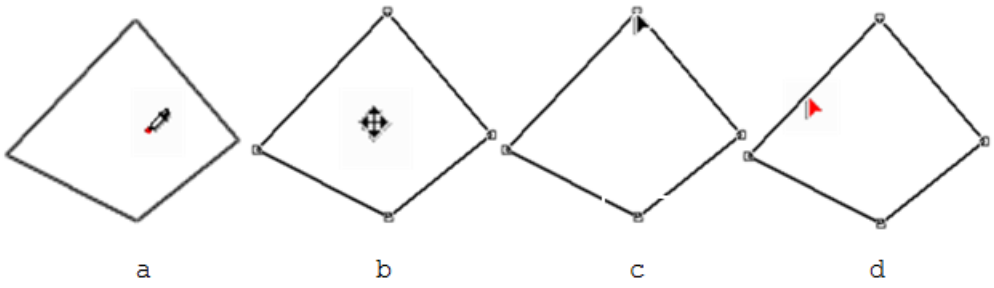
4.5 Drawing Polygons

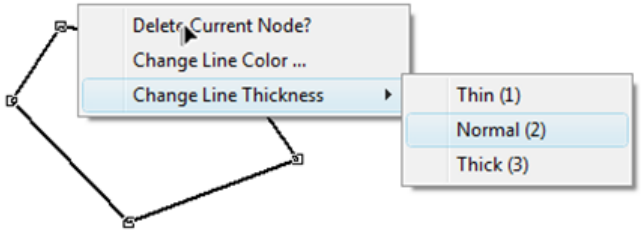
For several functionalities in iMOD you need to specify or draw polygons. For each of those, the methodology is similar and will be described here. After you click the *Draw Polygon* button you can add points of the polygon on the graphical window by clicking your left-mouse button sequentially.



	Click the left mouse button to place another point of the polygon. Click your right mouse button to stop

After the polygon has been drawn, the following options are available whenever you move the mouse in or near the polygon.



	This icon appears whenever you move the mouse inside a polygon. Then click the left mouse button to select the polygon. Once a polygon is selected the other options become available.
	Click the left mouse whenever this icon appears and drag the mouse over the graphical window to move the selected polygon.
	This icon appears whenever the mouse position is on one of the nodes of the polygon. Click the LEFT mouse button to move the selected node. Click the RIGHT mouse button to display the following menu options: 
	Delete Current Node? click this option to delete the current node. You can not undo this action.

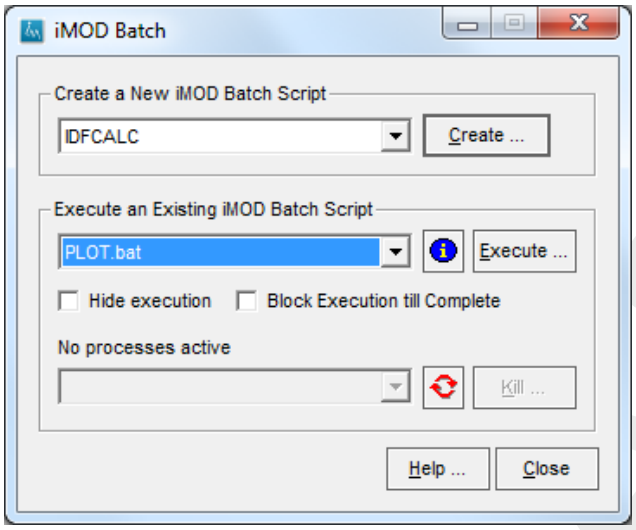
	Change Line Color . . .	click this option to change the colour of the polygon with the *Colour* window.
	Change Line Thickness	click this option to change the thickness of the line into *Thin (1)*, *Normal (2)* or *Thick (3)*.
	This icon appears whenever the mouse position is on a segment of the polygon. Click the left mouse and you can ADD a new node.	

4.6 Create an iMOD Batch file

The iMOD Batch functions include a variety of tools that can be used to execute iMOD data processes fast and repetitively. The Batch functions are described in detail in chapter 8. The Batch functions can be executed in command mode without starting iMOD. But these functions can be used also interactively from the iMOD main menu.

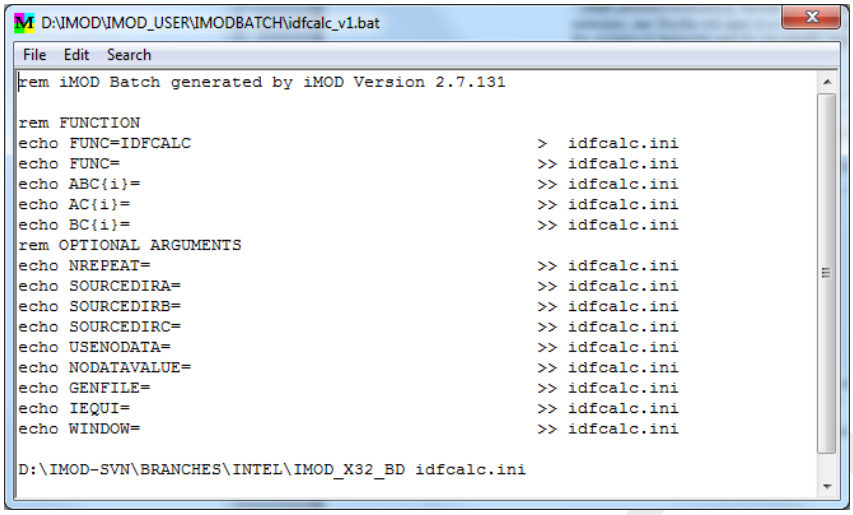
Select the main option *Edit* and then choose the option *iMOD Batch* to display the *iMOD Batch* window.

iMOD Batch window:



Create a New iMOD Batch Script	Select a Batch function from the dropdown menu.
Create	Click this button to create and save a batch file in the *.\USER\IMODBATCH* directory. The file automatically gets an extension *.bat*. The file will open in a text editor after it is saved. See below for an explanation of the file contents.
Execute an Existing iMOD Batch Script	Select a Batch file from the dropdown menu.
	Open a text editor with the Batch file contents.
Help ...	Click this button to start the iMOD Help Functionality (if available in the selected *.PRF file).
Execute	Click this button to start the execution of the Batch function selected in the dropdown menu.
Hide execution	Check this option whenever it is needed to start an iMOD Batch file in a hidden command window.
Block Execution till Complete	Check this option to block the execution of iMOD until the executed iMOD Batch file has terminated.
No processes active	This drop down menu lists the processes that are currently running.
	*Refresh* Select this button to refresh the drop down list of active processes.
Kill	Select this button to terminate the selected process from the drop down menu left.
Help ...	Click this button to start the iMOD Help Functionality (if available in the selected *.PRF file).
Close ...	Close the *iMOD Batch* window.

Example of an iMOD Batch file in a text editor window:



The file contains all keywords used for the specific batch function. The keyword contents can be added to complete the batch function. The command line to execute the batch file is at the bottom of the file.

5 View Menu options

This chapter describes the View Menu options, starting with on overview (section 5.1).

The following View Menu options are described in more detail:

- section 5.2: Goto XY.
- section 5.3: Add Background Image.
- section 5.4: iMOD Manager.
- section 5.4.1: iMOD Manager Properties.
- section 5.5: iMOD Project Manager.
- section 5.6: Subsurface Explorer.
- section 5.7: Lines and Symbols.

5.1 Overview of View Menu options

The *View* menu contains the following options:

- **Copy to Clipboard**
  Click this item to copy the content of the current graphical window to the Windows *Clipboard*. You can use the shortcut *Ctrl-C* instead.
- **Show Transparent IDFs**
  Check this item to draw IDF-files in a transparent mode. The used transparency is 50% and can not be altered.
- **Show Opaque IDF's**
  Check this item to draw all selected IDFs in opaque mode onto each other. This is helpful to plot IDFs with different dimensions onto each other, e.g. a smaller IDF on top of a larger one.
- **Apply NODATA Transparency**
  Check this item to draw those parts of IDF-files transparently that contain "missing" data (cell value is equal to the *NoDataValue* of the particular IDF).
- **Show IDF Features**
  - **IDF Raster Lines**
    Check this item to draw the line around each of the cells within an IDF.
  - **IDF Extent**
    Check this item to draw a single line around the boundaries of the IDF.
- **iMOD Manager . . .**
  Check this item to show the *iMOD Manager* window (section 5.4), this window will hold all active/loaded maps.
- **Project Manager**
  Check this item to display the *iMOD Project Manager* window; this window is able to read in a runfile and display its content in a tree view. From here the content can be ported to the *iMOD Manager* to quickly display model information.
- **Zoom Map**
  - **In**
    Click this item to zoom IN on the centre of the current graphical dimensions.
  - **Out**
    Click this item to zoom OUT on the centre of the current graphical dimensions.
  - **Rectangle**
    Click this item to zoom in for a rectangle to be drawn. Use the left-mouse button to determine the lower-left corner of the rectangle, click again for the upper-right corner (or vice-versa).
  - **Full Map**

Click this item to zoom in on the entire extent of the selected maps on the tab *Maps* on the *iMOD Manager* or on the selected overlay Maps in the tab *Overlay* on the *iMOD Manager*.

- **Mask Zoom: *.msk**
  Click this item to zoom to the zoom level in the last used or saved mask file.
- **Tag Zoom**
  This option is not available in the most recent iMOD version

◇ **Mask**

- **Save Mask . . .**
  Click this item to save the current zoom level to a *.MSK-file
- **Load Mask . . .**
  Click this item to load a *.MSK-file and zoom to the zoom level in that file.

◇ **Goto XY . . .**
Click this item to display the *Goto XY* window (see section 5.2) in which you can specify a location in coordinates or cell indices to zoom on to.

◇ **Graph**
This is used internally by iMOD and can not be manipulated.

◇ **Show Background Image**
Click this icon to display the default topographical overlay as defined by the KeyWord TOP25 in the selected *.PRF-file or display the overlays (*.BMP; *.png) as defined by the menu option *Add Background Image*.

◇ **Add Background Image . . .**
Click this item to specify bitmaps (*.BMP; *.png) to be used as background whenever the Show Background Image is selected, see section 5.3.

◇ **Transparent Background Image**
Click this item to display bitmaps that are used for background plotting in a transparent way.

◇ **Show Location in Google Earth**
Click this item to show the current zoom window within Google Earth. iMOD assumes that UTM-coordinates are used.

◇ **Accuracy**
This item computes the number of cells out of an IDF-file that are used to display a coloured image of the values within the IDF. The more cells are read, the more accurate the image will be displayed, however, the more time this will cost. iMOD computes the number of screen pixels necessary to display the image with the highest detail (i.e. the optimal detail). Thereafter, it depends on the choice of the user, how much of the optimal detail will remain:

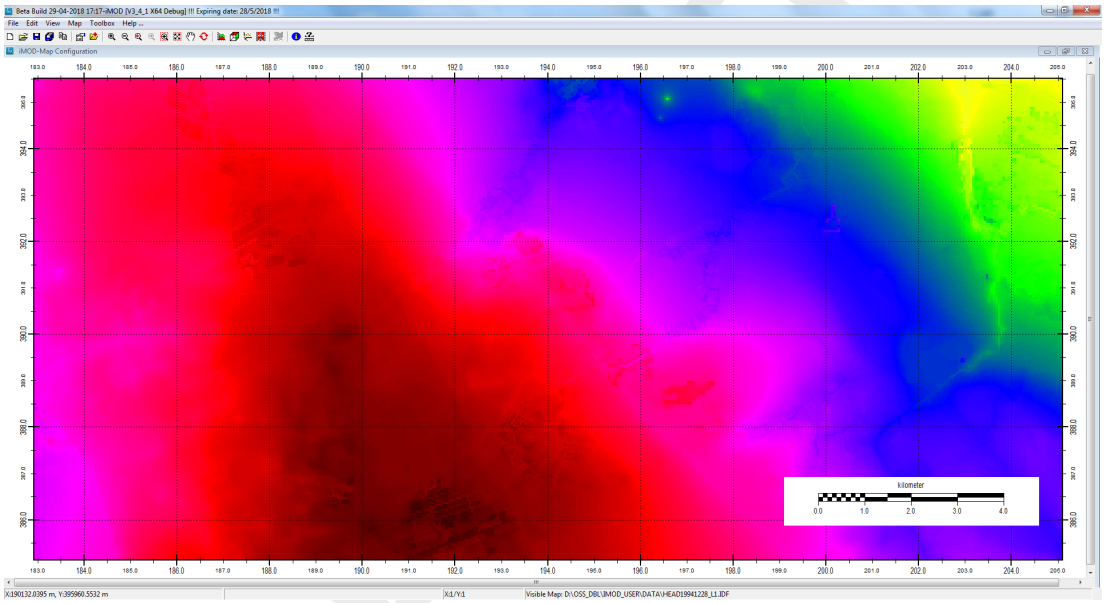
- **Low**
  Check this item to display IDF at a 10^{th} of the optimal detail.
- **Medium**
  Check this item to display IDF at a 5^{th} of the optimal detail.
- **High**
  Check this item to display IDF at a 3^{rd} of the optimal detail (default).
- **Excellent**
  Check this item to display IDF at full detail.

◇ **Layout**

- **Show Scalebar**
  Click this item to show a scalebar in the lower-right corner of the graphical window. The size and parts of the scalebar are determined automatically and will update whenever the zoomlevel changes. The location and size of the scale bar can be changed interactively via the mouse.

- **Show Axes**
  Check this option to show axes around the graphical window. The coordinates are trimmed to most logic values. The textsize, font and ticsizes can not be changed, though the location of the axes can be changed interactively via the mouse.
- **Show Rasterlines**
  Check this option to show raster lines around the graphical window from the major tic marks.
- **Show NorthArrow**
  Check this option to show a north arrow (or other image) that has been assigned to the keyword NORTHARROW in the selected *.PRF-file.

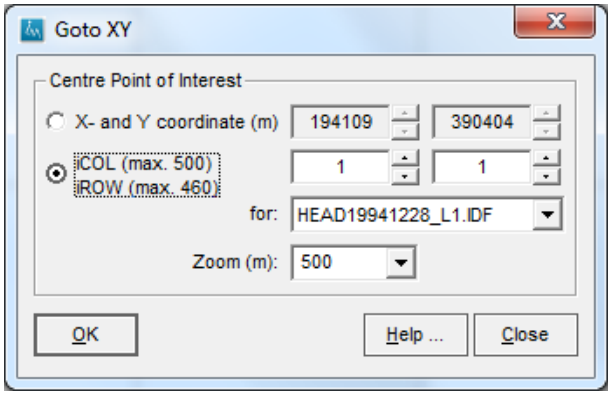
Example of usage of the options Show Axes and Show ScaleBar and pasted into this manual by the option Copy to Clipboard:



5.2 Goto XY

This functionality will offer the possibility to zoom on a centre point of interest. On the menubar click *View* and then choose the option *Goto XY* to open the corresponding window.

Goto XY window:

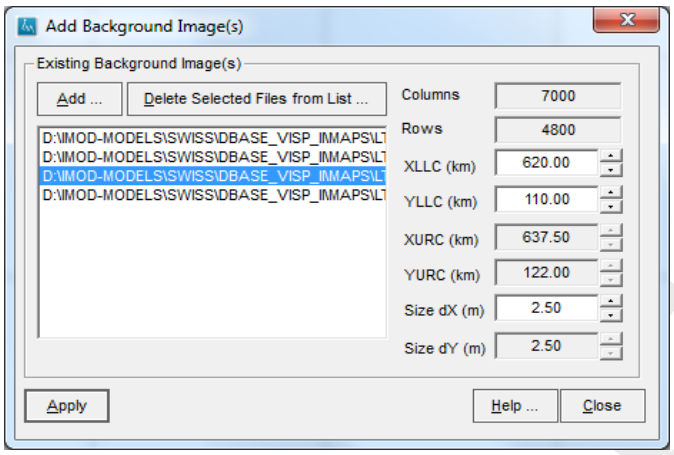


X- and Y coordinate (m)	Check this option to enter coordinates (X and Y) to zoom on.
iCOL (max. 5,800) *iROW (max. 6,680)*	Check this option to enter column and row indices to zoom for. In this case the limits for the column and row indices are 5,800 and 6,680, respectively. Those are based on the dimensions of the selected IDF.
for:	Select an IDF in the dropdown menu. You can select out of those listed in the *iMOD Manager*.
Zoom (m)	Select a range for the zoom level out of the dropdown menu. The final zoom level will be at the maximum amount of cells fitting in the y-direction in the map-window, e.g. Zoom level = 100 m, cell size = 25 m; at this zoom level a box of 4x4 cells will be completely shown in the middle of the map-window.
OK	Adjust the zoom level, closes the *Goto XY* Window and redraws the canvas
Help	Click this button to start the iMOD Help Functionality (if available in the selected *.PRF file).
Close	Leave the current zoom level unchanged and close the *Goto XY* Window.

5.3 Add Background Image

This functionality will offer the possibility to use your own background images instead of the one defined by the keyword TOP25 in the selected *.PRF-file. On the menubar click *View* and then choose the option *Add Background Image* to open the corresponding window.

Add Background Image window:



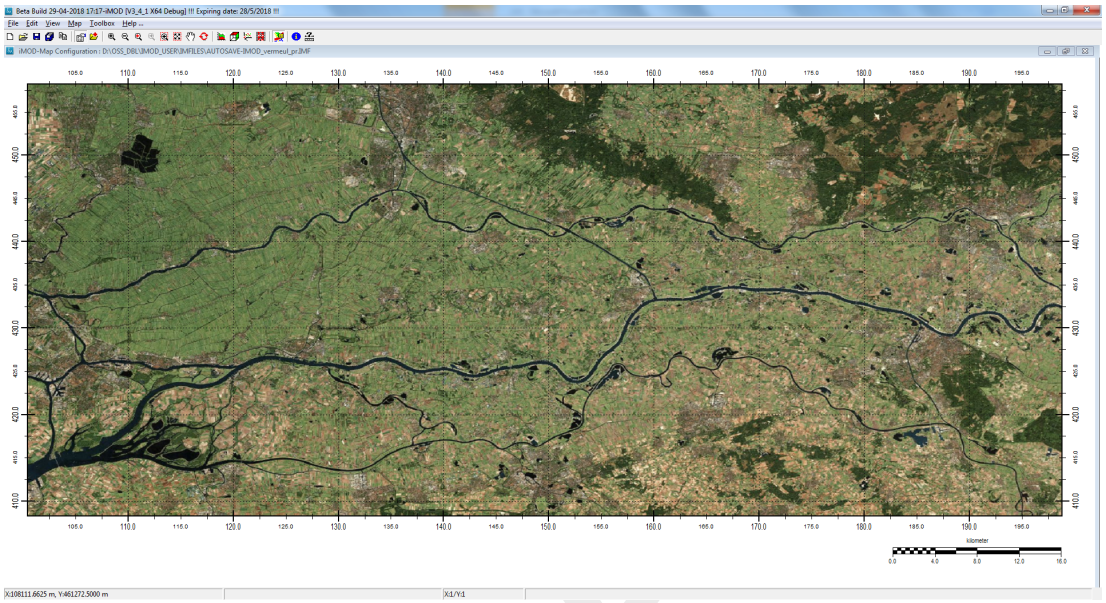
Add ...	Click this button to add a *.BMP or *.png file to the menu list of *Existing BMPs*. You are able to load all kinds of images, e.g. aerial photograph, satellite images (obtained by Google Earth). The PNG format is preferable to BMPs since it is significant smaller in size.
Delete Selected Files from List ...	Click this button to delete the selected file from the menu list of *Existing BMPs*.
Columns	Number of columns in the image
Rows	Number of row in the image
XLLC (km)	Enter the x-coordinate of the lower-left-corner (south-west) of the image.
YLLC (km)	Enter the y-coordinate of the lower-left-corner (south-west) of the image.
XURC (km)	Displays the x-coordinate of the upper-right-corner (north-east) of the image that will be computed from the XLLC and the bitmap width.
YURC (km)	Displays the y-coordinate of the upper-right-corner (north-east) of the image that will be computed from the YLLC and the bitmap height.
Size dX (m)	Bitmap width.
Size dY (m)	Bitmap height which will be equal to *Size dX*, automatically.
Apply	Click this button to show the background map. It also closes the *Add Background Image* window. The images (can be more than one) in the list of *Existing BMPs* will be shown when the menu option *Show Background Image* from the *View* menu is checked or when the corresponding icon from the tool bar is checked.
Close	Click this button to close the *Add Background Image* window.
Help . . .	Click this button to start the iMOD Help Functionality (if available in the selected *.PRF file).

Note: To reference the image topographically you can either enter the position of the image or add a so called worldfile in the same directory as the loaded image. iMOD will search for {ext}W files, e.g. BMPW or PNGW to read the information. The following information is listed in a worldfile.

Dx	Rastersize (m) x-direction (west-east).
RotX	Value of rotation along the y-axis, *RotX=0* for correct usage in iMOD.
RotY	Value of rotation along the x-axis, *RotY=0* for correct usage in iMOD.

Dy	Rastersize (m) in the y-direction, it should be a negative number, since it is measured from north to south. For correct usage in iMOD *Dx=-Dy.*
XULC	X-coordinate (m) for the center of the upper-left-corner (north-west).
YULC	Y-coordinate (m) for the center of the upper-left-corner (north-west).

Example of usage of a bitmap loaded via the option Add Background Image:



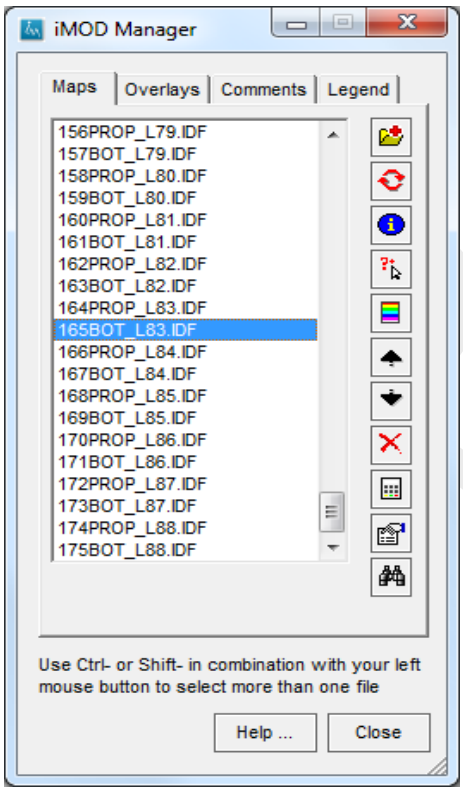
5.4 iMOD Manager

All active maps and overlays are managed by the *iMOD Manager*. On the menubar click *View* and then choose the option *iMOD Manager* to open the corresponding window.

The window has four tabs:

1 **Maps:**
This tab lists all maps loaded in iMOD. A large variety of maps can be loaded in the iMOD Manager, up to 500 files;
2 **Overlays:**
This tab lists all maps used as background only, these can be *.GEN and *.IPF files. Any *.SHP that will be read in this tab will be converted to a *.GEN file format;
3 **Comments:**
This tab lists comments that are *attached* to the selected map on the *Maps* tab;
4 **Legend:**
This tab displays the legend of the selected map which was drawn lastly from the selected list of maps on the *Maps* tab.

iMOD Manager window, Maps Tab:

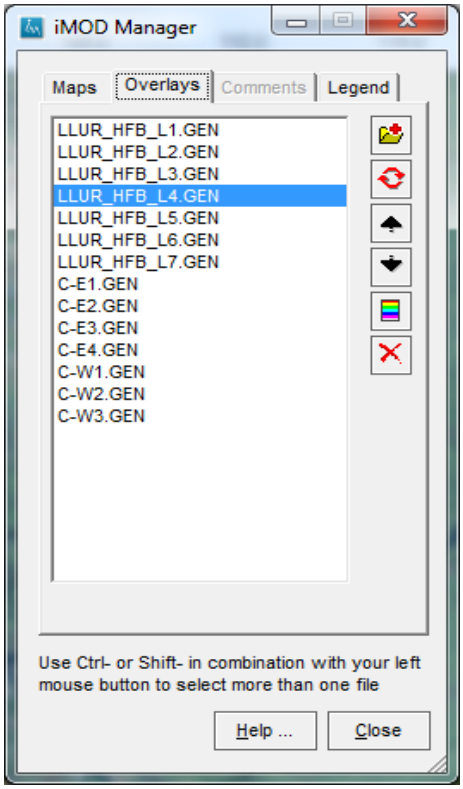


	Open Map Click this button to open a map. iMOD can read a variety of maps with known file types: *.IDF, *.IPF, *.IFF, *.ISG, *.ASC, *.SHP (will be converted internally to a GEN of IPF file), *.GEN, *.GEF (will be converted internally to a IPF file), *.NC (will be converted internally to a IDF file) and *.MAP (will be converted internally to a IDF file). Alternatively the shortcut F2 can be used, or select the menu option *Map* and then choose *Add Map*.
	*(re)Draw a Map* Click this button to redraw all selected maps.

	*MapInfo* Click this button to open the *MapInfo* window.
	*Map Value inspection* Click this button to open the *Map Value* window.
	*Legend* Click this button to open the *Legend* window.
	*Up* Click this button to move the selected files one position up in the list.
	*Down* Click this button to move the selected files one position down in the list.
	*Delete* Click this button to remove the selected files from the *iMOD Manager*.
	*Calculate* Click this button to open the *IDF Calculator* window.
	*Properties* Click this button to open the*Properties* window.
	*Find Files* Click this button to open the*Find Files* window, see section section 5.4.2.
Help	Click this button to start the iMOD Help Functionality.
Close	Hides the *iMOD Manager*. The iMOD Manager can be displayed again by choosing the menu option *View* and then choose the option *iMOD Manager*.

Note: To select more than one of the files in the tab *Maps* of the *iMOD Manager*, use the mouse-keyboard combination *Ctrl*- or *Shift*- combination. For many functionalities in iMOD it is necessary to select the desired files in the *iMOD Manager*, first. Bear in mind that several options are not available if these files are not selected.

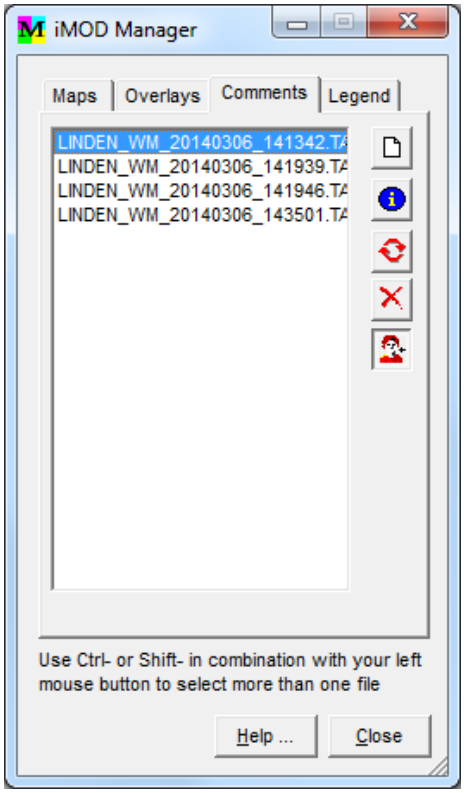
iMOD Manager window, Overlays Tab:



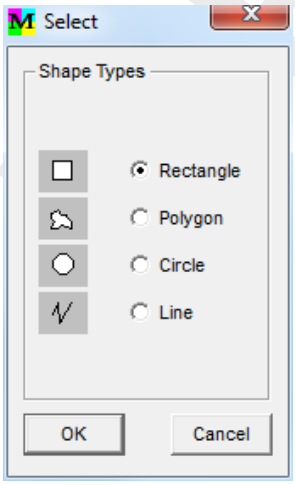
	*Open Overlay* Click this button to open an overlay map. iMOD can read a variety of overlays with known file types: *.GEN, *.SHP (will be converted internally to a GEN of IPF file), and *.IPF.
	*(re)Draw an Overlay* Click this button to redraw all selected maps.
	*Legend* Click this button to open the *Lines and Symbols* window, see section 5.7.
	*Up* Click this button to move the selected files one position up in the list.
	*Down* Click this button to move the selected files one position down in the list.
	*Delete* Click this button to remove the selected files from the *iMOD Manager*.

Note: Whenever the tab *Overlays* is selected in combination with the menu option *ZoomMap* and then *ZoomFull* (or the *ZoomFull* icon from the icon bar), iMOD will use the selected overlays to adjust the zoom level such that those files will be displayed fully.

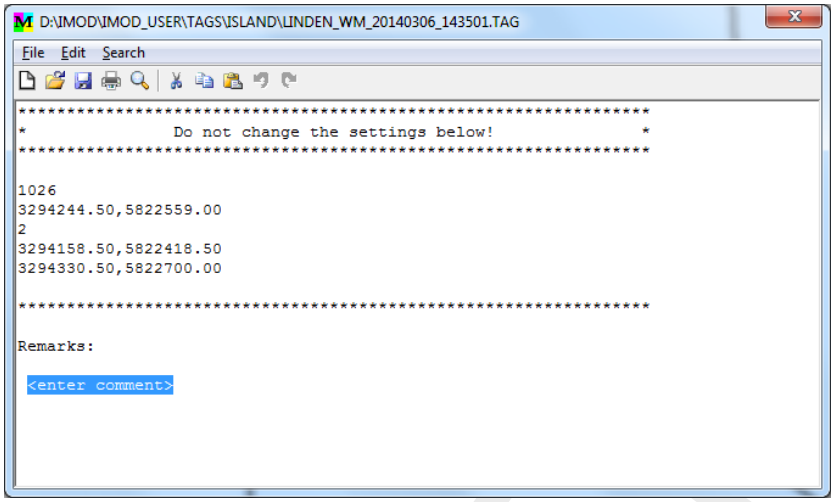
iMOD Manager window, Comments Tab:



Create tag
Click this button to create a tag. The tags are connected to the selected map (only one map may be selected). The tags may be defined as rectangle, polygon, circle or line. Select the shape type, click the *OK* button on the *Select* window and draw the shape on the graphical window (see section 4.4 for instructions).



A text file editor will open in which the coordinates of the drawn shape are shown and in which a comment can be added which will be tagged to the map. The comment is to be added at the location of the text: <*enter comment*>

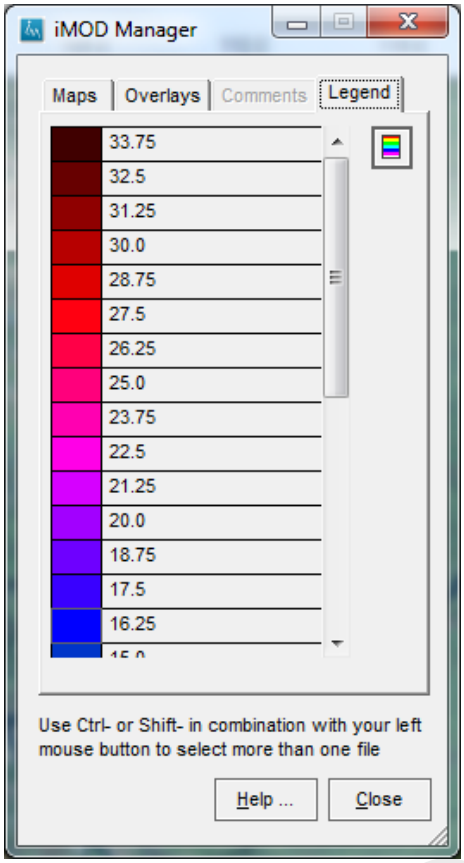


Save the comment and close the text editor window to proceed.

	*Info* Click this button to open a text editor window to view the comment.
	*(re)Draw a Map* Click this button to draw the tag and the comment id.
	*Delete* Click this button to remove the selected tag from the *iMOD Manager*. The button is active when the user button is activated
	*User* Click this button to allow the user to delete tags. Each user can delete his/her own tags only.
Help	Click this button to start the iMOD Help Functionality.
Close	Hides the *iMOD Manager*. The iMOD Manager can be displayed again by choosing the menu option *View* and then choose the option *iMOD Manager*.

Note: *The Comments tab on the iMOD Manager window is active when the TAGS variable is defined in the iMOD_INIT.PRF file, see section 9.1.*

iMOD Manager window, Legend Tab:



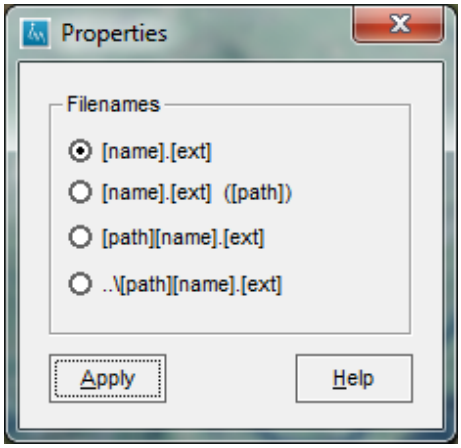
	Legend Click this button to open the *Legend* window (section 6.6).

Note: The legend will be shown for the file (map) that has been drawn last, i.e. the lowest selected file in the Map list on the *Maps* tab. There are different ways to plot the legend on the canvas.

5.4.1 iMOD Manager Properties

In the *Maps* tab of the *iMOD Manager* window, click the *Properties* button ( ) to open the Properties window. iMOD will use the selected file name properties to display the names of the files in all the tabs of the iMOD Manager.

Properties Window:

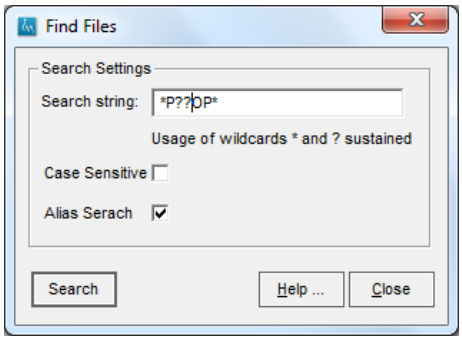


[name].[ext]	Click this option to display the names and their extent only, e.g. SURFACE.IDF.
[name].[ext] ([path])	Click this option to display the names and their extent together with the entire path, e.g. SURFACE.IDF (C:\IMOD).
[path][name].[ext]	Click this option to display full pathnames, e.g. C:\IMOD\SURFACE.IDF.
..\[path][name].[ext]	Click this option to display the relative pathnames, such that all files in the iMOD Manager can be still distinguished, e.g. ..\SURFACE.IDF.
Apply	Click this button to close the *Properties* window and apply the selected syntax.
Help . . .	Click this button to start the iMOD Help Functionality.

5.4.2 iMOD Manager Find Files

In the *Maps* tab of the *iMOD Manager* window, click the *Find Files* button ( ) to open the *Find Files* window. iMOD will select files in the iMOD Manager to the selection criteria specified in this window.

Find Files Window:



Search String	Specify the search string for which filenames need to be selected. Use the wildcards "*" (all characters and unknown amount) and "¿' (all character but amount is equal to number of "¿'-symbols).
Case Sensitive	Select this option to select filenames which are case-sensitive equal to the search string, in this case "E" $\neq$ "e".
Alias Search	Select this option to allow the search on the alias given to each filename in the iMOD Manager. If unselected, the search string is applied to the filename and the complete foldername.
Search	Click this button to select the filenames in the iMOD Manager for the given entries in this window. After that it closes this *Find Files* window.
Help . . .	Click this button to start the iMOD Help Functionality.
Close	Click this button to close the iMOD *Find Files* window.

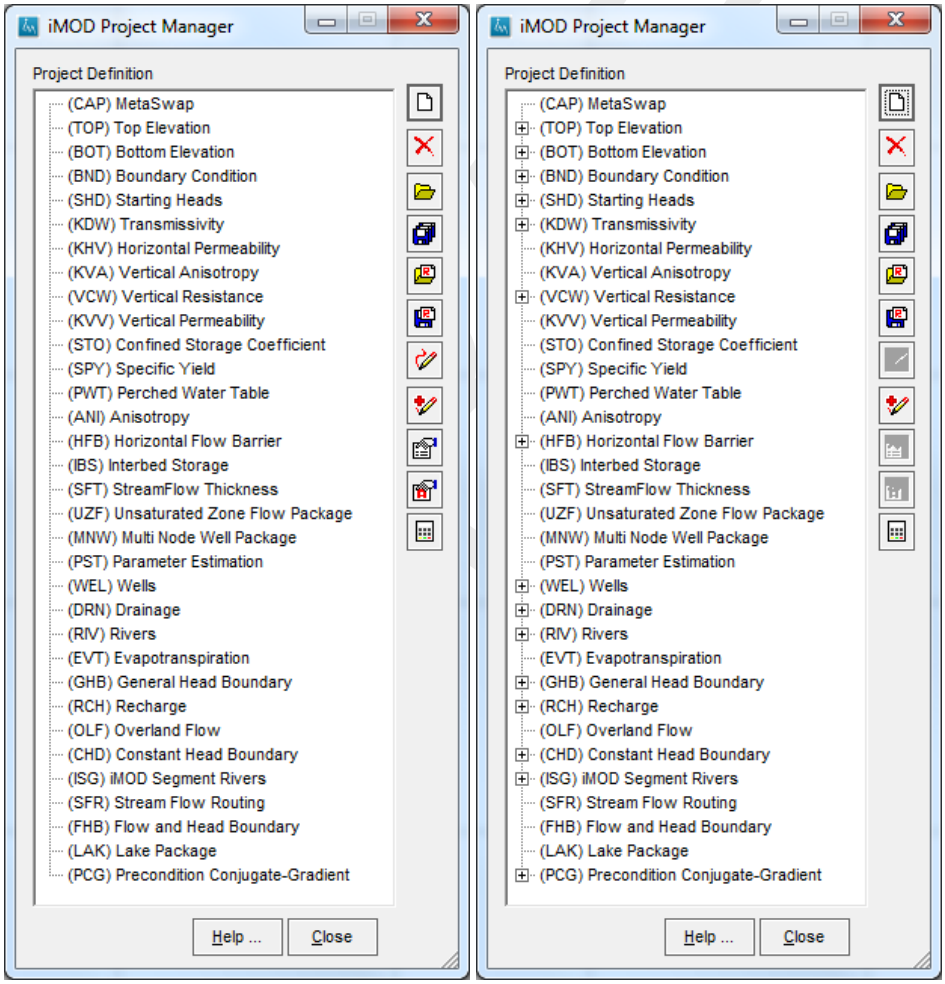
5.5 iMOD Project Manager

The files needed in a model simulation are defined in the *iMOD Project Manager*. On the menubar click *View* and then choose the option *iMOD Project Manager* to open the corresponding window.

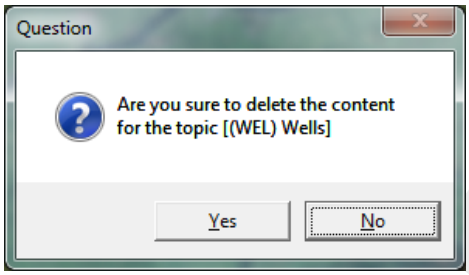
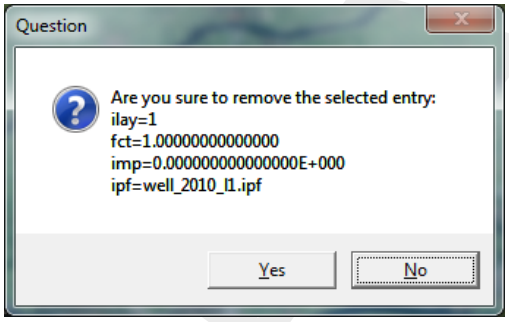
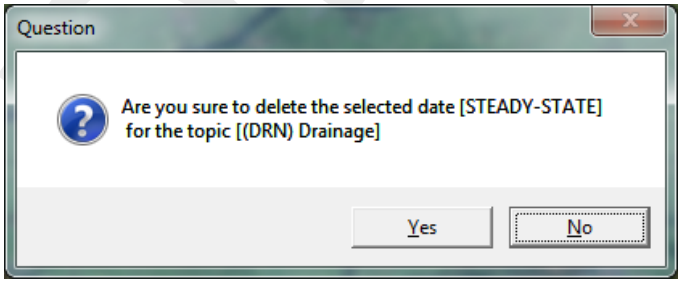
The *iMOD Project Manager* window shows a list of all possible model input topics. iMOD saves the characteristics in a project file, a so called *.PRJ file. From a project file a runfile (*.RUN) can be generated that will be used in the model simulation, see section 7.9 or a standard MF2005 configuration can be save (*.NAM), see section 5.5.4. Moreover, a runfile can be read and used to write a *.PRJ file. During the creation of a model configuration (*.RUN or *.NAM), it is possible to change the number of model layers and/or the time characteristics of the simulation, e.g. the begin and end time and/or sizes in stress-periods and/or the configuration of packages.

iMOD Project Manager window:

Initially: *After reading a project (*.PRJ) file:*



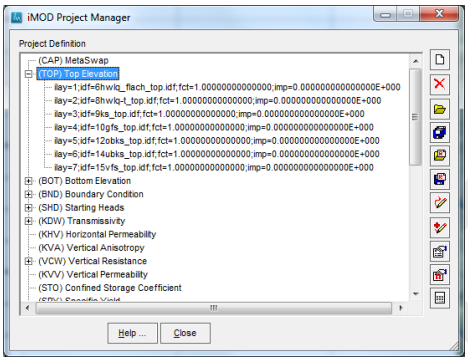
Note: The Project Manager does not yet support the PKS package.

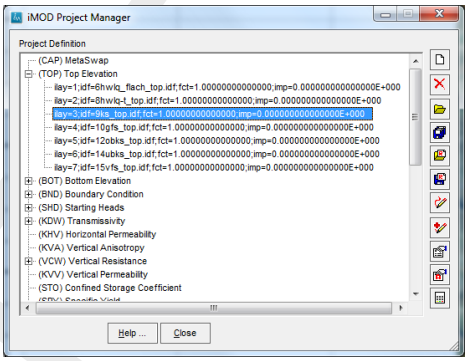
Project Definition	All acronyms bracketed are described in more detail in chapter 10. Once a project file *.PRJ (or a runfile *.RUN) has been read, the topics that contain model information (recognized by the small “plus” signs) can be expanded to access the underlying files/information.
	*Refresh* Click this button to refresh the *Project Definition* table. All definitions will be removed.
	*Delete* Click this button to delete the selected entry. If the main category is selected, the entire category will be deleted. If a single entry is selected, the corresponding package entry will be deleted. *Question to confirm the desired removal of a category.*  *Question to confirm the desired removal of a single entry.*  *Question to confirm the desired removal of all entries for a selected date.* 
	*Open Project file (*.prj)* Click this button to select a project file (*.PRJ). iMOD will read the project file and fills the treeview in the *Project Definition* table.
	*Save Project file (*.prj)* Click this button to save the project file (*.PRJ) on disk.
	*Open Runfile (*.run)* Click this button to select a runfile (*.RUN). iMOD will read the entire runfile and fills the treeview accordingly in the *Project Definition* table.

Save Runfile (.run / *.nam)*
Click this button to create a runfile (*.RUN) or MF2005 configuration (*.NAM) file. It starts the *Define Simulation Configuration* window, see section 5.5.4.

Draw
Click this button to port the files within the selected topic to the *iMOD Manager* and to display the files. The action of the *Draw* button depends on the selection in the tree view. It will port all files underneath the selected branch. Whenever a branch is expanded individual files can be selected that need to be ported to the *iMOD Manager*.

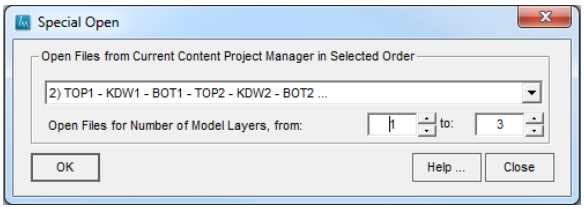
Draw all files in the expanded branch: *Draw the selected file only:*





Packages that are defined by constant values instead of spatial datasets, such as IDF, IPF, ISG and/or GEN-files (e.g. lay=3;fct=1.0;imp=0.0;constant=15.0) will not be ported to the *iMOD Manager*.

Special Open:



In this case, the files that are assigned to the 3 layers will be opened in the order top elevation, transmissivity and bottom elevation for each model layer, up to the 3^{rd} model layer. More options are:

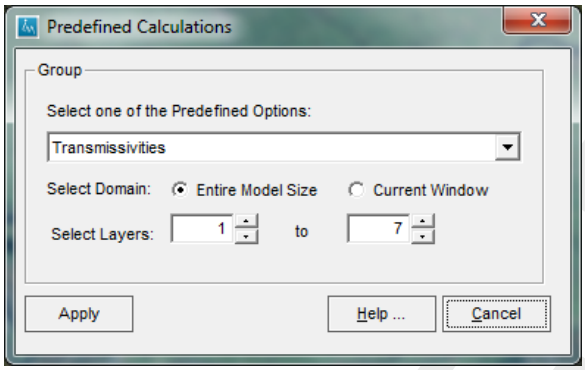
- TOP1 - BOT1 - TOP2 - ...
  Ports the IDF files for the top and bottom elevations to the iMOD Manager;
- TOP1 - KDW1 - BOT1 - TOP2 - KDW2 - BOT2 ...
  Ports the IDF files for the top, transmissivity and bottom elevations to the iMOD Manager;
- TOP1 - KDW1 - BOT1 - VCW1 - TOP2 - BOT2 - VCW2 - TOP3 ...
  Ports the IDF files for the top, transmissivity, bottom elevations and vertical resistance to the iMOD Manager;
- TOP1 - BOT1 - VCW1 - TOP2 - BOT2 - VCW2 - TOP3 ...
  Ports the IDF files for the top, bottom elevation and vertical resistance to the iMOD Manager;
- TOP1 - SHD1 - BOT1 - TOP2 - SHD2 - BOT2 ...
  Ports the IDF files for the top, starting head and bottom elevations to the iMOD Manager;
- TOP1 - KHV1 - BOT1 - TOP2 - KHV2 - BOT2 ...
  Ports the IDF files for the top, horizontal permeability and bottom elevations to the iMOD Manager;
- TOP1 - BOT1 - KVV1 - TOP2 - BOT2 - KVV2 - TOP3 ...
  Ports the IDF files for the top, bottom elevations and vertical permeability of the interbed to the iMOD Manager;

Define Characteristics:
Click this button to open the *Define Characteristics* window, see section 5.5.1.

Define Characteristics Automatically:
Click this button to open the *Define Characteristics Automatically* window, see section 5.5.2.2.

Calculator
Click this button to start the *Predefined Calculations* window in which it is possible to complete standard calculations on model parameters, such as computed vertical resistances between model layers based upon permeability values and top- and bottom of model interfaces.

Predefined Calculations:



The following predefined calculations are implemented:

- **Transmissivities**
  Based upon the horizontal permeability (m/d) values per modellayer and the top- and bottom of model layers, the transmissivity (m^2/d) is computed;
- **Vertical Resistances Aquitards**
  Based upon the vertical permeability (m/d) in between model layers, the vertical resistance (d) of the intermediate model layer (aquitard) is computed;
- **Total Vertical Resistances**
  Based upon the horizontal permeability (m/d), the top- and bottom, the vertical anisotropy (-) per model layer and the vertical permeability (m/d) in between model layers, the total vertical resistance (d) between model layers is computed.

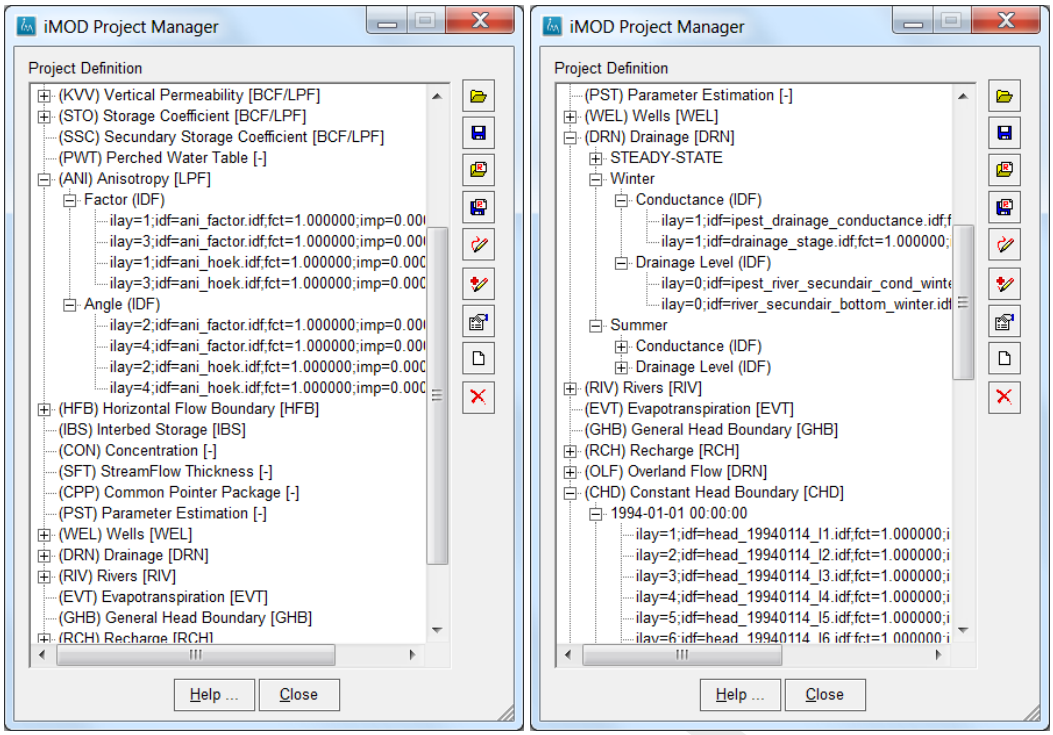
The computation can be carried out for the total modeling domain or for the current zoom window of the graphical canvas. Also, it can be done for a selected (sub)set of model layers.

Help ...	Click this button to start the iMOD Help Functionality.
Close	Click this button to close the iMOD Project Manager.

Note: For transient (time variant) topics, individual stressperiods (specific dates and/or periods) can be assigned to packages as a whole. They become accessible in the iMOD Project Manager. Those (and time invariant topics) may contain more than one levels (subtopics) of necessary input, such as ANISOTROPY that consists of FACTORS and ANGLES and DRAINAGE that consists of CONDUCTANCE and DRAINAGELEVEL. Use the *Define Characteristics Automatically* button () option to efficiently configure the time variant model input and/or multi-layered input.

Anisotropy with more subtopics:

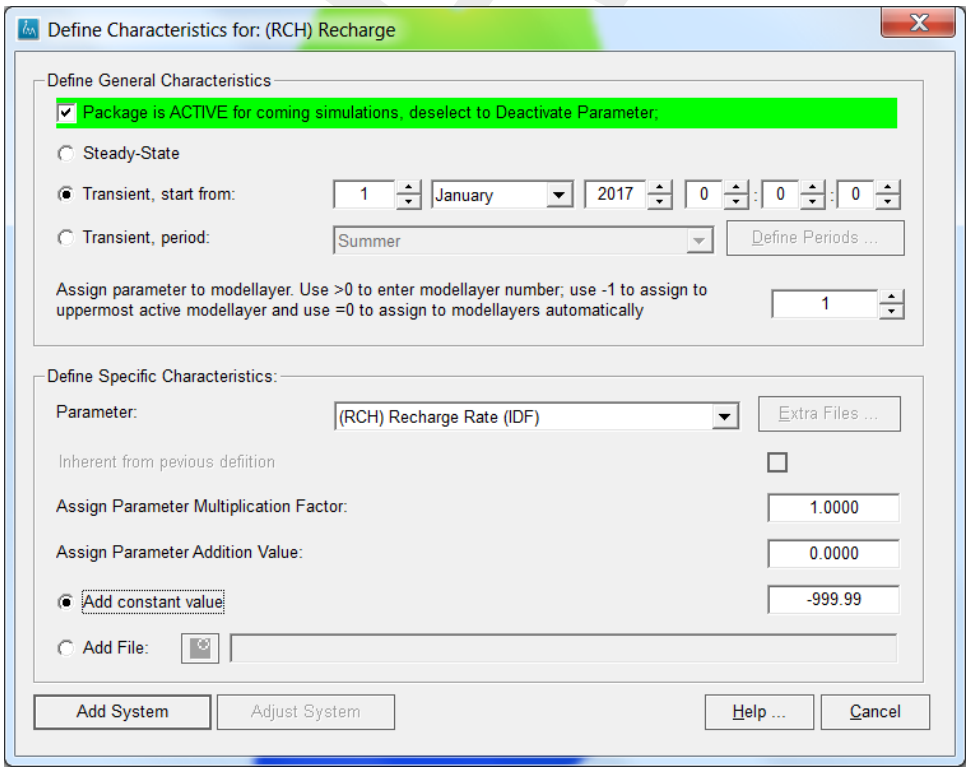
Drainage/Constant Head with time variant information:



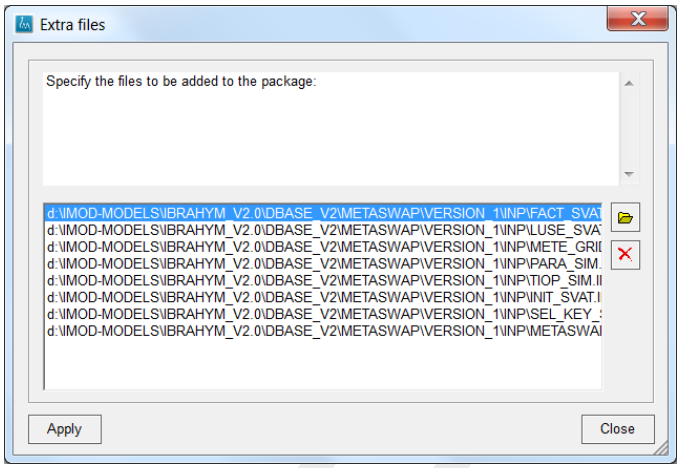
5.5.1 Define Characteristics

The *Define Characteristics* ( ) option of the *iMOD Project Manager* window opens a window which enables to define the characteristics of the model input topics as described below. **Note:** Whenever the PST package is selected a dedicated window is displayed (see section 5.5.5).

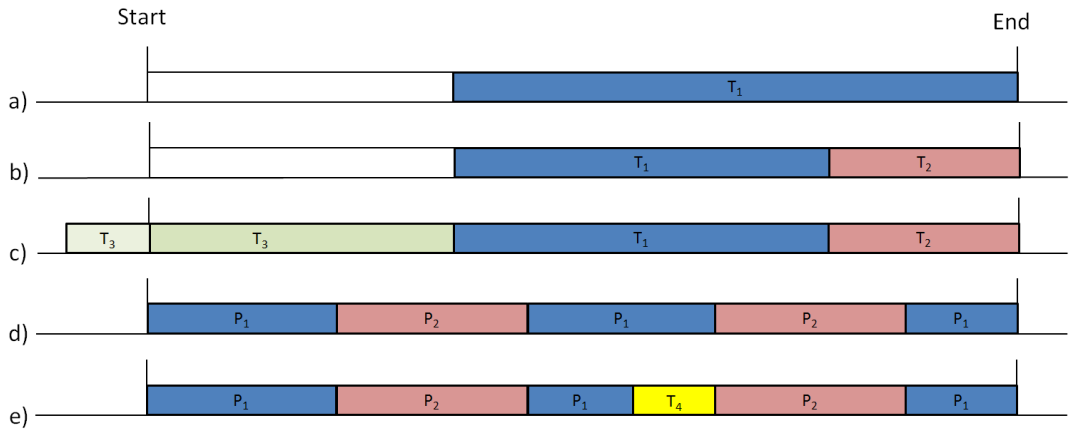
Define Characteristics window:



Package is ACTIVE for coming simulations, de-select to Deactivate Parameter	Check this option to exclude, in case it is ACTIVE, or include, in case it is DEACTIVE (red bar), this model topic when saving a runfile (*.RUN).
Steady-state	Check this button to define a steady state model input for the selected topic.
Transient, start from	Check this button to define input for use in a transient model. Enter the start date for the input in the input fields to the right. Besides defining the date there is also the option of defining the time [hh]:[mm]:[ss]. E.g. entering 10^{th} of June 2014 08 00 00 means that the input files as specified under *Define Specific Characteristics:* are starting at the 10^{th} of June 2014 at 08:00:00 am forward. The packages end, whenever another input is defined ahead of time.
Transient period:	Select a predefined period to indicate the start period for the current topic. This period will be endless in time, unless another input is defined ahead in time.
Define Periods...	Click this button to open the window in which it is possible to add and/or alter period definitions, see section 5.5.3.
Assign parameter to model layer . . .	Enter the model layer number to which the input is assigned in the model. Three options are possible: ◇ **Layer = 0** (for time variant input) A zero-value will assign the characteristics automatically to the model layers intersected by the depth of the model topic (e.g. the depth of the screen for wells or the depth of stage to bottom level for rivers); ◇ **Layer < 0** (for time variant input) A negative value will assign the characteristics to the upper most active model layer as defined in the Boundary Condition. Whenever the RCH and/or EVT packages are selected, the appropriate flags in the corresponding MF2005 packages will be set to 3 to indicate this automatic layer assignment; ◇ **Layer > 0** A positive value will assign the characteristics to the corresponding model layer.
Parameter:	Choose the parameter for which the specific characteristics will be defined. Depending on the model input topic the number of parameters is 1 (e.g. for WEL), 2 (e.g. for ANI), 3 (e.g. for EVT) or 4 (e.g. for RIV). See chapter 10 for detailed information about these different input per topic.

Extra files:	Click this option to add extra files to the topic. This option is only available whenever the MSW (Unsaturated Zone) package is selected. *Extra Files:*  In this case, the files are listed to be copied in a simulation with MetaSWAP.
Inherent from previous definition:	Select this option to inherent the input for the selected topic from the previous (in time) definition for this topic.
Assign Parameter Multiplication Factor:	Change the multiplication factor from the default value 1.0 in case the model input needs to be multiplied. Multiplication goes before the additional value.
Assign Parameter Addition Value:	Change the addition factor from the default value 0.0 in case the model input needs to be increased (added) with a constant value. Multiplication goes before the additional value.
Add constant value	Check this button and enter a constant value for the parameter for the whole model area.
Add file:	Check this option and enter a file name to be used for the parameter. It depends on the package whether an IDF, IPF, ISG or GEN file need to be entered.
	*Open File* Click this button to open the a Windows Explorer to locate the file name for the parameter, this can be an IDF, IPF, ISG or GEN file that depends on the topic considered.
	*Open* Click this button to select a folder, remember to add a wild-card to the folder name in order to select appropriate file names by iMOD.
Help ...	Click this button to start the iMOD Help Functionality.
Add System *Adjust System*	Check this button to save the entered input for the selected topic and return to the *Project Manager* window. This can mean that the parameter for an existing topic are adjusted, or a new system is added to a topic. The tree-view in this window will collapse all topics and expand the selected and modified topic. For large project configurations with many time-definitions it can take several minutes to (re)fill in the tree-view field.

Note: Packages are assigned to a particular date and time at which they start. They will never end, but can be overruled by another packages that is defined ahead of them. In the following figure it is explained what this means whenever more packages become available and interact whenever *periods* are defined.



In the figure shown above, a) describes the situation in which only a single package T_1 is defined half-way a simulation between *Start* and *End*. In b), another package T_2 is defined and overrules T_1 at the start of T_2. In c), a third package T_3 is defined before T_1 and even before the start of the simulation. Consequently, the package T_3 becomes active, directly at the start of the simulation. In d), a period P_1 and P_2 are defined at a certain moment within a single year, they will be repeated for each year, automatically. Finally, in e) a package T_4 is defined besides the period definitions P_1 and P_2. As a consequence, the package T_4 will *split* period P_1 halfway, up to the moment period P_2 is defined again.

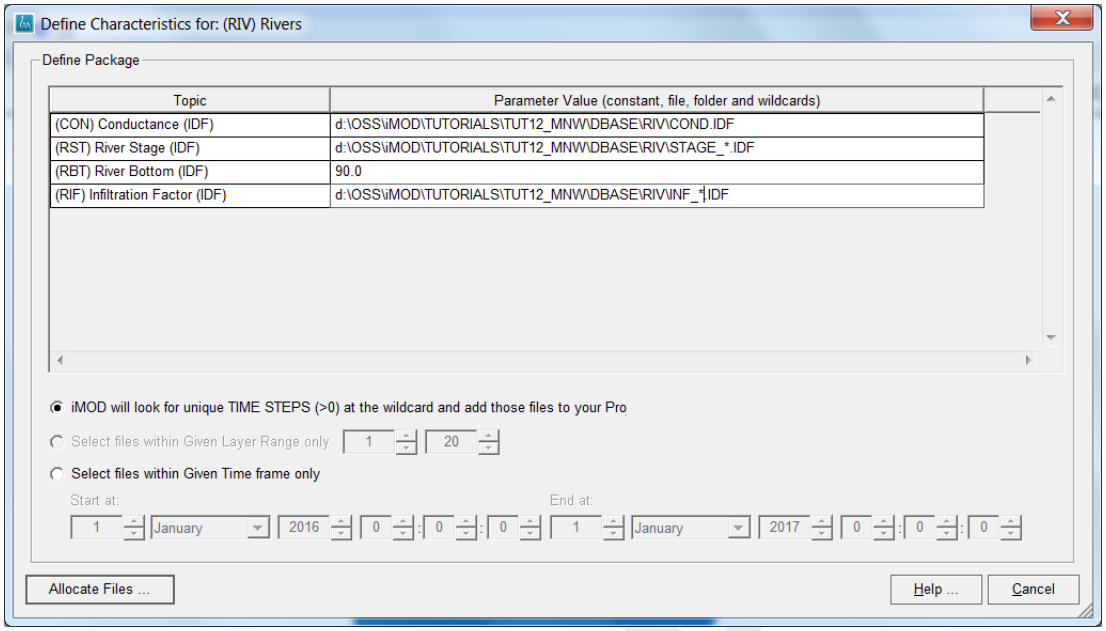
5.5.2 Define Characteristics Automatically

The *Define Characteristics Automatically* ( ) option on the *iMOD Project Manager* window opens a window which enables to define the characteristics of the model input topics in a more advanced way than the default *Define Characteristics* window (see section 5.5.1). The process consists out of two steps. In the first step you define the characteristic of the source for each topic, in the second step iMOD will list the found sources and it is possible to modify this list manually before adding them to the *Project Manager* window.

5.5.2.1 Define Source for Topics

The first step is to define the type of sources for each topic.

Define Characteristics for: window

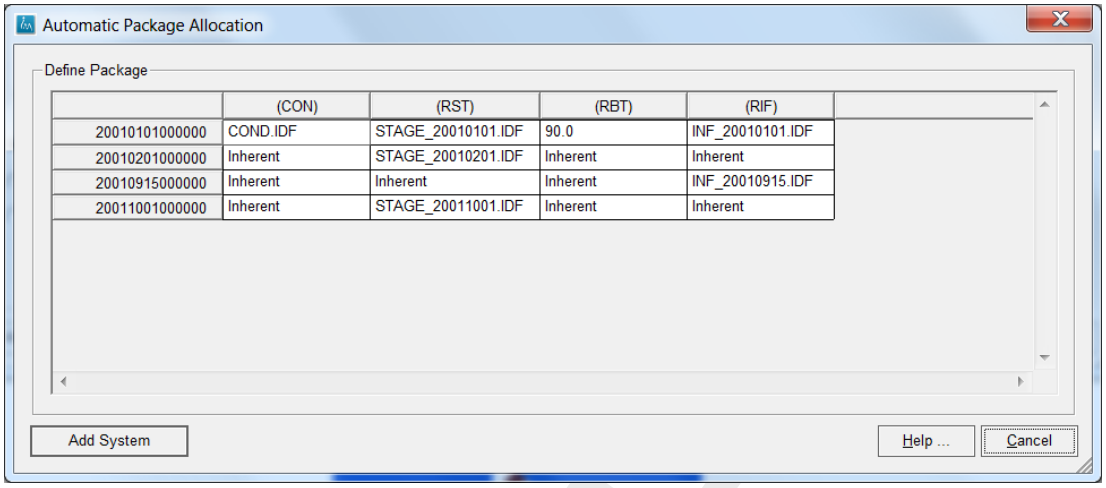


Define Package	This table show how each of the topics for the package (here the topics CON, RST, RBT and RIF for the RIV package) need to be specified. Three options are available ◇ Constant Value Specify a constant value for the topic to be used for all stress-periods or model layers, e.g. a value of 90.0 means that this topics is 90.0 for all stress-periods or model layers; ◇ File Name Specify a file name for the topic to be used for all stress-periods or model layers, e.g. a file name as D:\MODEL\DBASE\BOTTOM.IDF applies that this file is used for all stress-periods or model layers; ◇ Wild card Specify a file name with a wild card (*) to specify that all files that are part of this will be added. It depends whether a module or package is associated, for modules layers can be added this way, for packages multiple stress periods, e.g. D:\MODEL\DBASE\STAGE_*_.IDF will add all files that belong to this group.
iMOD will look for unique TIMESTEPS	Select this option to select unique timesteps and add the related files to your project.
iMOD will look for unique LAYERS	Select this option to only select unique layers and add the related files to your project.
Select files within given layer range	Select this option to be able to only select the files of the given layer range of your project.
Select files within given time frame only	Select this option to define the start- and end date for the input. Besides defining the date there is also the option of defining the time [hh]:[mm]:[ss]. E.g. entering 10^{th} of June 2014 08 00 00 means that the input files as selected at the 10^{th} of June 2014 at 08:00:00 am up to the specified end date.
Allocate Files ...	Click this button to apply the given sources for topics and pop-up a list of constant values and/or file namer per model layer or stress period.
Help ...	Click this button to start the iMOD Help Functionality.
Cancel ...	Click this button to close the *Define Characteristics Automatically* window and return to the *Project Manager* window.

5.5.2.2 Modify List of Topics

The second step is to inspect the list of found files based on the given sources for each topic.

Define Characteristics for window:

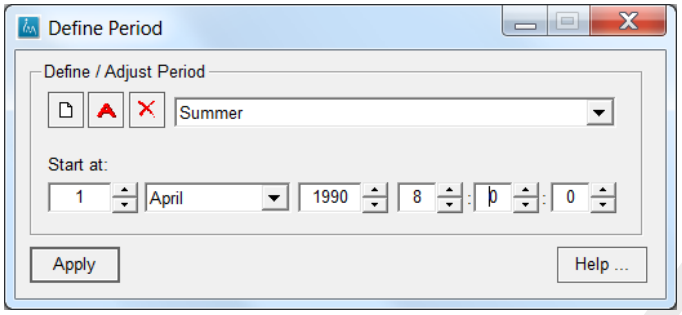


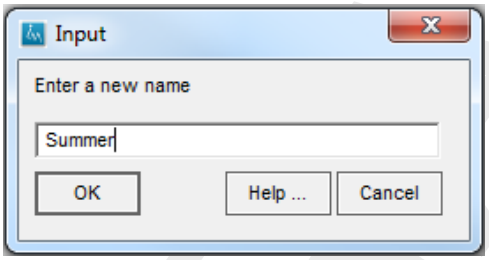
example time and layers

5.5.3 Define Periods

A period is defined by a starting date and time for which a package, assigned to that period, will be repeatedly included in the runfile. A period start at a particular moment in time, but it will never end repeating itself. The only way of ending a package defined by a period, is by defining another package after or equal to the starting date of the period.

Define Periods window:

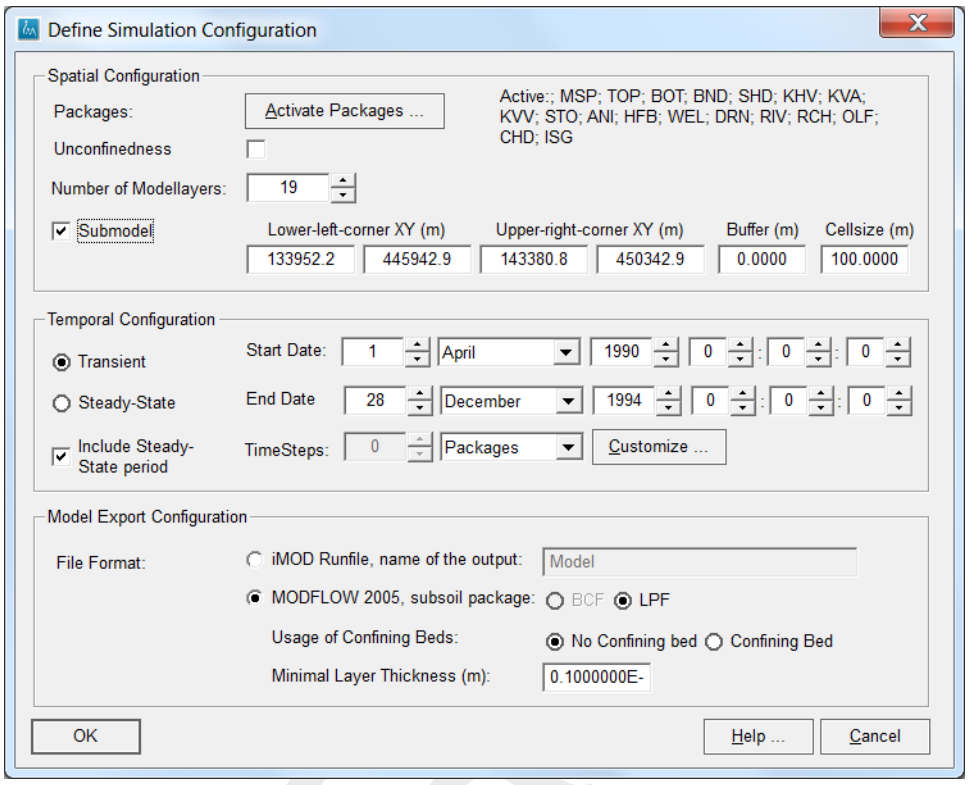


	*New* Click this button to define a new period. In the following window it is possible to enter a name for the period.  Use the *OK* button to agree with the entered period name, click the *Cancel* button to cancel the creation of a new period. Click *Help* to open the iMOD help functionality.
	*Rename* Click this button to rename an existing period name. Usage of the window as described by *New* will be used.
	*Delete* Click this button to delete the selected period. Whenever the last period is delete, iMOD will ask to enter a period name as described by *New* and if this cancelled, the *Define Period* window will be closed.
Summer	Select one of the predefined period from the dropdown list. If none available, create one first by means of the *New* button.
From	Modify the starting date and time for the selected predefined period. In this example the period *Summer* is a period starting from 1990 and thereafter each year at the 1^{st} of April at 08:00:00 am. Any modification will effect any systems for topics that are assigned to that period definition.
Help ...	Click this button to start the iMOD Help Functionality.
Apply	Click this button to close this window, any modification to a period definition will be stored.

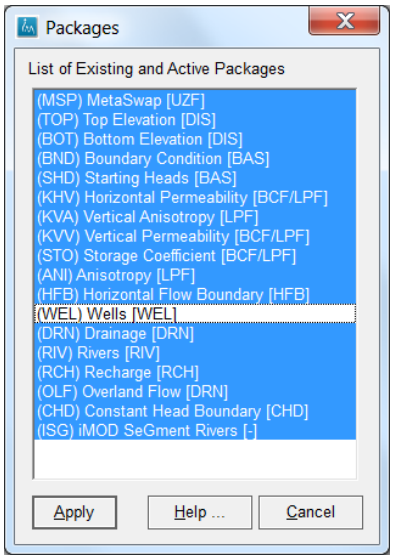
5.5.4 Define Simulation

Define Simulation window:

Within the *Define Simulation window* it is possible to configure the runfile and/or standard Modflow2005 configuration files. Several options are available, such as defining the location and size of clip models, time-discretisations and active packages.



Packages:	Click this button to select the packages that need to be active in de model.

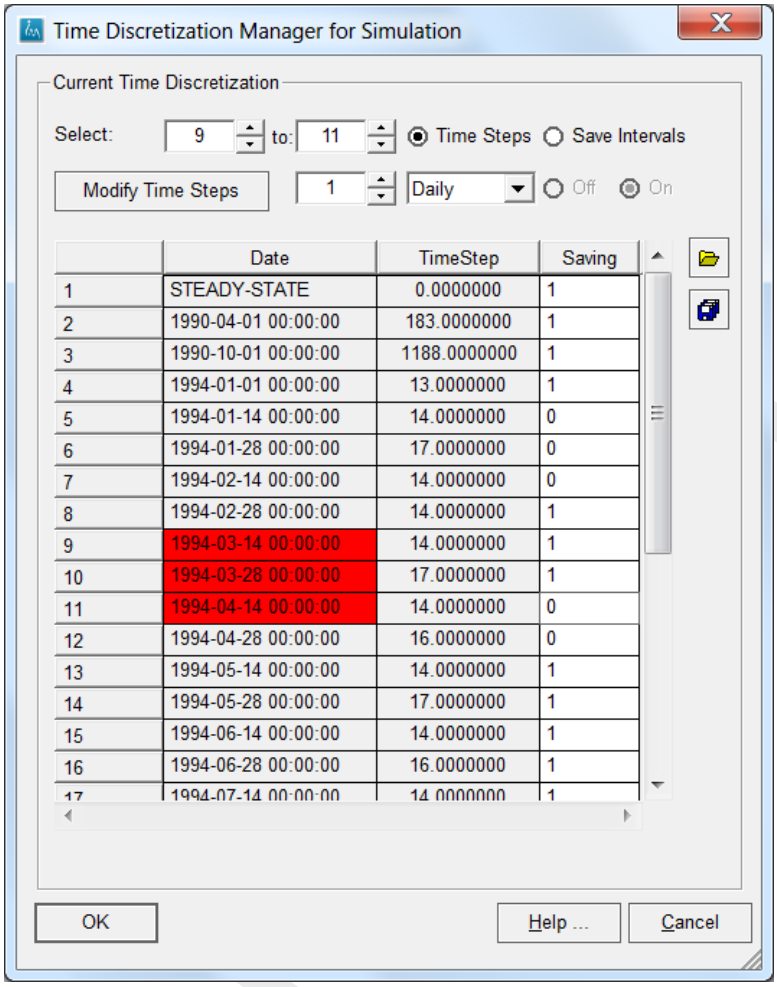


By default all packages are selected, those are listed on the uppermost right corner on the *Define Simulation window* as well. Use the CTRL-left-mouse button to selected and deselect packages. It is not recommended to deselect packages that are essential for a correct simulation of the model, such as the BND, SHD packages. Click the *Apply* button to accept your choice, click the *Cancel* to leave the *Packages window* without any changes in the selected package and click the *Help* button to start the Help-functionality.

Unconfinedness	Select this checkbox whenever a model configuration need to be prepared to simulate unconfined conditions whereby the transmissivity is a function of the computed hydraulic head. This option becomes selectable whenever the packages TOP, BOT, KHV and KVV are available.
Number of Modellayers	Enter a number of modellayers for which the model configuration need be built. iMOD will fill in the maximum number of modellayers based on the modellayers that have been filled in for the most important packages, BND, SHD (TOP and BOT), KDW or KHV and VCW or KVV.
Submodel	Check this option to define the lower- and upper-right-corner, buffersize and cellsize of a submodel or clipmodel to be included in the runfile or used to generate the Modflow2005 configuration files.
Transient	Check this option to generate a transient runfile (*.RUN) or Modflow2005 configuration files. iMOD will collect all packages that are within the specified *Start Date* and *End Date*. Besides defining the date there is also the option of defining the time [hh]:[mm]:[ss]. On default the start- and end time is set to 00:00:00.
Start Date	Enter a start date and time for the transient simulation, this date will be **start** of the first stress-period. iMOD will fill in this *Start Date* initially with the earliest defined date in the packages. Whenever the *Start Date* is decreased, before the initial value, the model will include packages that remain inactive until the first dat at which they are defined.
End Date	Enter an end date and time for the transient simulation, this date will be the **end** of last stress-period. iMOD will fill in this *End Date* initially with the latest defined date in the packages. The *End Date* is the date at which the simulate will terminate, so it is the end of the last stress period. It is allowed to increase the *End Date* beyond the initial value to enforce the latest stress period to be include in the model.
Steady-State	Check this option to generate a runfile (*.RUN) for a steady-simulation, the option becomes available only whenever at least one package is defined for a steady-state period. iMOD will collect all packages that are connected to a steady-state definition as specified in the *Define Characteristics* window.

Include Steady State period	Select this option to include an initial steady-state period, prior to the start of the transient simulation. This option becomes available whenever at least one package is defined for a steady-state period, see section section 5.5.1.
TimeSteps	Select one of the option from the drop down menu: ◇ **Hourly** Select this option to generate hourly stress-periods; ◇ **Daily** Select this option to generate daily stress-periods; ◇ **Weekly** Select this option to generate weekly stress-periods; ◇ **14/28** Select this option to generate stress-periods on the 14th and 28th day of each month; ◇ **Monthly** Select this option to generate monthly stress-periods; ◇ **Yearly** Select this option to generate yearly stress-periods; ◇ **Decade** Select this option to generate stress-periods per decade; ◇ **Packages** Select this option to generate stress-periods that are determined by the input data as specified by the available packages in the *Define Characteristics* window. It can yield a non-constant time sequence for stress periods, but will be most optimally to the amount of stress-periods; ◇ **Custom** Select this option to refine or inspect time configuration priorly defined. This option is selected automatically after the first time the option *Customize* is selected.

Customize... Click this button open the *Time Discretization Manager for Simulation* window to customize the time steps of the simulation to be included in the runfile and/or Modflow2005 configuration.



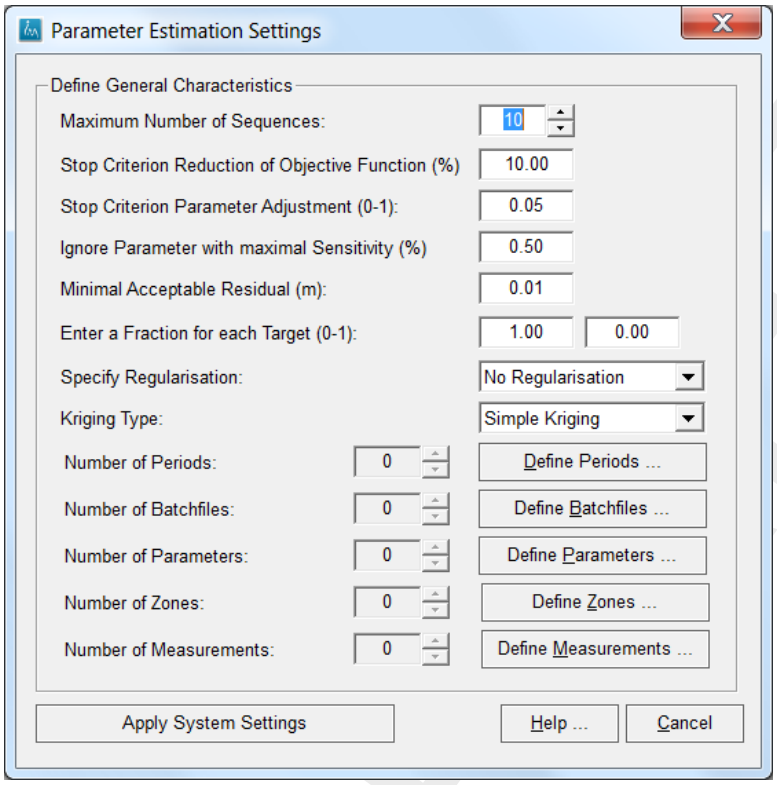
- *Select*
  Enter the row number within the table to be modified, e.g. 9 ad 11, those will be displayed by a red box to indicate the selected rows;
- *Time Steps*
  Select this option to modify time steps in the table;
- *Save Intervals*
  Select this option to modify the save intervals in the table;
- *Modify Time Steps / Modify Save Intervals*
  Click the button to adjust the selected rows according to the selected settings in the fields right of this button and explained below;
- *1*
  Enter a number to enter time steps sizes for the selected value from the drop down menu, next to it to the right, e.g. 1 means in this case 1 day.
- *Daily*
  Select from the drop down menu the appropriate time step size to be used for the selected rows. E.g. *Daily* will include time steps on a daily base for the selected rows. Other choices are *Hourly, Weekly, Monthly, Decade, 14 / 28, Yearly, Packages*
- *Off / On*
  Select on of the options to turn the Save interval for the selected rows on or off;
- *Save As*
  Click this button to save the current time step configuration to a *.TIM file, see section 9.4;
- *Open*
  Click this button to load the current time step configuration from a *.TIM file, see section 9.4;

File Format:	The file format of the model configuration can be defined with this option. There are two possibilities: 1. *iMOD Runfile* and/or standard *MODFLOW2005* files.
iMOD Runfile, name of the output:	Select this option if the iMOD Runfile format is preferred. Give the name of result folder to be created by the runfile, e.g. the result folder is finally located in the {USER}\MODELS \MODEL. After selecting the *OK* button, it is necessary to enter the name of the runfile to be created.
MODFLOW 2005	Select this option if the MODFLOW 2005 format is preferred. After selecting the *OK* button, it is necessary to enter the name of the namfile to be created.
BCF / LPF	Select on of the configuration options to denote the subsoil characteristics in the Modflow2005 files. The availability of the options depend on the active packages.
Usage of confining beds	Select the *No Confining bed* whenever the model configuration is 3-D, that is there are no confined beds, the bottom of each model layer is equal to the top of the underlying model layer. Select the *Confining Bed* whenever the bottom of each or at least a single model layer is not equal to the top of the underlying model layer.
Minimal Layer Thickness (m)	Enter a minimal thickness for model layers. For Modflow2005 it is important that the thickness of model layers is not zero. Whenever a minimal thickness of 0.01 is entered, the top- and bottom elevations will be adapted such that there is a minimal thickness of 0.01 meter. Moreover, the hydraulic properties will be adjusted for those areas to reflect the corresponding layer, that will be part of the model layers that increase in thickness.
OK	Click this button to select an existing or non-existing runfile (iMODFLOW) and/or namfile (Modflow2005), by default iMOD will save the runfile and/or namfile in the {USER}\RUNFILES folder. It is convenient to do that as well, however not obliged, but it will allows iMOD to start the runfile from the *Model Simulation* tool, see section 7.9.
Help ...	Click this button to start the iMOD Help Functionality.
Cancel	Click this button to close this window.

5.5.5 Parameter Estimation

Parameter Estimation Settings window:

Within the *Parameter Estimation Settings window* it is possible to configure the settings for a parameter estimation. Several options are available, such as defining the main settings, the number and characteristics of parameters, the zones and measurements, see section section 12.33 for more detailed information on parameter estimation. Most of the parameters are described in more detail in section section 10.14 as well.

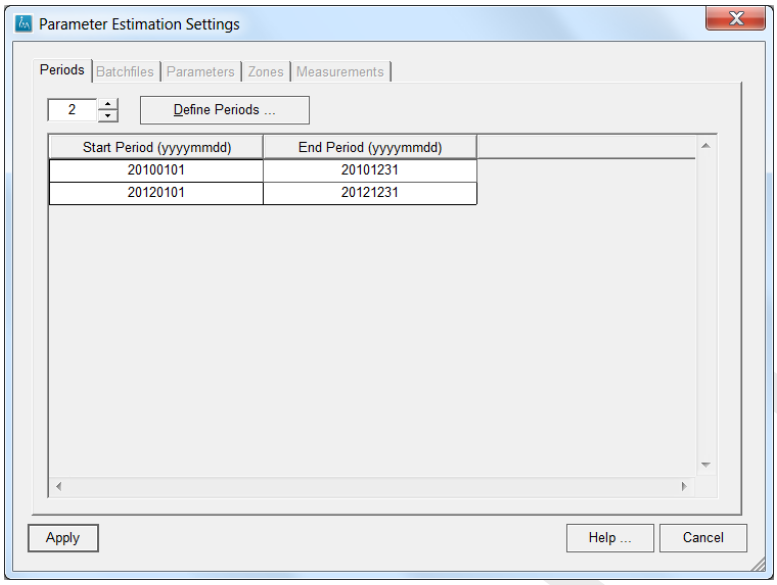


Maximum Number of Sequences	Enter the number of sequences (iteration) of the parameter estimation routine. Enter a zero will start a sensitivity analysis only.
Stop Criterion Reduction of Objective Function	Enter a percentage for which the objective function needs to be reduced between adjacent iteration in order to terminate the parameter estimation process. The percentage is computed as the ratio between the previous and current objective function value.
Stop Criterion Parameter Adjustment	Enter a value for this stop criterion between 0.0 and 1.0 to terminate the parameter estimation whenever it becomes less than the specified value. The stop criterion is computed as
Ignore parameter with maximal Sensitivity	Enter a percentage of the sensitivity of a parameter which will be excluded from the current parameter estimation cycle, whenever its sensitivity is less than the specified value.
Minimal Acceptable Residual	Enter a value to skip measurements in the formulation of the objectie function whenever the absolute residual is less than the specified value.

Enter a fraction for each Target	Enter a weight value for two targets of the objective function; the first target is the sum of the square root residuals, the second the sum of the square root residual of the dynamic in a measurement (see section 12.33.2). Its is not necessary to scale them to a sum of 1.0, internally the entered fraction will be scaled to a total sum of 1.0, this includes any Batchfiles included (see *NUmber of Batchfiles* further below).
Specify Regularisation	Regularisation is a process in which parameters can be *grouped* mathematically of excluded/ignored in the parameter optimization. This can be done manually (using the parameter sensitivity) or automatically by means of some mathematically expressions. Select of the following options to use a regularisation: ◇ **No Regularisation** No use of regularisation, that is no Scaling and/or Eigenvalue Decomposition; ◇ **Scaling** Use Scaling as a regularisation, use this option whenever the amplitude of different residuals differ significantly, such as measurements of groundwater level and discharge; ◇ **Scaling and Eigenvalues** Use Scaling and Eigenvalue Decomposition whenever it is very difficult to exclude parameters on their sensitivity or relevant contribution to the reduction of the objective function; ◇ **Eigenvalues** Use Eigenvalue Decomposition whenever the parameter sensitivity alone, is not enough the estimate the relevance of a parameter to the parameter optimization.
Kriging Type	Specify the type of Kriging (section 12.33.4.1) whenever the Pilot Point concept is used (section 12.33.4). Whether Pilot Points are used is steered by the fact that an IPF file will be entered by the zones (see *Number of Zones*) instead of IDF files. The following options are available: ◇ **Simple Kriging** (section 12.33.4.1) Simple Kriging assumes stationarity of the mean, all variables have the same mean over the entire domain; ◇ **Ordinary Kriging** (section 12.33.4.1) In Ordinary Kriging a unknown mean is assumed only over the search neighborhood, so the mean is recomputed for the values in the search neighborhood.

Define Periods ...

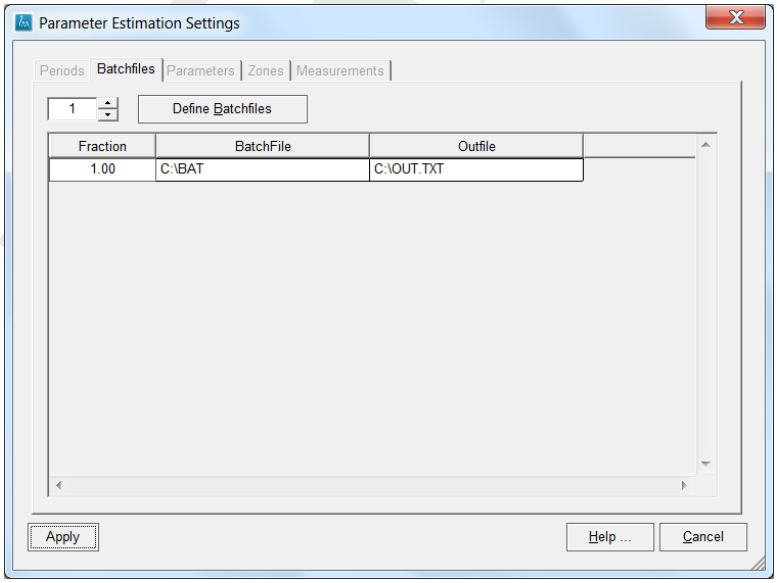
Click this button to define periods (see section 10.15) for which measurement need to be included in the computation of the objective function, e.g. 1^{st} of January 2010 to the 31^{st} of December 2010 and the 1^{st} of January 2012 to the 31^{st} of December 2012.



Click the *Define Periods* to increase or decrease the number of available rows (entry fields) in the table, e.g. 2. There is a maximum of 10 rows. Click the *Apply* button to accept your entry, click the *Cancel* to ignore any changes, click the *Help* button to start the Help-functionality.

Define Batchfiles ...

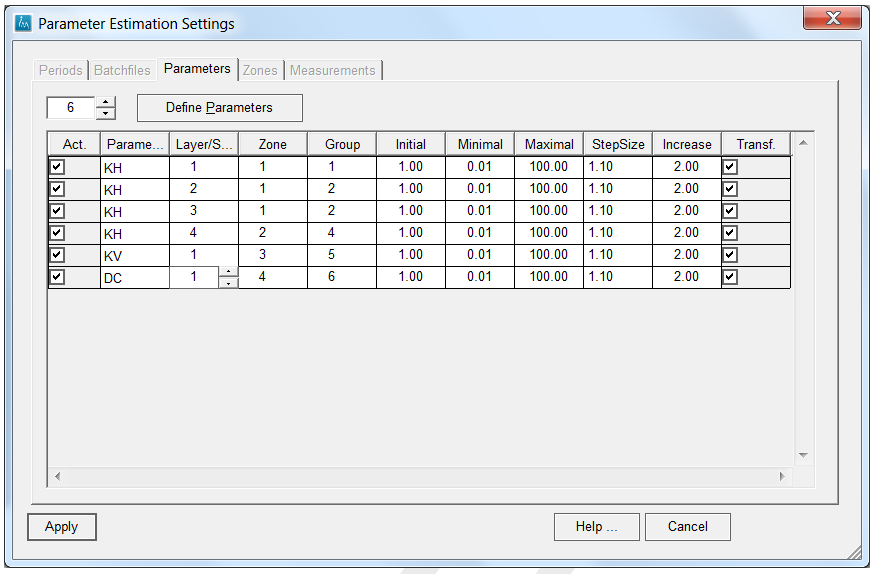
Click this button to define batchfiles (see section section 10.16) for which extra or additional measurement need to be included in the computation of the objective function.



Click the *Define Batchfiles* to increase or decrease the number of available rows (entry fields) in the table, e.g. 1. There is a maximum of 10 rows. Click the *Apply* button to accept your entry, click the *Cancel* to ignore any changes, click the *Help* button to start the Help-functionality.

Define Parameters ...

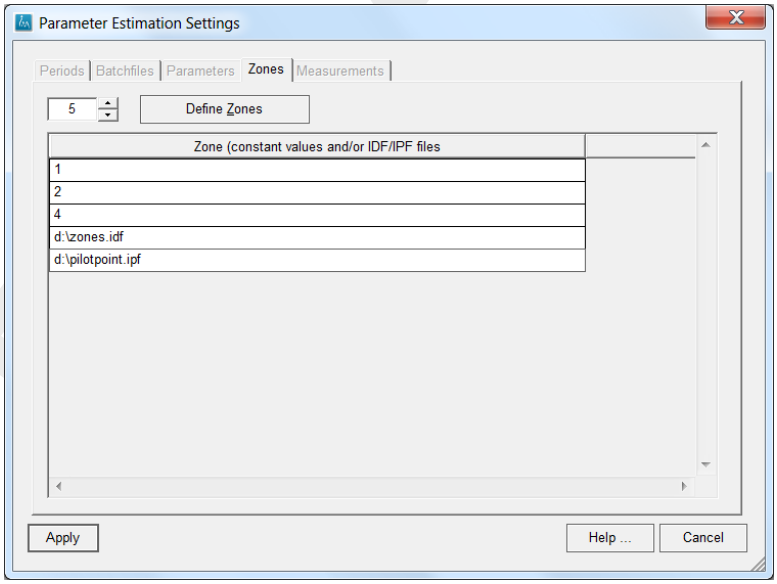
Click this button to define parameter to be optimized.



Click the *Define Parameters* to increase or decrease the number of available rows (entry fields) in the table, e.g. 2. There is a maximum of 1000 parameters, see section section 10.17 for the explanation of the variables. Click the *Apply* button to accept your entry, click the *Cancel* to ignore any changes, click the *Help* button to start the Help-functionality.

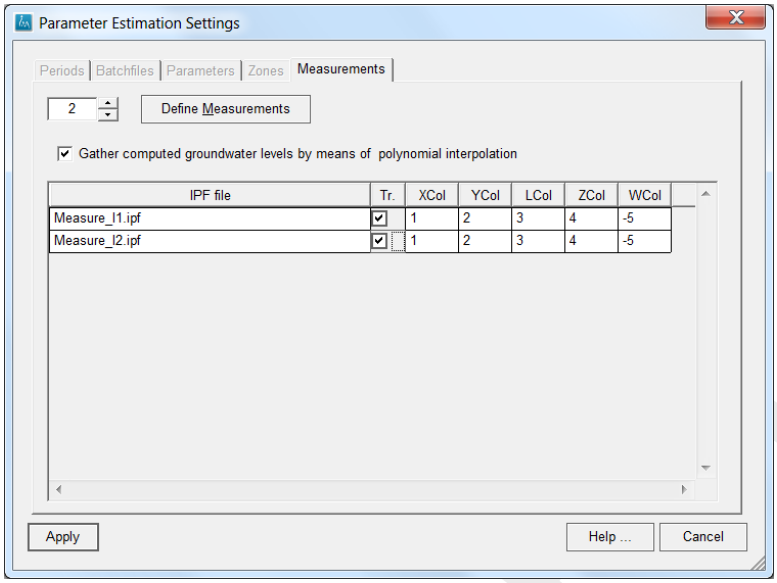
Define Zones ...

Click this button to define zones (see section section 10.19) for which parameters need to be adjusted.



Click the *Define Zones* to increase or decrease the number of available rows (entry fields) in the table, e.g. 5. There is a maximum of 1000 zones, constant values can be entered to denote a zone fore the entire model area, multiply zones can be combined in a single IDF file and/or IPF files can be entered for usage of Pilot Points (section 12.33.4). Click the *Apply* button to accept your entry, click the *Cancel* to ignore any changes, click the *Help* button to start the Help-functionality.

Define Measurements ...	Click this button to define measurements (see section section 10.4) which are needed to compute the objective function. It is also possible to exclusively use *Batchfiles* instead.



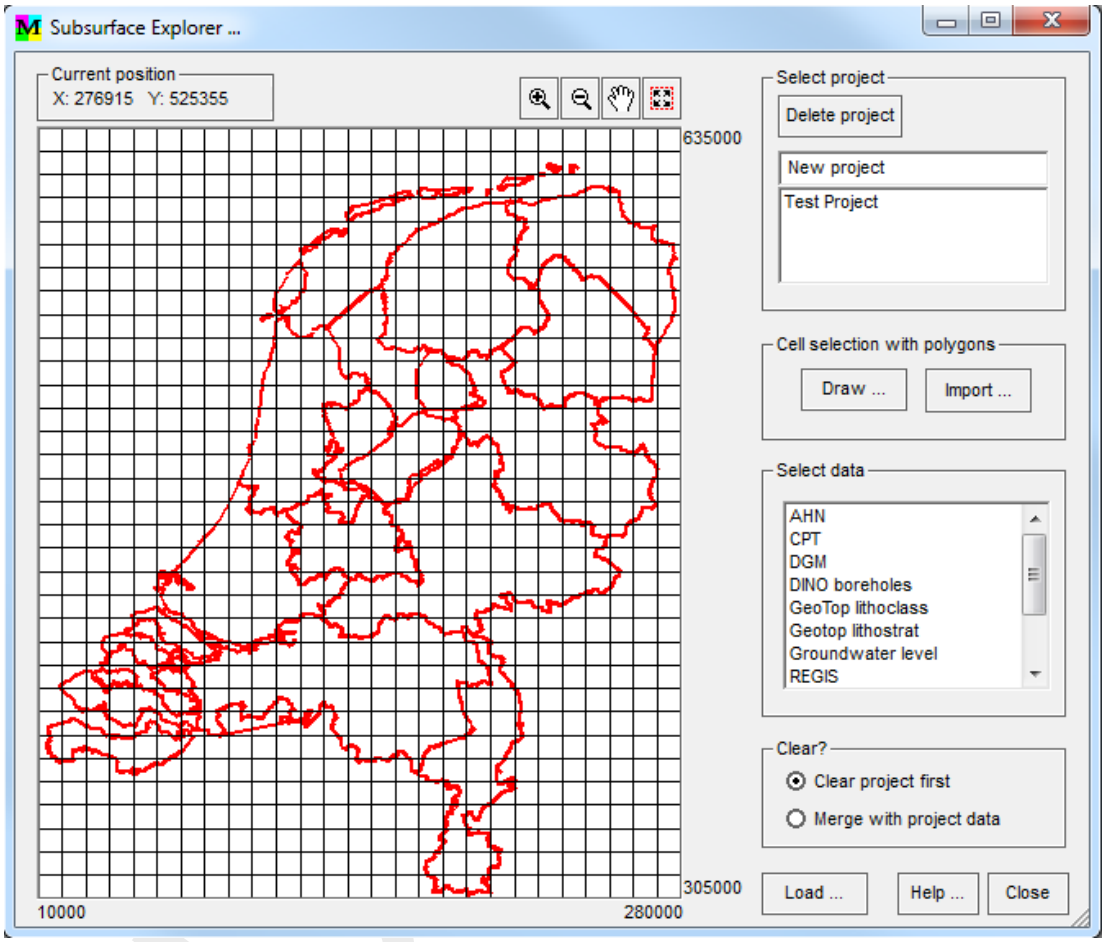
Click the *Define Measurements* to increase or decrease the number of available rows (entry fields) in the table, e.g. 2. There is a maximum of 50 zones, constant values can be entered to denote a zone fore the entire model area, multiply zones can be combined in a single IDF file and/or IPF files can be entered for usage of Pilot Points (section 12.33.4). Click the *Apply* button to accept your entry, click the *Cancel* to ignore any changes, click the *Help* button to start the Help-functionality.

Apply System Settings	Click this button to leave the *Parameter Estimation Settings window* and store the adjustments.
Cancel	Click this button to close the *Parameter Estimation Settings window*
Help ...	Click this button to start the Help-functionality

5.6 Subsurface Explorer

The *Subsurface Explorer* tool can be used to import prepared subsurface data of the Netherlands into iMOD. The data that can be loaded are stored in a database, of which the path has to be specified in the IMOD_INIT.PRF file using the keyword SUBSURFEXDBASE. Furthermore, the path to the 7-zip executable on the users computer also has to be specified in this file using the keyword 7ZIP, see section 9.1 for more information.

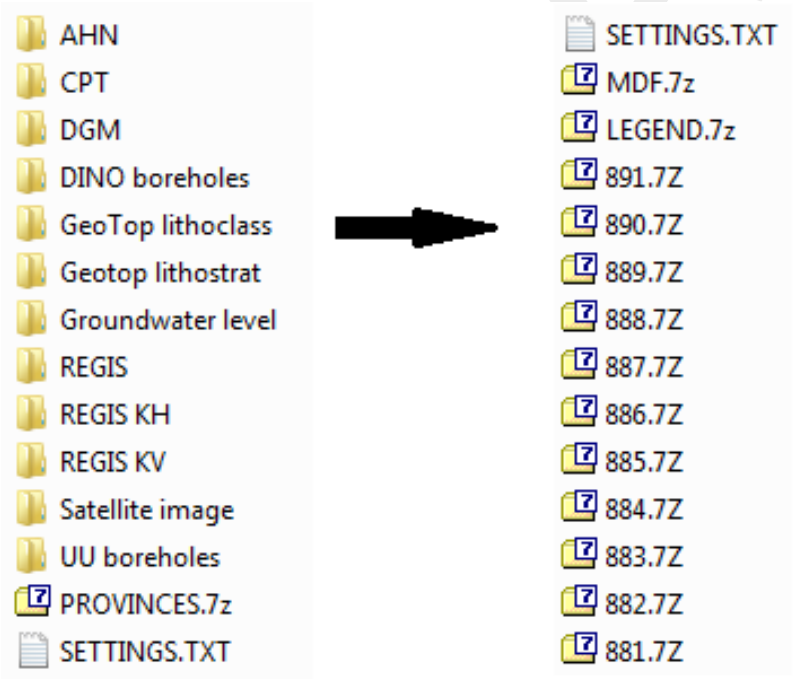
Subsurface Explorer window:



Current Position	Shows the current mouse position on the map in RD coordinates.
	*Navigation buttons for the map* Use these buttons to adjust the view of the map. The buttons can be used to zoom in, to zoom out, to move the map by dragging and to reset the map to the original view, respectively.
	*Delete a project* Select an existing project in the *Select project* listbox and click this button to delete the project.
Create a new project	To create a new project enter a new name in the *Select project* listbox.
Use an existing project	To continue using an existing project select it by clicking the name once in the *Select project* listbox.

Draw ...	*Draw a polygon* Click this button to be able to draw one or more polygons on the map. To draw a polygon, click the left mouse button to place points and click the right mouse button to stop drawing. When the polygon is finished the tool will select the cells corresponding to the polygon.
Import ...	*Import a polygon from file* Click this button to open a *.GEN file containing one or more polygons. The polygons will be drawn on the map and the corresponding cells will be selected.
Select data	This list shows the available data. Select one or more data types from this list you would like to load.
Clear project first	Select this option to first clear the data in an existing project before loading the selected data.
Merge with project data	Select this option to merge the selected data with the data already present in an existing project folder.
Load ...	*Load the data* Click this button to load the current selection. The tool will check whether all required information is present.

Note: The window can be resized by dragging the edges in order to adjust the size of the map. When using polygons to select cells, the data will not be clipped to the polygon, it is only used to select the cells. A *.SHP file can easily be converted to a *.GEN file in order to be able to use it in the *Subsurface Explorer*. Simply open the file in iMOD as an overlay (*iMOD Manager*) and the *.SHP file will automatically be converted to a *.GEN file, which will be placed in the same directory.



The database that contains the data that is imported using the *Subsurface Explorer* tool should be structured as the above figure shows:

- ◇ The data type folders, SETTINGS.TXT and PROVINCES.7Z are stored in the folder to which the IMOD_INIT.PRF file should refer (see section 9.1).
- ◇ The SETTINGS.TXT file contains the minimal X, minimal Y, maximum X, maximum Y and the grid size of the map that is visible on the tool. In this case the map of the Netherlands

is divided into cells using a grid with a cell size of 10 km^2, resulting in a total of 891 cells. An example of this SETTINGS.TXT is:
10000.0
305000.0
280000.0
635000.0
10000.0

- ◇ The map of the Netherlands is drawn using a GEN file stored in PROVINCES.7Z. The data type folders contain a zip file for each cell, containing only the data of that data type for that cell.
- ◇ Furthermore, when applicable, the data type folder can contain a legend in LEGEND.7Z archive, containing an iMOD format legend which will then be used when plotting the data (see section 9.15).
- ◇ For data types that consist of many raster files, e.g. REGIS, an MDF file stored in the database in MDF.7Z, is used to be able to handle these files more conveniently (see section 6.5).
- ◇ Finally each data type folder also contains a SETTINGS.TXT file, in which the extension (e.g. IPF), whether a legend is present (1 for yes, 0 for no) and (only when the data type has the IPF extension) a header which should be used for the IPFs of that data type are given.
- ◇ By using the SETTINGS.TXT files the tool can also be used for other datasets in other geographical regions and new data types can be added easily without editing the source code.
- ◇ The names of the files in the archives containing the data of cells are only the cell number followed by the extension of the file. For example the DINO borehole data of cell 55 are stored in 55.IPF (and corresponding text files). Only raster files have an extended name which also contains one string with information about the layer the file describes, this information can for example be which geological formation the layer describes or at which depth the layer is situated. For example the geotop layer at a depth of 50 centimeters below the surface in cell 55 is named -50_55.IDF. The string on the left of the underscore can contain any character supported by the operating system, except for an underscore, because this character is used to be able to separate the additional information about the layer from the cell number.
  The grid cells are numbered in the following way:

91	92	93	94	95	96	97	98	99	100
...	...	...	...	...	...	...	...	...	...
...	...	...	...	...	...	...	...	...	...
...	...	...	...	...	...	...	...	...	...
...	...	...	...	...	...	...	...	...	...
...	...	...	...	...	...	...	...	...	...
...	...	...	...	...	...	...	...	...	...
...	...	...	...	...	...	...	...	...	...
11	12	13	14	...	...	...	...	...	...
1	2	3	4	5	6	7	8	9	10

- ◇ The data that are loaded are downloaded from the database to the userś computer and are stored in a folder with the name of the project specified using the tool in the IMOD_USER\SUBSURFACE_EXPLORER\directory.
- ◇ The data that are loaded into iMOD using the *Subsurface Explorer* are plotted and shown in the iMOD manager, after which all iMOD functionalities can be used to analyse and edit the data.

Examples of dataset settings-files
Example AHN
IDF
0

This means that the AHN-dataset contains files of the filetype *.IDF and no predefined legend-file (*.DLF) is provided.

Satellite image
PNG
0

In this example satellite images are stored as *.png files and will be available via the iMOD TOPO-tool.

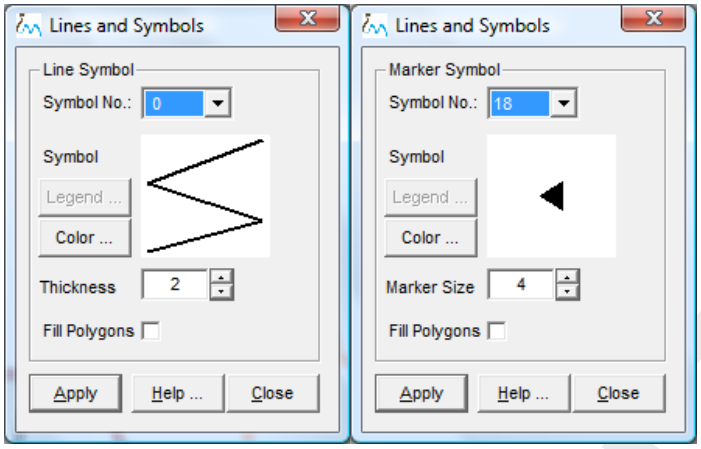
Example DINO boreholes - header file
IPF
1
5
"X-COORDINATE, M"
"Y-COORDINATE, M"
"IDENTIFICATIE"
"MAAIVELD, M+NAP"
"EINDDIEPTE, M+NAP"
3,TXT

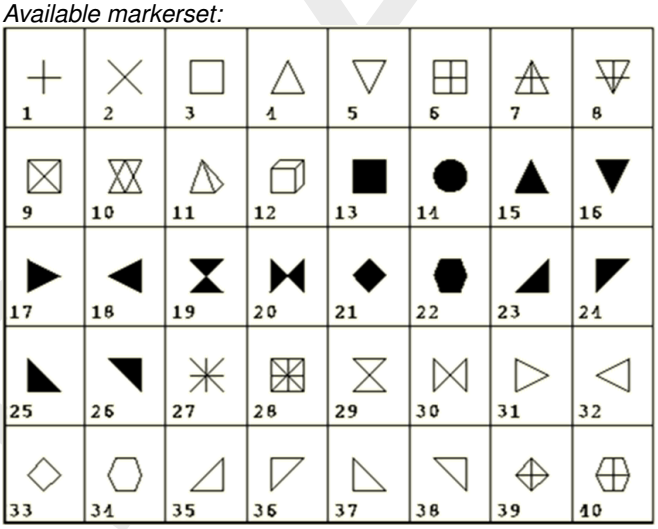
iMOD will merge all IPF-files to one large IPF-file and putting these header lines on top of the final IDF-file. In this example the header lines stand for: 1. filetype, 2. legend provided (1=yes, 0=no), 3. amount of IPF-files to be merged, 4.-8. column names, 9. IPF attribute file types with the needed information given in column 3.

5.7 Lines and Symbols

In the *Overlays* tab of the *iMOD Manager* window, click the *Legend* button to open the *Lines and Symbols* window, left for lines (GENs, IFFs, ISGs), right for points (IPFs).

Lines and Symbols window:



Symbol No.:	Select one of the symbol numbers out of the dropdown menu. For lines (GENs and SHPs) these types vary from solids, to dashed and stippled pattern (1-7). For point data (IPFs) these types vary from between circles, triangles, rectangles and other shape forms (1-40). *Available markerset:* 
Symbol	This field will display the symbol chosen from the dropdown menu *Symbol No.:*.
Color	Click this button to open the default *Colour Selection* window.
Thickness	Enter the value of the thickness of the line (GENs and SHPs).
Markersize	Enter the value of the size for the symbol (IPFs).
Fill Polygons	Select this checkbox to fill in the polygons (GENs and SHPs).
Close	Click this button to close the *Lines and Symbols* window without applying any changes.
Apply	Click this button to apply the configuration and close the *Lines and Symbols* window.
Help	Click this button to start the iMOD Help Functionality.

6 Map Menu options

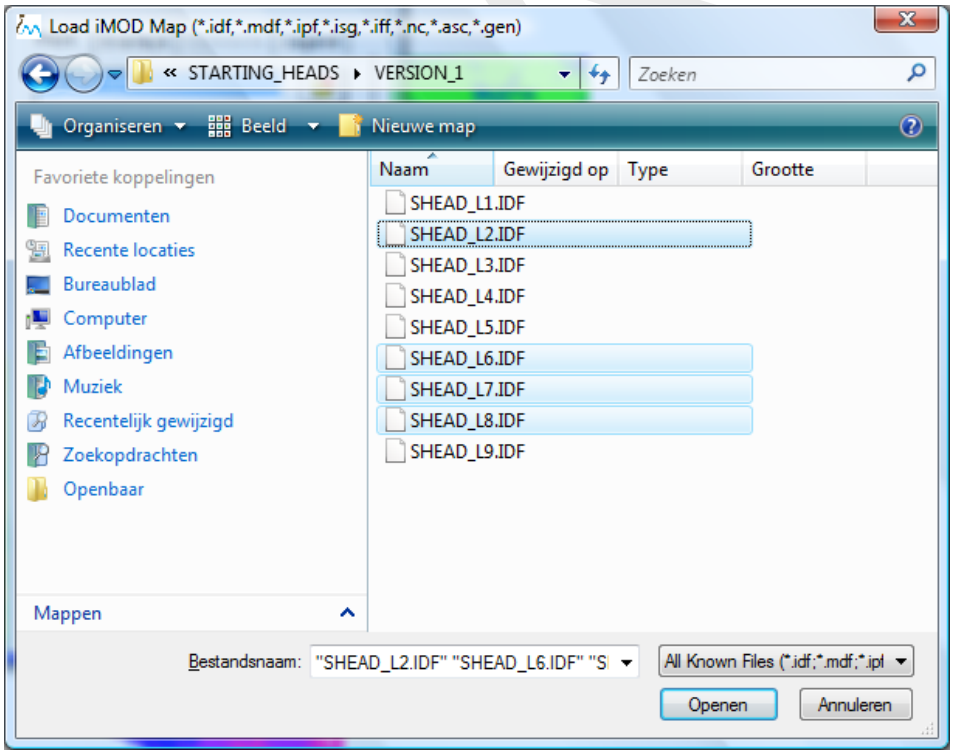
This chapter describes the Map Menu options:

- section 6.1: Add Map.
- section 6.2: Quick Open.
- section 6.3: Map Info.
- section 6.4: Map Sort.
- section 6.5: Grouping IDF Files.
- section 6.6: Legends.
- section 6.7: IDF Options.
- section 6.8: IPF Options.
- section 6.9: IFF Options.
- section 6.10: ISG Options.
- section 6.11: GEN Options.

6.1 Add Map

Click the option *Add Map* from the *Map* menu to open an IDF-, MDF-, IPF-, IFF-, ISG-, GEN- or ASC-file. Alternatively click on the *Add Map* button on the *iMOD Manager*, click on the icon *Open Map* on the toolbar, or use the shortcut F2 on the keyboard to open the *Load iMOD Map* window.

Load iMOD Map window:



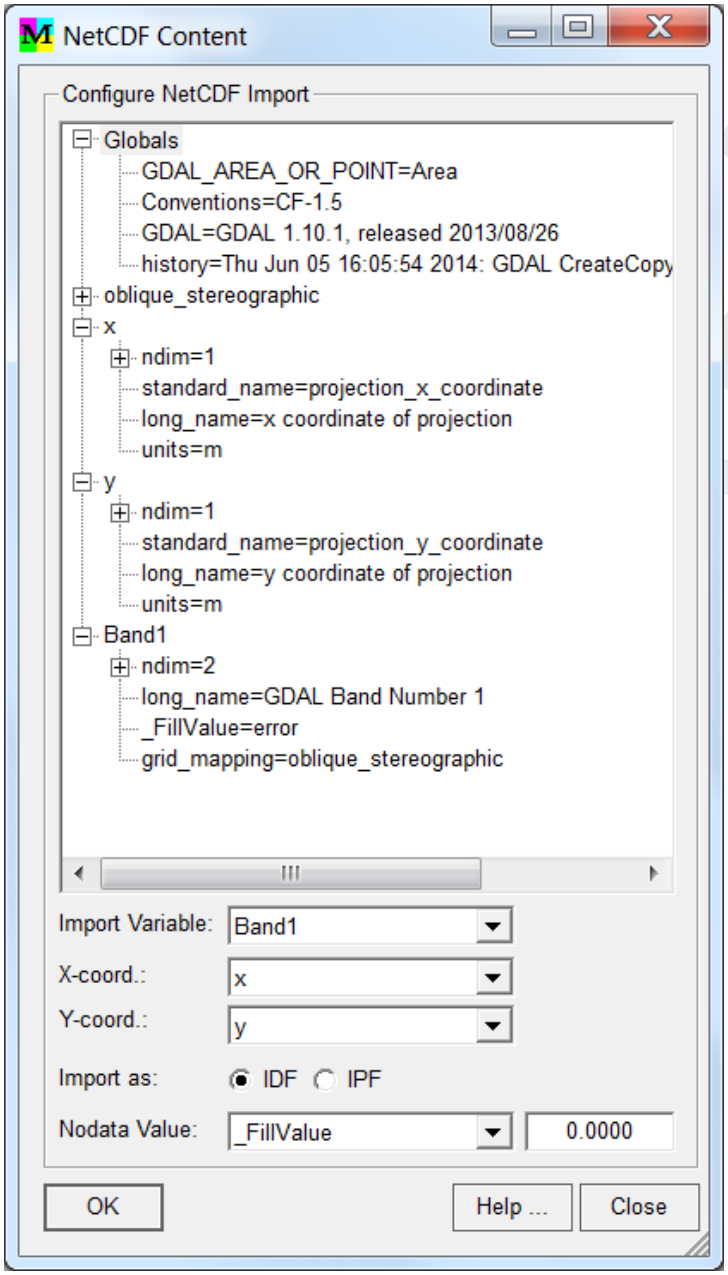
Note: To select multiple files, use the combination Shift-left mouse to select adjacent files or use the Ctrl-left mouse button to select files in any order.

Note: Whenever an ASC-file is opened (see section 9.13 for the syntax of an ASC-file), iMOD will convert these file into IDF format and write them in the same folder. Whenever such a file

exists, you will be asked to overwrite it.

Note: Whenever NC-file (NetCDF) is opened, iMOD will convert these file into IDF format and write them in the same folder. Since, a NetCDF file is general file format iMOD can not convert this type of a file without the interference of the user. Here for, iMOD will display the following window in which the user can specify the correct attributes for the x- and y-coordinate and the actual data block to be converted. **This function is only available in the X32-bits version of iMOD**.

NetCDF Content window:

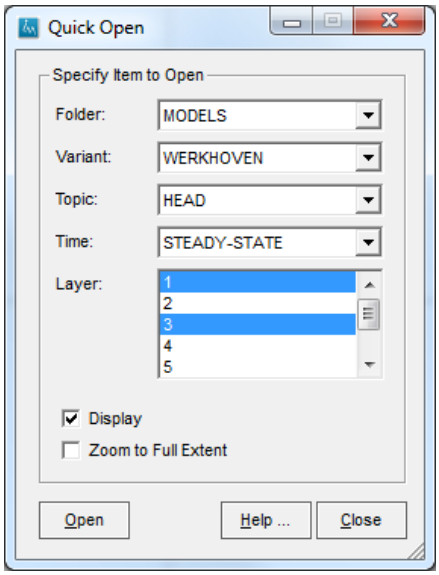


Configure NetCDF Import	This treeview lists the content of the selected NetCDF file. iMOD needs an attribute for the x- and y-coordinates and an actual data block. In the example presented above, these are the attributes *x*, *y* and *Band1*. The x- and y-coordinates (actual the mids of the gridcells) need to be stored in an one dimensional array (ndim=1) and the data block in a two dimensional array (ndim=2). iMOD will compute the cell size that is needed for the formulation of an IDF file, which can be equidistant en non-equidistant.
Import Variable	Select an available variable that is stored by a two dimensional array in the NetCDF, e.g. *Band1*.
X-Coord.:	Select an available variable that is stored by an one dimensional array in the NetCDF for the representation of the x-coordinate, e.g. *x*.
Y-Coord.:	Select an available variable that is stored by an one dimensional array in the NetCDF for the representation of the y-coordinate, e.g. *y*.
Import As:	Select one of the following options to store the converted NetCDF file: ◇ **IDF** Select this option to generate an IDF file; ◇ **IPF** Select this option to generate an IPF file, each point in the IPF file represents the location as described by the selected x- and y-variable;
NoData Value	Select an available variable to represent the *NodataValue* of the data block. iMOD will fill in the corresponding value in the input field to the right.
Close	Click this button to close the *NetCDF Content* window without converting the NetCDF file into an IDF.
OK	Click this button to convert the NetCDF into an IDF file and close the *NetCDF Content* window.
Help	Click this button to start the iMOD Help functionality.

6.2 Quick Open

This functionality offers the ability to search specific iMOD folders for particular IDF-files more quickly than by means of the default windows Explorer. Select the menu option *Map* and then choose *Quick Open* to open the *Quick Open* window.

Quick Open window:

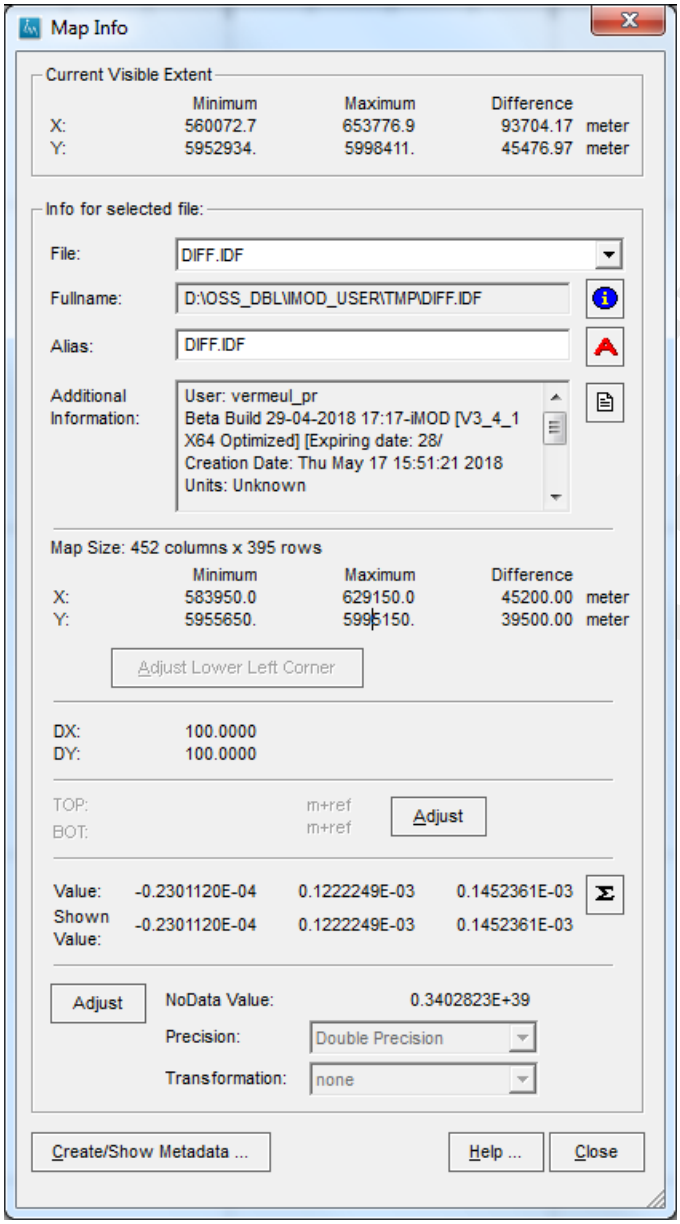


Folder	Select one of the existing foldernames from the dropdown menu. The foldernames MODELS, SCENARIOS and SCENTOOL refer to the corresponding folders below {USER}.
Variant	Displays the existing subfolders below *Folder*. In the situation that the option SCENTOOL is chosen from the *Folder* dropdown menu, it will display the scenario variants of the active scenario opened by the Pumping Tool.
Topic	Displays the existing subfolder below *Variant*. These will be the particular result folders.
Time	Displays all unique IDF-files in the *Folder**Variant**Topic* folder. iMOD will search in this folder for IDF-files that agree with *Topic*_*_L*.IDF.
Layer	Displays all existing layers in the *Folder**Variant**Topic* folder. iMOD will search in this folder for IDF-files that agree with *Topic_Time*_L*.IDF.
Display	Select this option to display the IDF-file(s). Unselect this option to add the IDF-file(s) to the drawing list in the *iMOD Manager* only.
Zoom to Full Extent	Select this option to adjust the zoom level to the maximum extent of the last selected IDF-file.
Open	Click this button to add the selected IDF-file(s) to the *iMOD Manager*.
Help . . .	Click this button to start the Help functionality.
Close	Click this button to close the *Quick Open* window.

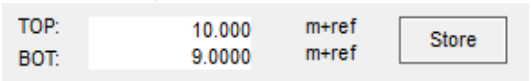
6.3 Map Info

The functionality *Map Info* will display information about the selected map. Select the menu option *Map* and then choose *Info Map* to open the corresponding window.

Map Info window:

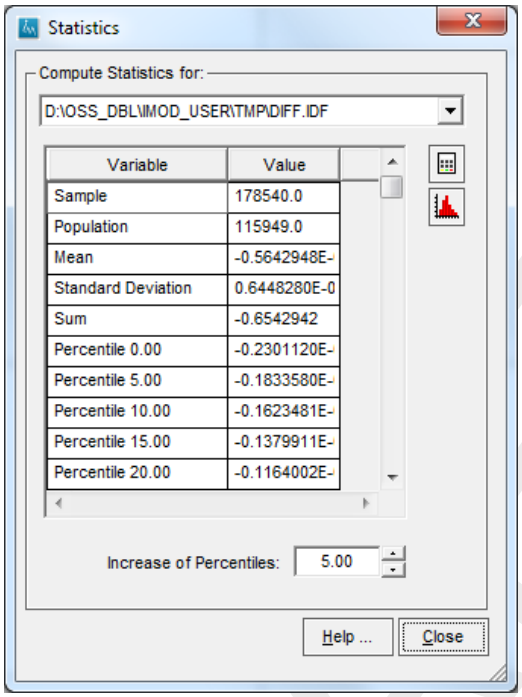


Current visible extent	This group displays the current extent of the graphical window. It shows the minimum and maximum x- and y coordinates and the delta-x and delta-y values (all in meters).
File:	This dropdown menu shows all the files that are opened in the *iMOD Manager*. Click this menu to select the file for which the information should be displayed.
Fullname:	This string field shows the full pathname of the selected map.

	*Map Information* The functionality depends on the selected type of map (IDF, MDF and GEN) and is as follows: ◇ **IDF** Select this option to open the *IDF Edit Table* window, see section section 6.7.4.3; ◇ **MDF** Select this option to open the *MDF Files Sorter* window, see section section 6.5; ◇ **GEN** Select this option to open the *Content of Associated Datafile* window, see section section 6.11.1.
Alias:	This string field shows the alias of the selected map. This name will be used in the *iMOD Manager* as well and offers the possibility to clarify identical file names. Any modification will be saved only whenever the *Rename* button will be clicked.
	*Rename* Click this button to rename the entered *Alias*.
Additional Information	This field shows any additional information that is attached to the selected IDF. For other file types, this field will be showing the string: “No additional information found”.
	*Edit* Click this button to edit the additional information in a regular text editor, e.g. Notepad. Whenever any modifications are saved from Notepad, iMOD will write the renewed additional information in the IDF file, automatically.
Map Size	This string will show the number of rows and columns in an IDF or IPF-file.
X: *Y:*	This field shows for the whole map the minimum and maximum values for the x- and y coordinates and their total delta-x and total delta-y values (all in meters).
Adjust Lower Left Corner	Click this button to allow editing of the lower left corner of the IDF file. Click this button (rename to *Save Adjustments*) again to save the modification in the IDF file, iMOD will adjust the upper right corner accordingly.
DX: *DY:*	For equidistant IDF-files, it shows the width and height of the rows and columns, respectively. For non-equidistant IDF-files, it shows the minimum and maximum dimensions in both directions (all in meters).
TOP: *BOT:*	These will show the internal TOP and BOT values for IDF-files for so called voxel-IDF file. These are used to present the IDF as a horizontal layer in the *Profile Tool* and/or 3*3D Tool* using the TOP and BOT values for the upper- and lower interface. If the values are greyed out, these values are not present in the current selected IDF file.
Adjust	Click this button to edit the TOP and BOT values. Once those values differ (TOP>BOT), the IDF-file will be treated as a voxel in the *3D Tool* and the *Profile Tool*. *Adjusting TOP and BOT values:* 
	Store Click this button to save the adjustments for TOP and BOT.
Value:	Displays the minimum and maximum data values of the IDF or IPF-file and the difference between them
Shown Value:	The minimum, maximum and difference between them of the current values plotted on the graphical canvas. This values represents the current statistics of data presented.

Statistics
Click this button to get the statistics of the selected IDF-file.

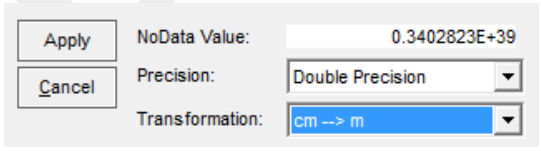
Statistics window:



Compute Statistics for:	Select one of the files from the dropdown menu to compute statistics for.
	*Compute* Click this button to (re)compute the statistics of the IDF-file.
	*Graph* Click this button to get a graph of the percentiles of the IDF-file.
Increase of Percentiles	Enter a value for the increment of the computed percentiles.
Help ...	Click this button to start the Help functionality.
Close	Click this button to close the *Statistics* window.

Adjust

Click this button the adjust the *NoDataValue*, type of *Precision* or internal *Transformation*.



Apply	Click this button to store the adjustments into the IDF.
Cancel	Click this button to cancel any adjustments.
NoData Value:	This field shows the *NoDataValue* of the IDF-file.
Precision:	Select a type of precision for IDF files. By default IDF files are saved in single precision (4 bytes per real/integer number). To maintain a level of accuracy in the representation of number, it could be desirable to change the single- to double precision (8 bytes per real/integer number). Bear in mind that the entire IDF file becomes double in size if switched to double precision.

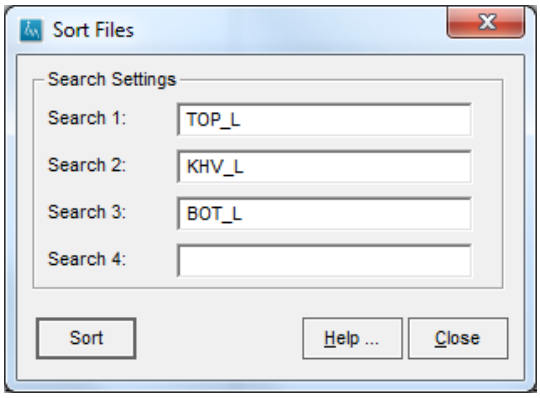
Transformation	Select a type of transformation for the values in the IDF. The data will be transformed internally. The available options are: ◇ **m** → **cm:** transforms meters into centimeters; ◇ **cm** → **m:** transforms centimeters into meter; ◇ **m** → **mm:** transforms meters into millimeters; ◇ **mm** → **m:** transforms millimeters into meters; ◇ **m3/day** → **mm/day:** transforms cubic meter per day into millimeters per day; ◇ **mm/day** → **m3/day:** transforms millimeters per day into cubic meters per day

Create/Show Metadata . . .	Click this button to display the metadata (.MET) file that might be associated to the selected map. For the example, iMOD will try to open the file HEAD_20050501_L1.MET. If the file does not exist, iMOD will create the file. This function is strongly discouraged for IDF files, use the *Edit* button to add additional info to an IDF file instead.
Help . . .	Click this button to start the iMOD Help Functionality.
Close	Hides the *iMOD Manager*. The iMOD Manager can be displayed again by choosing the menu option *View* and then choose the option *iMOD Manager*.

6.4 Map Sort

The functionality *Sort Selected Maps* will offer the possibility to sort/re-arrange selected (IDF) files in the *iMOD Manager* accordingly to a selected type of order. Select the menu option *Map* and then choose *Sort Selected Maps* to expand the following options:

◇ **Sort Using Keywords**
Select this option to sort the selected IDF file in a order that is defined by their individual internal values. E.g., use this option to re-arrange IDF files that describe top- and bottom elevations of interfaces;



Search 1: *Search 2:* *Search 3:* *Search 4:*	Enter a keyword that need to be used to sort the selected IDF files in the *iMOD Manager*. The entered example of three keywords will sort all files that match the following: TOP_L1.IDF, KHV_L1.IDF, BOT_L1.IDF, TOP_L2.IDF, KHV_L2.IDF, BOT_L2.IDF. The number of layers is determined automatically from the selected IDF files in the *iMOD Manager*, these files need to have a number after the character sequence of "_L".
Sort	Click this button to sort the selected IDF files in the *iMOD Manager* according to the entered search strings.
Help . . .	Click this button to start the iMOD Help Functionality.
Close	Closes the *Sort Files* window.

◇ **Sort Alphabetic Ascending Order (A-Z)**
Select this option to sort **any** the selected files upon their filename in an alphabetic ascending order, that is from A up to Z;

◇ **Sort Alphabetic Descending Order (Z-A)**
Select this option to sort **any** the selected files upon their filename in an alphabetic descending order, that is from Z up to A;

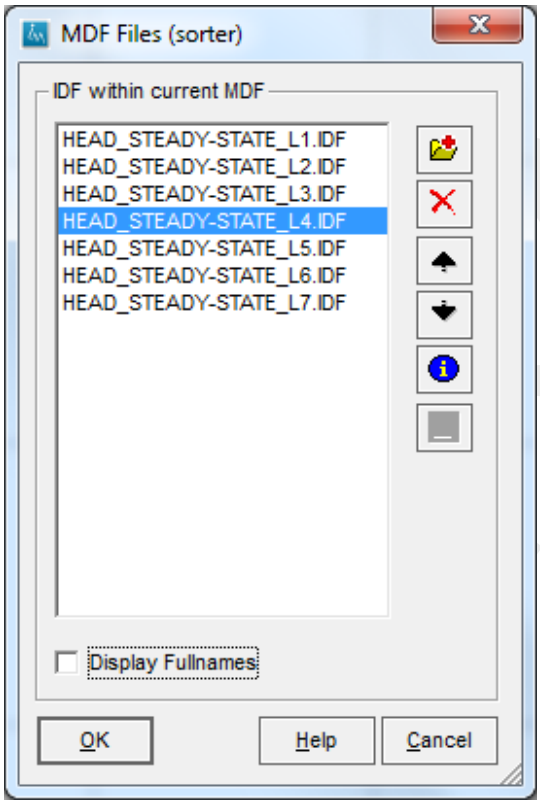
6.5 Grouping IDF Files

Different files in the *iMOD Manager* may be identified, analysed and/or displayed at the same time. For example, you want to analyse the differences between two model simulations and/or combine these with geohydrological cross-sections. To control the number of files in the *iMOD Manager*, it can be helpful to minimize (*grouping*) the number of files in the *iMOD Manager* in to a so-called MDF (Multi-Data-File). Use the following steps to create an MDF file:

1 Select the IDF-files that you want to *Group* in the menu list of the *iMOD Manager*;
2 Select the menu option *IDF Options* from the *Map* menu and choose the option *IDF Group* and enter an MDF file name in the *File Selector* window that pops-up.

After you entered a file name, iMOD will create an MDF-file. An MDF-file lists all the selected IDF-files in one single file. iMOD will remove all files from the *iMOD Manager* and reads in the created MDF file instead. The content of that file can be displayed via the option *Info* on the *Map Info* window. The following window will be displayed.

MDF Files (sorter) window:



	*Open IDF* Click this button to open an IDF-file, it will be added to the MDF-file whenever you click the *Ok* button.
	*Delete* Click this button to remove the selected files from the MDF-file.
	*Up* Click this button to move the selected files one position up in the list.
	*Down* Click this button to move the selected files one position down in the list.

	*Information* Click this button to display the description of the MDF-file.
	*Legend* Click this button to adjust the legend of the selected IDF-file
Display Fullnames	Click this option to display the entire pathnames for the IDF-files.
OK	Click this button to save the adjustments to the MDF-file. After that it will close the *MDF Files (sorter)* window.
Help . . .	Click this button to start the iMOD Help Functionality.
Cancel	Click this button to close the *MDF Files (sorter)* window without any changes.

To ungroup a MDF-file, select the menu option *Map* and choose the option *IDF Options* and then the option *IDF Ungroup MDF*.

Note: The IDF-file that is selected in the MDF-file will be used to plot on the graphical canvas. If multiple files are selected, only the first will be plotted though. The properties of the IDFs are known as all attributes (legend, cross-section types, colours, aliases) of the IDFs are copied into the MDF-file too.

Note: The order in which the IDF-files are listed is the same order as which they appear in the *Map Value*, *Cross-Section Tool* and *3DTool*. Moreover, MDF-files will be displayed in graphs separate from the files that are not in the MDF-file(s), see section 6.8.3.2.

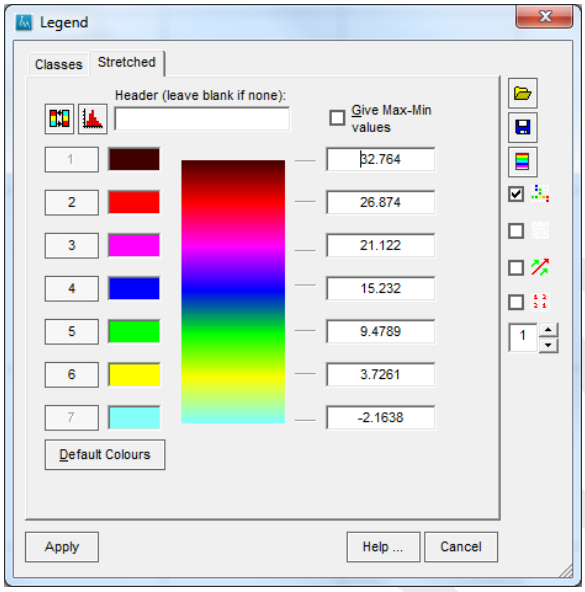
6.6 Legends

This section describes the usage of legends.

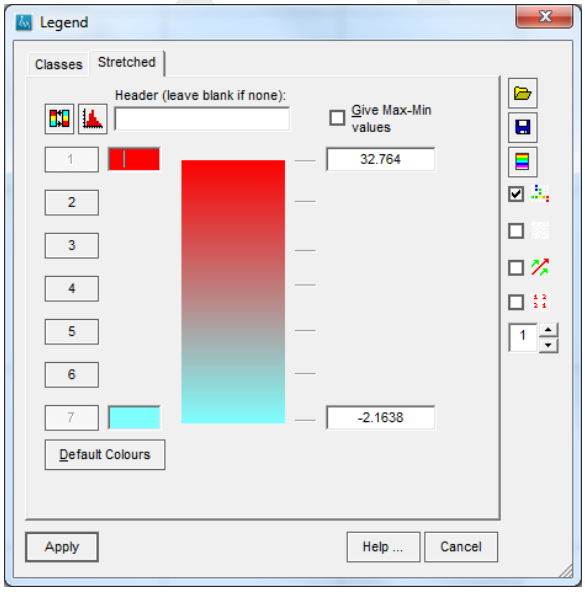
6.6.1 Adjust Legends

A legend is assigned to a map (*.IDF, *.IPF, *.IFF, *.GEN, *.MAP). Make sure you select one of them in the *iMOD Manager* to activate the *Legend* button on the *Maps* tab of the *iMOD Manager*. Alternatively you can click the right mouse button anywhere on the canvas and select the option *Legend* from the pop-up menu and then choose the option *Adjust Legend*. In both cases, the *Legend* window will appear, two examples of the *Stretched* tab on the *Legend* window, left using all color gradients (7 gradients), right using the first and last only (one gradient).

Legend window, Stretched Tab using all color gradients:



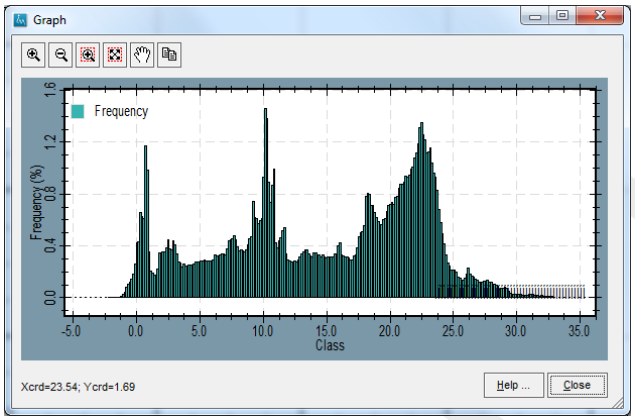
Legend window, Stretched Tab using two color gradients:



Flip Colours
Click this option to “flip” the colour sequence, e.g. red becomes blue and blue becomes red.

Histogram
Click this button to display a frequency distribution of the current legend classes. Move your mouse in the graph to show the value (X-crd) and the frequency in % (Y-crd).

Graph Window (with a Frequency Distribution for 250 stretched legend classes):



Zoom In
Click this button to zoom in at the position of the mouse cursor, repeatedly. Right click the mouse to stop.

Zoom Out
Click this button to zoom out at the position of the mouse cursor, repeatedly. Right click the mouse to stop.

Zoom Window
Click this button to zoom into a drawn rectangle. Left click the mouse to define the first corner of the rectangle. Left click again for the second corner. Right click to cancel the zoom operation.

Zoom Full
Click this button to adjust the zoom level to the full extent of the graph.

Move
Click this button to move the graph. Keep the left mouse button pressed and drag the mouse cursor to move the graph.

Copy to Clipboard
Click this button to copy the graph onto the clipboard of Windows. Paste the image into e.g. Word by the *Ctrl-V* key combination

Xcrd=
Ycrd= Display of the coordinates of the current mouse position.

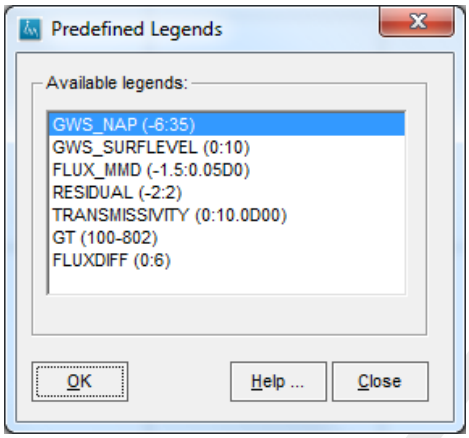
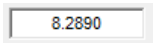
Help... Select this button to start the Help functionality.

Close Select this button to close the Graph window.

Header Enter a descriptive text for the corresponding legend. The text will be plotted on top of the legend whenever the legend is plotted on the graphical canvas. Leave the input field empty to ignore any legend header.

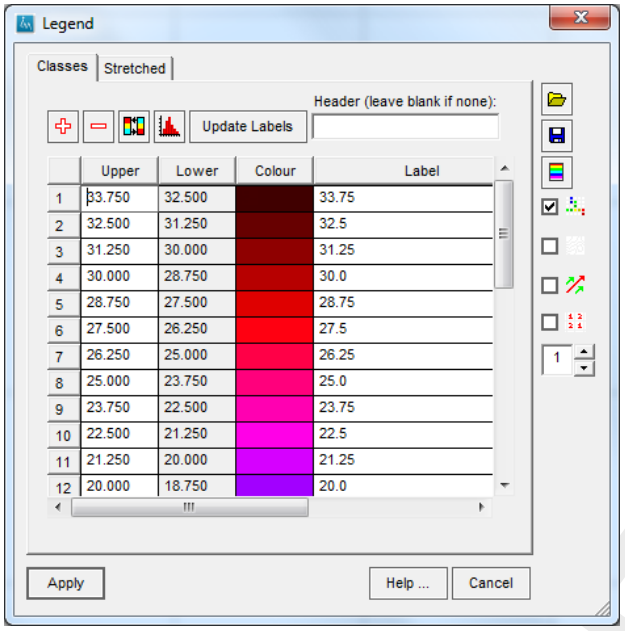
Open Legend File
Click this option to open an existing *.LEG-file for the syntax.

Save Legend File
Click this option to save the current legend in a *.LEG-file.

	*Predefined Legends* Click this option to select a predefined legend. Select one of the available legends and click the OK button to read the selected legend into the Legend window. *Predefined Legends:* 
	*GridCells* Click this option to display the IDF-file with filled gridcells.
	*ContourLines* Click this option to display the IDF as contourlines.
	*Flow direction* Click this option to display arrows indicating the direction of flow.
	*Data Numbers* Click this option to display the actual data on the mids of the selected IDF file. The size of the text is given by the entered *Line thickness*.
	*Line thickness* Click this option to set the line thickness of the contours and the direction of flow arrows. If you specify a line thickness of < 0, no labels will be plotted and the line thickness becomes the absolute given thickness, e.g. a line thickness of -2 means a line thickness of 2, without any label plotted.
	*ColorMark* Select the checkbox (2-6) to turn on/off the corresponding colour in the legend colour ramp. Clicking on the coloured field (red in this case) will show the default Colour window wherein you can specify another colour.
	*Class* Enter a class that corresponds to the colour, in this case the value *8.2890* corresponds to the red colour in checkbox *2*.
Give Min-Max values	Select this checkbox to enter a minimum and maximum value. As a result only the top and bottom input fields will be available to enter values.
Default Colours	Click this option to reset the current colours for the default colours.
Apply	Click this button to apply the legend setting to the current selected map file (IDF, IPF, IFF or GEN).
Cancel	Click this button to close the *Legend* window without applying any changes to the current legend settings.
Help	Click this button to start the iMOD Help Functionality.

Note: A stretched legend contains always 255 colours and classes. Whenever a legend contains 50 classes or less, the *Classes* tab will appear instead.

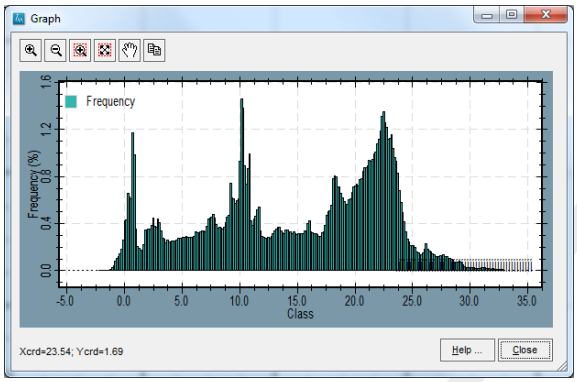
Legend window, Classes tab:



Upper	The first column shows the *Upper* limit for each class, enter different values if desired.
Lower	The second column shows the *Lower* limit for each class. It will be filled in automatically based on the filled in values for the *Upper* limit. The *Lower* limit should be filled in for the last record only.
Color	The third column displays the colour for the class. You can click on the colour to display the Colour window. The colour will be used to colour the values less than the *Upper* class and greater or equal to the *Lower* class.
Label	The fourth column shows the label that can be displayed in the *Legend* tab on the *iMOD Manager* window and/or plotted on the canvas. The label can be entered with a maximum of 50 characters.
Freq. (%)	The fifth column shows the frequency of occurrence of values within each class. It is only applicable to IDF-files.
	*Insert a Row* Click this button to insert a row in the table. It will be inserted one row above the row you select.
	*Delete a Row* Click this button to delete the selected row.
	*Flip Colours* Click this option to “flip” the colour sequence.

Histogram
Click this button to compute the frequencies of IDF values within each class of the current legend, a histogram is plotted, see above for an explanation). The computed values are temporary written in the fifth column. Once the *Legend* window has been closed, the values are removed.

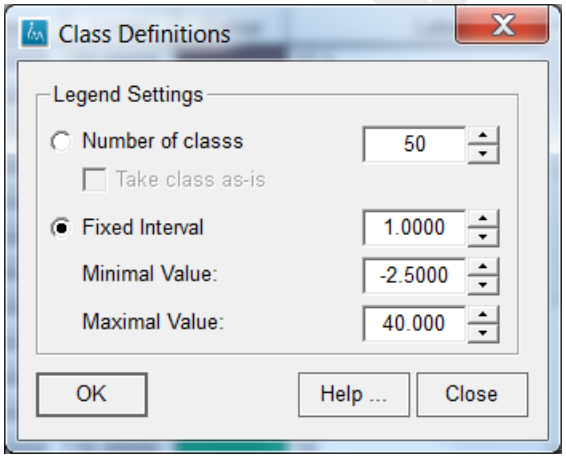
Graph window:



Update Labels	Click this button to update the labels in the *Label* column. These values will be used whenever the legend is plotted on the graphical canvas.
Header	Enter a descriptive text for the corresponding legend. The text will be plotted on top of the legend whenever the legend is plotted on the graphical canvas. Leave the input field empty to ignore any legend header.

Note: The maximum number of classes in a Classed Legend is 50. However, less is often desired. You are requested to specify beforehand the number of classes you want. When switching from the *Stretched* tab to the *Classes* tab, the following window will appear.

Class Definitions window:



Number of classes (1-50)	Enter the number of classes for which the stretched legend will be transformed.
Take class as-is	Select this option to sample from the classes as specified in *Streched* legend. If deselected, iMOD will try to round the class-interval to *nice* numbers that might - however - yield less classes.

Fixed Interval	Select this option to generate a legend with a fixed interval, e.g. 1.0 in between the specified *Minimal Value* and *Maximal Value*. iMOD starts at the minimal value and add another class with the specified interval to a maximum number of 50 classes or less if the maximal values is reached prior to that.
Minimal Value:	Enter a minimal value for the legend, e.g. -2.5.
Maximal Value:	Enter a minimal value for the legend, e.g. 40.0.
Ok	Click this button to continue to the *Classes* tab of the *Legend* window.
Close	Click this button to close this *Class Definition* window and return to the *Stretched* tab of the *Legend* window..
Help	Click this button to start the iMOD Help Functionality.

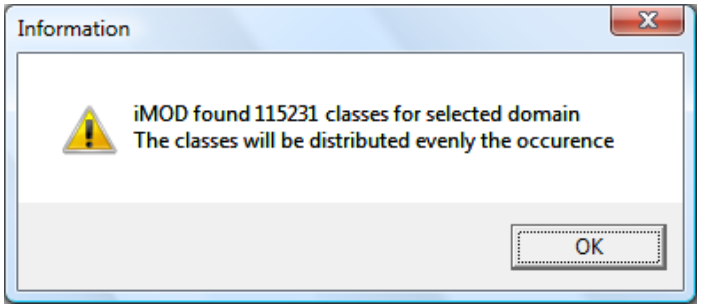
6.6.2 Generation of Legends

All active maps are accompanied by a legend. IDF-files are drawn standard by a legend, as well as IPFs, IFFs, ISGs and GENs. How to specify a legend is explained in sections 3.4.5, 4.2.1, 4.3.1, 4.4.1 and 4.5.2, respectively. However you can let iMOD assign classes and colours also. In that case you should select the respective **-options* under *Map,* or alternatively you can press the right mouse button anywhere on the canvas to see the following options:

- ◇ **Current Zoom Level**
  - □ **Percentiles**
    Click this option to build a non-linear legend based on the distribution of values in the selected IDF's for the current zoom level of these files (max. 2000 points).
  - □ **Linear**
    Click this option to build a linear legend based on the minimum and maximum values in the selected IDF's for the current zoom level of these files.
  - □ **Unique Values**
    Click this option to build a legend with unique values that appear in the selected IDF's for the current zoom level of these files (should be less or equal to 50).
- ◇ **Entire Zoom Extent**
  - □ **Percentiles**
    Click this option to build a non-linear legend based on the distribution of values in the selected IDF's for the entire zoom extent of these files (max. 2000 points).
  - □ **Linear**
    Click this option to build a linear legend based on the minimum and maximum values in the selected IDF's for the entire zoom extent of these files.
  - □ **Unique Values**
    Click this option to build a legend with unique values that appear in the selected IDF's for the entire zoom extent of these files (should be less or equal to 50).

Note: Whenever the number of unique classes exceeds the maximum of 50, iMOD will distribute the original number of classes to fit the maximum of 50. It will take the frequency of the original classes into account, such that the frequencies of the renewed classes are evenly distributed.

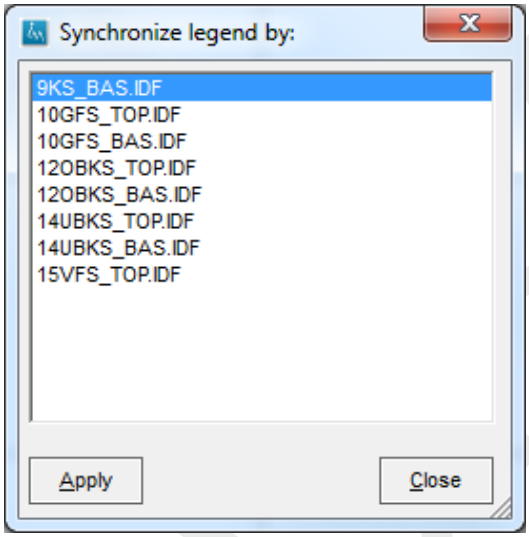
Information window:



6.6.3 Synchronize Legends

Whenever more than one map is selected in the *iMOD Manager* then the option *Synchronize Legends* becomes available to display the *Synchronize Legends* window. You can open this window by selecting the menu option *Map* and the option *Legend* or alternatively the same menu options from the popup menu whenever you click your right mouse button anywhere on the canvas.

Synchronize Legend by window:

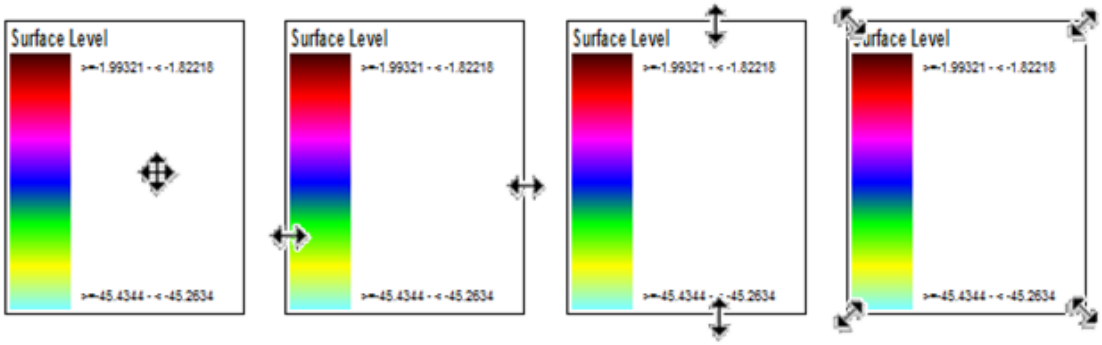


List menu	The menu list displays all the selected IDF-files from the *iMOD Manager*. The legend from the one that is selected (in this case the PWTHEAD_19890114_L1.IDF) will be used to be copied to the others.
Apply	Click this button to synchronize the listed IDF-files to the selected IDF and close the *Synchronize Legend* window.
Close	Click this button to close this *Synchronize Legend* window without changing any legend.

6.6.4 Plot Legends

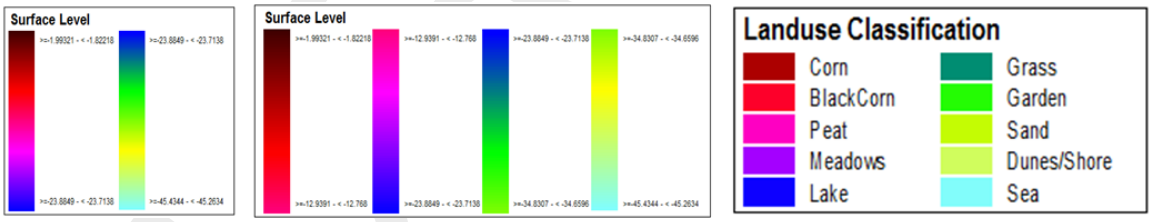
Right-click your mouse button anywhere on the canvas to select the menu option *Legend* and then choose the option *Plot Legend on Map* to plot the legend of the last drawn Map on the canvas.

How to move the legend:



The first picture shows that you can move the legend whenever the mouse symbol changes in a cross. The 2^{nd}, 3^{rd} and 4^{th} show that you can increase and decrease the size of the legend whenever the arrows appear (near the legend boundaries).

Whenever you right click the mouse button, you can specify the number of columns (*1-5*) that are used to display the legend, by selecting those in the popup menu *Legend Columns*. Whenever the legend contains less or equal 50 classes, the legend will be plotted such, that you can distinguish all the classes.



The left two pictures show a legend for two and four columns (both 255 classes) and the right picture shows a legend with 10 classes.

To remove the legend from the map, deselect the *Plot Legend on Map* option again.

Note: The size of the text will increase and decrease whenever the size of the legend area is changed. So, to increase the textsize, you should increase the area for the legend.

The following sections contains detailed descriptions of the Map options for different files that iMOD supports.

6.7 IDF Options

iMOD supports several basic functionalities that manipulate IDF-files:

- **IDF Value**,
- **IDF Export,**
- **IDF Calculator**, use this tool to apply simple algebra, rescale the IDF and/or merge several IDF-file into a single one.
- **IDF Edit**, use this tool to select areas for which computations are carried out. These can be simple algebraic computations and/or smoothing and/or interpolation.

6.7.1 IDF Value

WHY?

IDF-files are raster files with (non)-equidistant rastersizes. Interactive inspection of the raster with the *IDF Value* option is a quick and easy way to check the raster values.

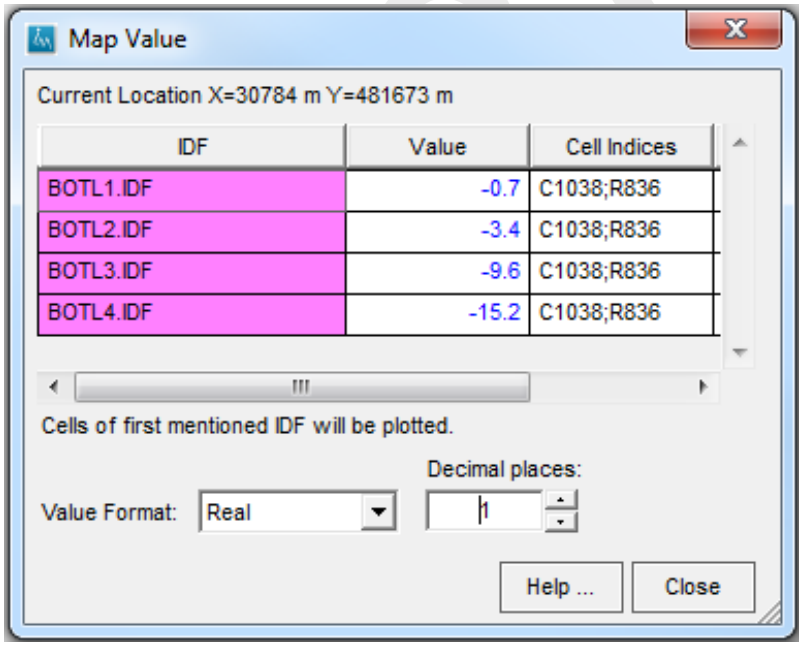
WHAT?

IDF Value allows you to view particular rastercell values for a single IDF-file, or multiple IDF-files. iMOD will read the IDF values underneath the current mouse position.

HOW?

In the *Map* tab of the *iMOD Manager* window, click the *Map Value* () button to open the *Map Value* window. Otherwise, use the shortcut F3 or right-click anywhere on the canvas to open the popup menu. Select the option *IDF Options* and then choose *IDF Analyse*. Or alternatively, select the menu option *Map*, choose the option *IDF Options* and then *IDF Analyse*.

Map Value Window:

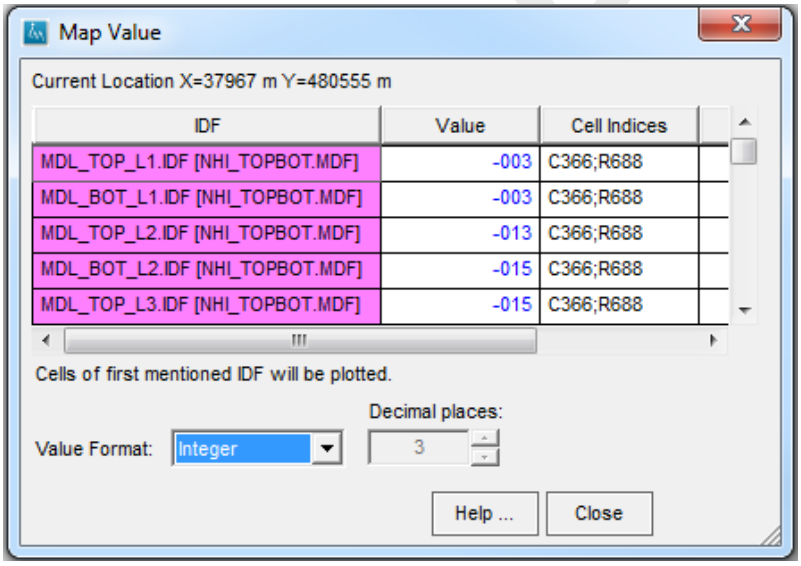


Current Location	This field shows the current location of the mouse on the canvas. Press the LEFT mouse button to freeze the current location. Restart inspection over the canvas by pressing the LEFT mouse button again.
IDF	This column in the table shows the IDF selected in the *iMOD Manager* and part of the inspection with *MapValue*.

Values	This column shows the value of the IDF (column 1) for the current location. Values that are greater than zero will be coloured red, less than zero become blue.
Transf.	This column shows the transformation that could take place, see for more information about this the syntax of the IDF-file and how to apply a transformation.
Cell Indices	This column shows the column and row number of the selected IDF-files, e.g. *Cell Indices=C32;R12* to represent column 32 and row 12 respectively. There is no need that the inspected IDF-files have identical dimension and/or raster discretization. Whenever the mouse is positioned outside the limits of the IDF, *Cell Indices=Outside*. In case the inspector works with a rectangle/polygon, the *Cell Indices* will show the minimum and maximum values for the column and row numbers that are within the rectangle/polygon, e.g. *Cell Indices=C40-32;R32-54*.
Value Format:	Choose the preferred value format from the dropdown menu. You can choose from: Real (e.g. 2.345), Integer (e.g. 002) and Scientific notation (0.2345E+01).
Decimal places:	Change this value to set the total amount of decimal values (values behind the point), the possible range is 0-15.
Help . . .	Click this button to start the iMOD Help Functionality.
Close	Click this button to close the *MapValue* window. Alternatively the *MapValue* window can be closed by clicking the RIGHT mouse button anywhere on the canvas.

Note: All selected files in the *iMOD Manager* will be listed in the *Map Value* window, however, IPFs, IFFs, GENs and ISGs will be left out automatically. There is no need to deselect them before clicking the *Map Value* button. Whenever a MDF-file is selected, all IDF-files within the MDF-file will be displayed and also the MDF-filename.

Map Value window filled with items from a MDF-file:



Note: The menu options *Move, Zoom In, Zoom Out, Zoom Full* and *Zoom Rectangle* are available during the *Map Value* exercise.

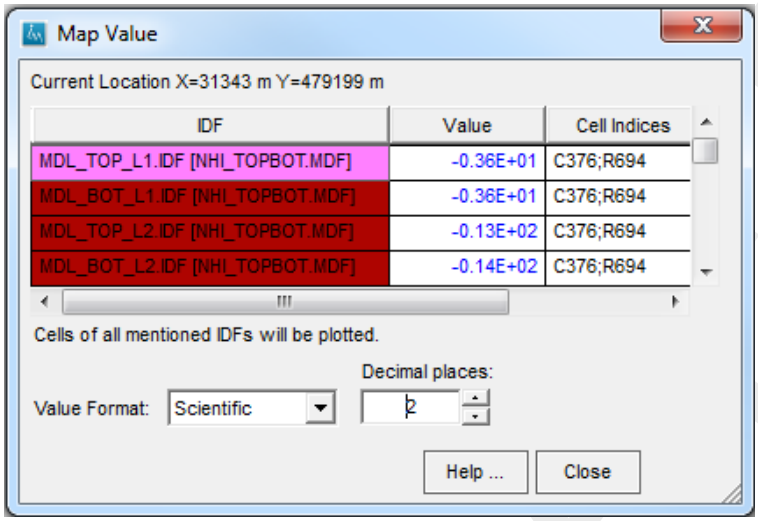
Note: Click the left mouse button to stop hovering over the graphical display. It will freeze the values in the *Map Value* window. Start hovering again by clicking the left mouse button again, terminate the entire functionality by clicking the right mouse button.

On default, the *Map Value* window will operate as a point inspector, in other words, the value will be read for the current position of the mouse. Click the menu option *Map*, the option *IDF*

Options and then the option *IDF Analyse* to display the following options to alter this:

- **Plot No Location**
  Check this item whenever no rastercells of the selected IDF-file need to be displayed.
- **Plot All Locations**
  Check this item whenever the rastercells of all selected IDF-files need to be displayed. Bear in mind that the performance will slow down whenever many IDF-files are included, and if IDF-files with non-equidistant rasters are included. Whenever this option is checked, all values in *Map Value* will be coloured differently.

  Map Value window in Plot All Locations mode:



- **Plot First Location Only**
  Check this item whenever the rastercells of the first IDF-file listed in the *Map Value* table, need to be displayed. This is the default.
- **Points**
  Check this item whenever the values for the current location of the mouse need to be listed. This is the default.
- **Rectangle**
  Check this item whenever the values need to be summed within a rectangle that you can draw. Use the left mouse button to locate the first position of the rectangle and the left/right mouse button to stop and close the *Map Value* window.
- **Polygon**
  Check this item whenever the values need to be summed within a polygon that you can draw. Use the left mouse button to locate the first position of the polygon and continue to add more points (as desired) to complete the polygon. Use the right mouse button to stop and close the *Map Value* window.
- **Circle**
  Check this item whenever the values need to be summed within a circle that you can draw. Use the left mouse button to locate the first position of the circle and expand the size of the circle while moving the mouse pointer away from the first position (centre of the circle). Click again on your left mouse button to stop and close the *Map Value* window.

6.7.2 IDF Export

WHY?

IDF-files have a specific format. The export to a format readable in other software applications makes it possible to use the IDF-files outside iMOD.

WHAT?

The IDF-files can be exported in several formats.

HOW?

Choose the option *Map* from the main menu, choose *IDF Options* and *then IDF Export* to display the following menu options:

- **ESRI ASC Format**
  Check this option to export the IDF into an ESRI ASCII format. Bear in mind that this ESRI ASCII format does not support non-equidistant cell sizes. Therefore, the export will result always in an equidistant ASCII file, see section section 9.13 for the exact syntax of an ESRI-ASCII file. Any ASCII file can be read in again in iMOD to convert is back to an IDF file, see section section 6.1.
- **NetCDF Format**
  Check this option to export the IDF into the NetCDF3 format. Any NetCDF file can be read in again in iMOD to convert is back to an IDF file, see section section 6.1. **This function is only available in the X32-bits version of iMOD**.

For both export formats the following options are available:

- **Export Total Extent**
  Click this option to export the current IDF for its total extent.
- **Export Current Extent**
  Click this option to export the current IDF for the current extent of the graphical window.
- **Export Given Extent**
  Click this option to export the current IDF for an extent to be entered.

6.7.3 IDF Calculator

WHY?

Arithmetic operations enable the creation of new IDF-files.

WHAT?

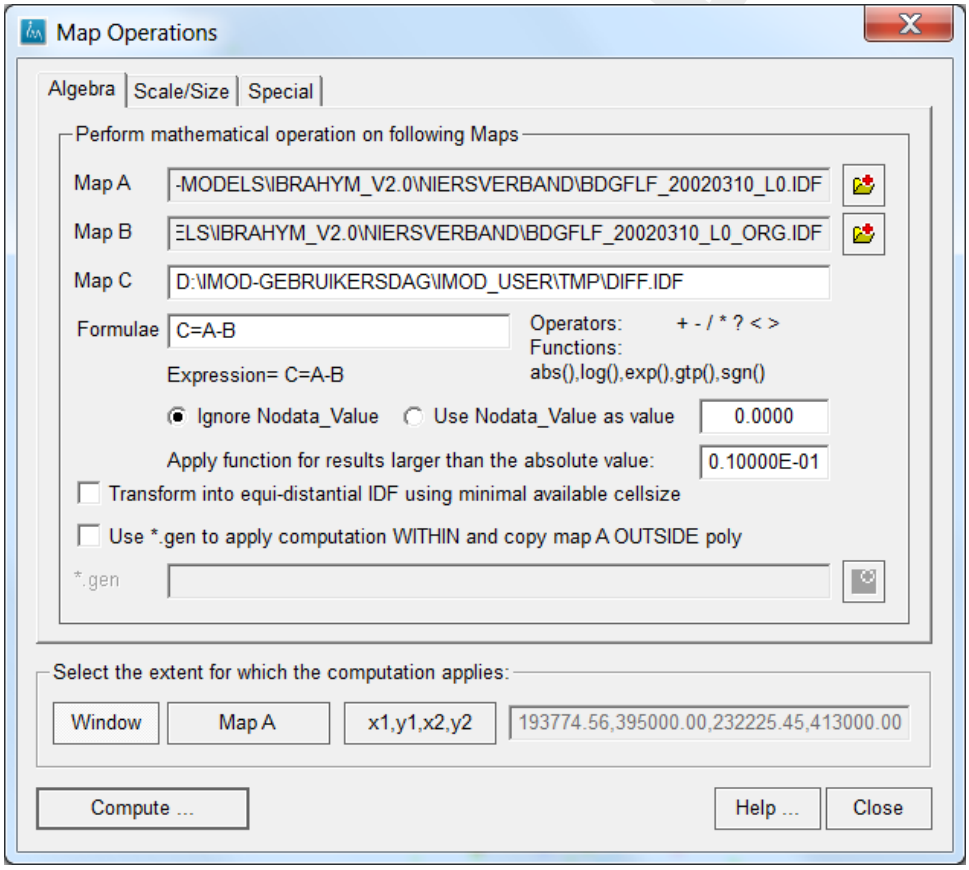
The *IDF Calculator* works with IDF-files as input and enables to:

- ◇ calculate a new IDF-file using an arithmetic expression on one or two IDF-files (the *Algebra* tab)
- ◇ change the rastersize of an IDF-file by upscaling or downscaling (the *Scale/Size* tab)
- ◇ merge a selected number of IDF-files into one IDF-files (the *Special* tab).

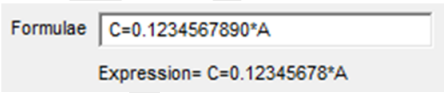
HOW?

The *IDF Calculator* can be displayed at any time from the *iMOD Manager*, click the *Calculator* button and/or use the *Map* option from the main menu (or the popup menu that displays whenever you click your right mouse button on the graphical window), choose *IDF Options* and then the option *IDF Calculate*. The following *Map Operations* window will be displayed.

Map Operation window, Algebra tab:

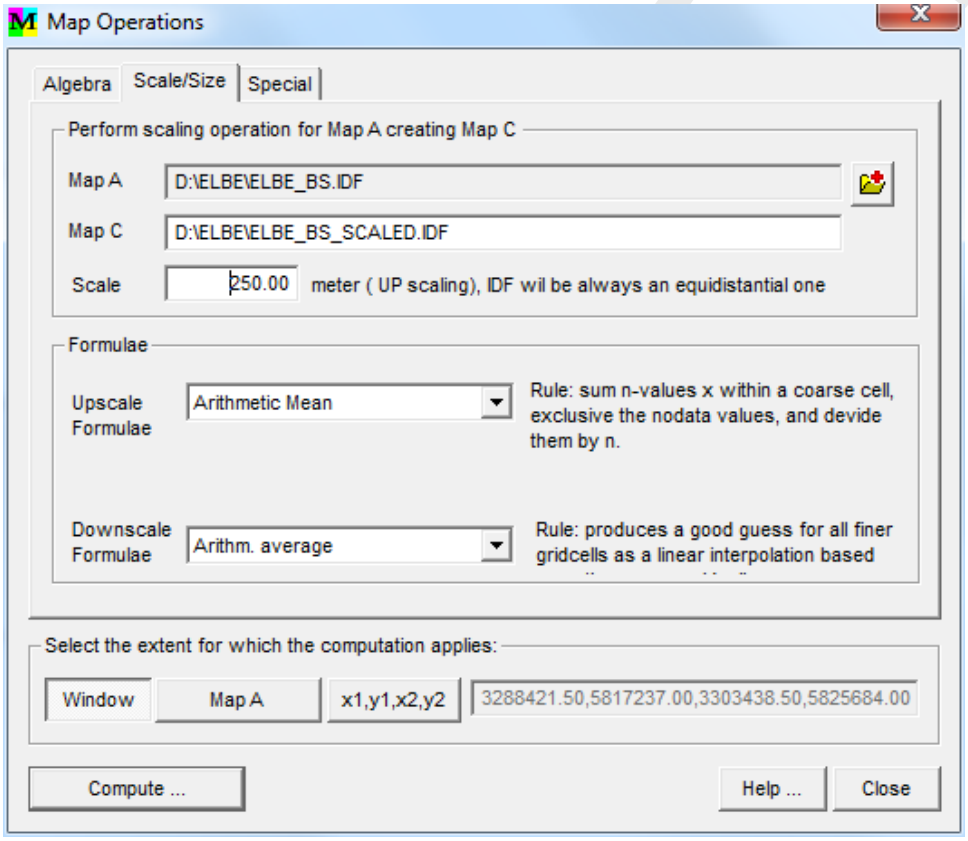


Map A	Displays the first IDF. On default it will display the first selected IDF from the *iMOD Manager*, click the button *Open IDF* to open a different one.
Map B	Displays the second IDF. On default it will display the second selected IDF from the *iMOD Manager*, click the button *Open IDF* to open a different one.
Map C	Enter the name of the resulting IDF. On default it will use the name DIFF.IDF that will be saved in the *TMP*-folder of the *USER*-folder.

	*Open IDF* Click this button to select an IDF-file from a folder
Formulae	Enter the formulae to be used. The syntax of the formulae is that the IDF-names entered as *Map A*, *Map B* and *Map C* are represented by the characters *A*, *B* and *C*, respectively. The example *C=A-B* means that the values of *Map A* will be subtracted by the values in *Map B*. The following operators are available in the formulae: ◇ "+" : adds values ◇ "-" : subtract values ◇ "/" : divides values ◇ "*" : multiplies values ◇ "?" : takes the first *NoDataValue* found in Map A then Map B ◇ "<": takes the smallest from both values between Map A and Map B ◇ ">": takes the largest from both values between Map A and Map B The following functions are available in the formulae: ◇ *ABS()* : takes the absolute value of the expression between brackets; ◇ *LOG()* : takes the logarithmic value of the expression between brackets; ◇ *EXP()* : takes the exponent of the expression between brackets; ◇ *GTP()* : computes the groundwaterclass ('grondwatertrap'). Enter a mean highest groundwaterlevel for Map A and the mean lowest groundwater level for Map B.
Expression	The *Formulae* entered is translated into an *Expression* which will be used, eventually. The expression can be used to check whether the formulae has been filled in correctly. **Note:** It is possible to include constant values in the *Formulae*, e.g. [C=1.5*A] which means that all values in *Map A* will be multiplied with a factor of 1.5. Be aware that the number should come before Map A as in the formulae [C=A*1.5] the constant value is ignored. This can be extended to another factor to be used for Map B, e.g. [C=1.5*A-0.5/B]. It should be noticed that these constant value are rounded to a single precision real (8 digit number), so the entered value *0.1234567890* becomes *0.12345678* in the *Expression*. *Example of the Formulae:* 
Ignore NoData_ Value	Check this option to ignore the *NoDataValues* in the IDF-files. On default cells equal to the *NoDataValue* of the IDF will be excluded in the computation.
Use NoData_ Value as value	Check this option to use the *NoDataValues* in the computation. Specify the value to replace the *NoDataValue* of the IDFs. On default the value 0.0 is used.
Apply function for . . .	After the calculation, this option gives values smaller than given/entered absolute value a nodata value, e.g. C=A-B if C<0.1 C=nodata.
Transform into . . .	Check this option to force that the resulting IDF (*Map C*) will have equidistant cellsizes. Whenever any of the selected IDF-files (*Map A* and/or *Map B*) is an IDF-file with non-equidistant cellsizes, the smallest cellsize in one of these will be used to determine the cellsize of *Map C*.
*Use *.gen*	Check this option to force that the results of Map C will be computed inside the polygons that are defined in the entered *.GEN-file (**.gen*).
**.gen*	Enter the *.GEN-file with polygons

	*Open File* Click this button to select a *.GEN file
Select the extent for which the computation apllies:	Click one of the following options: ◇ *Window*, to force that the results of Map C will be computed for the current ZoomLevel only; ◇ *Map A*, to force that the results of *Map C* will be computed for the extent of *Map A*. ◇ x1,y1,x2,y2, to enter the coordinates of a specific window
Compute...	Click this button to start the computation, it closes the *Map Operations* window afterwards.
Help ...	Click this button to start the iMOD Help Functionality.
Close	Click this button to close the *Map Operations* window.

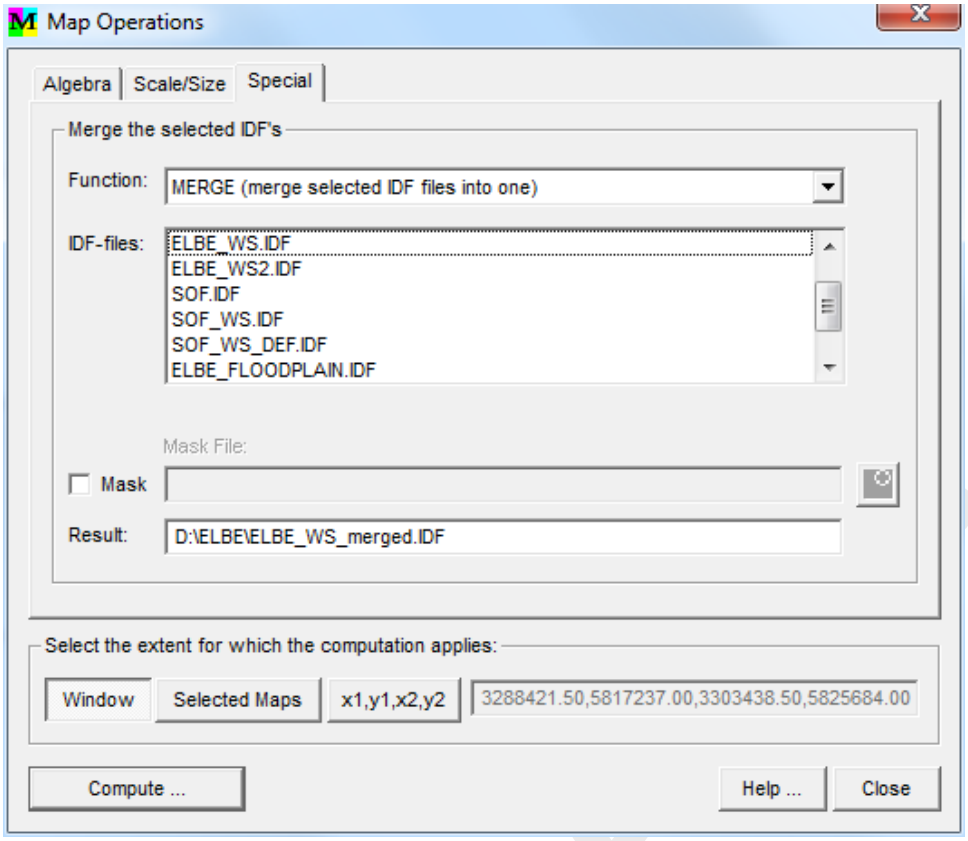
Map Operation window, Scale/Size tab:



Map A	Displays the IDF to be scaled. On default it will display the selected IDF in the *iMOD Manager*, click the *Open IDF* button to open a different one.
	*Open IDF* Click this button to select an IDF-file from a folder.
Map C	Enter the name of the resulting IDF. On default it will use the IDF-file from Map A and add the postfix SCALED to it, so the IDF-name will change from *.IDF into *_SCALED.IDF.
Scale	Enter the new cell size of the IDF. The resulting IDF will be an equidistant IDF with this cellsize.

Formulae	Select a *Formulae* to be used in the
Upscale Formulae	**Upscaling** (increased *cell size*): ◇ *Boundary* : minus values above positive values above zero values; ◇ *Arithmetic Mean* :sum cell values, excluding the *NoDataValues*, and divide them by the number of cells; ◇ *Geometric Mean* : take log()-function for cell values, excluding the *NoDataValues* and zero values, sum cell log-values, divide them by the number of cells and take the exp()-function; ◇ *Sum* : sum cell values, excluding the *NoDataValues*; ◇ *Sum Conductance* : sum cell values, multiplied with the ratio of ...., excluding the *NoDataValues*; ◇ *Inverse* : take the inverse ($1/x$) of cellvalues, excluding the *NoDataValues* and zero values, and divide them by the number of cells*;* ◇ *Most.Freq.Occur.* : take the cell value that occurs most frequently within a coarse cell, excluding the *NoDataValues*; ◇ *Sum Inverse*: take the sum of the inverse (x^{-1}) of *the cell* values, excluding *NoDataValues* and zero values; ◇ *Percentile* : take the cell value that occurs for a given *percentile (0-1)* within a coarse cell, excluding the NoDataValues;  ◇ *Block value* : the value of the centre cell.
Downscale Formulae	**Downscaling** (decreased *cell size*): ◇ *Arithm. Average:* produces a good guess for all finer gridcells as a linear interpolation based on the coarse gridcells. ◇ *BlockValue:* assign the value of the coarse gridcell to all finer gridcells.
Select the extent for which the computation apllies:	Click one of the following options: ◇ *Window*, to force that the results of Map C will be computed for the current ZoomLevel only; ◇ *Map A*, to force that the results of *Map C* will be computed for the extent of *Map A*. ◇ x1,y1,x2,y2, to enter the coordinates of a specific window
Compute...	Click this button to start the computation, it closes the *Map Operations* window afterwards.
Help ...	Click this button to start the iMOD Help Functionality.
Close	Click this button to close the *Map Operations* window.

Map Operation window, Special tab:



Function	Select one of the following functions: ◇ **MERGE** use this option to merge the selected IDF-files into a single one; ◇ **SUM** use this option to sum all values for the selected IDF-files into a single one. ◇ **MEAN** use this option to compute the mean for all values for the selected IDF-files; ◇ **MIN** use this option to compute the minimum value for all values for the selected IDF-files; ◇ **MAX** use this option to compute the maximum value for all values for the selected IDF-files.
IDF-files:	Select more than one IDF-file to be merged. iMOD will calculate values for the overlapping areas based on the inverse distance to the extent border of the IDF-files. The weigh-factor for an IDF-file, increases whenever the point considered, lies further away from the extent border of the IDF-file. Exact in the middle between two IDF-files, the weigh-factors for both will become 0.5.
Mask	Select this option to include a mask that determines the area to be merged, e.g. a boundary of catchment area. This option is only available whenever *Function*=MERGE.
Mask File:	Enter the IDF-file of the mask. All data in the IDF, not equal to the *NoDataValue* will be used to identify the size of the mask.
	*Open IDF-file* Click this button to open an IDF-file as *MaskFile*.
Result:	Enter the name of the IDF-file to be created.

Compute...	Click this button to start the computation, it closes the *Map Operations* window afterwards.
Help ...	Click this button to start the iMOD Help Functionality.
Close	Click this button to close the *Map Operations* window.

Note: It is allowed only to merge equidistant IDF-files (IEQ=0, see section 9.5).

6.7.4 IDF Edit

WHY?

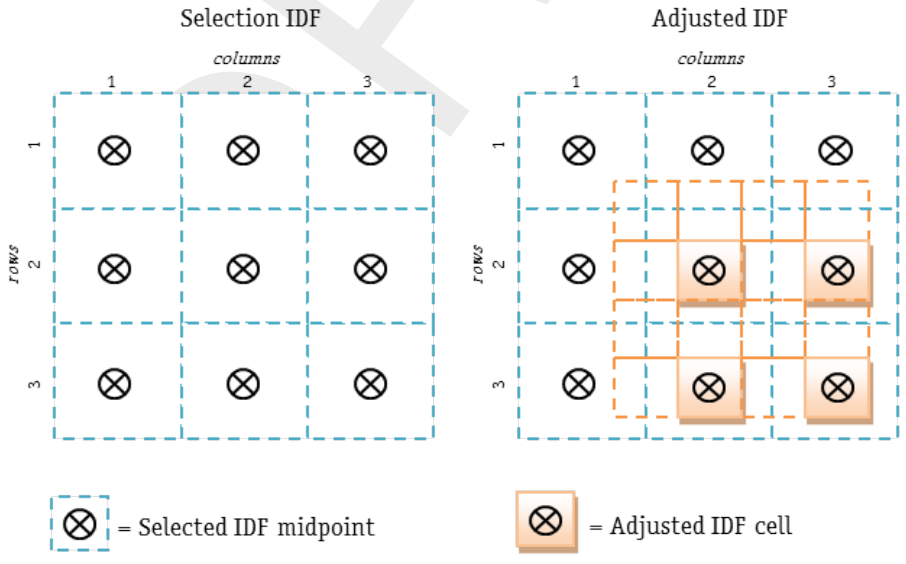
IDF-files are raster files with (non)-equidistant rastersizes. Normal GIS systems can not manipulate raster files in an easy way, moreover, it is difficult to edit individual rastercells.

WHAT?

The values of individual rastercells can be easily altered by selecting the rastercells and assigning a new value using the *Calculate* option. Also cellsizes of existing IDFs can be altered easily by grid refinement using upscaling and downscaling techniques.

IDF Edit allows you to select particular rastercells inside an IDF-file based on a maximum of two logical expressions that optionally operate inside a polygon. Logical expressions can be carried out sequentially. Once a selection has been made, different values can be assigned to them directly, or an interpolation and/or smoothing algorithm can be applied. A selection is carried out for mid points of rastercells for a particular IDF (Selection IDF). Those midpoints will be used also to perturb corresponding rastercells for IDF-files with different rastersizes and/or position. For these cases, it should be known that not all of the rastercells are affected by the alteration due to intermediate cells that are in-between midpoints of the Selection IDF.

Methodology of selecting and calculating with IDF Edit:

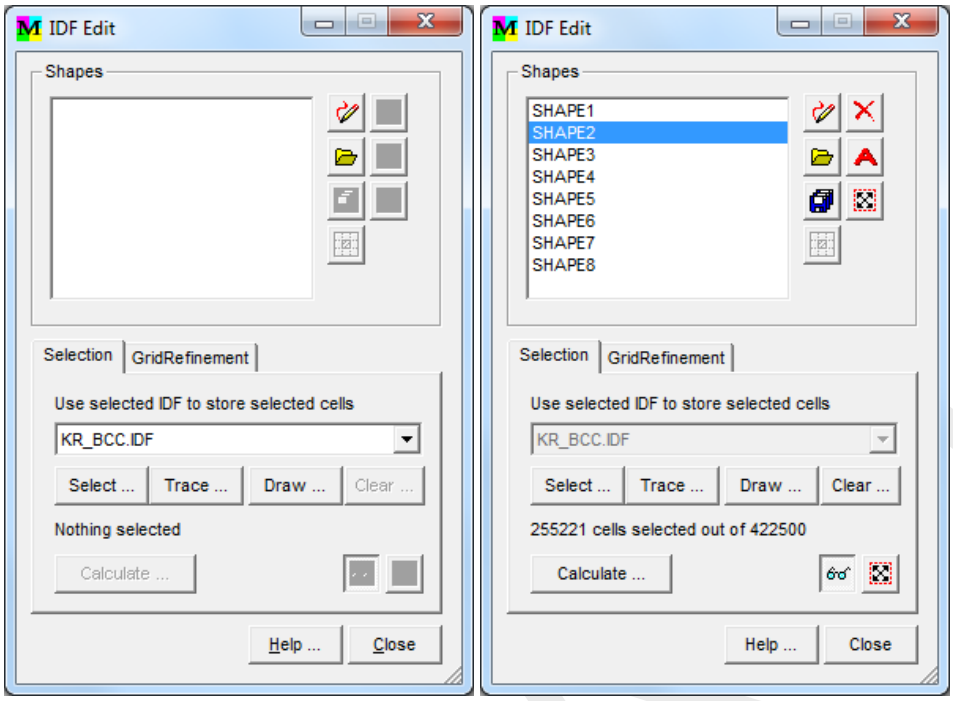


HOW?

To start *IDF Edit* select *Map* from the main menu, choose *IDF Options* and then *IDF Edit*. Alternatively, you can select the menu item from the popup menu that appears when you right-click your mouse in the graphical window. In both cases, you should select at least one

IDF from the *iMOD Manager*, the *IDF Edit* window will appear.

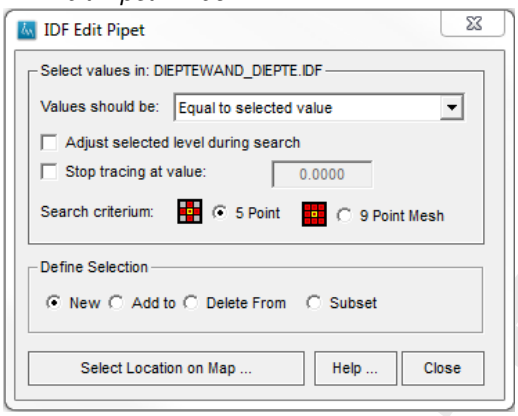
IDF Edit window, Selection tab, (left) initial window, (right) after selection:



Shapes	This groupbox contains several functionalities that are needed to draw and open polygons. The functionalities are explained in detail.
Use selected IDF to store selected cells	Select one of the IDF-files listed in the dropdown menu. The content of the list is based on the list of IDF-files in the *iMOD Manager*. The dimensions of the file that you select will be used to display the selection. This file will be the *Selection IDF*. If you select a *Selection IDF* with *CellSizes* of 25x25 meter, the selection will be displayed on that dimension. It is only possible to change the *Selection IDF* if the current selection is cleared and/or is empty.
Select . . .	Click this button to start the *IDF Edit Select* window.

Trace . . . Click this button to start the *IDF Edit Pipet* window.

IDF Edit Pipet window:



The trace selection will select cells in the IDF on the basis of their value and their connection to the identified cell.
Values should be: choose the selection condition from the pull down list.
Search criterium: check one of the buttons:

5 point: iMOD will search directly connecting cells on a five-point pattern
9 point: iMOD will search connecting cells on a nine-point pattern

Select Location on Map . . .: click this button to start the selection. The cursor will change in a pipette. Left click your mouse to select the cell from where you want to find the connecting cells which fulfill the selection condition. The *IDF Edit Pipet* window will close and the number of selected cells is indicated in the *IDF Edit* window. Click this button again if you like to start a new selection with new criteria and cell value (shown in the message bar at the bottom of your window *IDF-value:{value}*).

Define Selection:

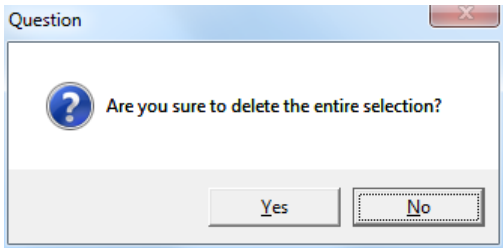
New Select this option to start a **new** selection.
Important Note: After a selection is made a new (clean) selection can only be started by clicking on the *Select Location on Map . . .* button. The selected cells will only than be deselected and a new selection can be made.

Add to Select this option to **add** the results of the evaluation to the current selection set.

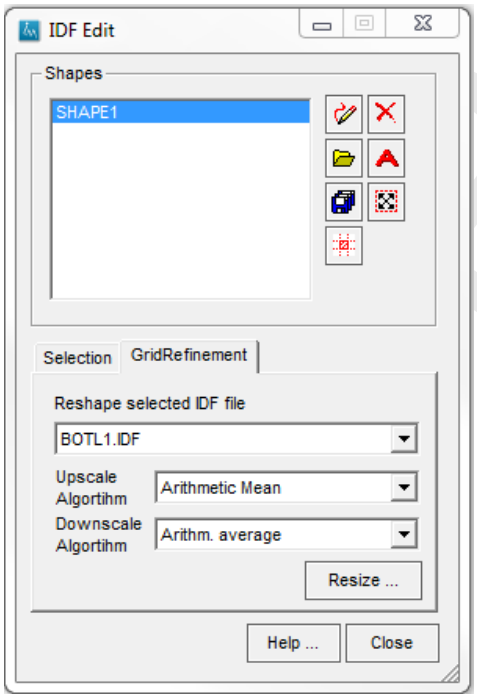
Delete From Select this option to **delete** the results of the evaluation from the current selection set.

Subset Select this option to **take** the result of the evaluation from the selection that is already in the selection set.

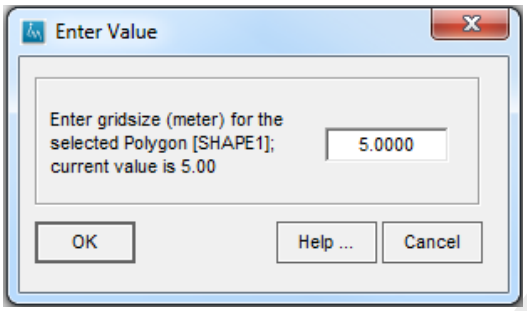
Draw . . . Click this button to start the *IDF Edit Draw* window (see section 6.7.4.2).

Clear	Click this button to clear the entire selection for the *Selection IDF*. This button is only available whenever cells are selected. You will be asked whether you are sure to continue. *Question window:* 
Calculate . . .	Click this button to start the *IDF Edit Calculation* window (see section 6.7.4.3).
	*Show Selection* Click this checkbox to display the selection.
	*Zoom Selection* Click this button to adjust the zoomlevel to the selected cells.
Help . . .	Click this button to start the iMOD Help Functionality.
Close	Click this button to close the *IDF Edit* window.

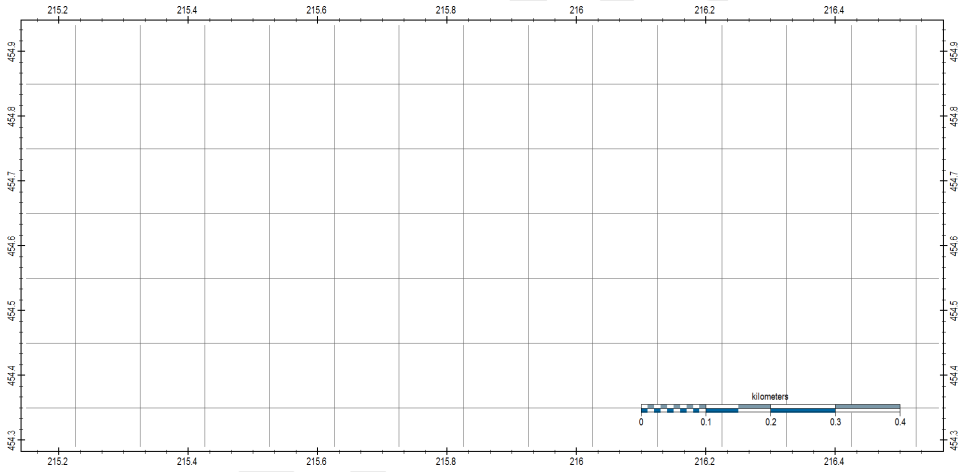
IDF Edit window, GridRefinement tab:



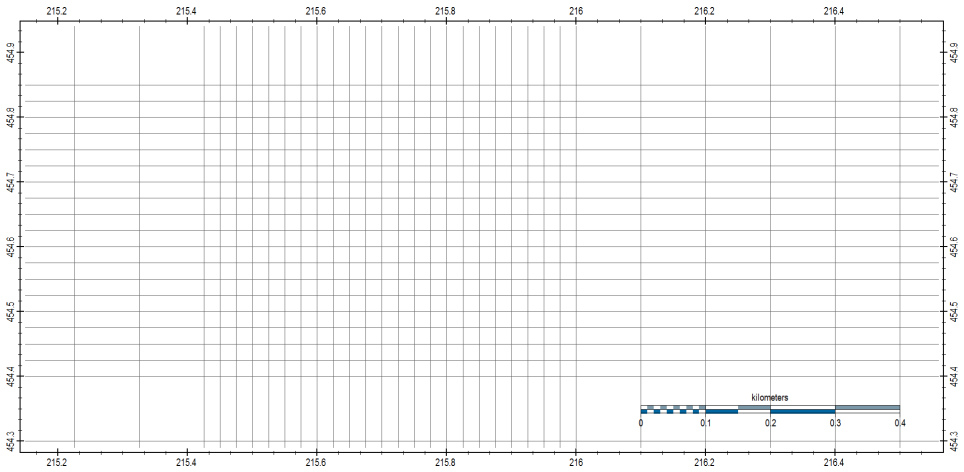
Reshape selected IDF file	Select the IDF file from the dropdown menu.

	Click this button to assign a new grid size. The *Enter Value* window will open.  Enter the new gridsize. This gridsize will be used within the selected polygon. It will be used for the entire extent of the IDF in case no polygon is selected. The new values of the gridcells are calculated by applying the up- or downscaling methodology selected in the dropdown menus.
Resize . . .	Click this button to resize and save the IDF-file.
Upscale algorithm	Select the upscaling methodology from the dropdown menu; more detail about the available methodologies is given in section 6.7.3
Downscale algorithm	Select the downscaling methodology from the dropdown menu; more detail about the available methodologies is given in section 6.7.3
Help . . .	Click this button to start the iMOD Help Functionality.
Close	Click this button to close the *IDF Edit* window.

Example of grid refinement:



Before refinement: grid with 100 m cell size

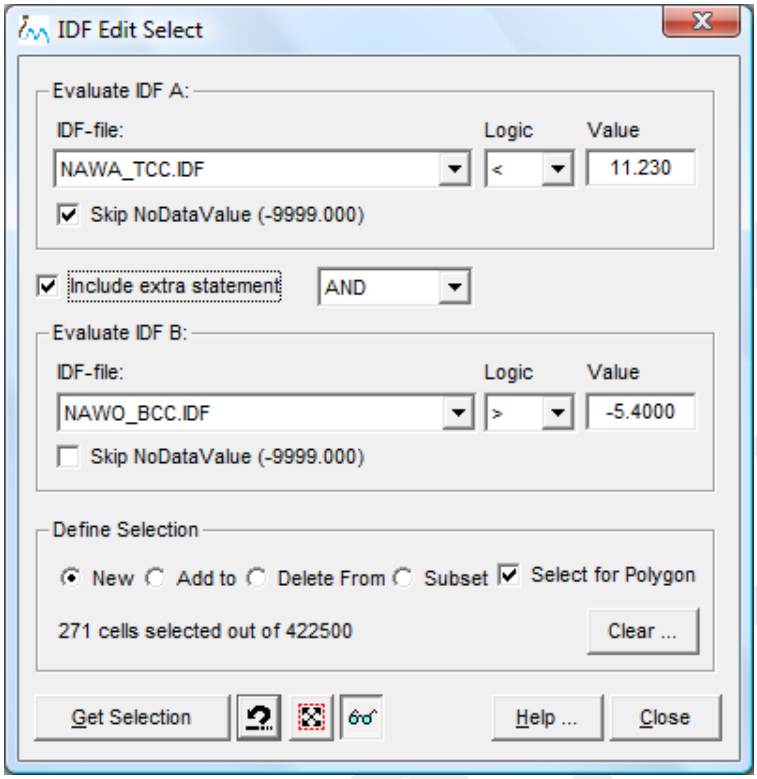


After refinement: grid with 100 m cell size and refined 25 m cell size; note that the cells on the right side are shifted.

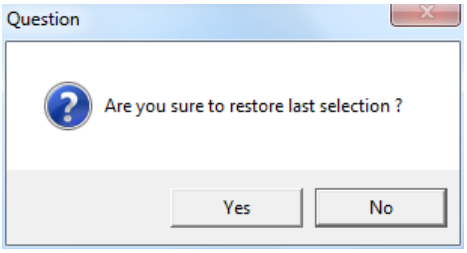
6.7.4.1 IDF Edit Select

Click the option *Select* on the *IDF Edit* window, to display the *IDF Edit Select* window.

IDF Edit Select window:

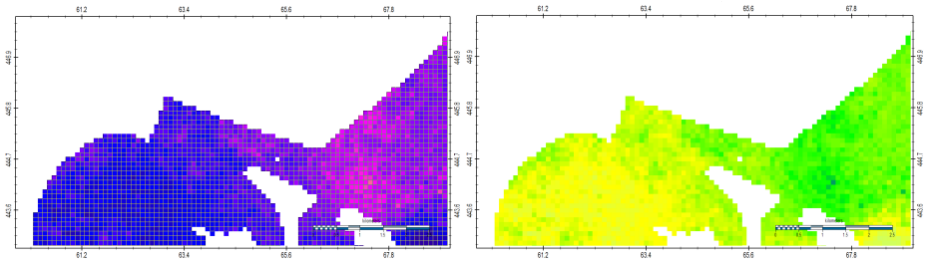


IDF-file:	Select one of the IDF-files listed in the dropdown menu.
Skip NoDataValue value (....)	Select this checkbox to “skip” the *NoDataValues* in the selection. On default this will be turned on, so cells with *NoDataValues* will not be selected, discarding their values. For the selected IDF-file, the current *NoDataValue* will be displayed between brackets.

Logic	Select one of the logic variables out of the dropdown menu: "=" : Equal to *Value* "<>" : Not equal to *Value* "<" : Less than *Value* "<=" : Less than or equal to *Value* ">" : greater than *Value* ">=" : Greater than or equal to *Value* "BND" : Selects the cells at the boundary of an area with cells equal to *Value* "SPIKE(4)": selects spikes with difference by *Value* relative to surrounding cells in four directions "SPIKE(3)": Selects spikes with difference by *Value* relative to surrounding cells in at least three directions "SPIKE(2)": Selects spikes with difference by *Value* relative to surrounding cells in at least two directions "ALL": Selects all cells "NaN": Select values equal to a so-called NaN-value in the IDF, these numbers contains value that are beyond the range of a single precision real "Inf": Select values equal to an infinite value. E.g. infinity is caused by dividing by zero "NodataValue": Select values equal to the *NodataValue* of an IDF-file **Note:** The selection of SPIKE(i) may be used to smooth outliers in the IDF-file using the smooth option in the *IDF Edit Calculation* window.
Value	Enter the value to be evaluated, e.g. *11.230* or *-5.400*
Include extra statement	Select this checkbox to add an extra IDF to be evaluated for the selection. You can choose the keywords: *AND*, click this option to make the selection set fulfill both the settings in *Evaluate IDF A* **and** *Evaluate IDF B*; *OR*, click this to make the selection set fulfill at least one of the settings in *Evaluate IDF A* **or** *Evaluate IDF B.*
New	Select this option to start a **new** selection.
Add to	Select this option to **add** the results of the evaluation to the current selection set.
Delete from	Select this option to **delete** the results of the evaluation from the current selection set.
Subset	Select this option to **take** the result of the evaluation from the selection that is already in the selection set.
Select for Polygon	Select this checkbox to apply the evaluation inside the current polygon(s) only. This option is active whenever a shape is selected in the *IDF Edit* window.
271 cells selected out of 422500	This shows the current number of selected cells (*271*) in the *Selection IDF* that consists of *422500* cells.
Clear . . .	Click this button to clear the entire selection for the *Selection IDF*. This button is only available whenever cells are selected. You will be asked whether you are sure to continue.
Get Selection	Select this button to get the selection. Each selection will be saved in the file: {USER}\tmp\{username}tmpselected{i}.dat.
	*Undo* Click this button to restore the previous selection. *Question window:* 

	*Zoom to Selection* Click this button to adjust the zoomlevel to the selected cells.
	*Show Selection* Click this checkbox to display the selection (see below).
Help ...	Click this button to start the iMOD Help Functionality.
Close	Click this button to close the *IDF Edit* window.

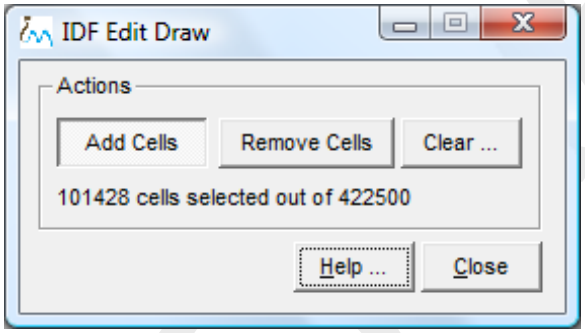
Example showing (left) and deshowing (right) the selected cells:

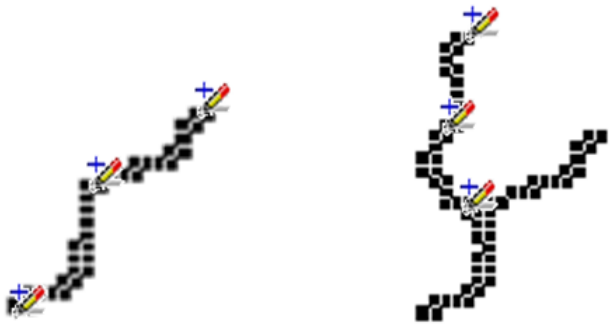


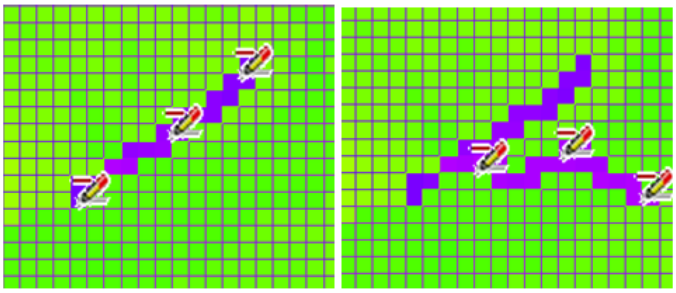
6.7.4.2 IDF Edit Draw

Click the option *Draw* on the *IDF Edit* window, to display the *IDF Edit Draw* window. Cells from the selected IDF can be selected or deselected interactively in the graphical window.

IDF Edit Draw window:



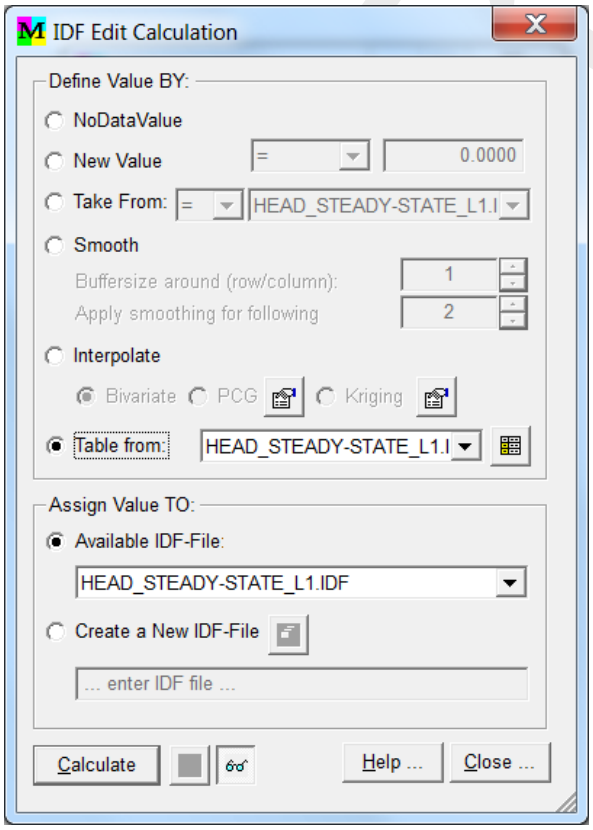
Add Cells	Click this button to add cells to the selection. Use the left mouse button and drag the cells you want to select on the graphical window. Release the left mouse button to stop drawing and click the left mouse again to add more cells to the selection. *Draw a selection in the Selection IDF:* 

Remove Cells	Click this button to remove current cells from the selection. Use the left mouse button to deselect cells from the current selection. Release the left mouse button to stop deselecting cells and click the right mouse button again to deselect more. *Deselect cells for the Selection IDF:* 
Clear . . .	Click this button to clear the entire selection for the *Selection IDF*. You will be asked whether you are sure to continue.
101428 cells selected out of 422500	This shows the current number of selected cells (*101428*) in the *SelectionIDF* that consists of *422500* cells.
Help . . .	Click this button to start the iMOD Help Functionality.
Close	Click this button to close the *IDF Edit Draw* window.

6.7.4.3 IDF Edit Calculate

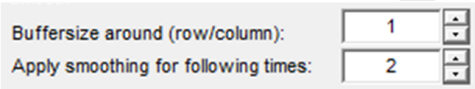
Click the option *Calculate* on the *IDF Edit* window, to display the *IDF Edit Calculate* window. This option is not active whenever nothing is selected.

IDF Edit Calculation window:

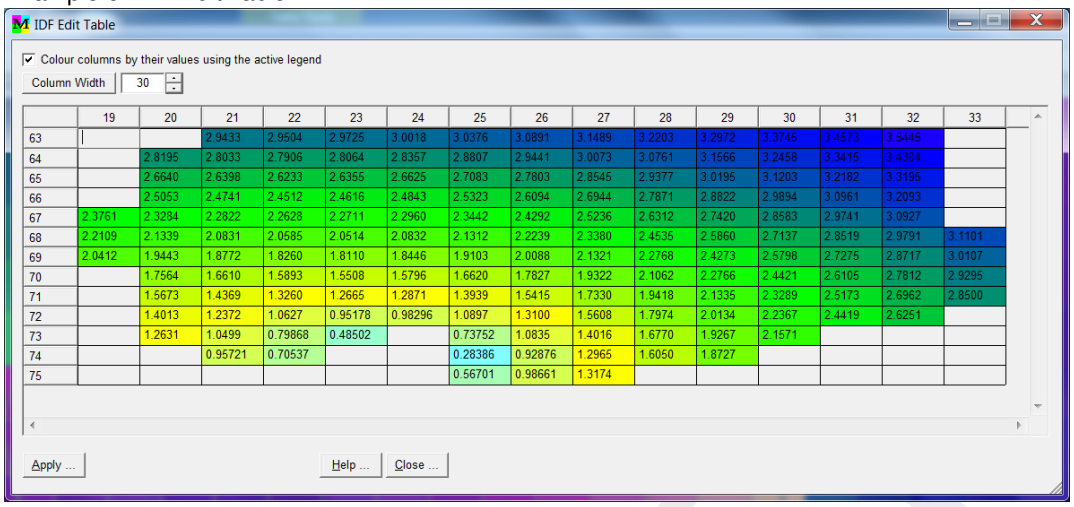


The calculation method is defined in the upper part of the window: *Define Value BY:*.

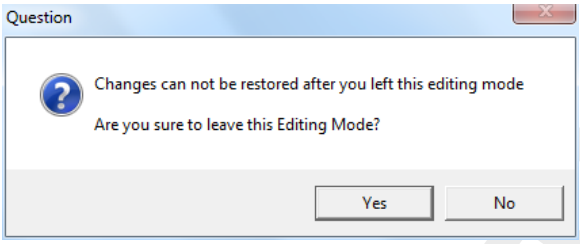
The IDF on which the calculation is applied is selected in the lower part of the window: *Assign Value TO:*.

NoDataValue	Select this option to apply NoDataValues
New Value	Select this option to enter a new value. Choose one of the following options from the dropdown menu: "=" : equal "+" : add a value "-" : subtract "*" : multiply "/" : divide
Copy Values From:	Select this option to copy the values from the selected IDF-file in the dropdown menu. The IDF-files are those that are available in the menu on the*iMOD Manager*.
Smooth	Select this option to smooth the values within the selection only. *Smooth options:* 
Buffersize around (row/column):	Enter the number of rows/columns that need to be taken into account in the smoothing. The value *1* means that around a cell that needs to be smoothed, one row/column will be used.
Apply smoothing for following times:	Enter the number of smooth operations. The more times smoothing is done the more smoother the result will be.
Interpolate	Select one of the following options to interpolate: ◇ **BIVARIATE** apply a bivariate method; ◇ **PCG** apply a preconditioned conjugate gradient method; ◇ **KRIGING** apply a normal Kriging procedure.
	*Properties* Click this button to display a properties window for the selected interpolation option. Whenever the PCG is selected the *PCG Settings* window will be displayed (section Table 4.1), in case a Kriging method is selected, the *Kriging Settings* window is displayed (Table 4.1.
Table from:	Click this option to open a table that shows the cell values in the selected cells for the IDF-file selected in the dropdown menu (right).
	Click this button to display an overview of the cell values in a table. The *IDF Edit Table* window opens.

Example of *IDF Edit Table:*



	The values shown in the table can be edited.	
	Colour columns . . .	Select this checkbox to colour the table cells by their corresponding values. Deselect this checkbox to display the table uniformly white. Each time a table value is changed, the colouring will be adjusted accordingly.
	Column Width	Click this button to uniformly change the table column width by the entered column width
	Apply . . .	Click this button to confirm to copy the values. Be aware that you need to click the *Calculate* button to actually copy those values into the IDF-file selected in *Assign Value TO*.
	Close . . .	Select this button to discard any entered values and to close the *IDF Edit Table* window
Available IDF-file:	Select the IDF-file that need to be adjusted from the dropdown list.	
Create a New IDF-file	Select this option to enter a different IDF file (not yet loaded in the iMOD Manager and/or to be created	.
	*Save As* Click on the button and enter a name for the new IDF. The name of the new IDF is displayed.	
Calculate	Select this button to execute the calculation and to adjust the values in the selected *IDF-file*.	
	*Undo*Click this button to undo the last calculation. Each calculation will be saved in the file: {USER}\TMP\{USERNAME}TMPCOMPUTED{i}.DAT.	
	*Show Selection* Click this checkbox to display the selection of the graphical window.	
Help . . .	Click this button to start the iMOD Help Functionality.	

Close . . .	Click this button to close the *IDF Edit Calculation* window. If you have adjusted one/more files, you will be asked whether you are sure to leave the *IDF Edit Calculation* window. Changes can not be restored once you have agreed upon closing the *IDF Edit Calculation* window. *Question window to leave the IDF Edit Calculation window:* 

Note: Whenever you click the *Calculation* button, the IDF-file will be adjusted and the new results will be saved directly in the IDF-file. You can use the *Map Value* option, to inspect the adjusted results, without leaving the *IDF Edit Calculation* window.

6.8 IPF Options

iMOD supports several basic functionalities that manipulate IPF-files:

- **IPF Configure,**
- **IPF Analyse**, use this tool to analyse the content of the IPF-files (including the associated files, if available).
- **IPF Extract**, use this tool to extract points out of an IPF-file to be saved independently.

6.8.1 IPF Configure

WHY?

IPF-files in iMOD represent point data that can be displayed in different ways.

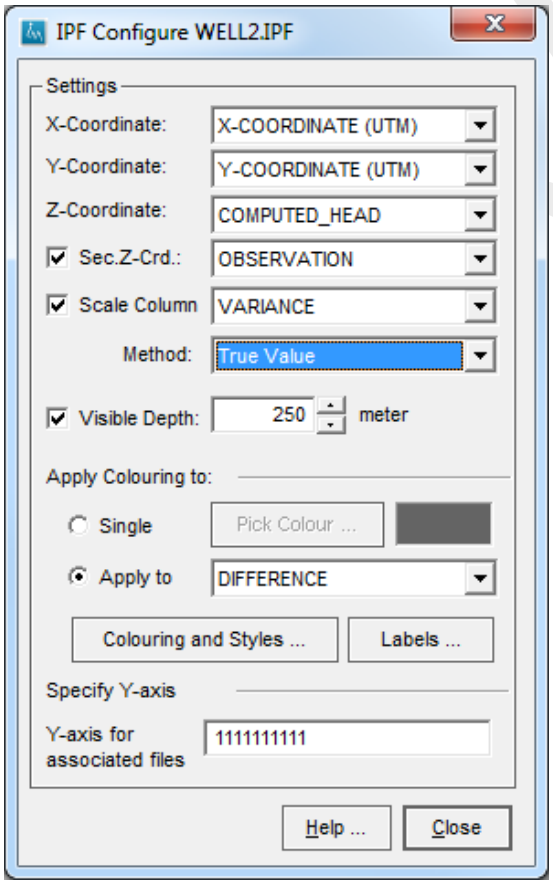
WHAT?

IPF Configure is used to define the settings for display and assigns a symbol, colour or label to the point data.

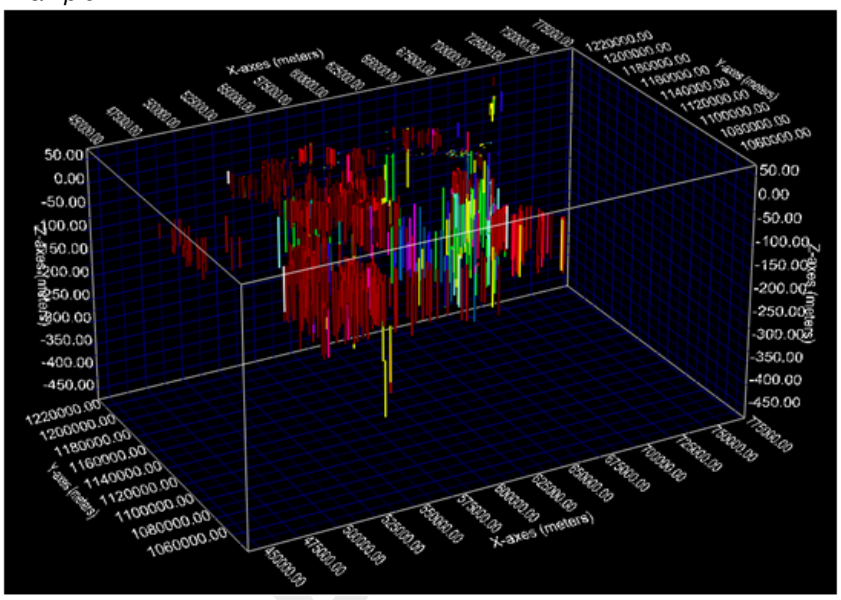
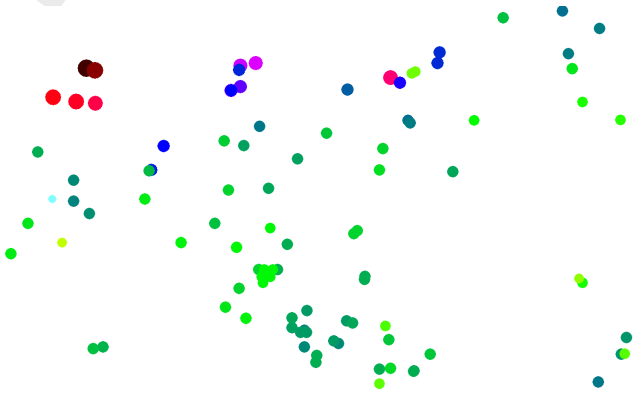
HOW?

Select the menu option *IPF Configure* from the *IPF-options* menu in the *Map* menu to display the *IPF Configure* window. Or, use right-click anywhere on the canvas to open the popup menu. Select the option *IPF-options* and then choose *IPF Configure*.

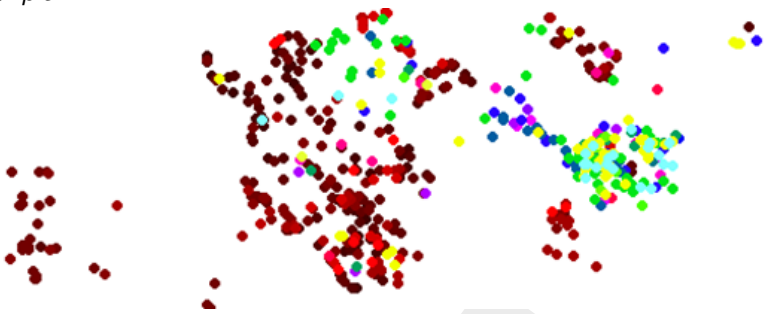
IPF Configure window:



X-Coordinate:	Select the column in the IPF that represents the X coordinate.
Y-Coordinate:	Select the column in the IPF that represents the Y coordinate.

Z-Coordinate:	Select the column in the IPF that represents the Z coordinate. This data column will be used in the *Cross-Section Tool* and/or *3D Tool*.
Sec.Z-Crd.:	Select the column in the IPF that represents the secondary Z coordinate. This data column will be used in the *Cross-Section Tool* and/or *3D Tool* in combination with the *Z-Coordinate*. In these cases, iMOD will draw a line between the *Z-Coordinate* and *Sec.Z-Crd.*. You can use this for the top and the bottom of the screen-depth of a well. *Example:* 
Scale Column:	Select the column in the IPF to scale the symbol size. This option *scales* points that have larger values than others, those will be displayed as an increased marker symbol. iMOD will scale the values linearly from large up to small and displays them accordingly, such that small values will be plotted upon large values. As NO legend is specified, the internal values of the appropriate column will be used, otherwise the minimal and maximal values of the specified legend will be used to scale the symbols. *Example:* 

Method	Select a method.
Visible Depth	Select and specify an interval in meters, over which points are visible.
Single	Select this option to display all points with the same colour.
Pick Colour . . .	Select this button to display the default *Colour* window in which a colour can be specified. The current colour is displayed to the right of this button.

Apply to	Select this option to colour the points according to the selected attribute that is chosen in the dropdown menu at the right. *Example:* 
Colouring and Styles...	Select this option to display the *Lines and Symbols* window, section 5.7.
Labels...	Select this button to display the *Select Label to be Printed* window.
Y-axes for associated files	Insert an indication which vertical axis to use for maximal 10 attributes that are available in associated files. The value 1, means that the attribute is plotted against the first y-axis, 2 means that the attribute is plotted against the second y- axis (on the right of the final graph).
Help...	Click this button to start the iMOD Help Functionality (if available in the selected *.PRF file).
Close	Select this button to apply the settings and close the *IPF Configure* window.

Note: On default, the classes for a legend will be computed linearly between the minimum and maximum values for the selected attribute. Use the options described to adjust this legend in order to plot proper colours.

6.8.2 IPF Labels

WHY?

The attributes of point data are displayed as labels.

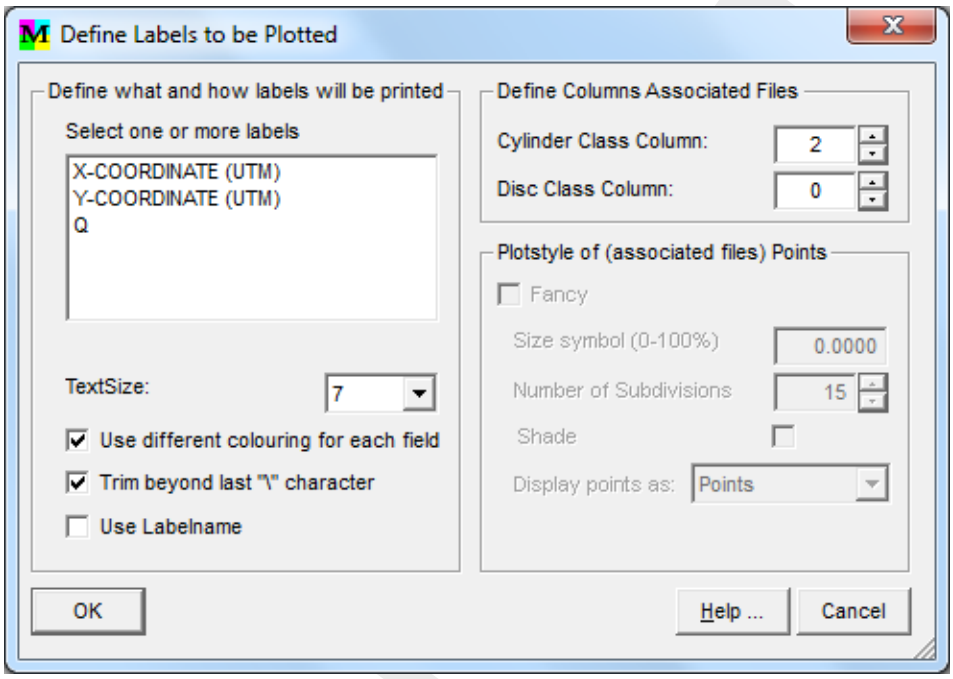
WHAT?

At the location of each point one or more labels can be displayed.

HOW?

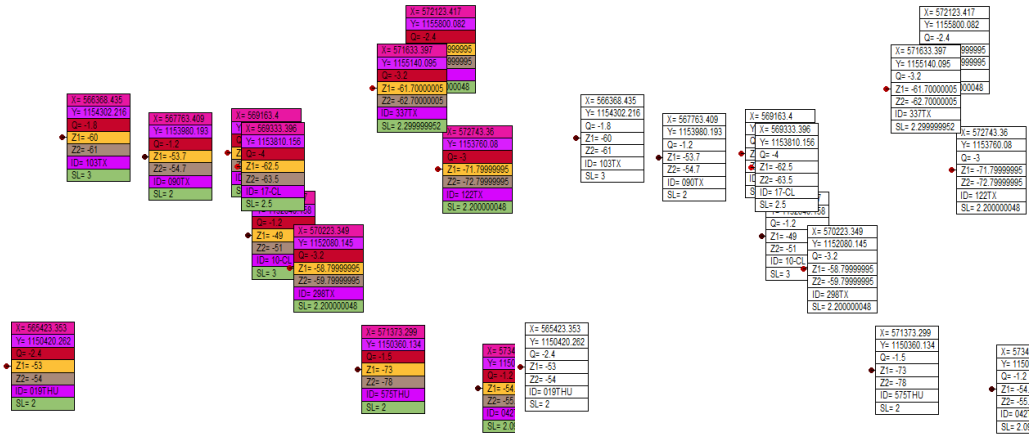
Choose the option *Define Labels to be Plotted* from the *IPF Configure* window to display the following window.

Define Labels to be Plotted window:

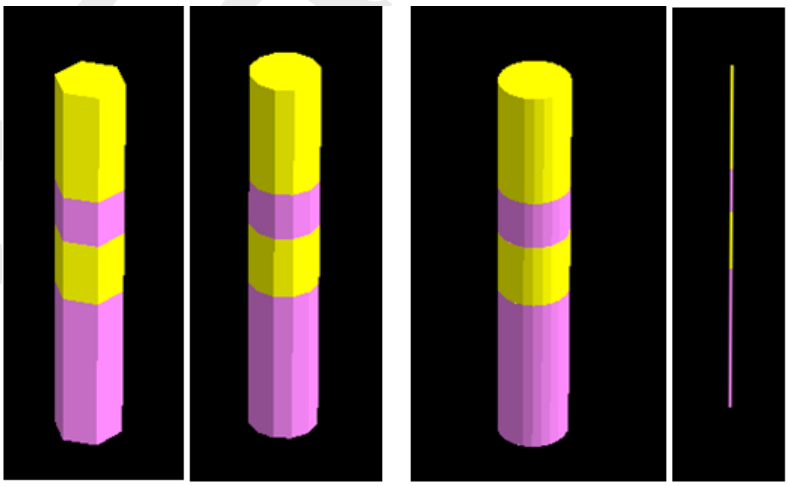


Select one or more labels	Select one of more labels from the menu to display near the point of the IPF file.
TextSize	Select one of the available textsizes from the dropdown menu.
Use different colouring for each field	Select this option to use different colour for each attribute value. Uncheck this option to use a white fill for each label.
Trim beyond last "\"character	Select this option to trim the attribute value beyond the last "\"character, so the attribute value *boreholes\east\NH45* will be displayed as *NH45.*
Use Labelname	Select this option to display the label name and the label value.
OK	Click this button to close the *Define Labels to be Plotted* window.
Help. . .	Click this button to start the iMOD Help Functionality.
Cancel	Click this button to cancel the chosen definition and to close the *Define Labels to be Plotted* window.

Example of the usage of different colouring for attribute fields:



The following options are available from the 3D Tool only:

Cylinder Class Column	Enter the number of the column in the associated files that need to be used for the size of the cylinders plotted for the boreholes.
Disc Class Columns	Enter the number of the column in the associated files that need to be used for the vertical location of a disc around a borehole
Fancy	Check this item to improve the display of a borehole.
Size Symbol	Enter a value to increase the size of the borehole
Number of subdivisions	Enter a number of subdivisions to be used to display a borehole. A large number of subdivisions will improve the appearance. *Examples of a fancy (first three on the left) display with different number of subdivisions and a non-fancy appearance (utmost right):* 
Shade	Check this item to apply a shade on the boreholes.
Display points as:	Choose one of the options: Points, lines, silhouette, fill

6.8.3 IPF Analyse

WHY?

The attribute and time series data of selected point data are displayed interactively.

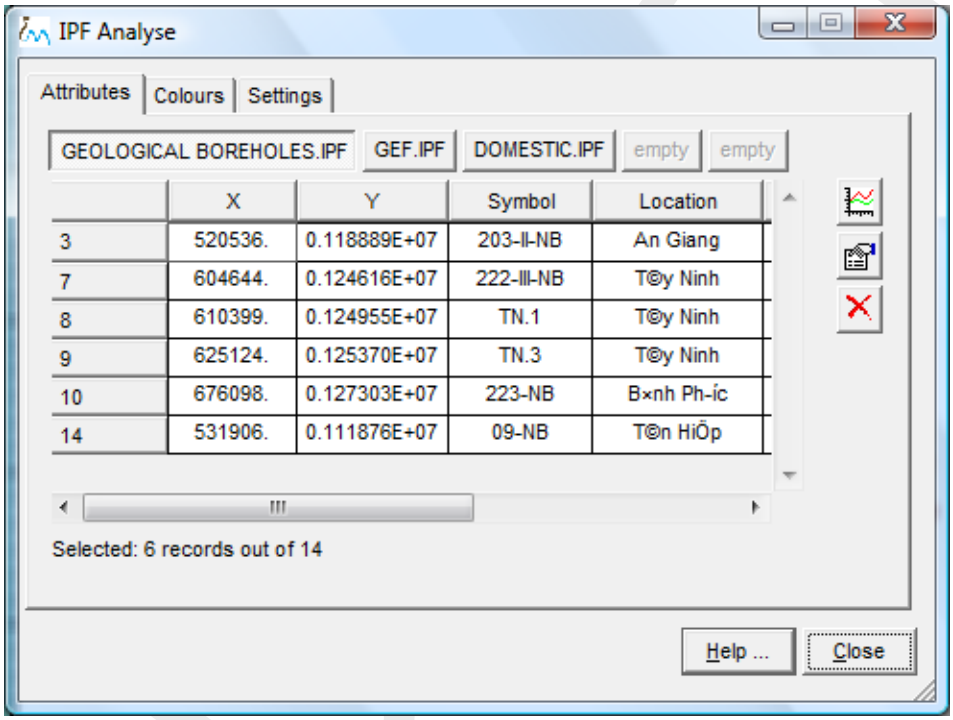
WHAT?

The *IPF Analyse* function is used primarily to display the timeserie data of selected point locations in a graph on the map or separately.

HOW?

To display the *IPF Analyse* window, click the *Map* option from the main menu, then the option *IPF-options* and then the option *IPF Analyse* (this is similar from the popup menu that displays whenever you click your right mouse button on the graphical window). The selected IPF-file(s) from the *iMOD Manager* will be used in the *IPF Analyse* window.

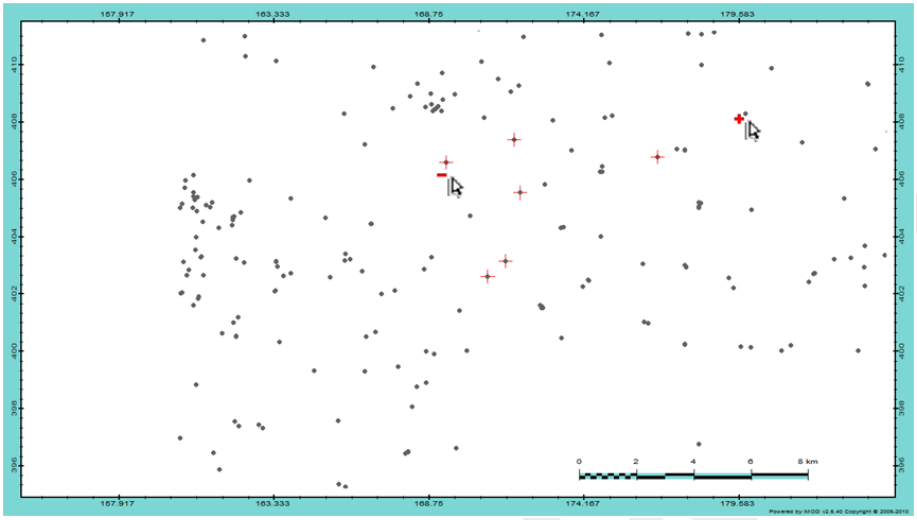
IPF Analyse window, Attributes tab:



Geological-boreholes.ipf, Gef.ipf, Domestic.ipf	These buttons show the IPF-files that are currently available in *IPF Analyse*. It is possible to select maximal 5 IPF-files before entering the *IPF Analyse* window. iMOD will switch to the correct tab if points are selected from other IPF-file(s).
Table	The table shows all attributes for the IPF-file, in this case 5 attributes are available (*X, Y, Symbol, Location, FinalDepth*). To add a point to the selection, left-click the mouse button at the location of the desired point in the graphical window. The cursor of the mouse will show a “plus”- or “minus”-sign to indicate whether you add or delete the current location to/from the selection. All selected points will appear on the graphical window as a small red cross. The first column in the table shows the record number, i.e. the row number in the data field of the IPF-file.
Selected 6 records out of 95	This shows the current number of selected point in the table.
	*IPF Figure* Click this button to display the content of the associated files for the selected point in the *IPF Analyse Plot* window.

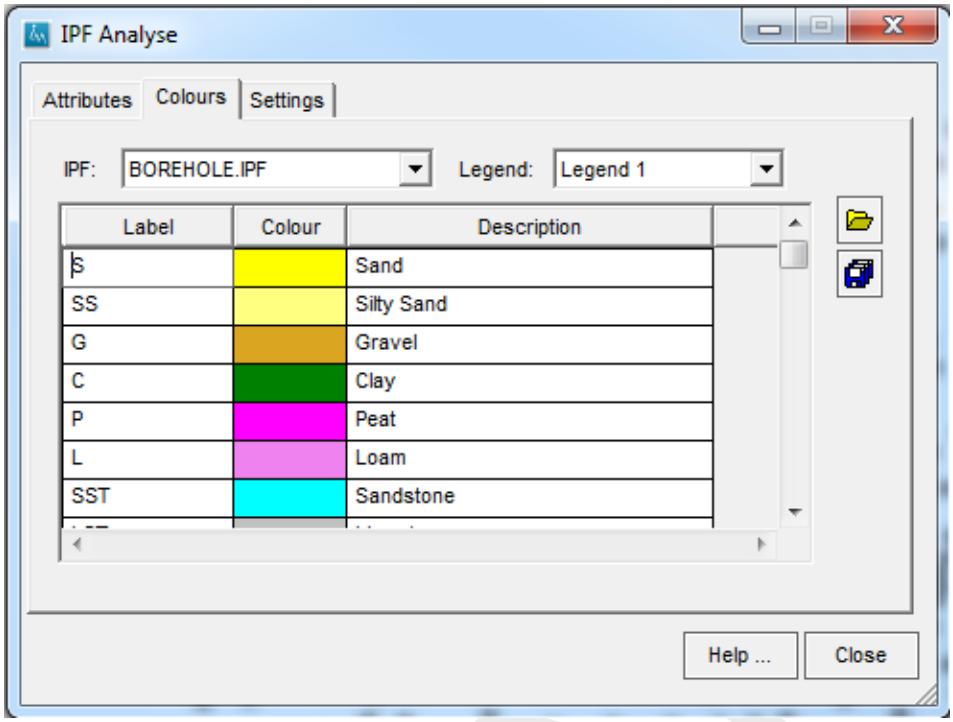
	*Properties* Click this button to display the*IPF Configure* window.
	*Delete* Click this button to remove the selected row from the selection displayed in the table.
Help. . .	Click this button to start the iMOD Help Functionality.
Close	Click this button to close the *IPF Analyse* window.

Example of selected points in IPF Analyse:



Note: Only 50 records can be displayed in each table. However, more records can be selected, but they will not appear in the table.

IPF Analyse window, Colours tab:

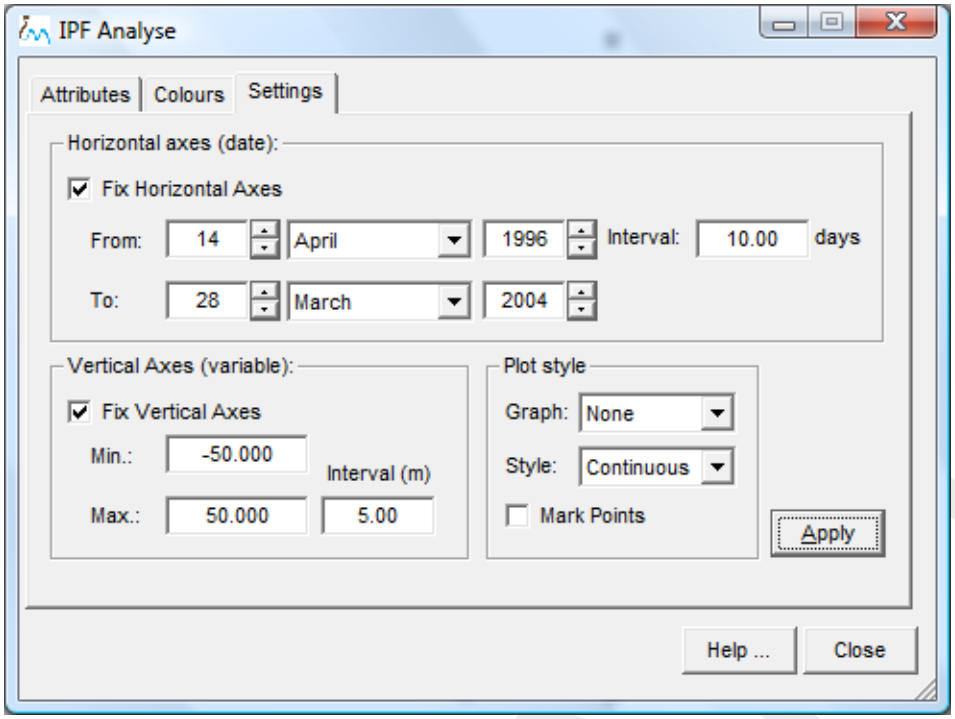


The *Colours* tab is only used when IPF-file(s) are selected that contain associated files that describe **borehole** information. The settings from the *Colours* tab will be used to colour the individual zones in the boreholes. The chosen and/or defined legend(s) will be saved in the IMF-file for later use. The next time you enter the *IPF Analyse window* the same legends will appear.

IPF:	This dropdown menu shows the IPF-files that are currently available in the *IPF Analyse.* For each IPF-file a different legend can be defined by making use of the *Legend:* dropdown menu.
Legend:	This dropdown menu allows you to select and/or define a different legend per available IPF-file.
Label	Enter the label that matches the second field (column) in the associated file that contain borehole information.
Color	Displays the colour of the field. You can specify a different colour by selecting the *Color* column for the appropriate row. The default *Colour* window will appear.
Description	Enter a descriptive string that will be used in the display of the legend.
	Open Click this button to open a *.DLF file that describes the colouring info.
	SaveAs Click this button to save the current colouring information to a *.DLF-file.

Note: On default the file {USER}\SETTINGS\DRILL.DLF will be used.

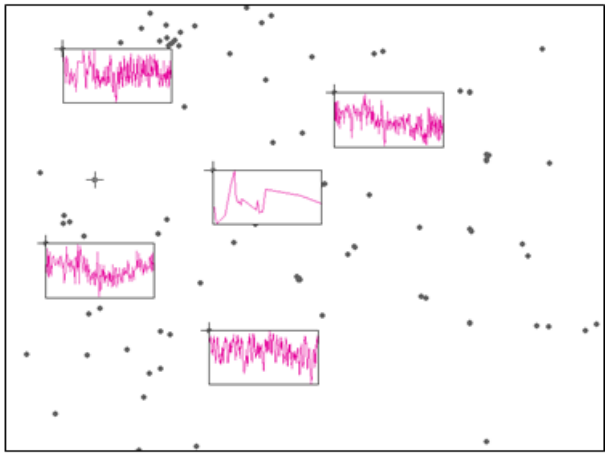
IPF Analyse window, Settings tab:

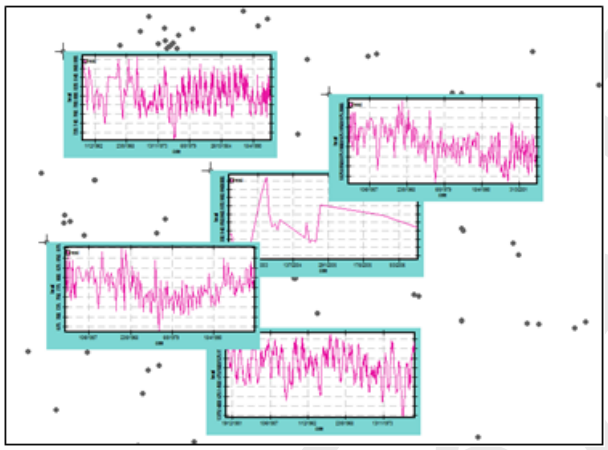


The *Settings* tab is only used when IPF-file(s) are selected that contain associated files that describe **timeseries** information. The settings from the *Settings* tab will be used to plot the individual timeseries that are associated with the selected points.

Fix Horizontal Axis	Select this checkbox to specify the dimensions of the X-axis. Whenever unselected, iMOD will determine the dimensions of the X-axis automatically.
From:	Enter the start date for the X-axis.
Interval:	Enter the interval of the X-axis in days.
To:	Enter the end date for the X-axis.
Fix Vertical Axes	Select this checkbox to specify the dimensions of the Y-axis. Whenever unselected, iMOD will determine the dimensions of the Y-axis automatically.
Min.:	Enter the minimum value for the Y-axis.
Max.:	Enter the maximum value for the Y-axis.
Interval (m):	Enter the interval of the Y-axis in meters.
Graph:	Select one of the following options: ◇ **None:** No timeserie will be plotted at the selected points; ◇ **Simple:** A simple timeserie-graph will be plotted at the selected points, without any axes; ◇ **Extended:** A timeserie-graph will be plotted at the selected points with x- and y-axes.

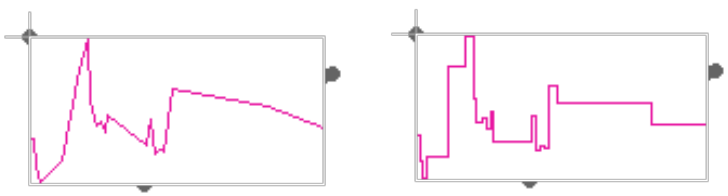
Example of the plotstyle Simple (top) and Extended (bottom)





Style:	Select one of the following options: ◇ **Continuous:** The individual data points in the timeseries will be connected directly from one point to the other. This assumes that the intermediate unknown data points will be on a straight line between the two known data points. e.g. use this option to display timeseries of groundwaterhead; ◇ **Blocked:** The individual data points in the timeseries will be connected as horizontal line. This assumes that the intermediate unknown data points will have the same value as the previous known data point. e.g. use this option to display timeseries of extraction rates.
Mark Points	Select this checkbox to place markers (cross) at the location of data points.
Apply	Select this button to apply the settings to the data points selected in the table on the *Attributes* tab.

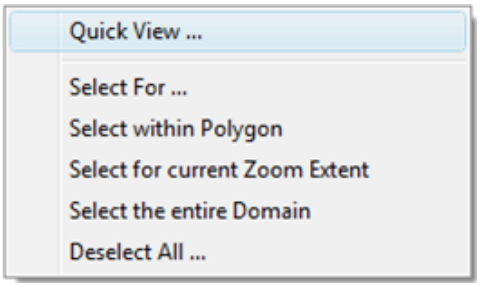
Example of Continuous (left) and Blocked (right) lines:



6.8.3.1 Drop down menu

Click the right mouse button on the graphical display to show the following option.

Drop down menu IPF Analyse:

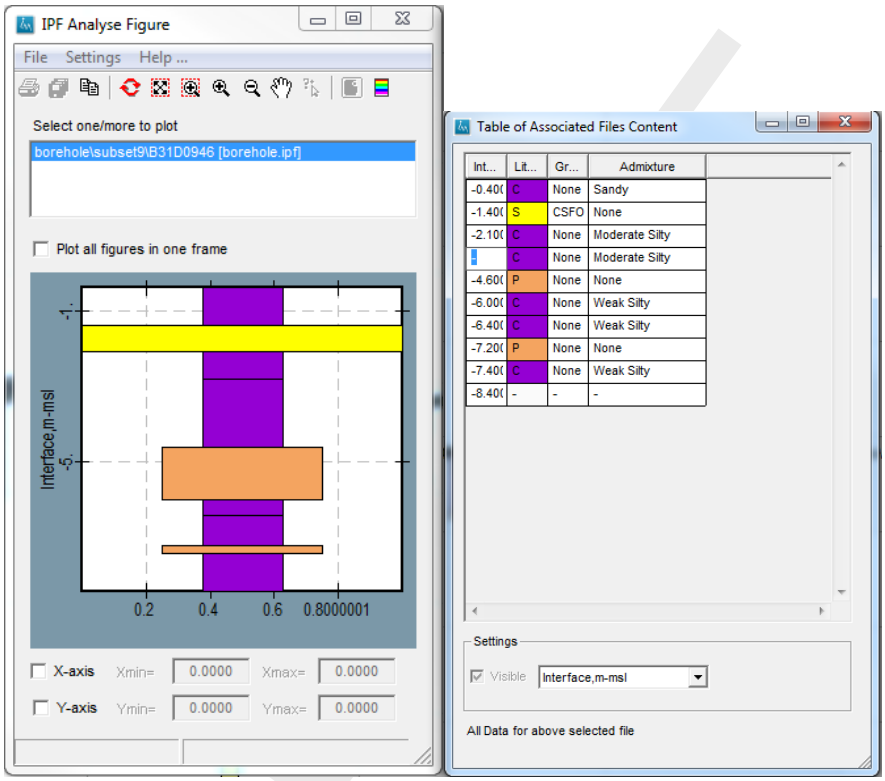


Quick View . . .	Select this option to display the *IPF Analyse Figure* window. The content of the *IPF Analyse Figure* window will change whenever you select a different data point with the mouse. Whenever you click the left-mouse button, the current location remains unchanged and you can use the *IPF Analyse Figure* window. If you click the left-mouse button again on the graphical display, the current location changes again according your mouse position. Use the right mouse button to stop *Quick View*.
Select For . . .	Select this option to display the *Select For* window to select data points that meet a specific criteria.
Select within Polygon	Select this option to draw a polygon on the graphical display, to select all data points that are inside that polygon.
Select for current Zoom Extent	Select this option to select all data points present in the current zoom extent.
Select the entire Domain	Select this option to select all data points from the current IPF.
Deselect All . . .	Select this option to delete all selected data points from the table in the *Attributes* tab on the *IPF Analyse* window. You will be asked to confirm this operation beforehand.

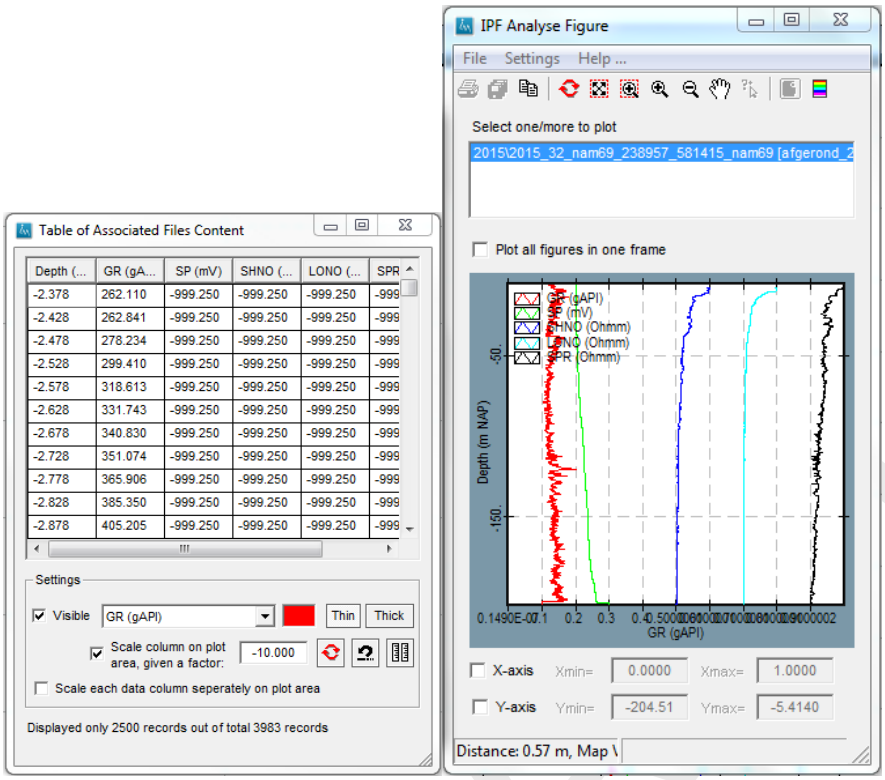
6.8.3.2 IPF Analyse Figure

Click the option IPF Figure  from the IPF Analyse window to start the IPF Analyse Figure window. By moving with your cursor over the boreholes you are able to navigate through the characteristics of the boreholes. Depending on the position of the cursor on the borehole, the specific layer will be highligthed in the *Table of Associated Files Content* (see figures below).

Example 1: Borehole (IPF option=2) representation
IPF Analyse Figure window: Table of Associated File Content



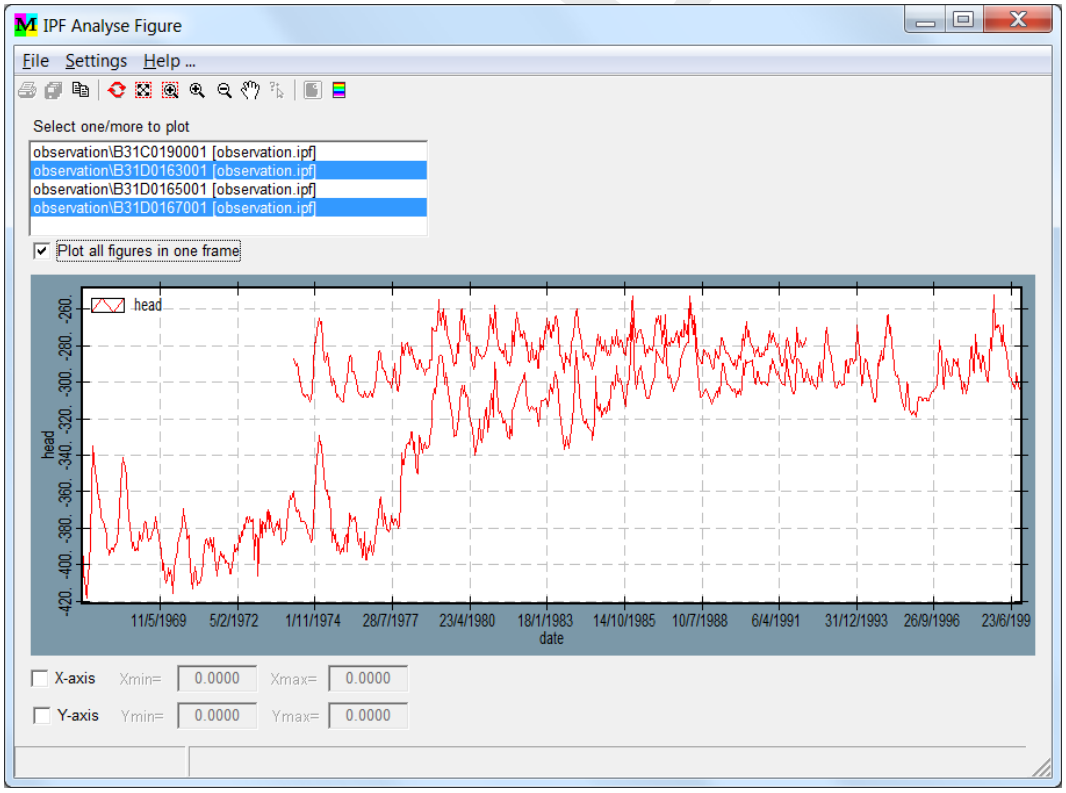
Example 2: Well locs (IPF option=3) representation
IPF Analyse Figure window: Table of Associated File Content



File – Print	Print the selected figure to the default Windows external Printer.
File – Quit	Click this option to close the *IPF Analyse Figure* window.
Settings – Continuous Lines	Click this option to display timeseries as continuous lines. The individual data points in the timeseries will be connected directly from one point to the other. This assumes that the intermediate unknown data points will be on a straight line between the two known data points. e.g. use this option to display time-series of groundwaterhead.
Settings – Block Lines	Click this option to display timeseries as block lines. The individual data points in the timeseries will be connected as horizontal line. This assumes that the intermediate unknown data points will have the same value as the previous known data point. e.g. use this option to display timeseries of extraction rates.
Help . . .	Click this button to start the iMOD Help Functionality.
	*Print* Click this icon to print the current Graph(s) to a printer.
	*Export* Click this icon to export the current Graph(s) to an ASCII-file (*.csv).
	*Copy to Clipboard* Click this icon to copy the current Graph(s) to the windows *Clipboard*. Use the shortcut *Ctrl-C* , alternatively
	*Redraw* Click this button to redraw to the graphical content.
	*Zoom Full* Click this button to zoom to the entire extent of the Graph(s).
	*Zoom Rectangle* Click this button to zoom in for a rectangle. Use the left-mouse button to determine the lower-left corner of the rectangle, click again for the upper-right corner (or vice-versa). All graphs will be adjusted accordingly.

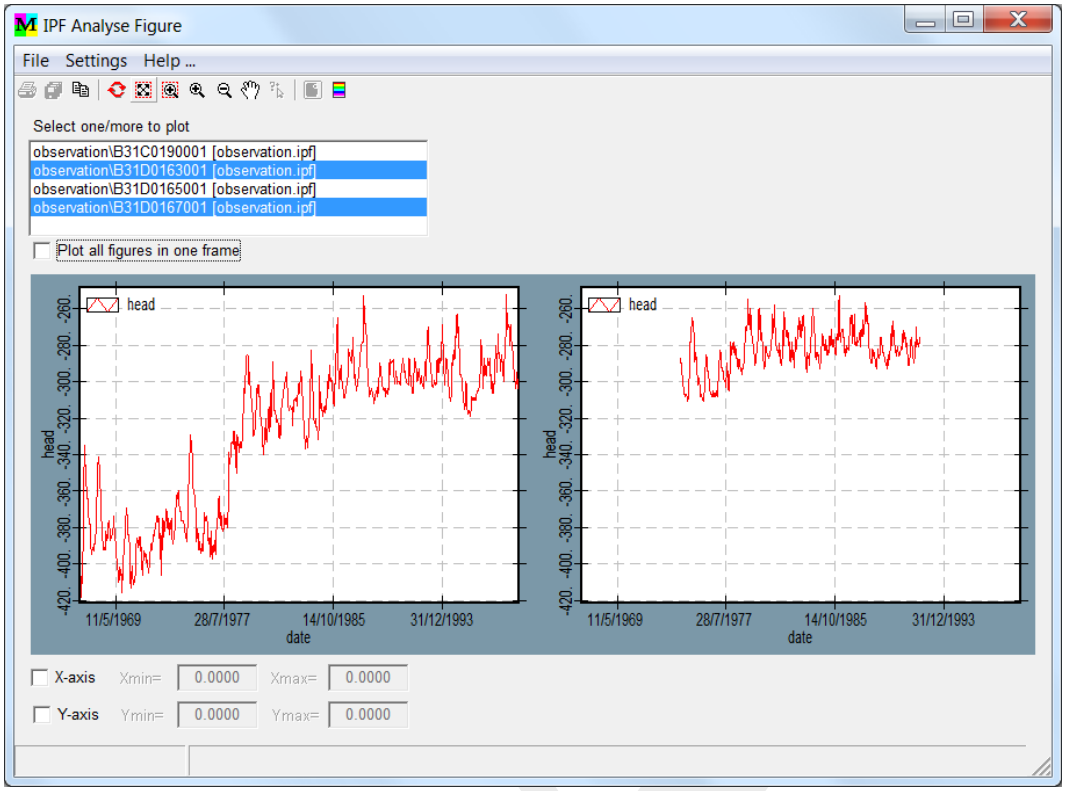
	*Zoom In* Click this button to zoom IN on the centre of the current Graph(s).
	*Zoom Out* Click this button to zoom OUT on the centre of the current Graph(s).
	*Move* Click this button to move the current Graph(s). Click the left-mouse button on that location where you want to move from, repeat this after the display has been refreshed (automatically). Use the right mouse button to stop the moving process.
All data for above selected file	The maximal number of records that can be displayed in the table is 500. Whenever, more data is found in the associated file, iMOD will display this warning (e.g. displayed only 500 out of total 812 record) and will not present data that exceeds the number of 500 records. However, they will be presented in the graph.
Select one/more to plot	Select one or more of the listed files to plot. For each file that is selected, a new graph will be displayed.

Example of two timeseries displayed:



Plot all figures in one frame	This option is available whenever more than one file is active in the *IPF Analyse Figure* window. It allows you to combine the selected files in a single graph.

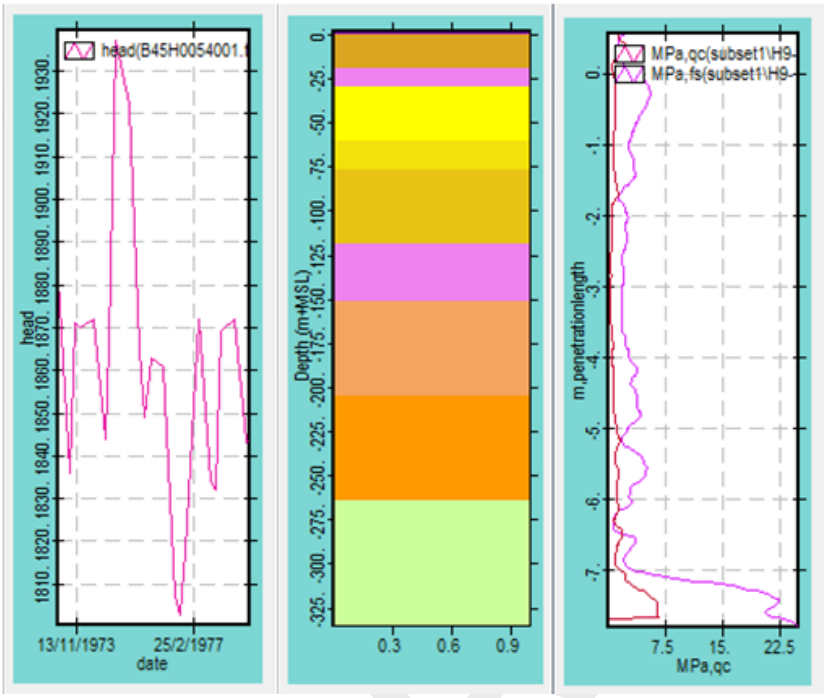
Example combining different files together in a single frame:



X-axis	Select this option to specify the minimum and maximum values for the x-axis. These values remain active for graphs that are plotted on the graphical canvas as well as specified on the *Settings* tab on the *IPF Analyse* window.
Y-axis	Select this option to specify the minimum and maximum values for the y-axis.These values remain active for graphs that are plotted on the graphical canvas as well as specified on the *Settings* tab on the *IPF Analyse* window.
Visible	Select a attribute from the dropdown menu to adjust the plotting setting associated with it.
Red colour box	Click in this field to open a *Window Colour* Window in which a colour can be depicted to be used for the selected field from the dropdown menu.
Thin	Select this option at the *Table of Associated Files Content* window to apply a thin line thickness.
Thick	Select this option at the *Table of Associated Files Content* window to apply a thick line thickness.
Scale column on plot area, given a factor:	Select this option to define the multiplication factor per data column selected in the dropdown menu.
	*Update* Click on this button to update the *IPF Analyse Figure* plot window with the defined factor(s).
	*Reset* Click on this button to reset all the defined factors (for all columns) to 1.0 and refresh the *IPF Analyse Figure* plot window to the initial values.
	*Scale All* Click on this button to apply the defined factor to all columns at once and refresh the *IPF Analyse Figure* plot window with the defined factor(s).
Scale each data column seperately on plot area	Select this option to plot each data column on a different x-axis next to the previous data column. This can make it easier to analyze each data column seperately. This option only works in case of cone penetration or well log data.

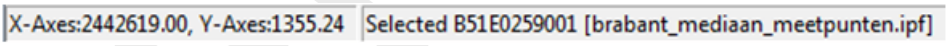
It depends on the type of the selected associated files what kind of figure will be displayed.

Example of (left) timeseries, (middle) boreholes and (right) borelogs in an IPF Analyse Figure window:



Note: Whenever you move the mouse in the graphical area, the coordinates of the current graph will be displayed underneath. If more than one file is selected, the current selected graph will be displayed too.

Example of display of the current position in the graph that shows the timeseries of B15E0259001:



6.8.4 IPF Extract

WHY?

An IPF-file containing a subset of point data is extracted from an IPF-file.

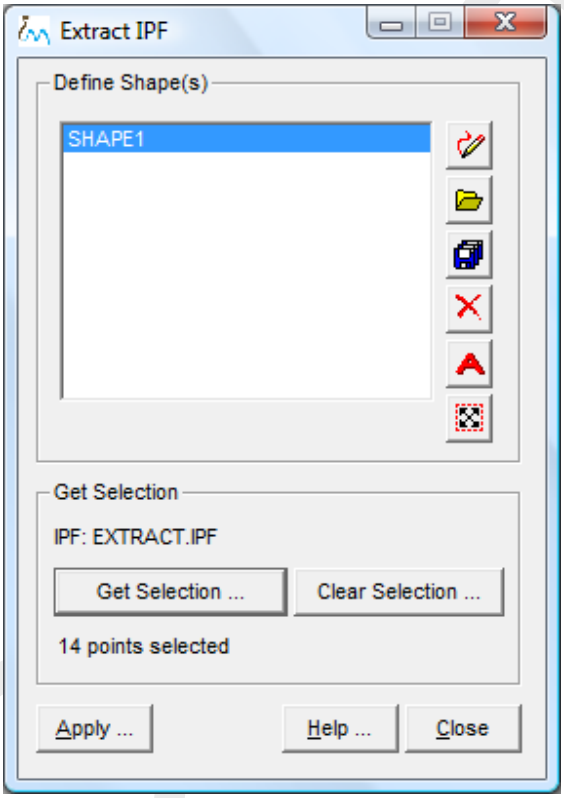
WHAT?

A new IPF-file is created by selecting point locations from an existing IPF-file.

HOW?

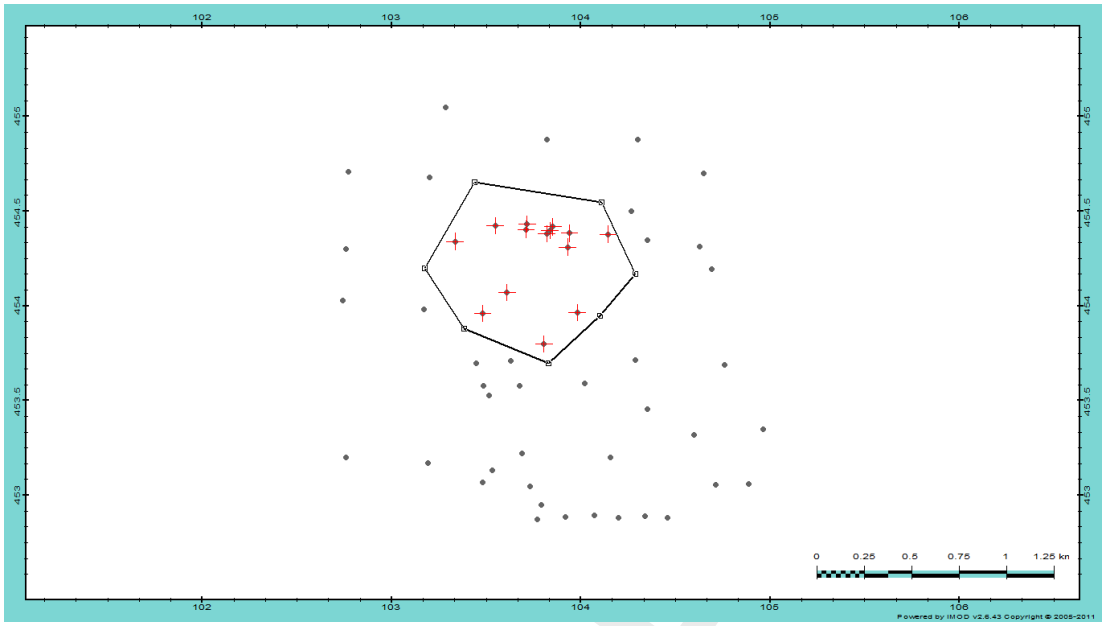
Select a single IPF file in the *iMOD Manager* for which points need to be extracted and saved in a new file. Select the option *Map* from the main menu, choose the option *IPF-options* and then the option *IPF Extract* to display the *IPF Extract* window.

IPF Extract window:



	Click these buttons to draw, open, save, delete or rename a shape. More detailed information can be found in section 4.2
IPF:	Displays the IPF filename selected in the iMOD Manager.
Get Selection ...	Click this button to start the IPF Find window (section 6.8.5). Whenever you have specified a polygon, bear in mind that only those points will remain that are inside the selected polygons. All selected points will be marked by a red cross.
Clear Selection ...	Click this button to remove the current selection. You will be asked to confirm this action.
14 points selected	Shows the number of selected points.
Apply . . .	Click this button to select a new IPF file to save the selected IPF points. If this action is successfully, the IPF Extract window will close and the new created IPF will be added to the iMOD Manager.
Help . . .	Click this button to start the Help functionality.
Close	Click this button to close the IPF Extract window.

Example of selected IPF Points:



Note: Associated files will be copied too. If the original IPF relates to a relative folder, e.g. “asfiles\”, the new IPF will copy those associated files to a relative folder “asfiles\” below the folder of the new IPF file.

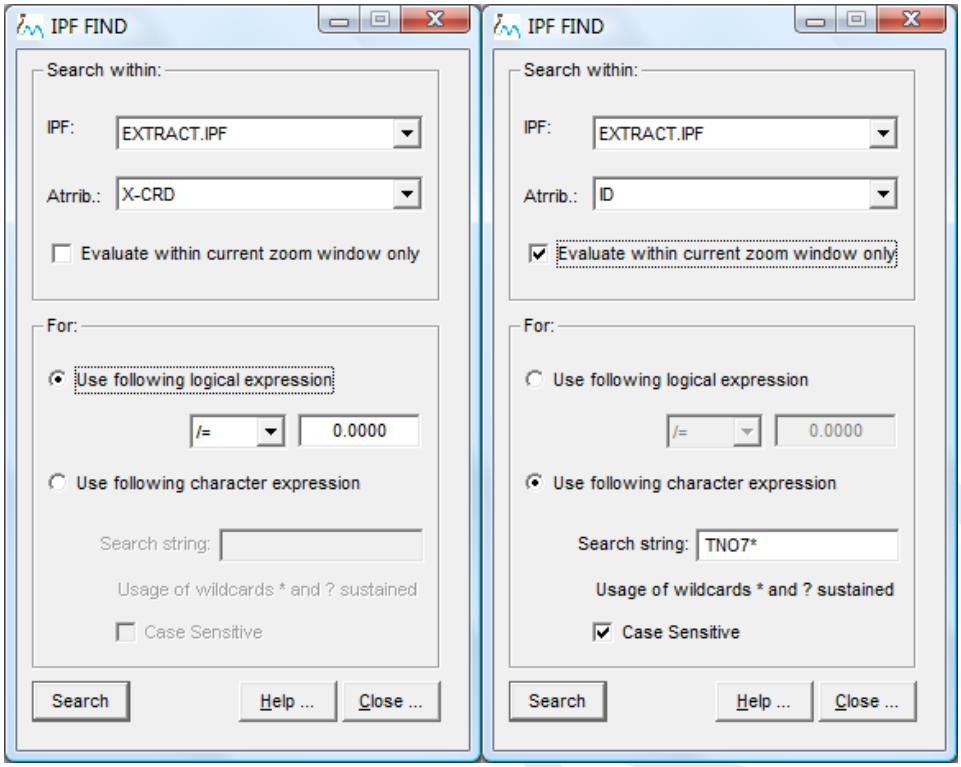
6.8.5 IPF Find

WHY?
To create a new IPF-file by extracting points from an existing IPF-file.

WHAT?
The selection of point data is made inside an interactively defined rectangle or polygon using a logical expression for numerical data or using a character expression for alphanumeric data.

HOW?
Select the option *IPF Find* from the *Extract IPF* window to start the *IPF Find* window.

IPF Find window, (left) using a logical expression and (right) using a character expression:



IPF:	Displays the active IPF file.
Attrib.:	Select one of the attributes within the IPF file to be used for the selection of points.
Evaluate within current zoom window only	Select this checkbox to force a search of points within the current zoom level only.
Use following logical expression	Select this option to specify a logical operator and enter a numeric value to be evaluated. Select one of the following expression (only for numeric values!): **<:** Less than **<=:** Less or equal to **=:** Equal to **>:** Greater than **>=:** Greater or equal to **\=:** Not equal to
Use following character expressions	Select this option to specify a search string, e.g. *TNO7**. Use the character "*" to identify that any character is valid and the "?" to denote that any character is valid but for that number of positions equal to the number of "?"-marks.
Case sensitive	Select this item to apply a case sensitive search on characters.
Search	Click this button to start the search process.
Help . . .	Click this button to start the Help functionality
Close	Click this button to close the *IPF Find* window.

6.9 IFF Options

6.9.1 IFF Configure

WHY?

IFF-files (iMOD Flowpath Files) in iMOD represent line data generated by the Pathline Simulation function (see section 7.14). This function uses IMODPATH to compute flowlines based on the budget terms that result from an iMODFLOW computation.

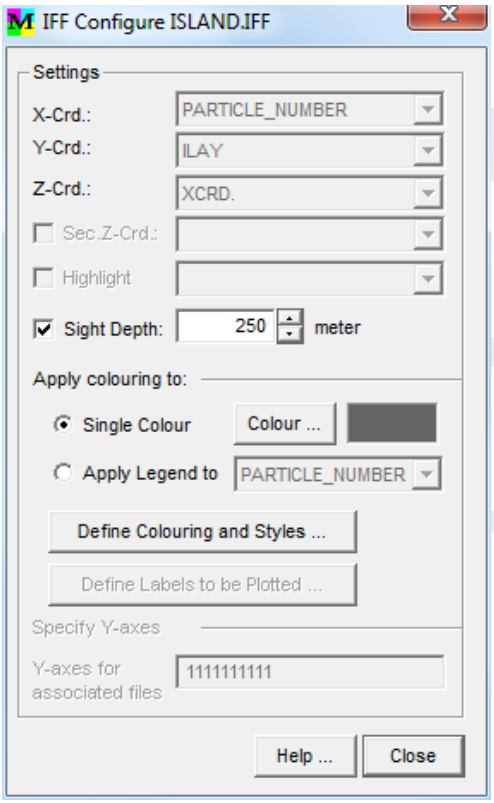
WHAT?

IFF Configure is used to define the settings for display and assigns a symbol, colour or label to the flowlines. The options are similar to the *IPF Configure* function (see section 6.8.1).

HOW?

Select the menu option *IFF Configure* from the *IFF-options* menu in the *Map* menu to display the *IFF Configure* window. Or, use right-click anywhere on the canvas to open the popup menu. Select the option *IFF-options* and then choose *IFF Configure*.

IFF Configure window:



The IFF-configure window is comparable to the IPF-configure window (see section 6.8.1) except that some functions are not active.

X-Crd.:	Not active
Y-Crd.:	Not active
Z-Crd.:	Not active
Sec. Z-Crd.:	Not active
Highlight	Not active
Sight Depth	Select and specify an interval in meters, over which points need to be displayed.

Single Colour	Select this option to display all points with the same colour.

Colour . . .	Select this button to display the default *Colour* window in which a colour can be specified. The current colour is displayed to the right of this button.
Apply Legend to	Select this option to colour the lines according to the selected attribute that is chosen in the dropdown menu at the right.
Define Colouring and Styles. . .	Select this option to display the *Lines and Symbols* window, section 5.7.
Define Labels to be Plotted	Not active
Y-axes for associated files	Not active
Help. . .	Click this button to start the iMOD Help Functionality (if available in the selected *.PRF file).
Close	Select this button to apply the settings and close the *IPF Configure* window.

6.10 ISG Options

ISG-files contain all necessary information to simulate a river segment, such as: location, time dependent waterlevels, cross-sections, structures. iMOD supports functionalities that manipulate these ISG-files:

- **ISG Configure**,
- **ISG Edit**, use this tool to analyse and/or adjust the content of the ISG-file.
- **ISG Show**, specify what attribute need to be plotted.

6.10.1 ISG Configure

WHY?

ISG-files are used in iMOD to simulate the location of the surface water system.

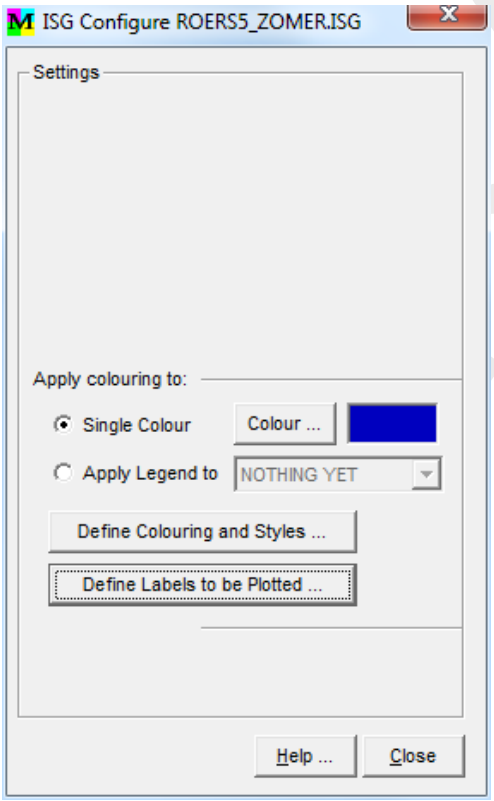
WHAT?

ISG-files in iMOD represent line and point data that can be displayed in different ways.

HOW?

Select the menu option *ISG Configure* from the *ISG-options* menu in the *Map* menu to display the *ISG Configure* window. Or, use right-click anywhere on the canvas to open the popup menu. Select the option *ISG-options* and then choose *ISG Configure*.

ISG Configure window:



Settings	Not yet implemented
Apply colouring to:	The lines can be presented in a single colour or in different colours according a defined legend
Single colour	Select this option to display all lines with the same colour. The colour can be changed by clicking the *Colour . . .* button.

Apply legend to	Not yet implemented
Define Colouring and Styles...	Select this option to display the *Define Colouring and Styles* window
Define Labels to be plotted	Select this button to display the *Select Label to be Printed* window. Not yet implemented.
Help...	Click this button to start the iMOD Help Functionality (if available in the selected *.PRF file).
Close	Select this button to apply the settings and close the *IPF Configure* window.

6.10.2 ISG Show

WHY?

To show the location of point data of ISG-files.

WHAT?

ISG-files contain six types of point data which are connected to the river segments: nodes, segment nodes, cross-sections, calculation nodes, structures and QH-relationships.

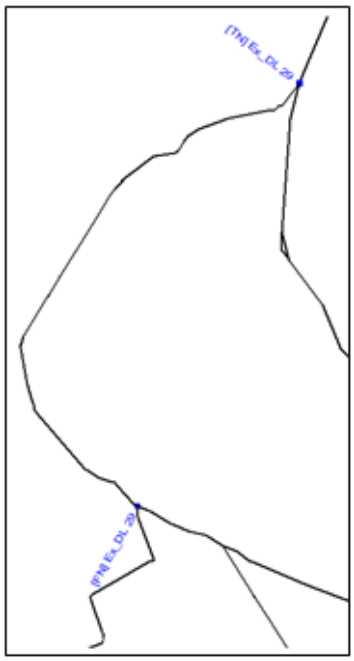
HOW?

Select at least one ISG-file in the *iMOD Manager* and use the option *Map* from the main menu (or right click your mouse on the graphical canvas), choose the option *ISG-options*, and then *ISG Show* to list the following categories (see also next page)

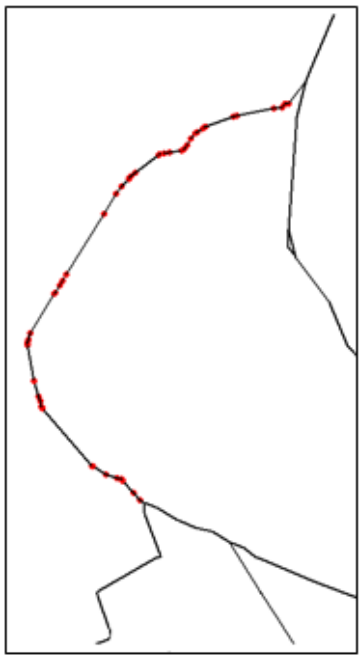
- **Nodes**
  Check this item to display the FromNode (FN) and ToNode (TN) of the river segment (symbol = rectangle) and their labels;
- **Segment Nodes**
  Check this item to display the nodes (symbol = solid circle) that define the river segment;
- **Calculation Nodes**
  Check this item to display the calculation nodes (symbol = square with a crossline) that contain information on waterlevels, bottom levels, infiltration resistance and infiltration factors, all time dependent.
- **Cross-Sections**
  Check this item to display the cross-sections (symbol = polygon shape of cross-section) that are available on the river segment containing information on the shape of the river bed;
- **Structures**
  Check this item to display the locations (symbol = triangle) that contain information on waterlevels before and after weirs/structures, both time dependent;
- **QHW-relationships**
  Check this item to display the nodes (symbol = square with two colours) with a discharge-head relationship;
- **Flow Direction**
  Check this item to display the flow direction (symbol = arrow in the direction of the flow). The flow direction is the order in which the coordinates for the segment are entered. This can be changed by the *Rotate*-button on the *Coordinates* -tab on the *ISG Attributes*-window.

Note: Colouring of all above mentioned attributes can be defined in the *ISG Edit* window.

Examples of the symbols used for all categories available in ISG-files:

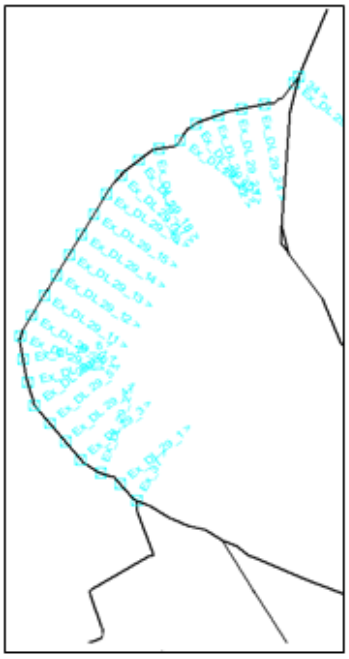


Nodes



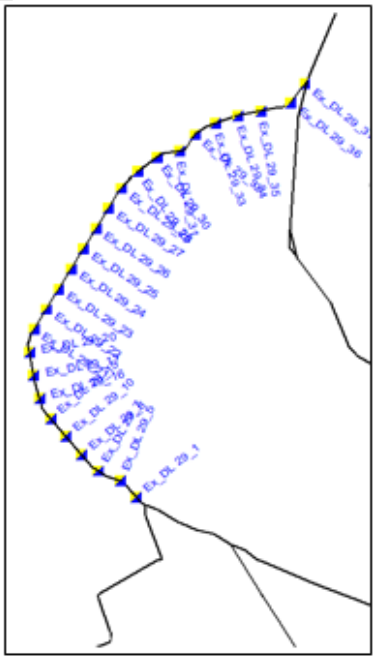
Segment nodes

Cross-sections



Calculation nodes

Structures



Q-H relationships

6.10.3 ISG Edit

WHY?

To add, delete or adjust interactively the line and point (attribute) data stored in an ISG-file.

WHAT?

The *ISG Edit* window has seven tabs:

- ◇ *Segments*: to select one or more segments from the list; actions on the segments can be executed, such as viewing in a profile or conversion to a raster;
- ◇ *Polygons*: to define or load a polygon to use in the selection;
- ◇ *Attributes*: to remove or adjust one or more of the attributes;
- ◇ *Calc. Points*: to define the value to be adjusted of the attribute Calculation points; calculation points are the points on a segment where a water level is calculated;
- ◇ *Structures*: to define the value to be adjusted of the attribute Structures; Structures are the weirs on a segment where a (fixed) water level is maintained;
- ◇ *Cross-sections*: to define the value to be adjusted of the attribute Cross-sections; Cross-sections are the points on a segment where a cross-section is defined;
- ◇ *QWD relationships*: to define the value to be adjusted of the attribute Q-Width-Depth relationships; Q-Width-Depth relationships are the points on a segment where the relation between the discharge and the width and depth of the water level is defined.

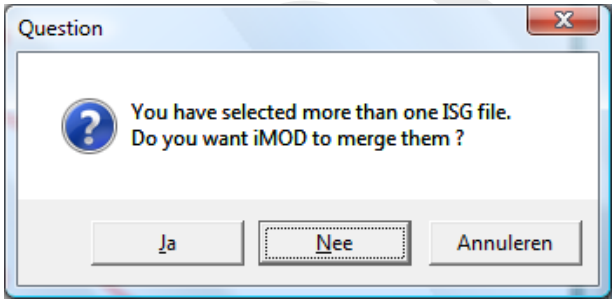
The bottom part of the ISG Edit window shows the display settings of the segment attributes.

HOW?

Select at least one ISG-file in the *iMOD Manager* and use the option *Map* from the main menu (or right click your mouse on the graphical canvas), choose the option *ISG-options*, and then *ISG Edit* to display the *ISG Edit* window.

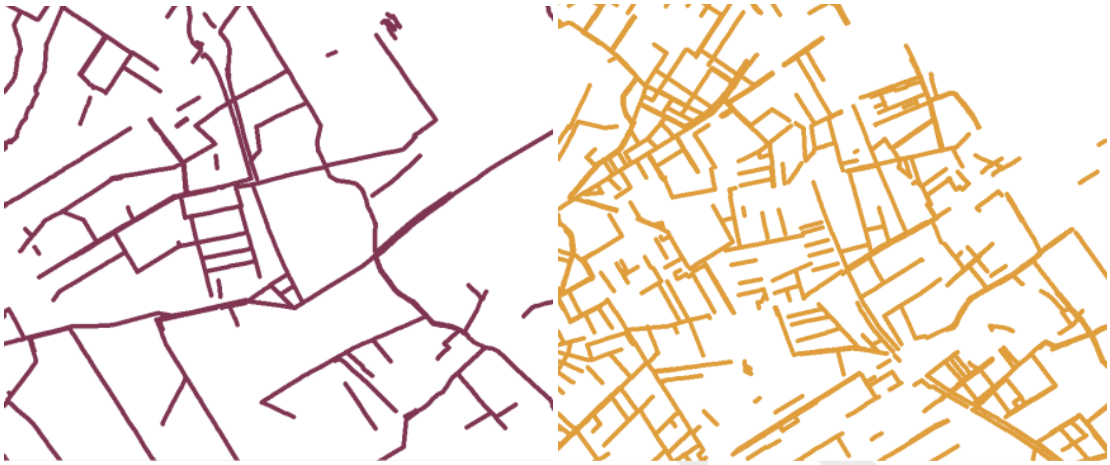
Note: In case, more ISG-files are selected in the *iMOD Manager*, prior to starting the *ISG Edit* option, iMOD will offer the possibility to merge all selected ISG-files into a single ISG-file.

Question window:



The example below shows how two different ISG-files would be merged into a single one.

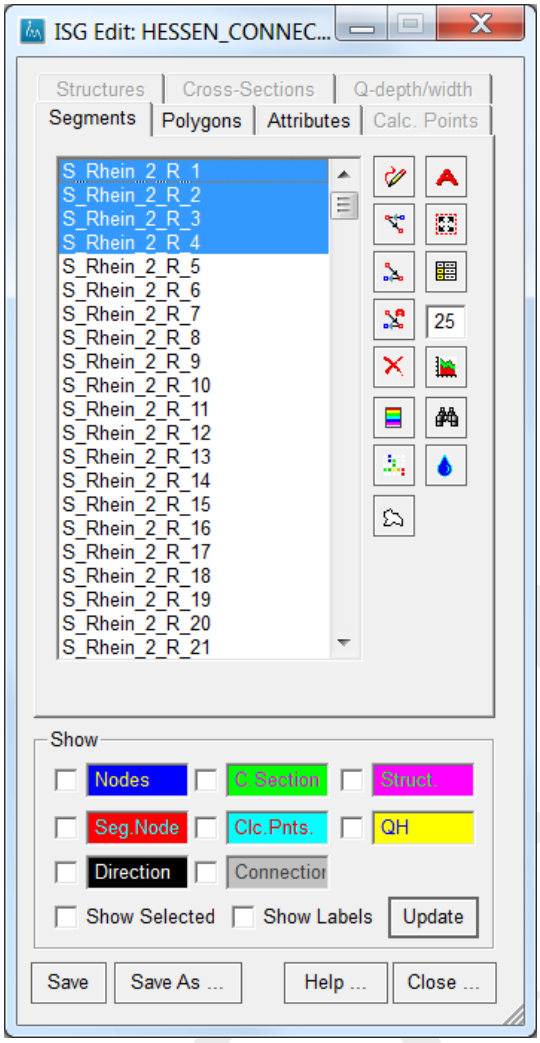
Example of ISG's of a primary and secondary surface water system



Example of an ISG capturing the combined primary and secondary surface water systems

6.10.3.1 ISG Edit window, Segments tab:

Use this window to select segment for modification, create new segments, delete existing segments and or start a variety of analyse functionalities, such as visuals of length-profiles, colouring of attributes on segments, rasterize parameters and so on.



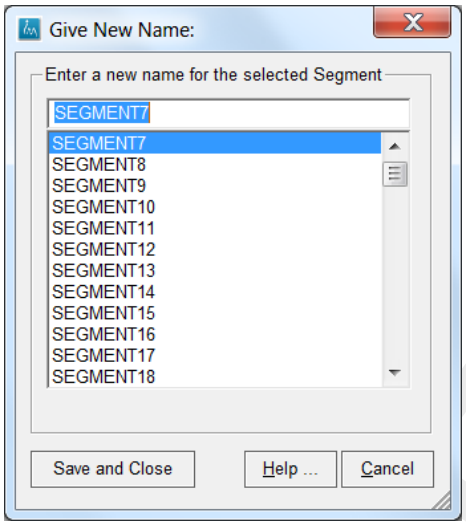
Segment List	This list shows all the available river segments in the current opened ISG-file, e.g. 17.ISG. Select at least one river segment from the list to activate the *Delete, Profile* and *ZoomSelect* options. Moreover, a segment can be selected on the graphical canvas whenever the mouse is left-clicked near a Segment Node. A segment that is selected will be presented as a green line and can be edited moving your mouse on the line. Several options are available using the right-mouse button.

Example of a selected segment,
(left) moving an existing node, (right) adding a new node:

Rename

Click this button the rename the selected segment point. The *Give New Name:* window will appear.

Give New Name window:



ZoomSelect

Click this button to adjust the zoom level to fit the selected segments.

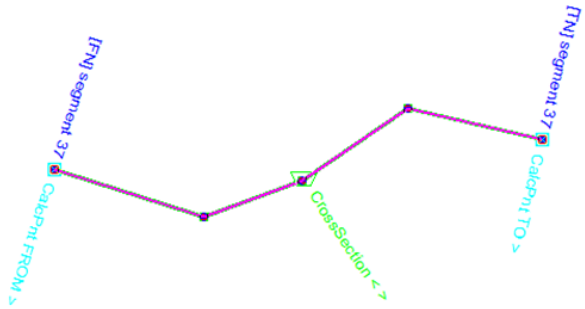
Attributes

Click this button to start the *ISG Attributes* window, see section 6.10.3.9.

DrawSegment

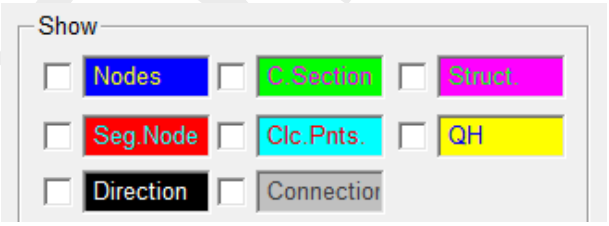
Click this button to start drawing a new segment. On default, the name of the segment will be Segment{number}, it has two calculation nodes (one at the beginning and one at the end, that are both compulsory) and a single cross-section in the middle (one cross-section is compulsory for each segment).

Drawing a new segment:



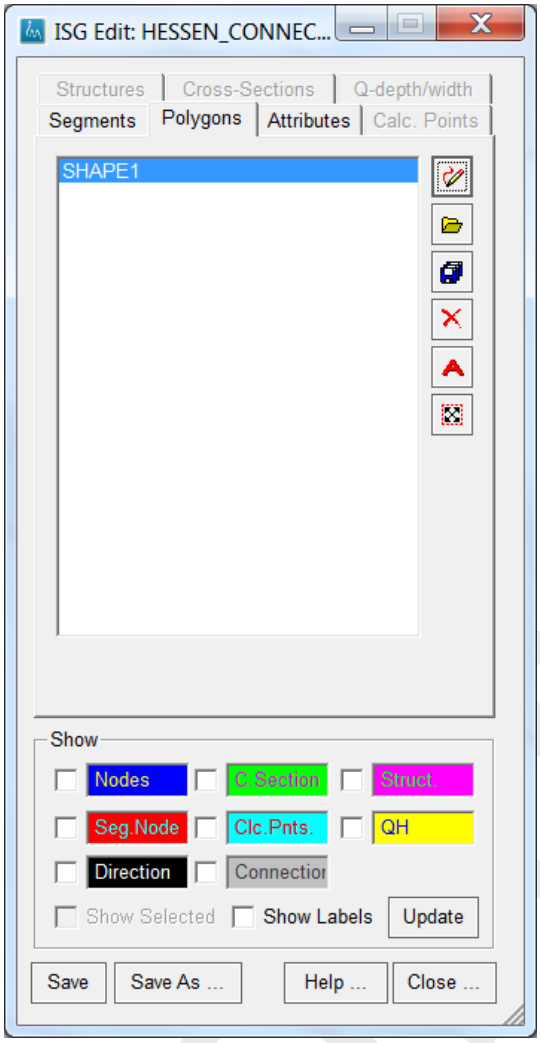
Connect Upstream

Click this button to select the upstream segment interactively, a *Pipet* cursors appears ( ) and any segment that is near your mouse position will be highlighted. Use your left mouse button to select the segment that need to be used from the upstream segment. Remove an entered upstream by clicking next to any of the segments. Terminate the selection process by the clicking the right or middle mouse button.

	*Connect Downstream* Click this button to select the downstream segment interactively, a *Pipet* cursors appears () and any segment that is near your mouse position will be highlighted. Use your left mouse button to select the segment that need to be used from the downstream segment. Remove an entered upstream by clicking next to any of the segments. Terminate the selection process by the clicking the right or middle mouse button.
	*AutoConnect Downstream* Click this button select the downstream segments, automatically. iMOD avoids any recursive connections, so whenever the first point of a nearby segment is within the selected distance (*Snap Distance*) but it connects (in)direct to the selected segment, it is not used as a downstream segment. Eventually iMOD picks the nearest of all suitable segment within the *Snap Distance*.
	*Drip* Click this button to select all connected, downstream segments from the selected isg segments, automatically. iMOD select all connected segments from the initial selected segment.
	*ISG Search* Click this button to start the *ISG Search* window, see section 6.10.3.11.
	*ISG-Rasterize* Click this button to start the *ISG Rasterize* window, see section 6.10.3.13.
	*Profile* Click this button to start the *ISG Profile* window, see section 6.10.3.12.
	*Legend* Click this button to start the *ISG Legend* window, see section 6.10.3.10.
	*Delete* Click this button to remove the selected river segment(s) from the current list of segments. You need to confirm this action. Unless the *Save* option is applied, the segment is removed from memory only.
	*Select in Polygon* Click this button to draw a polygon and select all segment inside the polygon. If at least one point of the segment falls within the polygon, the segment is selected.
Save	Click this button to save the loaded ISG to disc, using the original ISG-filename.
SaveAs ...	Click this button to save the loaded ISG by another filename.
Show Labels	Click the checkboxes for the attributes to be plotted.  Click on the input field (*Nodes, Seg.Nodes, C.Sections, Clc.Pnts, Struct, QH* or *Direction*) to start the default *Colour* window in which the colour can be changed.
Show Selected	Check this checkbox to display the attributes as defined by *Show* for the selected segment(s) only.
Update	Click this button to redraw the ISG.
Help ...	Click this button to start the Help functionality.
Close ...	Click this button to close the *ISG Edit* window. Whether you've changed the ISG or not, you'll be asked to confirm this.

6.10.3.2 ISG Edit window, Polygons tab:

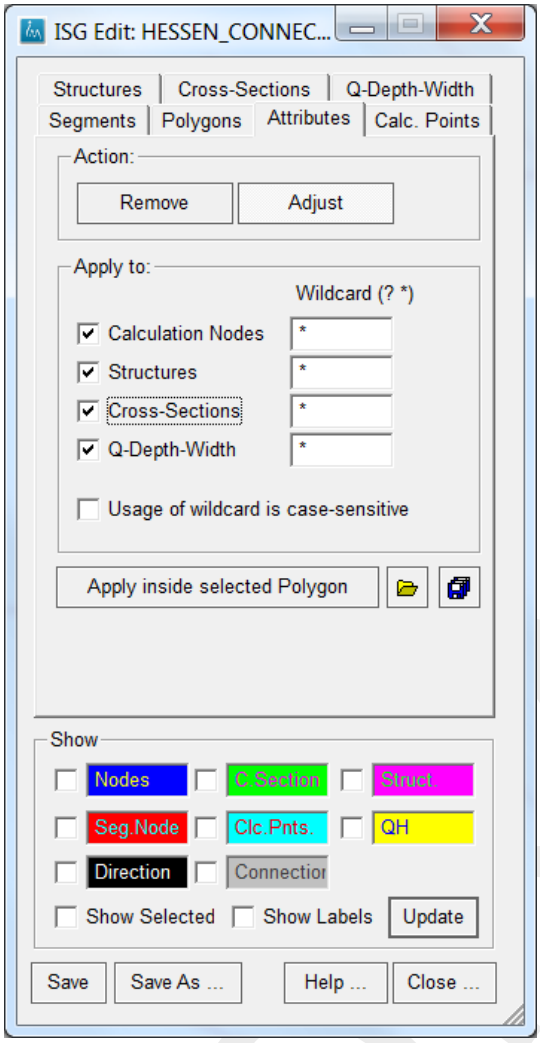
Use this window to enter polygon(s) for which selection and modifications need to be carried out for a variety of attributes of the ISG file.



Click one of these buttons to draw, open, save, delete, rename a shape and/or zoom into the selected shape(s). More detailed information can be found in section 4.2.

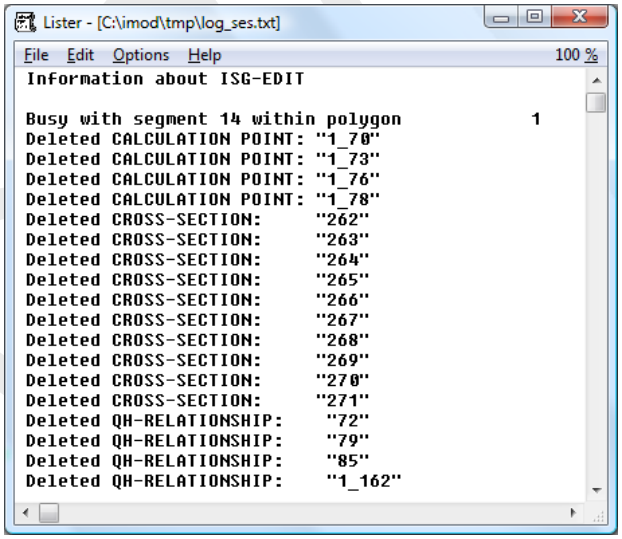
6.10.3.3 ISG Edit window, Attributes tab:

Use this window, to specify what attributes of the ISG file need to be modified and whether this need to be done for selected segments or within the entered polygons.



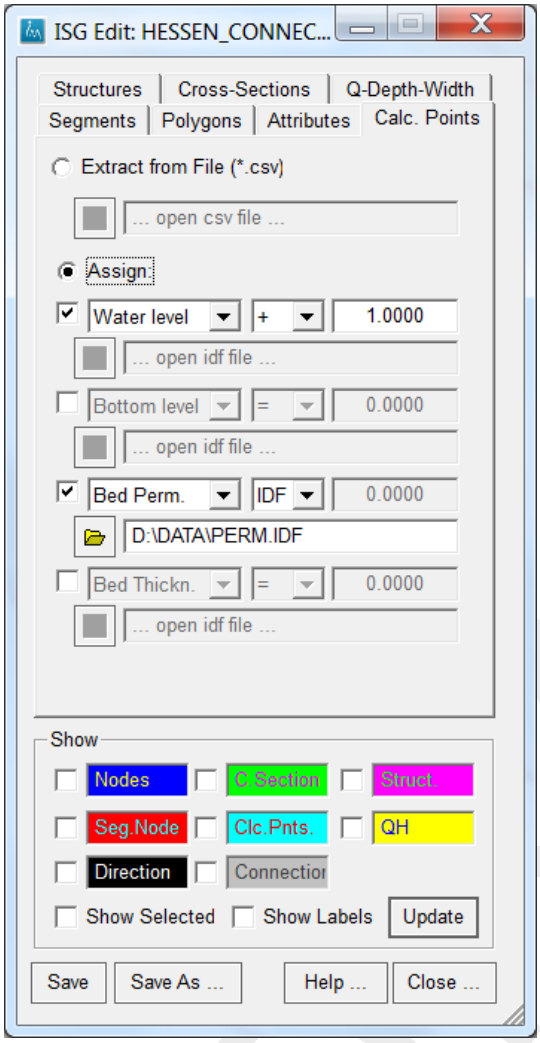
Action Select the following options:

- ◇ *Remove:*
  Click this option to **remove** all selected attributes that match the given search string;
- ◇ *Adjust:*
  Click this option to **adjust** all selected attributes to match the given search string.

Apply to	Select the following options: ◇ *Calculation nodes:* Select this option to apply modification on attributes for calculation nodes, such water levels, inflow, stream widths; ◇ *Structures:* Click this option to apply modification on attributes for structures, such as up- and downstream water levels; ◇ *Cross-Sections:* Click this option to apply modification on attributes for cross-sections, such as widths, depth, Manning's coefficients; ◇ *Q-Depth-Width:* Click this option to apply modification on attributes for discharge-depth-width relation ships, such as discharge, width and depth values. The appropriate tabs (more than one if needed) become available depending on the choice of your selection.
Wildcard (?)*	Enter a search string; e.g. *Test** selects all labels that start with *Test* and end with anything; *??Test* selects all labels that start with two characters followed by *Test* only.
Usage of wildcard is case-sensitive	Select this checkbox to evaluate the search string as case sensitive.
Apply inside selected polygon	Select this button to apply the removal or adjustment to the loaded ISG-file. Whenever at least one polygon is selected on the *Polygons* tab, the operation will affect attributes inside those selected polygons. If no polygons are selected, the operation will affect the selected segment on the *Segments* tab only. **Bear in mind that all these adjustments are stored in the loaded ISG.** A log file (LOG_SES.TXT) will be created and stored in the {user}\tmp folder. This file shows all adjustments to the ISG-file. *An example of the LOG_SES.TXT file:* 
	*Open SES-file* Click this button to open a Segment Edit Settings *.SES file for a more detailed description of this type of file see section 9.22.
	*Save SES-file* Click this button to save the current settings on the *Attributes* tab to a Segment Edit Settings *.SES file. Whenever such a files has been saved, it can be used via the iMOD Batch function ISGADJUST (see section 8.3.5).

6.10.3.4 ISG Edit window, Calc. Points tab:

Use this window to specify what parameters need to be modified and how that need to be carried out. A maximum of four multiply (duplicate) parameters can be applied.



Extract from File (.CSV)*	Select this option to read adjustments for all variables from a CSV File (see section 9.12). The list of parameters depends on the type of ISG (see section 9.9.2)
Assign	Select this option to specify adjustments for all variables separately. Select the checkbox to activate a specific parameter. There is a maximum of four (duplicate) attributes to be modified at the same time. The list of parameters depends on the type of ISG (see section 9.9.2). Specify the kind of adjustment that manipulates the selected variable. Choose an operator from the dropdown menu: ◇ Select "=" to use the entered value; ◇ Select "+" to add the entered value; ◇ Select "-" to subtract the entered value; ◇ Select "/" to divide by the entered value; ◇ Select "*" to multiply by the entered value; ◇ Select "IDF" to sample a value out of the entered IDF-file at the location of the calculation point.

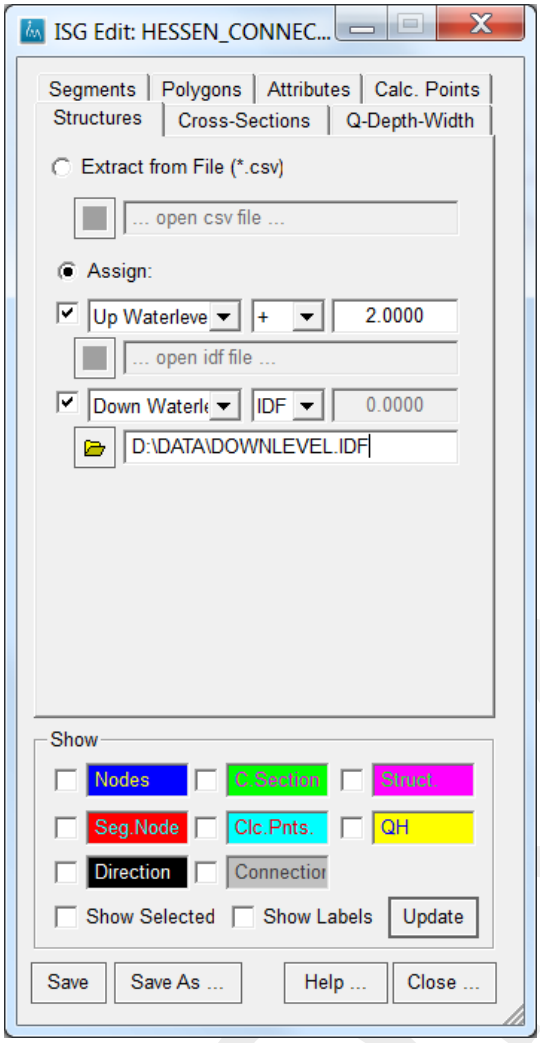
	*Open IDF* Click this button to open an IDF-file. This becomes available whenever the option “IDF” is selected from the dropdown menu for the type of operator.

Note: Adjustments to the number of calculation points can be made by the iMOD Batch function ISGSIMPLIFY (see section 8.3.4).

Note: Adjustment to calculation points can be carried out by the iMOD Batch Function ISGADDSTAGES (see section 8.3.7). This function is suitable of modifying water levels in the ISG via IPF files.

6.10.3.5 ISG Edit window, Structures tab:

Use this tab to specify any adjustment for the up- and downstream water levels at structure, these are simple weirs.

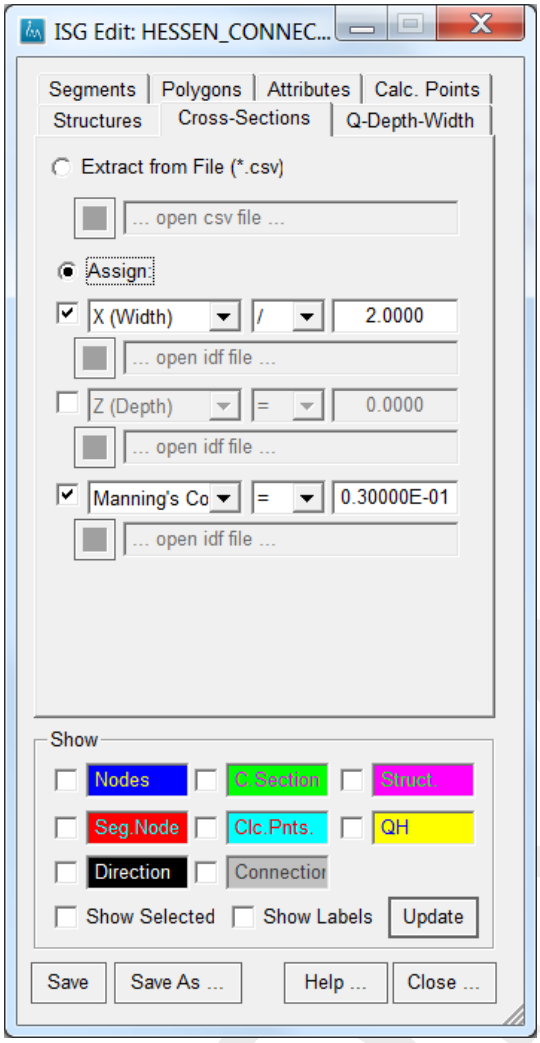


Extract from File (.CSV)*	Select this option to read adjustments for all variables from a CSV File (see section 9.12). The list of parameters is described in section 9.9.4.
Assign	Select this option to specify adjustments for all variables separately. Select the checkbox to activate a specific parameter (“Up Waterlevel” and “Down Waterlevel”). There is a maximum of two (duplicate) attributes to be modified at the same time. Specify the kind of adjustment that manipulates the selected variable. Choose an operator from the dropdown menu: ◇ Select “=” to use the entered value; ◇ Select “+” to add the entered value; ◇ Select “-” to subtract the entered value; ◇ Select “/” to divide by the entered value; ◇ Select “*” to multiply by the entered value; ◇ Select “IDF” to sample a value out of the entered IDF-file at the location of the calculation point.

	*Open IDF* Click this button to open an IDF-file. This becomes available whenever the option “IDF” is selected from the dropdown menu for the type of operator.

6.10.3.6 ISG Edit window, Cross-Sections tab:

Use this tab to specify the modification on parameters for cross-section, such as Width, Depth and/or Manning's coefficient (MRC).



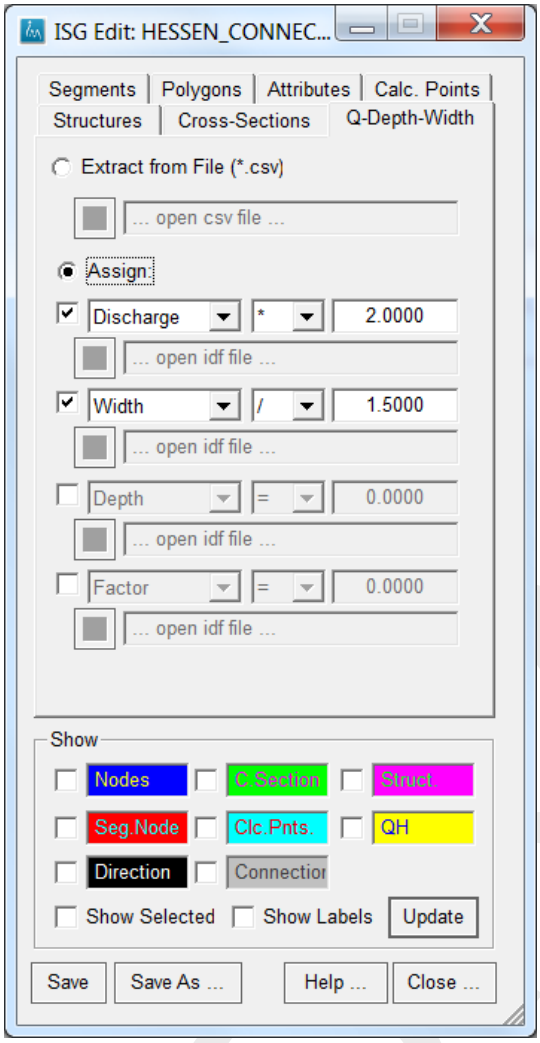
Extract from File (.CSV)*	Select this option to read adjustments for all variables from a CSV File (see section 9.12). The list of parameters is described in section 9.9.3.
Assign	Select this option to specify adjustments for all variables separately. Select the checkbox to activate a specific parameter ("X (Width)" and "Z (Depth)" and "Manning's Coefficient"). There is a maximum of three (duplicate) attributes to be modified at the same time. Specify the kind of adjustment that manipulates the selected variable. Choose an operator from the dropdown menu: ◇ Select "=" to use the entered value; ◇ Select "+" to add the entered value; ◇ Select "-" to subtract the entered value; ◇ Select "/" to divide by the entered value; ◇ Select "*" to multiply by the entered value; ◇ Select "IDF" to sample a value out of the entered IDF-file at the location of the calculation point.

	*Open IDF* Click this button to open an IDF-file. This becomes available whenever the option “IDF” is selected from the dropdown menu for the type of operator.

Note: Cross-sections can be added also by the iMOD Batch function ISGADDCROSSSECTION (see section 8.3.3).

6.10.3.7 ISG Edit window, Q-Depth-Width tab:

Use this tab to specify the modification on parameters for the discharge (Q), -width (W) and -depth (D) relation ships. These can be used for ISG files that are compliant to the SFR packages, see section 9.9.



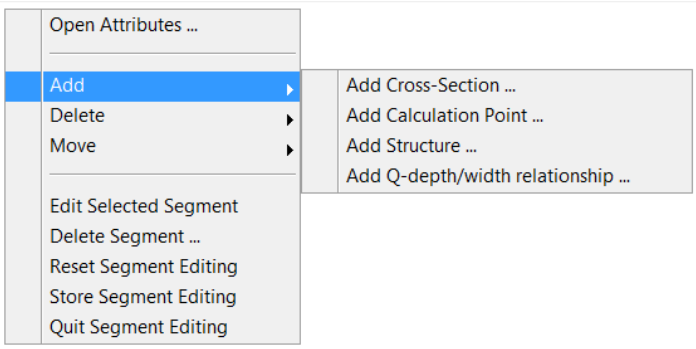
Extract from File (.CSV)*	Select this option to read adjustments for all variables from a CSV File (see section 9.12). The list of parameters is described in section 9.9.5.
Assign	Select this option to specify adjustments for all variables separately. Select the checkbox to activate a specific parameter (“Discharge”, “Width ”, “Depth” and “Factor”. There is a maximum of four (duplicate) attributes to be modified at the same time. Specify the kind of adjustment that manipulates the selected variable. Choose an operator from the dropdown menu: ◇ Select “=” to use the entered value; ◇ Select “+” to add the entered value; ◇ Select “-” to subtract the entered value; ◇ Select “/” to divide by the entered value; ◇ Select “*” to multiply by the entered value; ◇ Select “IDF” to sample a value out of the entered IDF-file at the location of the calculation point.

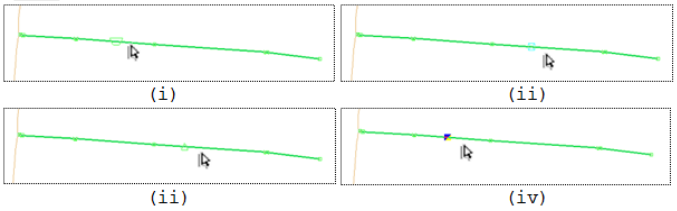
	*Open IDF* Click this button to open an IDF-file. This becomes available whenever the option “IDF” is selected from the dropdown menu for the type of operator.

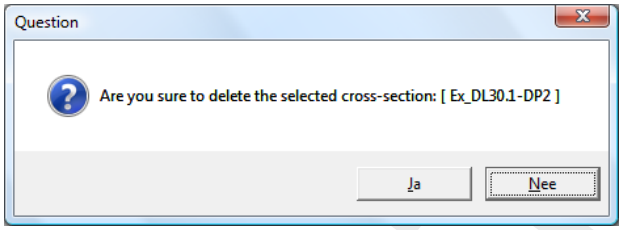
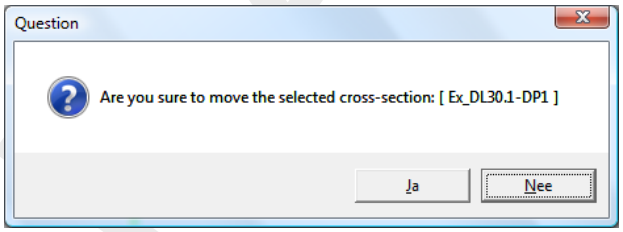
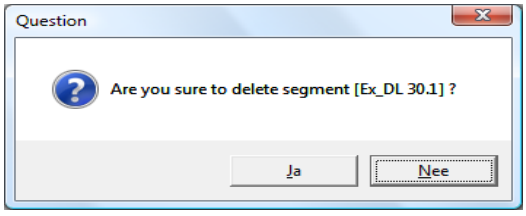
6.10.3.8 Dropdown menu

Once you have selected a segment by either picking it in the menulist in the *Segments* tab on the *ISG Edit* window, or alternatively by clicking your left mouse button at any segment node on the graphical canvas, the following options will be available whenever you press the right mouse button anywhere on the graphical canvas.

Dropdown menu:



Open Attributes . . .	Click this button to start the *ISG Attributes* window.
Add Cross-Section *Add Calculation Point* *Add Structure* *Add Q-Depth-Width Relationship*	Click this button to add a Cross-section, Calculation point, Structure and/or Q-Depth-Width relationship to the selected segment. Click your left mouse button to add the attribute along the selected segment; depending on the type a icon will move along the segment. Click your right mouse button to cancel the operation. *Example of adding different attributes to a segment, (i) cross-section, (ii) calculation nodes, (iii) structure and (iv) Q-Depth-Width relationship:*  (i) (ii) (ii) (iv) Bear in mind that you cannot add additional calculation point, cross-sections whenever the ISG is compliant to SFR.

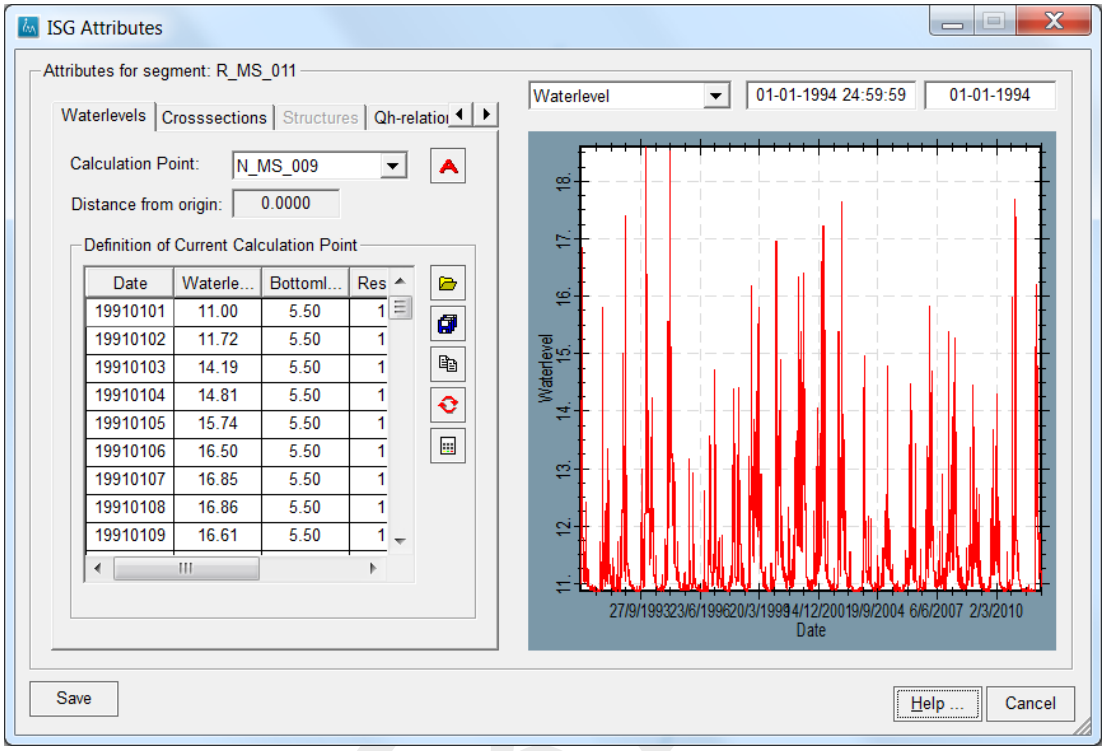
Delete Cross-Section *Delete Calculation Point* *Delete Structure* *Delete Q-Depth-Width Relationship*	Click this button to delete a Cross-section, Calculation point, Structure and/or Q-Depth-Width relationship from the selected segment. Select the feature by moving your mouse in the neighbourhood of the feature and press the left mouse button. iMOD will select the feature that is nearest to the current location of the mouse. You need to confirm any delete action via a *Question* window: *Question window:*  Bear in mind that the begin- and end calculation node may **never** be deleted and a single cross-section is at least obligatory for each segment.
Move Cross-Section *Move Calculation Point* *Move Structure* *Move Q-Depth-Width Relationship*	Click this button to move a Cross-section, Calculation point, Structure and/or Q-Depth-Width relationship from the selected segment. Select the feature by moving your mouse in the neighbourhood of the feature and press the left mouse button. iMOD will select the feature that is nearest to the current location of the mouse. You need to confirm any move action. *Question window:* 
Start Segment Editing	Click this item to start editing the layout of the segment. Whenever in this mode, you can alter the location of each individual node of the segment. The method is similar as used for altering nodes for polygons, see section 4.5. Until you select the option *Save ISG Editing* all modifications to the selected segment are not stored. You need to actively click the *Save* button on the *ISG Edit* window to save the modified segment on disc.
Delete Segment . . .	Click this item to delete the selected segment. You will be asked to confirm this delete operation. *Question window:* 
Reset Segment Editing	Click this item to restore the layout of the segment to that one prior to the moment that the option *Start ISG Editing* was selected.

Store Segment Editing	Click this item to store the current layout of the segment. Bear in mind that as long as the ISG has not been saved on disk using the option *Save* and/or *SaveAs* on the *ISG Edit* window, all changes are stored in memory only.
Quit Segment Editing	Click this item to quit the segment editing. Bear in mind that as long as the ISG has not been saved on disk using the option *Save* and/or *SaveAs* on the *ISG Edit* window, all changes are stored in memory only.

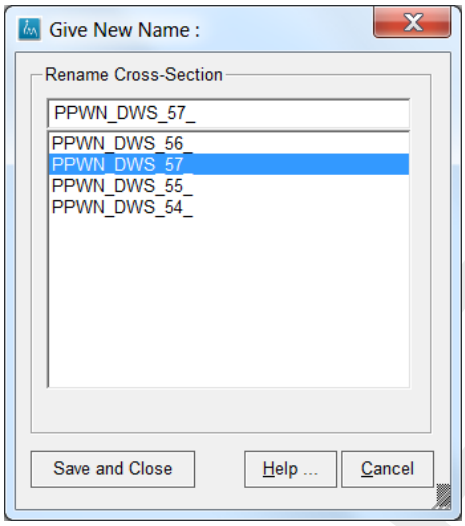
6.10.3.9 ISG Attributes

All attributes that appear on segments, can be analysed and adjusted in the *ISG Attributes* window. Click the *ISG Attributes* button on the *Segments* tab on the *ISG Edit* window to open the *ISG Attributes* window.

6.10.3.9.1 ISG Attributes window, Waterlevels tab:

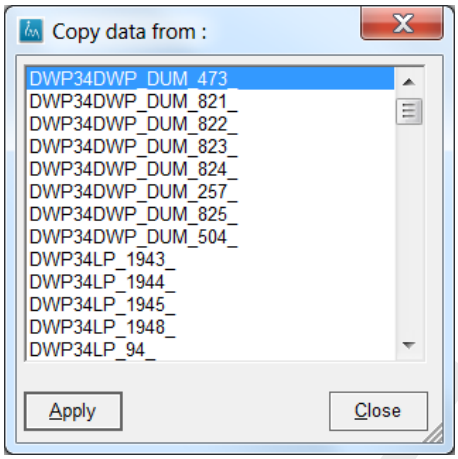


Calculation Point:	Select one of the listed calculation points from the dropdown menu. Any change in this dropdown menu will change the content of the underlying table and update the figure if possible.

	*Rename* Click this button the rename the selected calculation point via the *Give New Name:* window: *Give New Name window:* 
	Save and Close Click this button to store the changed name for the selected segment, the *Give New Name:* window will be closed.
	Help Click this button to start the Help functionality.
	Cancel Click this button to leave the name of the selected segment unchanged and close the *Give New Name:* window
Distance from origin	This field displays the distance of the selected calculation point from the origin of the segment (FromNode) in meters.
Definition of current Calculation Point	This table shows the current values for the current selected calculation point. Use the slide bars to manoeuvre through the table and enter new values for any gridcell if desired. A new record can be entered by filling in all columns. They will be sorted by date automatically after you select the *Redraw* option. It depends on the type of ISG what attributes need to be available, see section 9.9.2 for the exact content and meaning of variables.
	*Open CSV-file* Click this button to open a CSV-file, see section 9.12 for more detailed information about CSV-files.
	*SaveAs CSV-file* Click this button to save to a CSV-file, see section 9.12 for more detailed information about CSV-files.

Copy
Click this button to open the *Copy Data from* window.

Copy data from window:



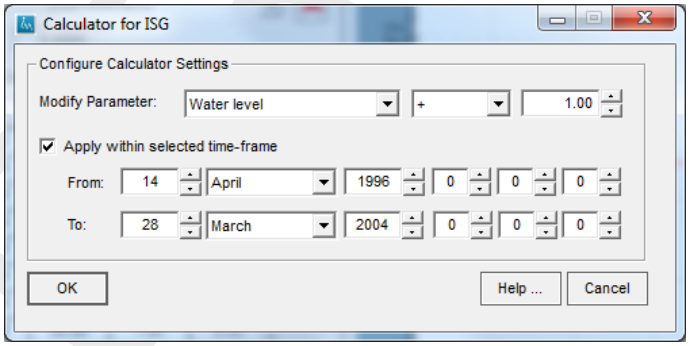
Apply
Click this button to copy the data from the selected calculation point onto the current calculation point and close the *Copy data from* window.

Close
Click this button to close the *Copy data from* window.

Redraw
Click this button to redraw the graphical display.

Calculator
Click this button to start the attribute calculator.

Attribute Calculator window:

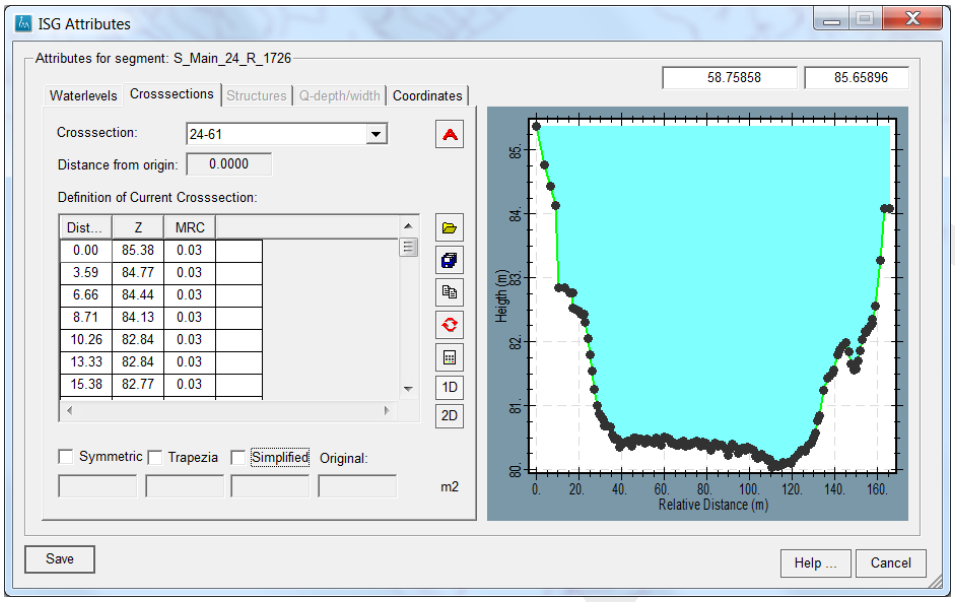


	Modify Parameter Select one of the available parameters from the dropdown list. The content of the dropdown menu depends from which tab the *Attribute Calculator* is started. Select the appropriate modifier from the second dropdown list, choose from: ◇ **+** Select this to add a value to all existing values; ◇ **-** Select this to subtract a value to all existing values; ◇ ***** Select this to multiply a value to all existing values; ◇ / Select this to divide a value to all existing values; ◇ **=** Select this to set a constant value to all existing values. Enter a value in the right most entry field.
	Apply within selected time-frame Click this option to ensure that the modification will be applied within the entered time-frame that can be specified in the underlying entry fields.
	From Specify the date from which any modification will take place.
	To Specify the date to which any modification will take place.
	OK Click this button to modify the parameter as configured and close the *Attribute Calculator* window.
	Help Click this button to start the Help functionality.
	Cancel Click this button to close the *Copy data from* window.
Segment Name:	This field shows the name of the current selected segment.
Current value:	Select one of the attributes (WLevel, Bottom, Resistance, Inf.Factor) from the dropdown menu to be shown in the graph.
Current {}-value:	This field shows the current value in the graph at the position of the mouse cursor.
Save	Click this button to store any modifications and close the *ISG Attribute* window.
Help ...	Click this button to start the Help functionality.
Cancel	Click this button to close the *ISG Attribute* window. You will **not** be asked to confirm to take any adjustments made.

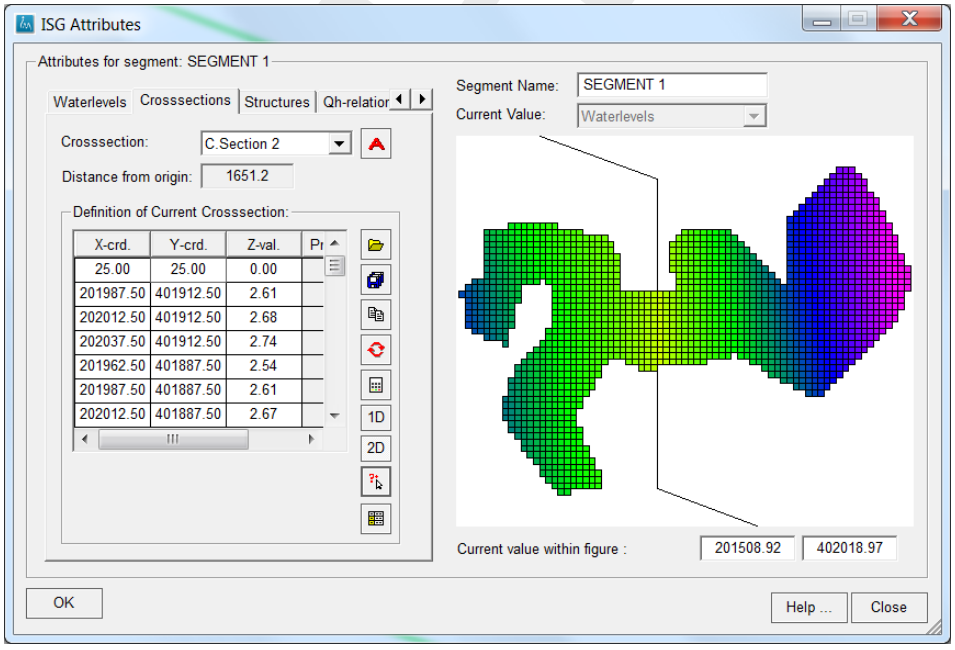
6.10.3.9.2 ISG Attributes window, Crosssections tab:

The *ISG Attributes* window, *Crosssections* tab can have two differences appearances. It depends on the usage of 1-D and 2-D cross-sections. A 1-D cross-section describes the bathymetry of a stream as a representation of a cross-section perpendicular to the direction ot the stream. A 2-D cross-section describes the bathymetry of a stream as a 2-D representation, a grid.

1D-representation



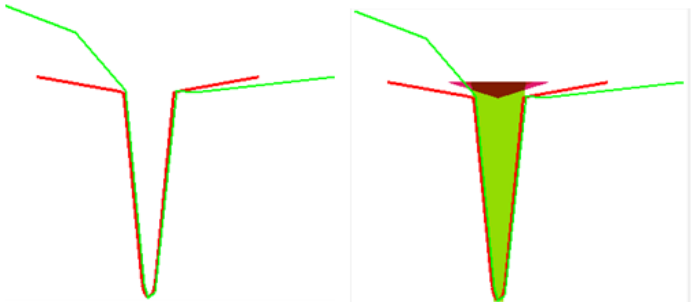
2D-representation



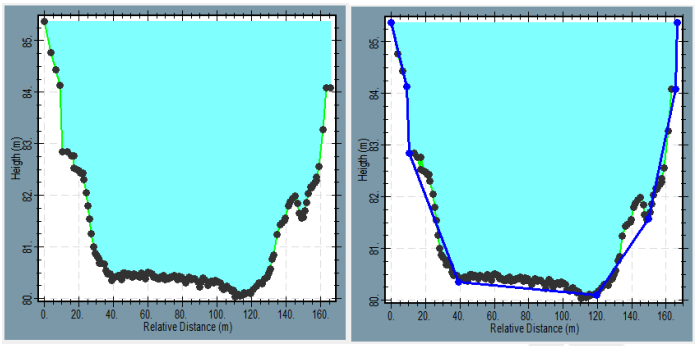
Crosssection:	Select one of the listed cross-sections from the dropdown menu.

	*Rename* Click this button the rename the selected cross-section. The *Give New Name:* window will appear (see previous page).
Distance from origin	This field displays the distance of the selected cross-section from the origin of the segment (FromNode) in meters.
Definition of current Crosssection	This table shows the current values for the current selected cross-section point. Use the slidebars to manoeuvre through the table and enter new values for any gridcell if desired. A new record can be entered by filling in all columns. They will be sorted by date automatically after you select the *Redraw* option. Column names within the table: - DIST: Distance of the cross-section measured from the centre of the riverbed (minus to the left en positive to the right). - Z: Relative level of the riverbed (meter), whereby zero will be assigned to the lowest riverbed level - MRC: Manning's roughness coefficient (-).
	*Open CSV-file* Click this button to open a *CSV* -file, see section 9.12.
	*SaveAs CSV-file* Click this button to save to a *CSV*-file, see section 9.12.
	*Copy* Click this button to open the *CopyAttribute* window.
	*Redraw* Click this button to redraw the graphical display.
	*Calculator* Click this button to start the attribute calculator.
1D	*1D representation* Click this button to view the 1D cross-section coupled to the cross-section point.

The following 3 options are only available in combination with the 1D cross-section representation:

Symmetric	Click this checkbox to determine a symmetric cross-section and compute the area that belongs to it.
Trapezia	Click this checkbox to determine a double multiply trapezia that represents the area computed by the symmetric cross-section most optimally.
	Example of a symmetric cross-section, green is the original cross-section, red the symmetric one: 

Simplified Click this checkbox to determine an eight-point cross-section. This option is used for an SFR compliant ISG file during the creation of the SFR package, see section 12.28.

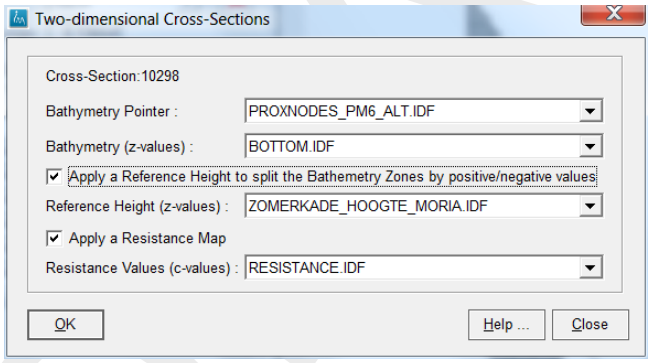


2D *2D representation*
Click this button to view the 2D cross-section coupled to the cross-section point.

The following options are only available in combination with the 2D cross-section representation:

Add/adjust Cross-section point
Click this button to add or adjust a specific cross-section point. The following window will be opened:

Two-Dimensional Cross-section window:



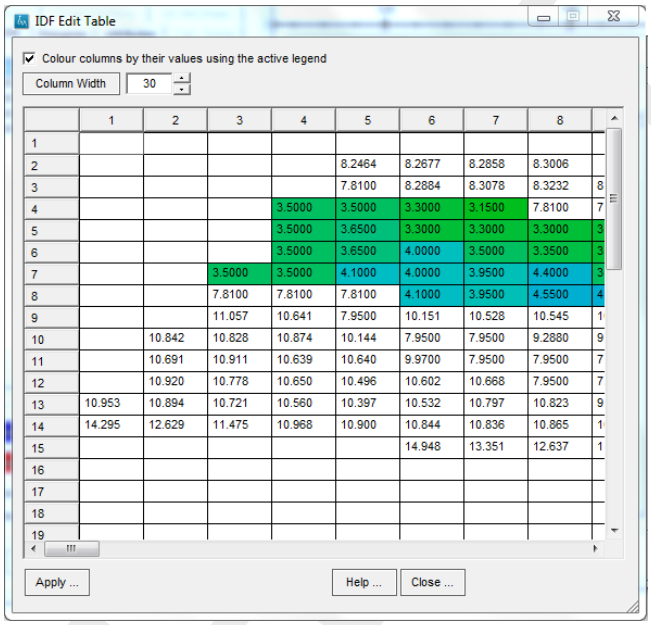
Cross-Section:	This value shows the number of the cross-section point.
Bathymetry Pointer:	Enter the name of an IDF-file describes the spatial distribution of two-dimensional cross-sections.
Bathymetry (z-values):	Select the name of an IDF-file that describes the bathymetry for the riverbed.
Apply a reference height. . .	Select this option in case both negative and positive values in the bathymetry pointer IDF-file needs to be taken into account at a certain reference level in assigning cross-sections.
Referenceheight (z-values)	Select the name of an IDF-file that describes the reference height for which positive and negative pointer values given at *Bathymetry Pointer* need to be applied. A single value is obtained per 2-D cross-section.

Apply a Resistance map	Select this option to insert a spatial variable resistance value for the bathymetry of the 2-D cross-section.
Resistance Values (c-values)	Select the name of an IDF-file that describes the resistance value for the 2-D cross-section. This is an integer multiplication factor with a maximal value of 256.

Edit Cross-Section grid

Click this button to view and edit the cross-section grid related to the selected cross-section point. This option will open the following window:

IDF Edit Table window:

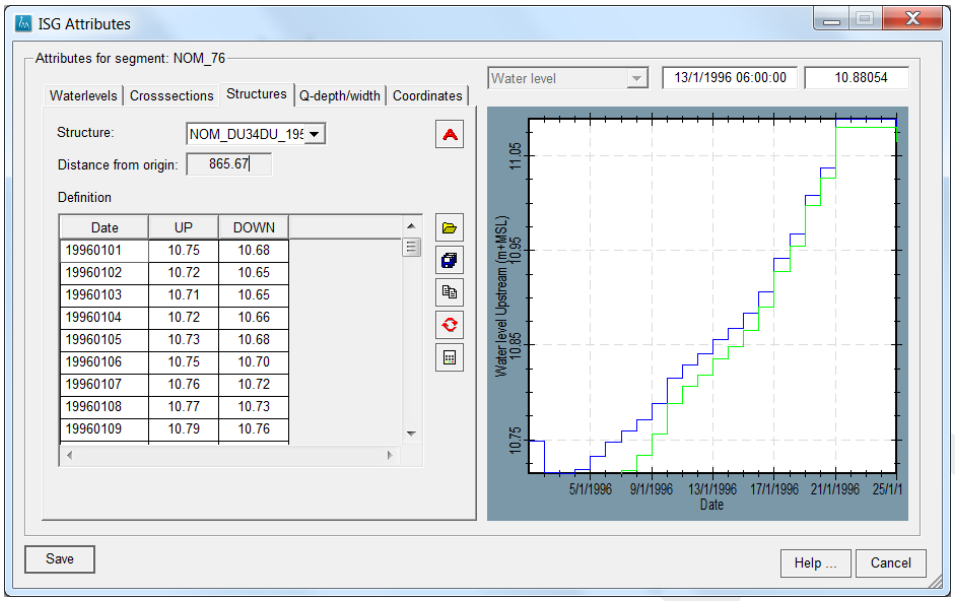


Colour columns. . .	Select this option when coloring the grid cells by the given legend is preferred.
Column width	Click on this button to refresh the column width with the value given in de box to the right of this button.

Note: Cross-sections can be applicable for different sections on a segment. It depends whether it is a one-dimensional cross-section or a two-dimensional one.

6.10.3.9.3 ISG Attributes window, Structures tab:

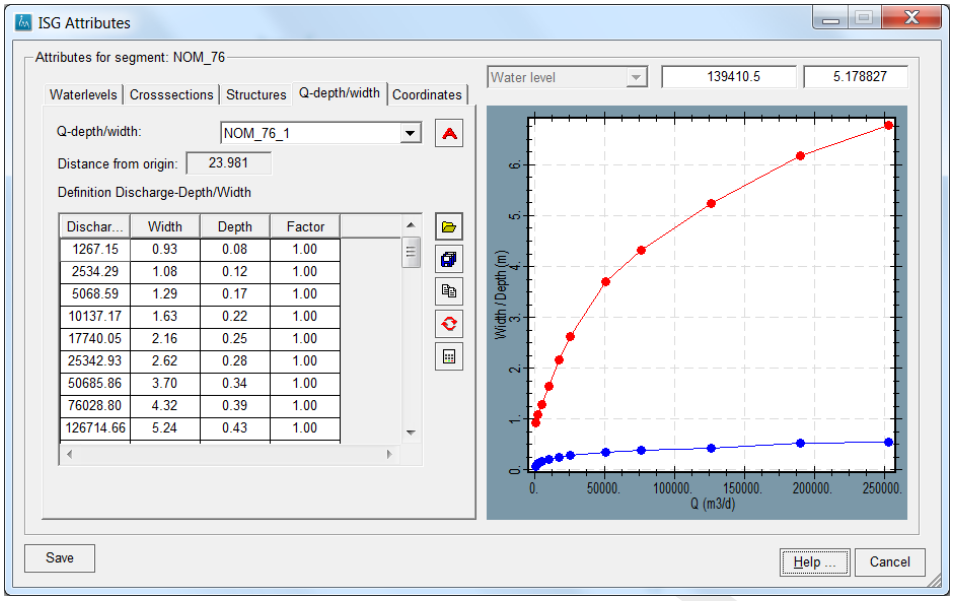
Use this tab to enter information on structures.



Structure:	Select one of the listed structures on the current segment from the dropdown menu.
	*Rename* Click this button the rename the selected structure point. The *Give New Name:* window will appear.
Distance from origin	This field displays the distance of the selected structure point from the origin of the segment (FromNode) in meters.
Definition Structures	This table shows the current values for the current selected structure point. Use the slidebars to manoeuvre through the table and enter new values for any grid cell if desired. A new record can be entered by filling in all columns! They will be sorted by date automatically after you select the *Redraw* option.
	*Open CSV-file* Click this button to open a CSV-file, see section 9.12 for more detailed information about CSV-files.
	*SaveAs CSV-file* Click this button to save to a CSV-file, see section 9.12 for more detailed information about CSV-files.
	*Copy* Click this button to open the *Copy Data from* window.
	*Redraw* Click this button to redraw the graphical display.
	*Calculator* Click this button to start the attribute calculator.
Value boxes on the top-right	These fields shows the current value in the graph at the position of the mouse cursor.

6.10.3.9.4 ISG Attributes window, Q-Depth-Width tab:

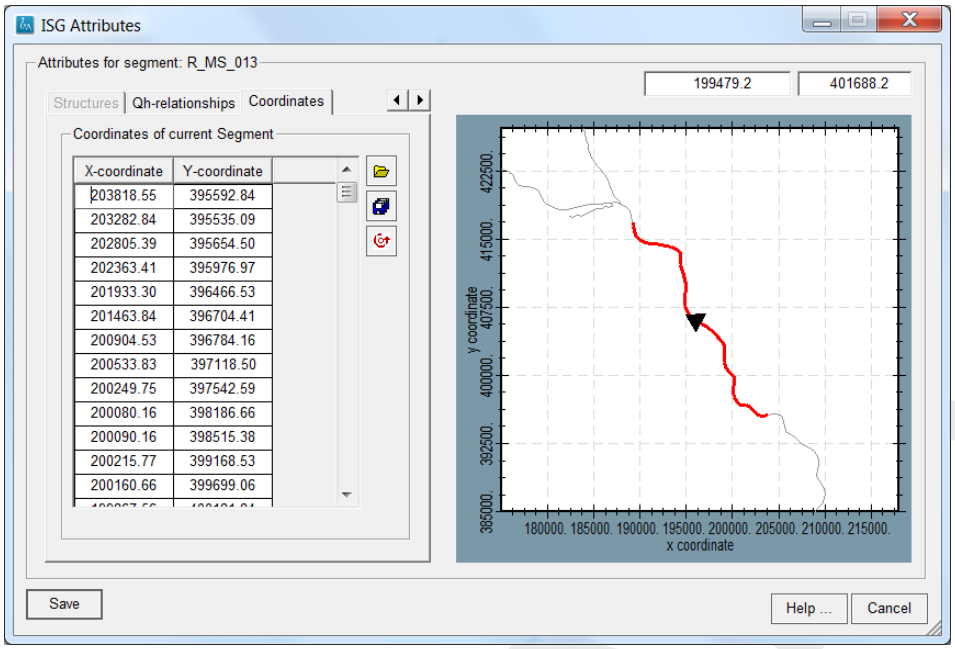
Use this tab to enter information on Q-Depth-Width relationships. This information is used for SFR compliant ISG files.



Q-Depth-Width:	Select one of the listed Q-Depth-Width relation points from the dropdown menu.
	*Rename* Click this button the rename the selected Q-Depth-Width relation point. The *Give New Name:* window will appear.
Distance from origin	This field displays the distance of the selected Q-Depth-Width relation point from the origin of the segment (FromNode) in meters.
Definition Q-Depth-Width Relationships	This table shows the current values for the current selected Q-Depth-Width Relationships point. Use the slidebars to manoeuvre through the table and enter new values for any grid cell if desired. A new record can be entered by filling in all columns!
	*Open CSV-file* Click this button to open a CSV-file, see section 9.12 for more detailed information about CSV-files.
	*SaveAs CSV-file* Click this button to save to a CSV-file, see section 9.12 for more detailed information about CSV-files.
	*Copy* Click this button to open the *Copy Data from* window.
	*Redraw* Click this button to redraw the graphical display.
	*Calculator* Click this button to start the attribute calculator.
Value boxes on the top-right	These fields shows the current value in the graph at the position of the mouse cursor.

6.10.3.9.5 ISG Attributes window, Coordinates tab:

Use this tab to display the coordinates of the selected segment, read in a different set of coordinates and/or switch the order in which the coordinates are entered.

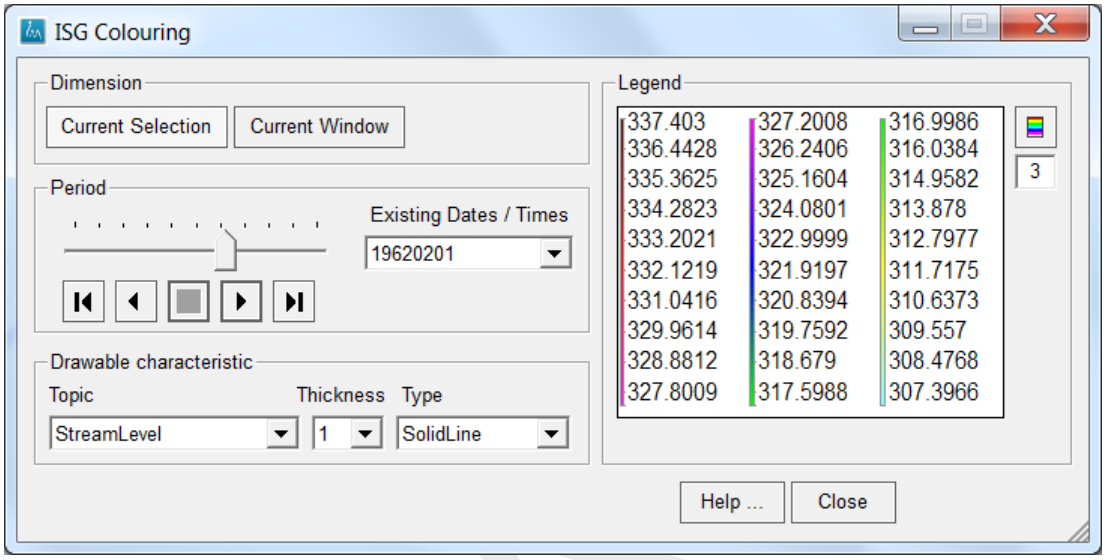


Coordinates of current Segment	This table shows the current coordinates of the reaches in the current selected segment. Use the slidebars to manoeuvre through the table and enter new values. A new record can be entered by filling in all columns!
	*Open GEN-file* Click this button to open a GEN-file, see section 9.10 for more detailed information about GEN-files.
	*SaveAs GEN-file* Click this button to save to a GEN-file, see section 9.10 for more detailed information about GEN-files.
	*Rotate* Click this button to rotate the coordinates. The order of the coordinate determines the flow direction whenever this type of ISG is used for the SFR package, see section 12.28.
Value boxes on the top-right	These fields shows the current value in the graph at the position of the mouse cursor.

6.10.3.10 ISG Colouring

Use this functionality to visualize the parameters on the ISG file as lines, so colouring the individual lines according a specified legend. Click the *Legend* option ( ) on the *ISG Edit* window to display the *ISG Colouring* window.

ISG Colouring window:



Note: At the start of the *ISG Colouring* window the legend for all parameters are initialised based on their minimal and maximal values.

Current Selection	Select this option to colour segment from the current selection only.
Current Window	Select this option to colour all segments disregard whether they are selected.
Period	Use this trackbar to select a different period that is available in the dropdown menu with *Existing Dates/Times*. This option is disabled whenever a single date/time step is available.
	Legend Click this button to start the *Adjust Legend* window in which the legend for the current *Topic* can be modified. Those modified legends will be stored as long as the *ISG Colouring* window remains active. Whenever the window is restarted, all legend will be initialised again.
3	*Number of Columns* Enter the number of columns to represent the legend.
Existing Dates/Times	Select one of the existing dates/times from the dropdown menu. This option is disabled whenever a single date/time step is available.
	Complete Backward Click this button to go the first existing date and move the trackbar to the utmost left.
	Backwards Click this button to step a single date step against time, repeatedly. The trackbar moves accordingly.
	Stop Click this button to stop the actions *Backwards* or *Forwards*.
	Forwards Click this button to step a single date step in time, repeatedly. The trackbar moves accordingly.

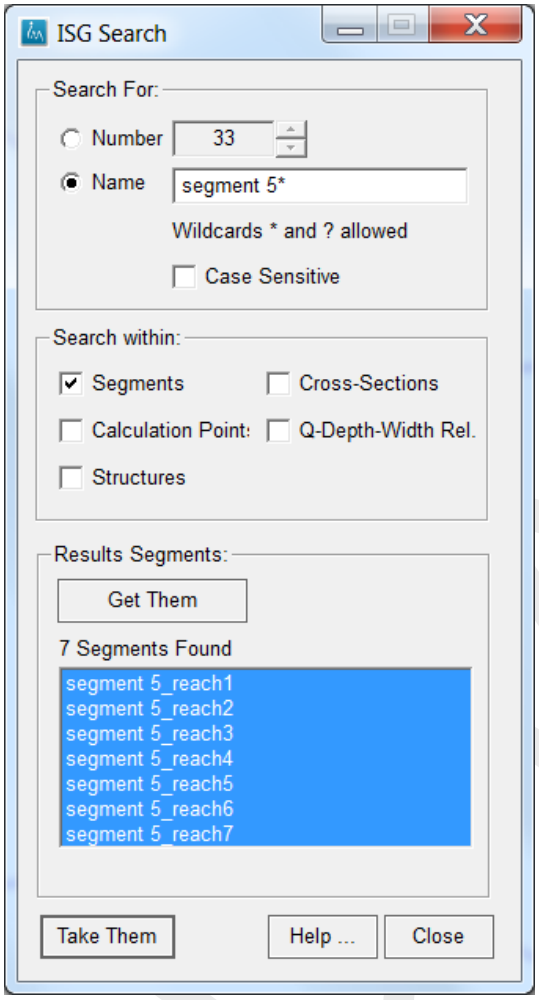
	*Complete Forward* Click this button to go the utmost existing date and move the trackbar to the utmost right.
Topic	Select one of the existing topic from the dropdown menu to display. The amount of topics depend on the type of ISG (see section 9.9.2).
Thickness	Select one of the existing line thicknesses to display the lines.
Type	Select one of the existing line types. The following line types are available: ◇ SolidLine; ◇ Dotted; ◇ Dashed; ◇ DotDash; ◇ DotDotDash; ◇ LongShort; ◇ ShortDash; ◇ LongShortShort.
Help ...	Click this button to start the Help functionality.
Close	Click this button to close the *ISG Colouring* window.

Note: All other functionalities of ISG Edit remain active whenever the *ISG Colouring* window is active.

6.10.3.11 ISG Search

Each segment and their attributes (Cross-sections, Calculation nodes, Structures and Q-Depth-Width Relationships) have labels. With*ISG Search* it is possible to search segments for specific labels in their segment and attributes. Click the *ISG Search* option from the *ISG Edit* window to start the ISG Search window.

ISG Search window:

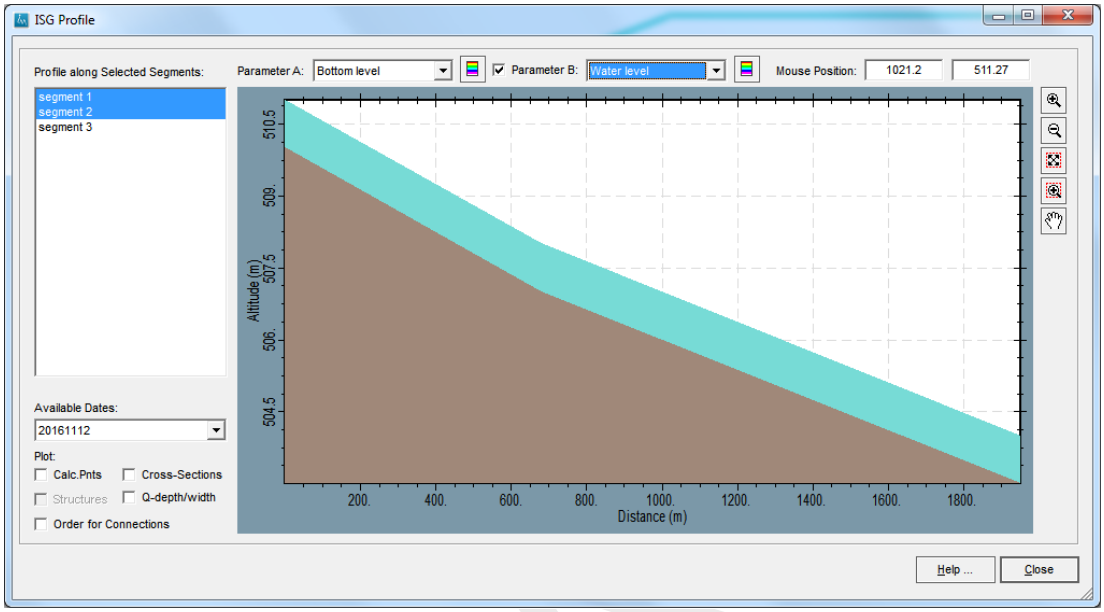


Number:	Select this option to enter a segment number to search for explicitly.
Name:	Select this option to enter a search string. Use the asterix (‘*’) as wildcard and the questionmark (‘?’) to define the number of positions. e.g. *Ex_** means, select all that match the *Ex_* at the start and a search string of *??Ex_** means that all are selected that start with two characters prior to *Ex_*.
Case Sensitive	Click this checkbox to make the search string case sensitive.
Search within:	Select those variables for which the search string will be applied to in order to find those segments that yield a match.
Get Them	Click this button to apply the search string for the selected variables.
Results Segments:	The menulist displays all segments that meet the entered search string.
Take Them	Click this button to take the selection to the main*ISG Edit* window.
Help . . .	Click this button to start the *Help* functionality.
Close	Click this button to close the *ISG Search* window.

6.10.3.12 ISG Profile

Click the menu option *Profile* from the ISG *Edit* window to display the *ISG Profile* window. A length-cross section is presented for the selected ISG segment.

ISG Profile window:



Profile along Selected Segments:	Select one or more segments from the menu list to visualize their length profile. Use your mouse wheel to switch quickly between segments.
Existing Dates:	Select an available date from the dropdown menu to visualize any of the selected items at *Polygon A:* and *Polygon B:* for the selected date.
	Zoom functions: see section 2.6.2 for an explanation
Graphical display	In the graphical display the profile along the selected segment is presented. In cyan, any parameters at *Polygon B:* is presented, in brown the parameter from *Polygon A:*.
Mouse Position	Displays the current position of the mouse in units of the display.
Plot: *Calc. Pnts* *Cross-Sections* *Structures* *Q-Depth-Width*	Click one of the checkbox next to those keywords to display the position of these in the graph.
Order for Connections	Click this keyword to sort the individual segment based on their connections. Segments can be randomly sorted in the segment list and can distort the graphical presentation whenever the graph of the individual segments is not in the correct order. This option sorts them before plotting.
Help . . .	Click this button to start the *Help* functionality.
Close	Click this button to close the *ISG Profile* window.

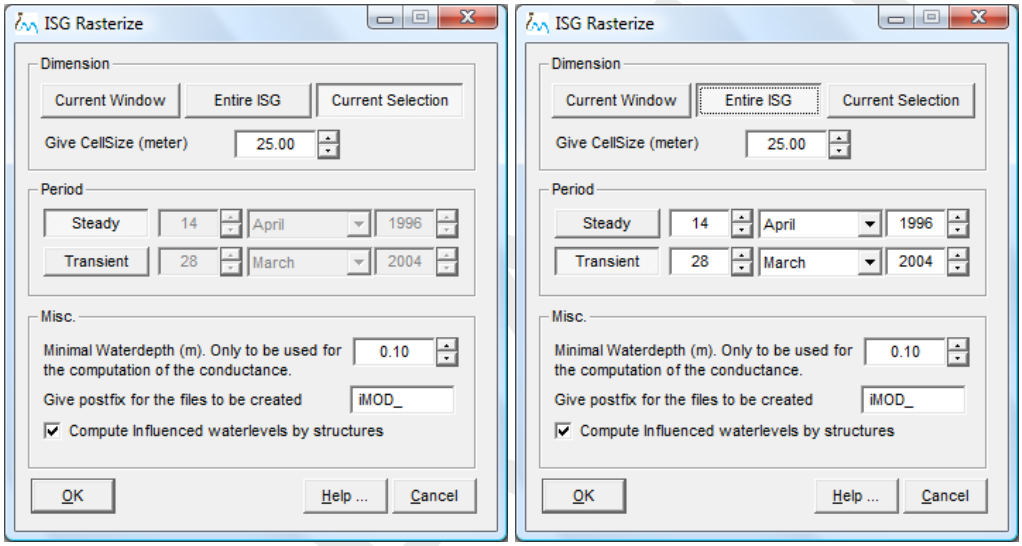
6.10.3.13 ISG Rasterize

For reasons of visual inspection (are all waterlevels correct?) as well as usage in a runfile (see runfile description), iMOD can perform a rasterization of the ISG-file in total or for individual river segments within the ISG-file. It yields several IDF-files (STAGE.IDF, BOTTOM.IDF, INFFCT.IDF and COND.IDF) that can be used and analysed using the standard iMOD functionalities.

Note: IMPORTANT to note is that a minimal resistance is applied of 0.001 days to avoid extraordinary conductance (COND) values.

Click the *ISG Rasterize* button in the *ISG Edit* window to start the *ISG Rasterize* window.

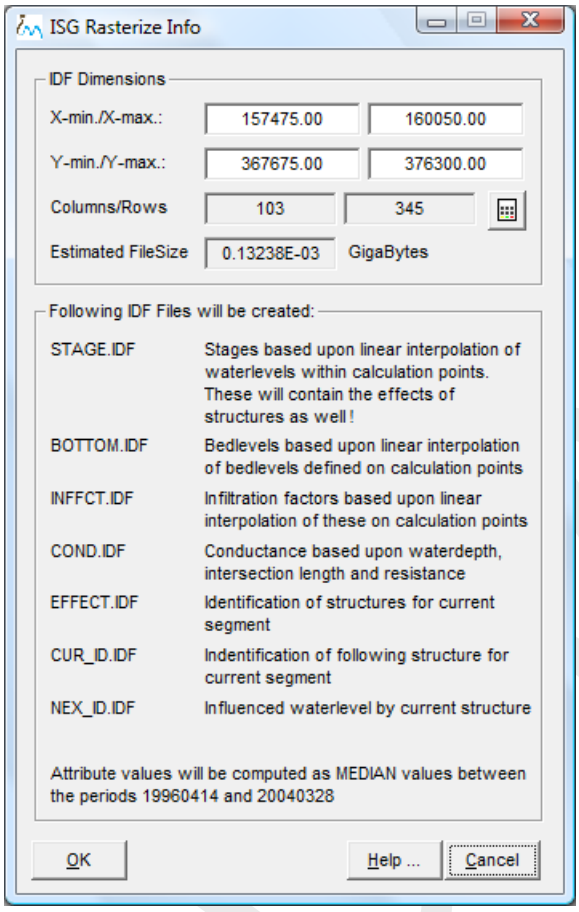
ISG Rasterize window, (top) for a Steady State configuration for the Current Selection and (bottom) for a Transient configuration for the Entire ISG:



Dimension	Select one of the following options: ***Current window:*** Click this option to rasterize all river segments within the current graphical zoom level. ***Entire ISG:*** Click this option to rasterize all river segments in the entire ISG. ***Current Selection:*** Click this option to rasterize the selected river segment(s) only.
Give CellSize (meter)	Enter the cellsize of the resulting IDF after the rasterization.
Period	Select one of the following options: ***Steady:*** Click this option to rasterize all timevariant input variables (e.g. waterlevels, bottom levels, infiltration factors, riverbed resistances, weirlevels) as mean values over all input. ***Transient:*** Click this option to rasterize all timevariant input variables as mean values over the selected period that can be entered in the input fields at the right.
Minimal waterdepth . . .	Enter a minimal waterdepth to be used when computing conductances, which in principal depend on waterdepth. By entering a minimal waterdepth > 0.0, the conductance will not become zero.
Give postfix . . .	Enter the postfix to be added to the default names after the rasterization, e.g. iMOD_ yields the IDF-filename iMOD_STAGE.IDF

Compute influenced waterlevels by structures	Check this item to compute the waterlevel as it is influenced by a weir structure. Upstream from each weir the waterlevel is horizontal until it reaches the level of the river plus waterdepth. From there the waterlevel follows the gradient of the river segment.
OK . . .	Click this button to open the *ISG Rasterize Info* window.
Help . . .	Click this button to start the Help functionality.
Cancel	Click this button to close the *ISG Rasterize* window.

ISG Rasterize Info window:



X-min/X-max	Enter the minimum and maximum X coordinates. On default these are filled in depending on the chosen *Dimension* on the *ISG Rasterize* window.
Y-min/Y-max	Enter the minimum and maximum Y coordinates. On default these are filled in depending on the chosen *Dimension* on the *ISG Rasterize* window.
Columns/Rows	Computed number of columns and rows that the yielding IDF-files receive.
	*Calculator* Click this button to recompute the number of columns and rows.
Estimated FileSize	Display of the filesize in Gbytes.
Following IDF Files will be created	This shows a list of all IDF-files that will be created by the rasterization, together with a brief description. The list depends on the choices made on the *ISG Rasterize* window.
OK	Click this button to start the rasterization. After successful rasterization, both the*ISG Rasterize* and *ISG Rasterize Info* window will be closed. All resulting IDF-files will be added to the *iMOD Manager*.
Help ...	Click this button to start the Help functionality.

Cancel	Click this button to close the *ISG Rasterize Info* window and to return to the *ISG Rasterize* window.

6.11 GEN Options

iMOD supports several basic functionalities to display GEN-files. These GEN-files should be loaded in the *Map* tab on the *iMOD Manager* (see section 5.4) instead of the *Overlays* tab. Whenever a GEN-file is selected the following options can be used:

- ◇ **GEN Info**, use this tool to analyse the content of the associated DAT file, if available.
- ◇ **GEN Configure,**
- ◇ **GEN Extract**, not supported yet!

6.11.1 GEN Info

WHY?

To display the information from a datafile associated to the GEN-file.

WHAT?

The attribute data is stored in a DAT-file which has the same name as the GEN-file. The attribute data is linked by the ID of the features (see section 9.11).

HOW?

Click the *Map Additional Info*  *button* from the *Map Info* window (see section 6.3) to display the *Content of associated datafile* window.

Content of associated datafile window:

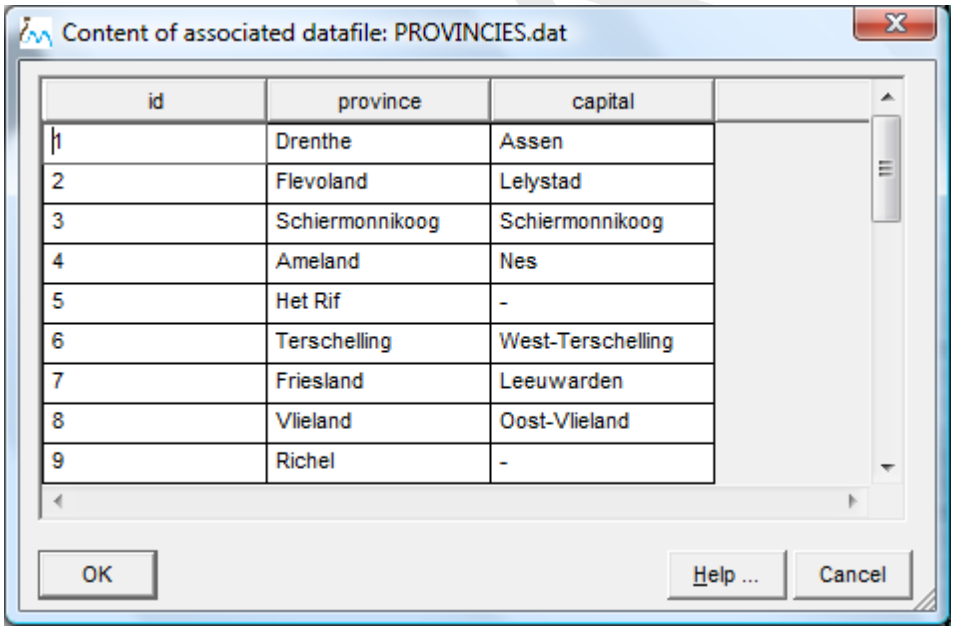


Table	Displays the content of the {name}.dat associated to {name}.gen. The first column is compulsory to enter values to relate to the *ID* of the lines or polygons.
OK	Click this button to close the *Content of Associated Datafile* window.
Help ...	Click this button to start the Help functionality.
Cancel	Click this button to close the *Content of Associated Datafile* window.

6.11.2 GEN Configure

WHY?

To display the GEN-file.

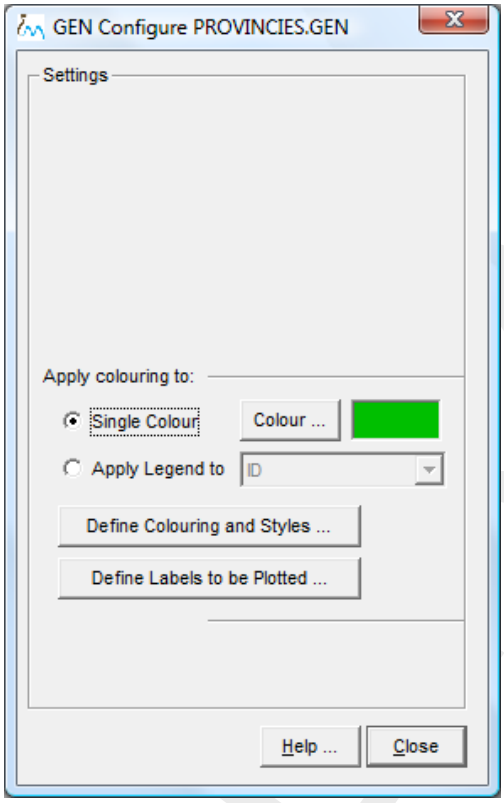
WHAT?

GEN Configure is used to define the settings for display and assigns a symbol, colour or label. The options are similar to the *IPF Configure* function (see section 6.8.1).

HOW?

GEN-files in iMOD represent polygon or line data that can be displayed in different ways. Select the menu option *GEN Configure* from the *GEN Options* menu in the *Map* menu to display the *GEN Configure* window.

GEN Configure window:

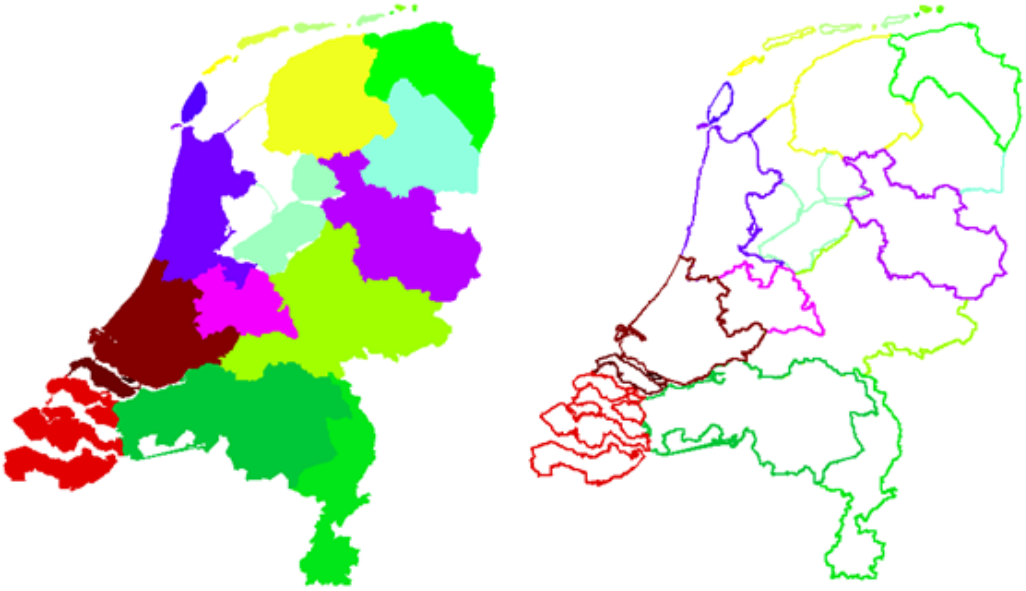


Single Colour	Select this option to display all polygon(s) or lines with the same colour. Depending whether you've selected the *Fill Polygons* option on the *Define Colouring and Styles* window, see section 5.7, the polygon will be filled in or the polygon will be outlined.
Colour . . .	Select this button to display the default *Colour* window in which a colour can be specified. The current colour is displayed to the right of this button, green is used in the example above.
Apply Legend to	Select this option to colour the polygons according to the selected attribute that is chosen in the dropdown menu at the right. A legend can be assigned identical to other iMOD files, e.g. IDF, IPF.
Define Colouring and Styles. . .	Select this option to display the *Lines and Symbols* window
Define Labels to be plotted	Select this button to display the *Define Labels to be Plotted* window.
Help. . .	Click this button to start the iMOD Help Functionality.

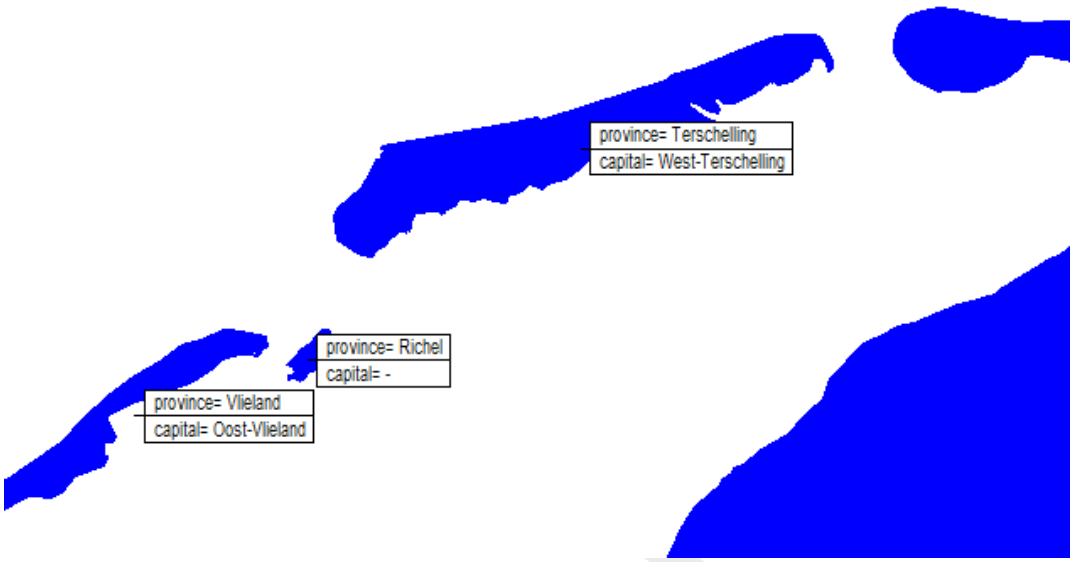
Close	Select this button to apply the settings and close the *GEN Configure* window.

Example of uniform colouring of a GEN-file, using the Fill Polygons option (left) or outline option (right):

Example of legend colouring of a GEN-file, using the filled in option (left) or outline option (right):



Example of labeling polygons of a GEN-file:



7 Toolbox Menu Options

This chapter contains a detailed description of a variety of Tools that are available in IMOD:

- section 7.1: Cross-Section Tool.
- section 7.2: Timeseries Tool.
- section 7.3: 3D Tool.
- section 7.4: Solid Tool.
- section 7.5: Movie Tool.
- section 7.6: GeoConnect Tool.
- section 7.7: Plugin Tool.
- section 7.8: Import Tools.
- section 7.9: Model Simulation.
- section 7.10: Quick Scan Tool.
- section 7.11: Pumping Tool.
- section 7.12: RO-tool.
- section 7.13: Define Startpoints.
- section 7.14: Start Pathline Simulation.
- section 7.15: Interactive Pathline Simulator.
- section 7.16: Waterbalance Tool.
- section 7.17: Compute Mean Groundwaterfluctuations (GxG).
- section 7.18: Compute Mean Values.
- section 7.19: Compute Timeseries.
- **??**: Compute Time-variant Statistics.

7.1 Cross-Section Tool

WHY?

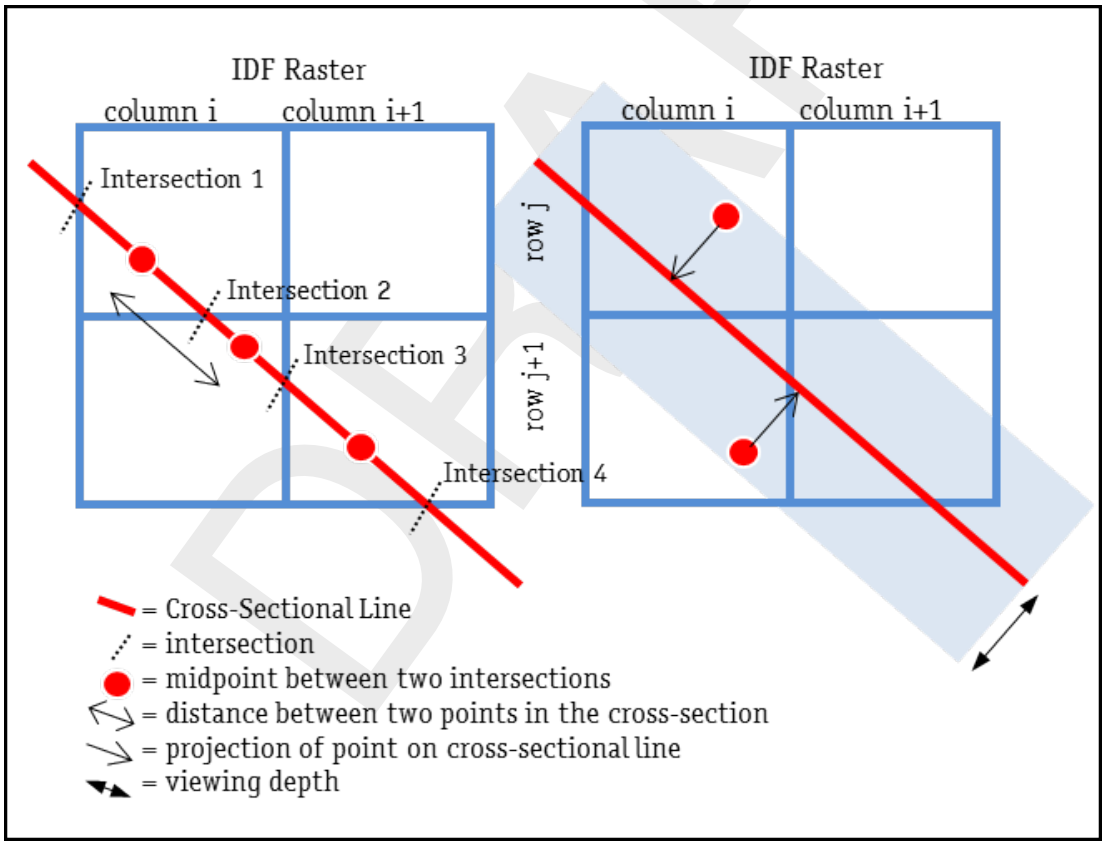
The *Cross-Section Tool* can be used to display cross-sections over a variety of data types, such as IDF's, IPF's with associated files (e.g. boreholes, timeseries) and IFF's (flowlines).

WHAT?

The *Cross-Section Tool* allows you to draw any line (Cross-Sectional Line) that will intersect the rastercells of any of the selected IDF-files. The IDF cell value in-between two raster cell intersections will be assigned to the cross-section points (the midpoints). Consequently, the distances between different points of a cross-section can vary, especially in case the Cross-Sectional Line is chosen to be diagonally.

IPF points and/or IFF lines, can be projected perpendicular on the Cross-Sectional Line within a chosen viewing depth. Bear in mind that breakpoints will cause points and/or lines to be projected twice (in the inside) or not at all (in the outside). For points, the nearest line will be used to be projected upon. Moreover, points that are too close to each other will become narrower, this can be overruled.

Methodology used by the Cross-Section Tool:

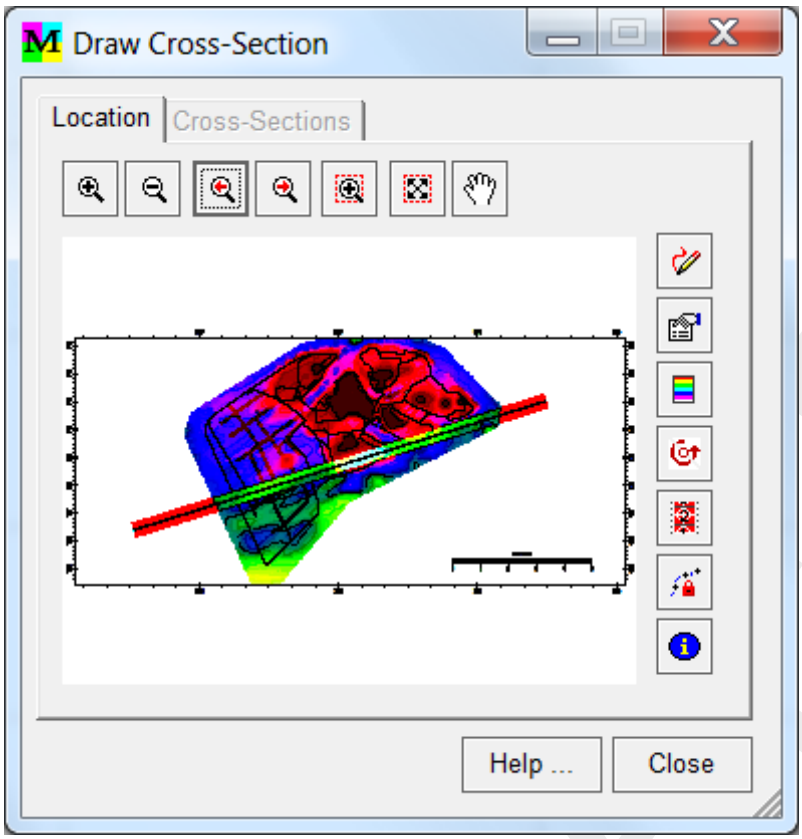


HOW?

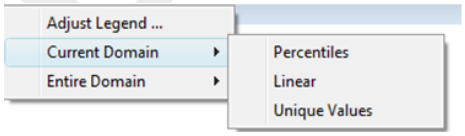
To start the *Cross-Section Tool* select *Toolbox* from the main menu, choose *Cross-Section Tool*. Alternatively you can select the *Cross-Section Tool* icon from the main toolbar. In both cases, all the selected files in the *iMOD Manager* will be activated in the *Cross-Section Tool*. The order in which these files are arranged, might affect the way they are displayed.

The *Cross-Section Tool* consists of two windows: *Draw Cross-Section* window and the *iMOD Cross-Section CHILD* window.

Draw Cross-Section window:



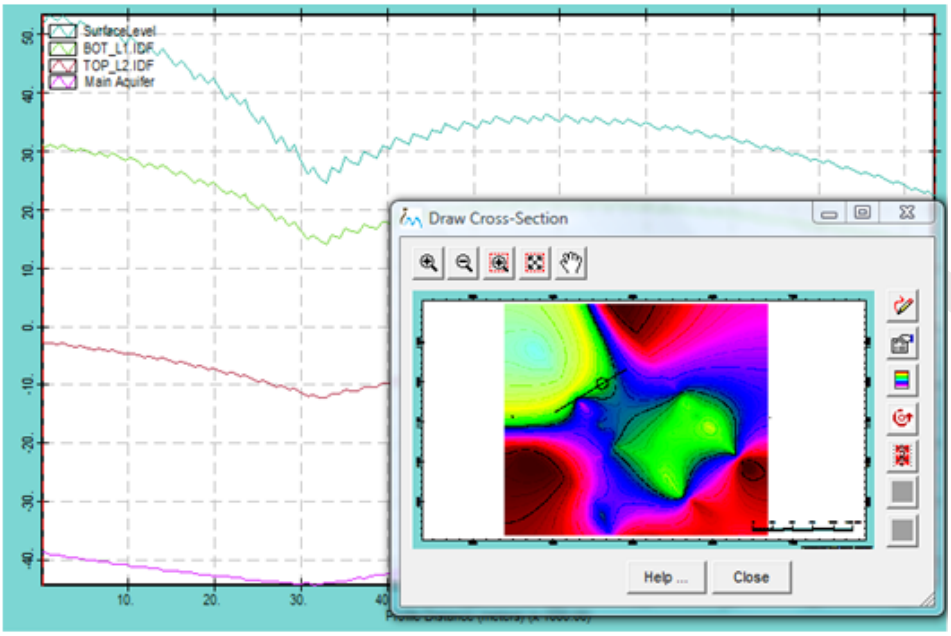
	*Zoom In* Click this button to zoom IN on the centre of the current graphical dimensions.
	*Zoom Out* Click this button to zoom OUT on the centre of the current graphical dimensions.
	*Go Back to Previous Extent* Click this icon and the map will return to the previous map extent and view. This view becomes the last view automatically whenever any other zoom button will be used.
	*Go to Next Extent* Click this icon and the map will go to the next extent viewed after the current view. This option becomes available whenever the *Zoom to Previous Extent* button has been selected priorly.
	*Zoom Rectangle* Click this button to zoom in for a rectangle to be drawn. Use you the left-mouse button to determine the lower-left corner of the rectangle, click again for the upper-right corner (or vice-versa).
	*Zoom Full* Click this button to zoom in on the entire extent of the selected maps on the tab *Maps* on the *iMOD Manager* or on the selected overlay Maps in the tab *Overlay* on the *iMOD Manager*.
	*Move* Click this button to move the current display. Click the left-mouse button on that location where you want to move from, repeat this after the display has been refreshed (automatically). Use the right mouse button to stop the moving process.

	*Draw Line of the Cross-Section* Click this button to draw the line of the cross-section on the *Graphical Area* on the *Draw Cross-Section* window. Click the left-mouse button to define the first point of the line and click this left-mouse button to insert intermediate points, if desired. Click the right-mouse button to stop the line drawing. If you've defined one point only, the last location will be added to the line, in other cases this last point will not be used! The used coordinates can be displayed on the cross-sectional view and/or within the *Cross-Section Properties* window. The line of the cross-section may consist of 250 points, maximally.
	*Cross-Section Properties* Click this button to open the *Properties* window (see section 7.1.1).
	*Cross-Section Legend* Click this button to open the *Adjust Legend* window (see section 6.6.1).
	*Flip Cross-Section* Click this button to "flip" the current Cross-Section
	*Cross-Section Movie* Click this button to open the *Movies* window.
	*Snap Coordinates* Click this button to "snap" the coordinates of the Cross-Section to the coordinates in the selected IPF-file. This functionality is therefore only available whenever an IPF is selected.
	*IPF-info* Click this button to open the *IPF-Info* window.
Graphical Display	This presents the display from the *Main iMOD* window. Anything that has been drawn before entering the *Cross-Section Tool* will display here. In this area you can specify the location of the Cross-Section. The location will appear as a black line. When you move the mouse in this *Graphical Area* your current coordinates will be displayed in the lower-left corner of the *Cross-Section* window. Moreover, the following symbols might occur whenever you move the mouse near the Cross-Section line:  Click your left mouse button and hold it, to move the entire Cross-Section line. Stop this by releasing the mouse button.  Click your left mouse button and hold it, to move an individual node of the Cross-Section line. Stop this by releasing the mouse button. iMOD will update your cross-section immediately. Click your right-mouse button anywhere on the *Graphical Display* to popup the following dropdown menu. *Popup menu:*  The functionalities are described in section 2.6.3. The popup menu becomes available only when IDF-files are selected in the *iMOD Manager*.
Help ...	Click this button to start the iMOD Help Functionality.
Close	Click this button to close the *Cross-Section Tool*; the *Draw Cross-Section* and *iMOD Cross-Section* windows will also close.

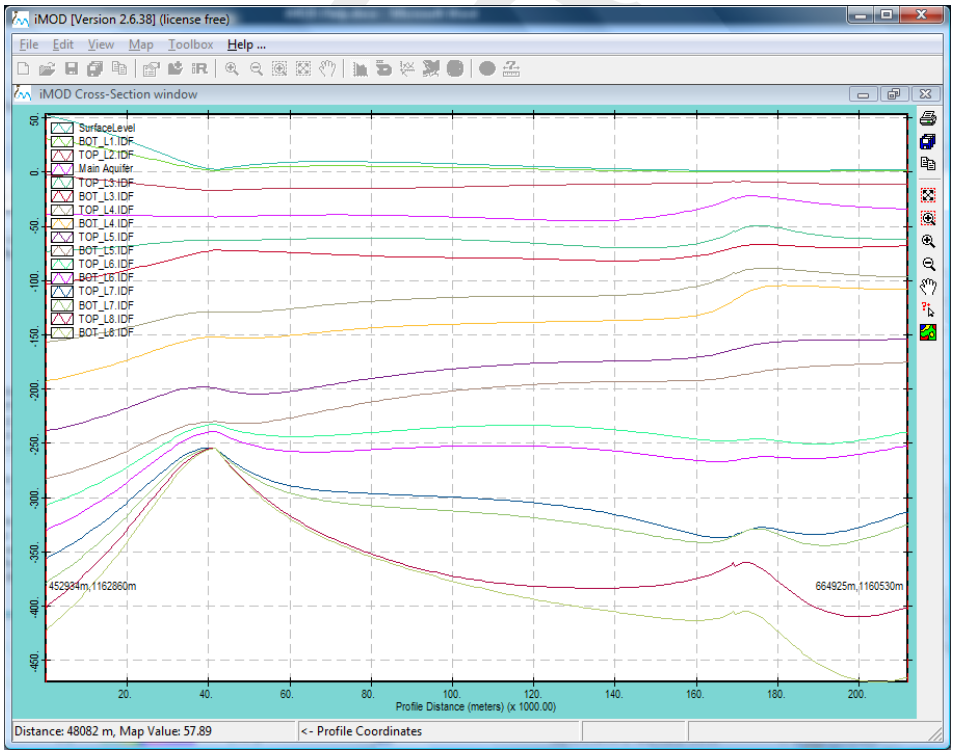
Note: All adjustments in the zoomlevel on the *Graphical Area* will be used whenever you leave the *Cross-Section Tool* again.

Note: iMOD will intersect the cross-section line with the raster cellvalues of the IDF-files. Since different IDF-files may be used (constant- and variable rastersizes), each cross-section can have different results at the intersections. After the intersection, iMOD determines the IDF values for the midpoints that are in the centre between two intersection points. Due to this, diagonal lines may display cross-sections with jagged lines.

Example of cross-sections that are jagged:



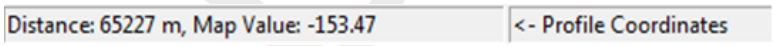
iMOD Cross-Section window:



	*Print* Click this icon to print the current Cross-Section on a printer.
	*Export or Save As Demo . . .* Click this icon to export the current Cross-Section to an ASCII-file (*.csv) or to save the current Cross-Section as an iMOD-demo in a new IMF-file.
	*Copy to Clipboard* Click this icon to copy the current Cross-Section to the windows *Clipboard*. Use the shortcut *Ctrl-C* , alternatively
	*Zoom Full* Click this button to zoom in on the entire extent of the Cross-Section.
	*Zoom Rectangle* Click this button to zoom in for a rectangle. Use the left-mouse button to determine the lower-left corner of the rectangle, click again for the upper-right corner (or vice-versa).
	*Zoom In* Click this button to zoom IN on the centre of the current Cross-Section.
	*Zoom Out* Click this button to zoom OUT on the centre of the current Cross-Section.
	*Move* Click this button to move the current Cross-Section. Click the left-mouse button on that location where you want to move from, repeat this after the display has been refreshed (automatically). Use the right mouse button to stop the moving process.
	*Cross-Section Inspector* Click this icon to use the *Cross-Section Inspector*.
	*Add a bitmap as background* Click this icon to select a BMP-file to be shown as background map. Whenever the Profile Tool is started via the Solid Tool (see section section 7.4, the position and background bitmap will be saved in the SPF file, see section section 9.21.
	*Legend* Click this icon to display the legend, see section section 7.1.2.

Note: Whenever you move the cursor over the *Draw Cross-Section* window, the coordinates are displayed in the lower-left corner of the *iMOD Cross-Section* window. Moreover, your position in the cross-section will be displayed in the *Draw Cross-Section* window as a small circle on the line for the cross-section.

Example of cursor location in the Cross-Section:

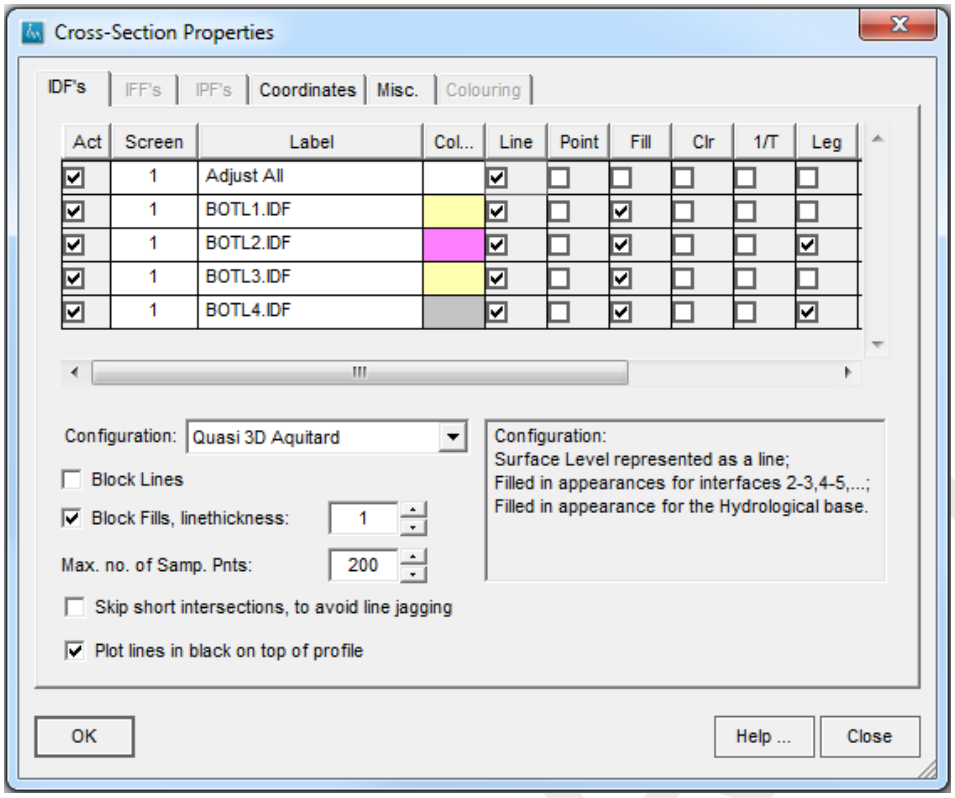


Note: Each point that determines the line for the cross-section is displayed as a red, vertical dashed line in the graph.

7.1.1 Properties

Click the option *Cross-Section Properties*  on the *Draw Cross-Section* window to open the *Cross-Section Properties* window. The properties are grouped for each filetype in the Cross-Section (IDF's, IPF's and/or IFF's) and their corresponding tabs become available when the filetype is present in the Cross-Section.

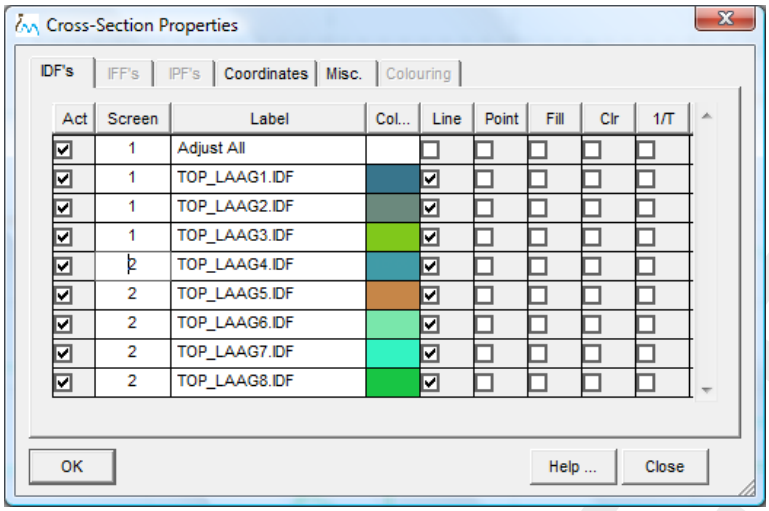
Cross-Section Properties window, IDF's tab:



The display mode of the IDFs is defined in the table. Several quick display configurations are available for layer models.

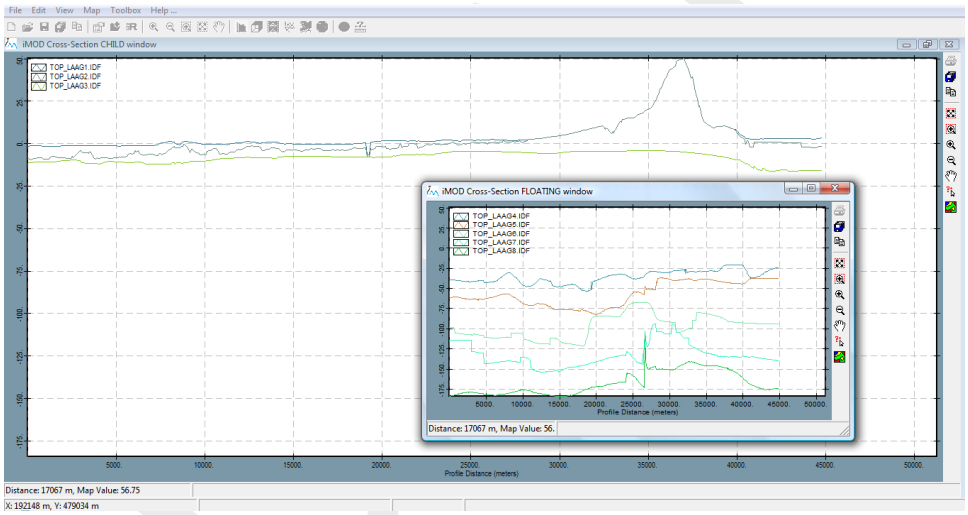
Act	Select the checkbox in this column to include the IDF in the cross-section.
Screen	Number of *Graphical Windows* (screens 1-50) for display; e.g. you can specify a separate *Graphical Window* (screen) for each IDF. All screens are synchronized, which means that all zoom and/or pan actions will be carried out for all screens simultaneously.
Label	Insert a text for a label.
Col. . . (Colour)	Displays the colour used to display the Cross-Section for each IDF. Select the column to open the *Colour* window to change the colour.

Cross-Section Properties window, IDFs tab for multiple Graphical Windows:



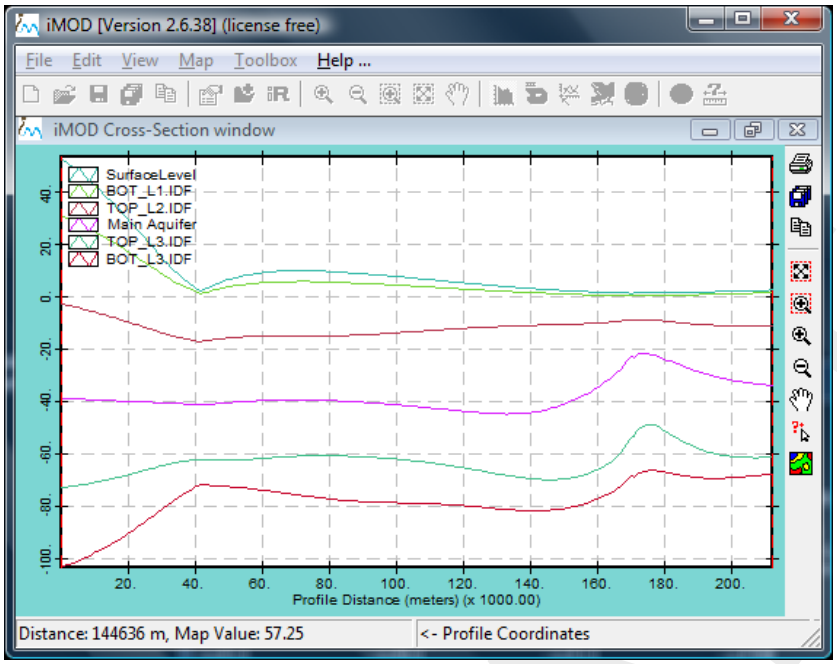
This is also possible for the loaded IPFs by adjusting the number in the *Screen* column, e.g. 1 means plot on first (=main) screen, 2 means plot on a second screen.

Example of using multi-screens to display the cross-section:



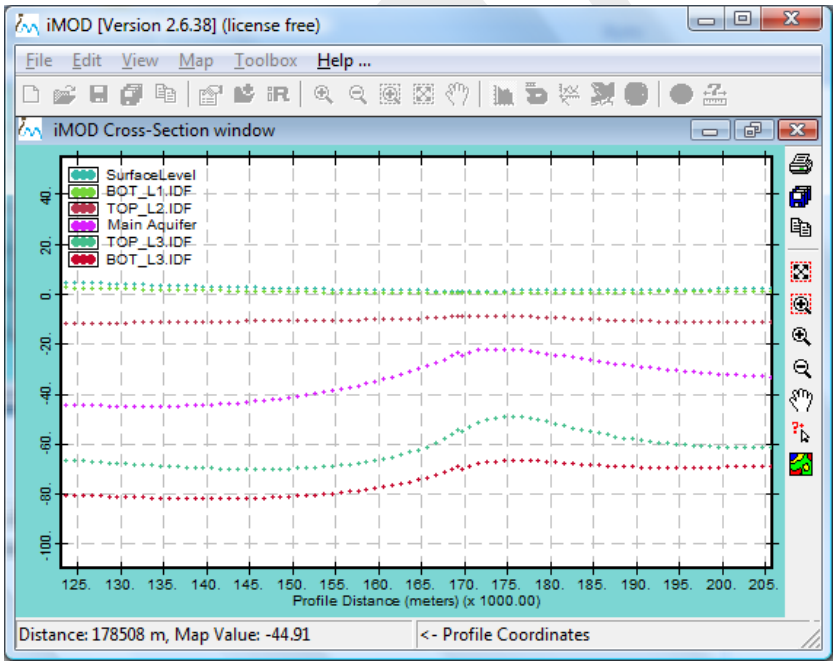
Line	Select the checkbox in this column to present the cross-section as solid lines.

Example of cross-section using the Line option:



Point	Select the checkbox in this column to present the cross-section as individual points.

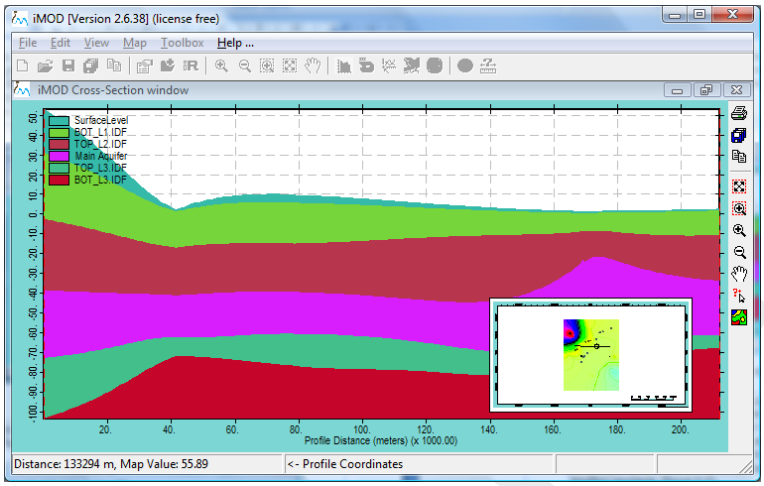
Example of a cross-section using the Point option:



Fill

Select the checkbox in this column to present the cross-section as a filled area. The cross-section is bounded by the surfacelevel at the top, and the minimum z-value of the cross-section at the bottom. It is important to know that IDFs with lower z-values will be “painted” over by IDFs with higher z-values, whenever the IDFs with higher z-values appear below the IDFs with lower z-values in the *iMOD Manager*.

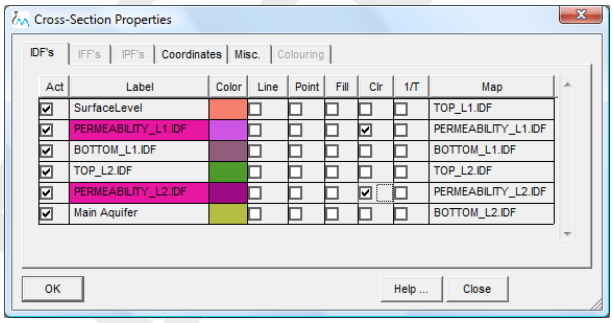
Example of a cross-section using the Fill option:



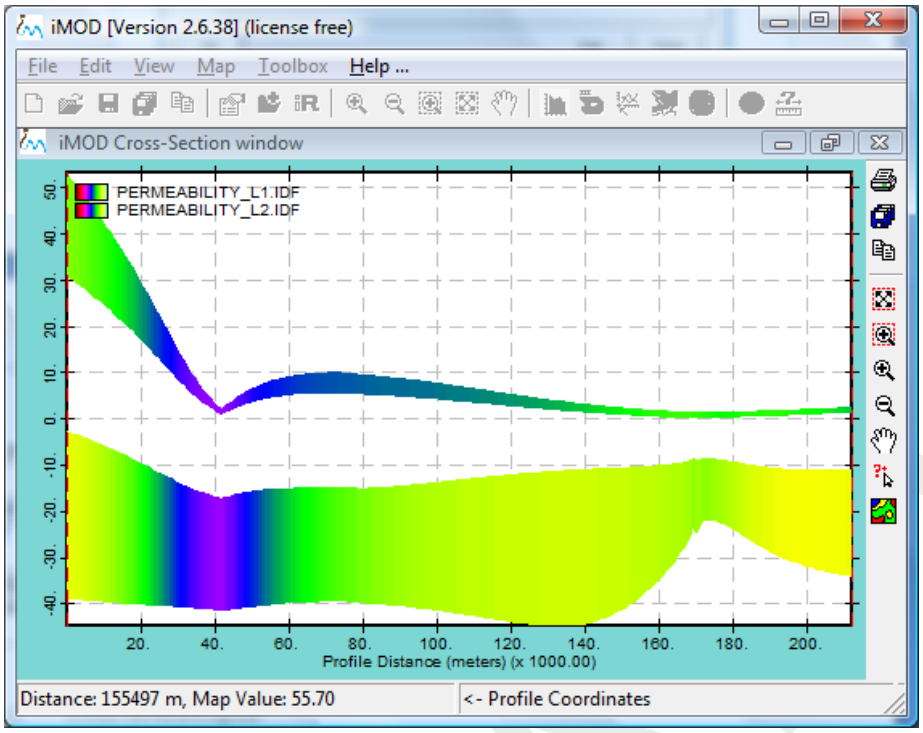
Clr

Select the checkbox in this column to present the cross-section as filled surfaces coloured by the values of the IDF-file. It uses the previous- and next IDF to determine the top- and bottom boundaries of the filled area. Whenever you choose this option, the *Act* option is selected for the previous and next IDF, automatically. E.g., use this option to display different information such as heads, transmissivities between boundaries of aquifers/aquitards.

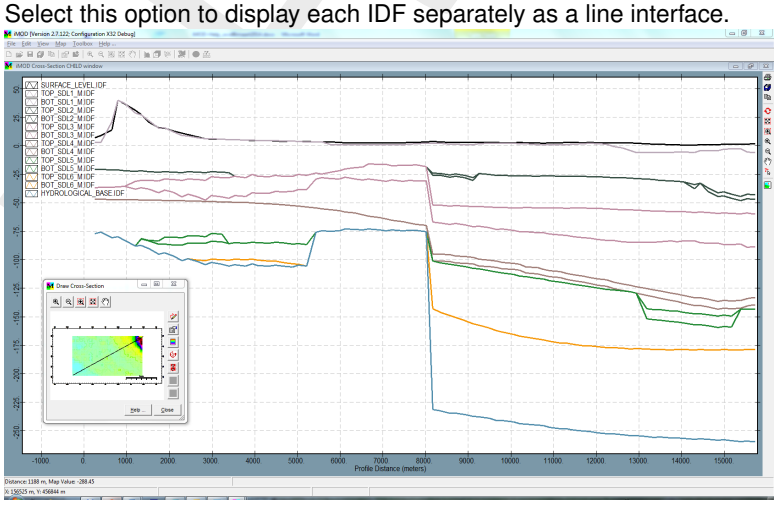
Example of using the Clr-option:



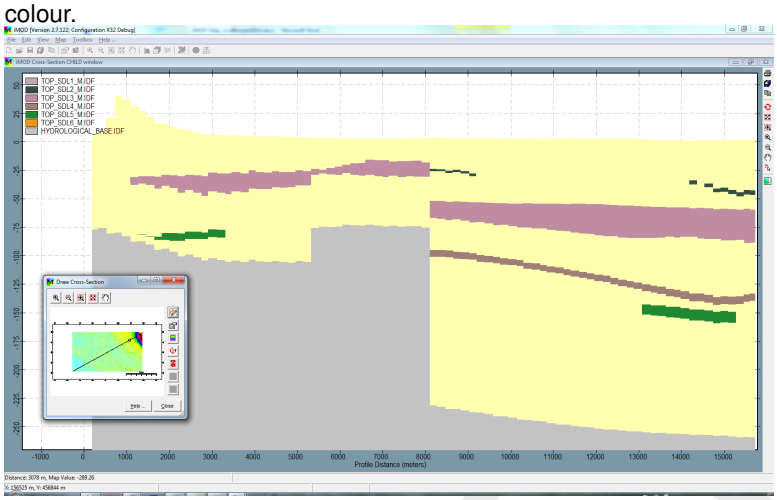
Example of a cross-section using the Clr option:



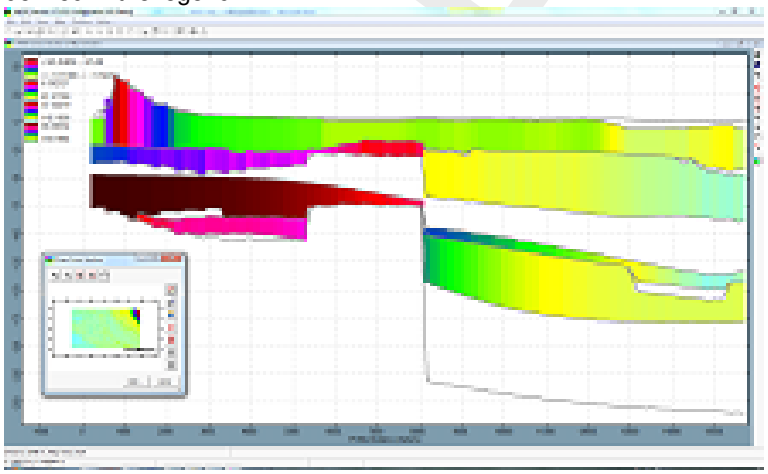
1/T	Select the checkbox in this column in combination with the option *Clr*, to divide the value of the IDF by the thickness.
Map	Displays the IDF-filename. You can not adjust this field.
Configuration	Select an option from the drop down list to display the layers in a predefined display mode. *Interfaces* Select this option to display each IDF separately as a line interface.



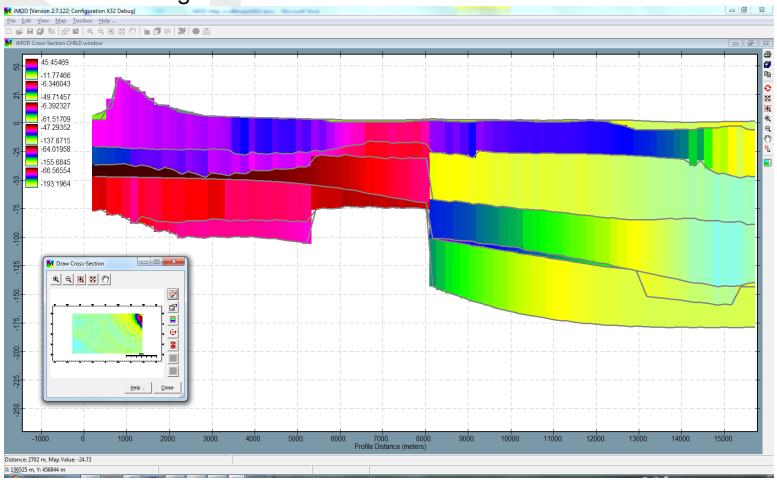
Quasi 3D model
Select this option to display the aquifers in yellow and the aquitards in a contrasting colour.



Coloured Quasi 3D Model
Select this option to display the aquifers coloured by the values of the IDF-file as defined in the legend.



Coloured 3D Model
Select this option to display the aquitards coloured by the values of the IDF-file as defined in the legend.

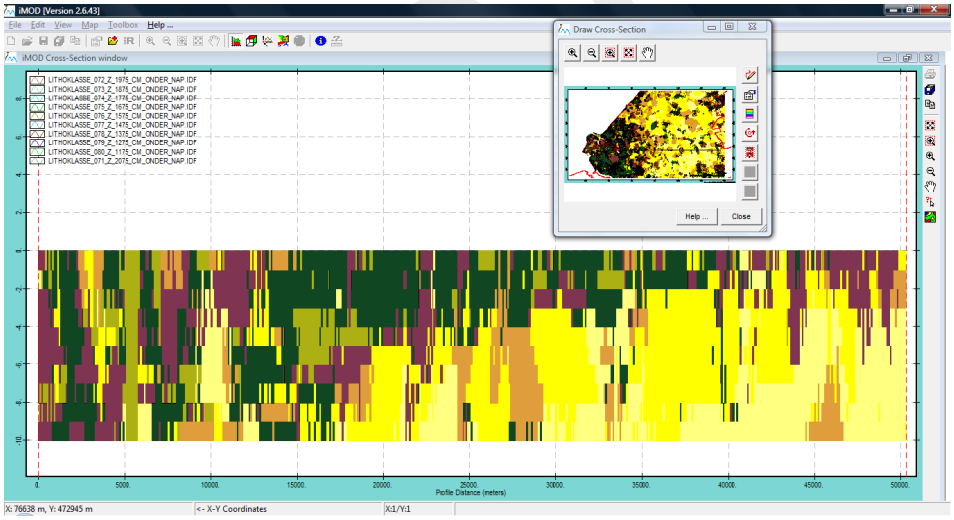


Block lines	Check the box to display the cross-section with lines that connect the individual data points with horizontal lines.
Block Fills	Check the box to display the cross-section with lines that connect the individual data points directly.
Linethickness	Choose the linethickness to display the layer interfaces
Plot lines in black. . .	Check the box to plot the lines in black on top of the drawn profile, e.g. to be able to distinguish multiple aquifers when no in between aquitards are available.
Max. no of Samp. Pnts.	Maximum number of sampling points used to construct the cross-section.
OK	Click this button to accept any changes; the *Cross-Section Properties* window is closed.
Help. . .	Click this button to start the iMOD Help Functionality.
Close	Click this button to close the *Cross-Section Properties* window, without any changes to the properties of the cross-section.

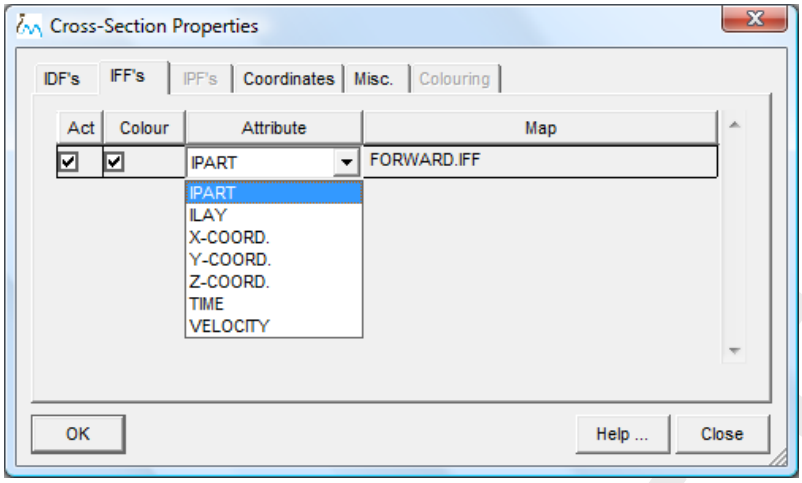
Note: The checkbox in the first row of the table is used to adjust all rows simultaneously. So, whenever you select the option *Fill* on the first row, all rows will inherit this setting. It also applies to the *Colour* option.

Note: Alternative to the *Clr* option, IDF-files can have top- and bottom information stored internally (see IDF syntax). If those IDF-files (voxels) are plotted in the *Cross-Section Tool* the actual IDF value (e.g. permeability) will be coloured (using the associated legend as with the *Clr* option) between the stored top and bottom elevation inside the IDF.

Example of a coloured plot of IDF with internal top and bottom elevations:



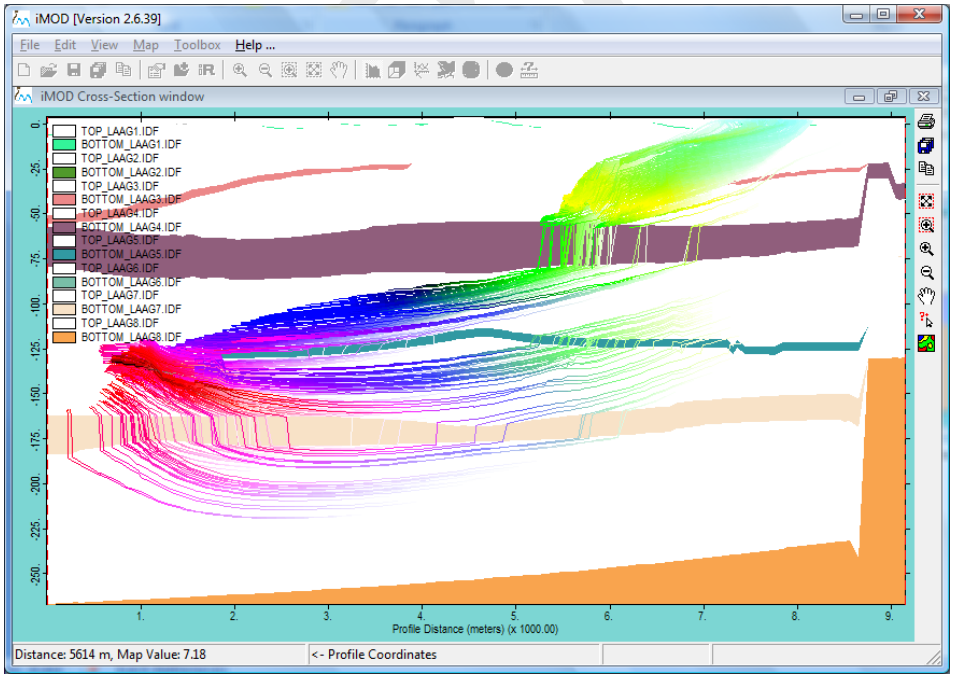
Cross-Section Properties window, IFF's tab:



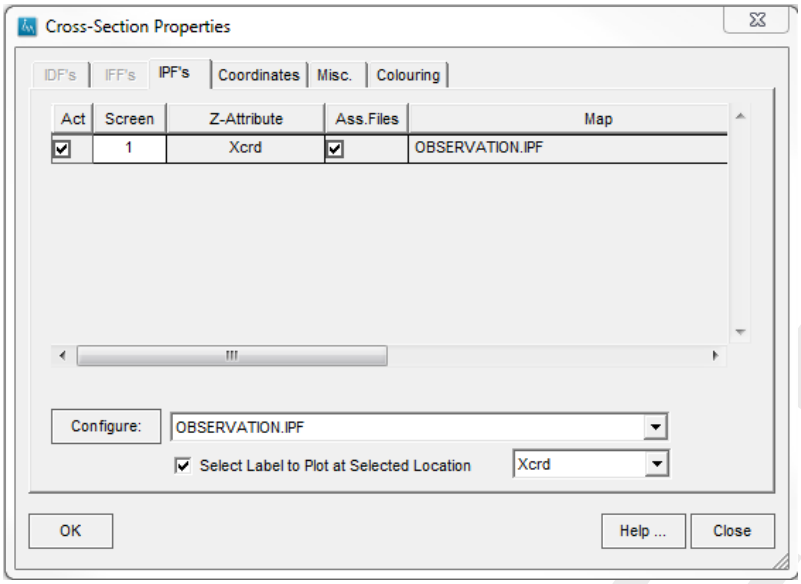
Act	Select the checkbox in this column to include the IFF in the cross-section.
Colour	Select this checkbox to colour the lines according to the selected *Attribute* item.
Attribute	Select an item from the dropdown menu to be used to colour the lines.
Map	Displays the IFF filename. You can not adjust this field.

The example below shows a cross-section presenting an IFF-file (flowpath) in combination with IDF-files that represent the top and bottom of aquitards.

Example of a Cross-Section showing flowlines from an IFF file:



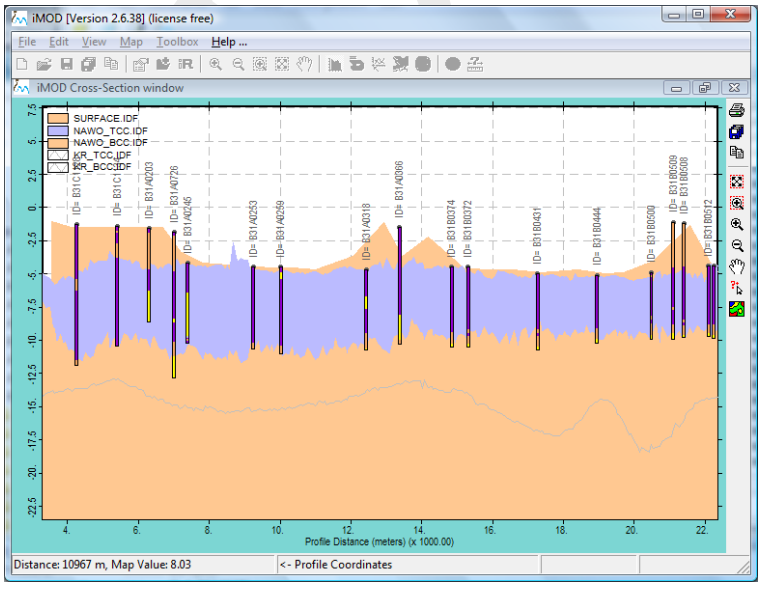
Cross-Section Properties window, IPF's tab:



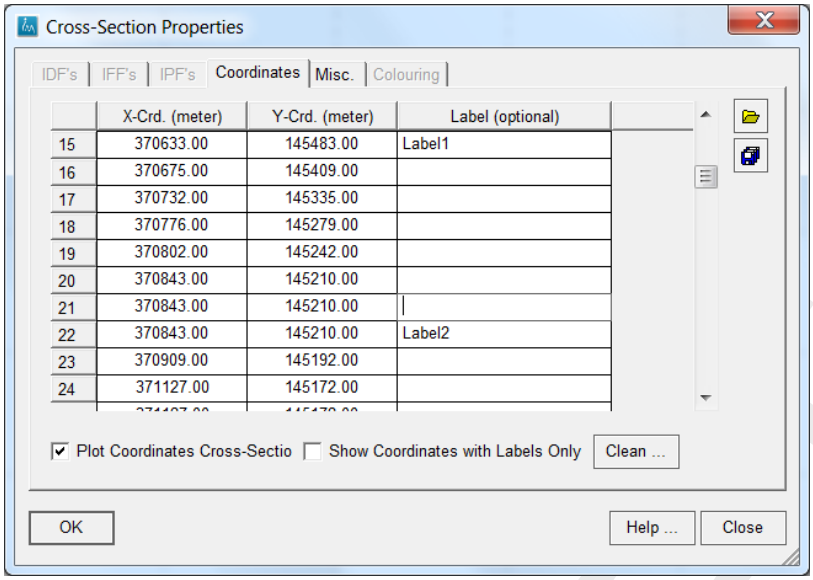
Act	Select the checkbox in this column to include the IPF in the cross-section.
Z-Attribute	This is the attribute in the IPF-file that will be used to position any label. In case timeseries are presented, this attribute value is used too. For others (boreholes, borelogs) this value is irrelevant.
Ass-Files	Select this checkbox to use the associated file of the IPF. These can be time-series, boreholes and/or borelogs.
Map	Displays the IPF-filename. You can not adjust this field.
Configure:	Select this button to set the configuration for the IPF-file (see 129).
Select label to Plot at Selected Location	This dropdown menu allows you to set the location of the labels of the boreholes.

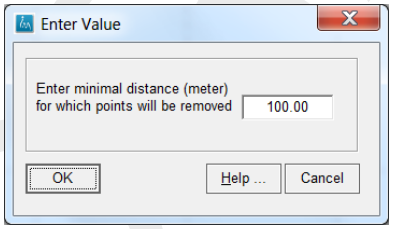
Below an example is given of IPF-files displayed in combination with the Fill option.

Example of a cross-section with boreholes associated to an IPF-file:

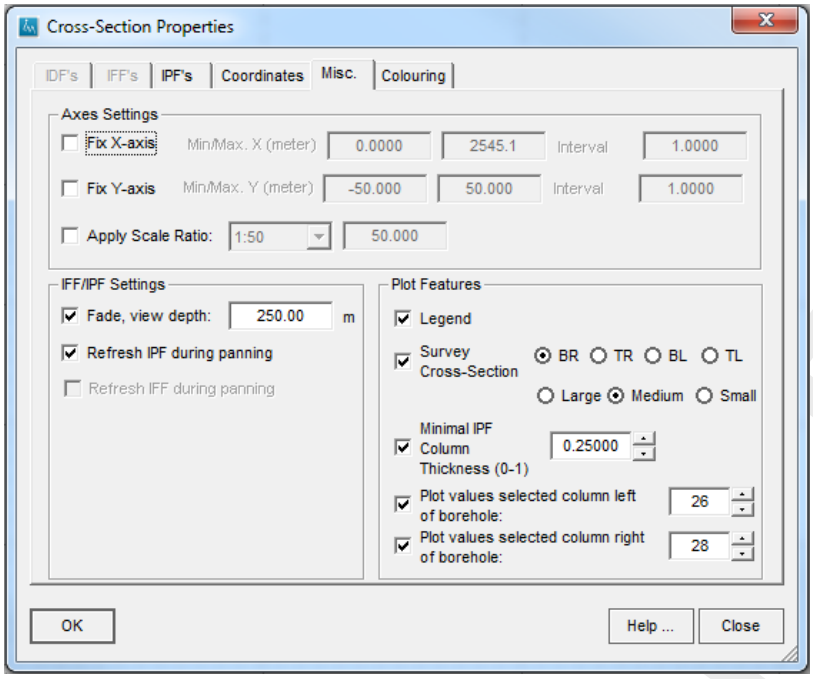


Cross-Section Properties window, Coordinates tab:

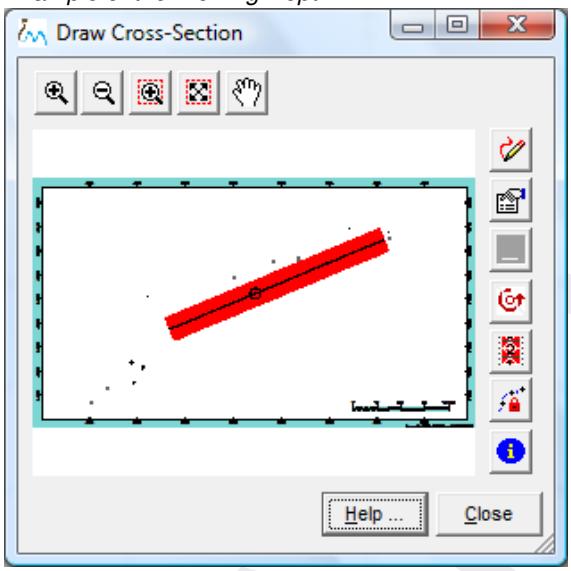


	*Open* Click this button to use the coordinates from an existing GEN-file.
	*Save As* Click this button to save the current coordinates into a GEN-file format.
Plot Coordinates Cross-Section	Select this option to display the coordinate of the line of the cross-section within the graph of the cross-section.
Show Coordinates with Labels Only	Select this option to only display the coordinates with a label within the graph of the cross-section.
Clean...	Starts a new window *Enter Value*:  In this window a value can be filled in for the minimal distance that needs to be between coordinate points. The points in a profile will be depleted; points that are closer to each other than the given distance will be removed.

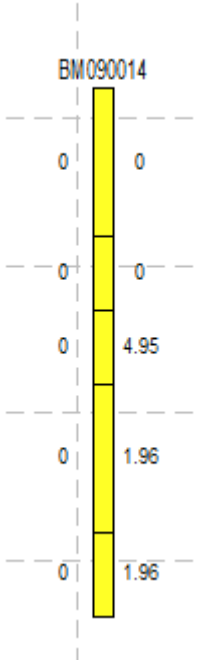
Cross-Section Properties window, Misc. tab:



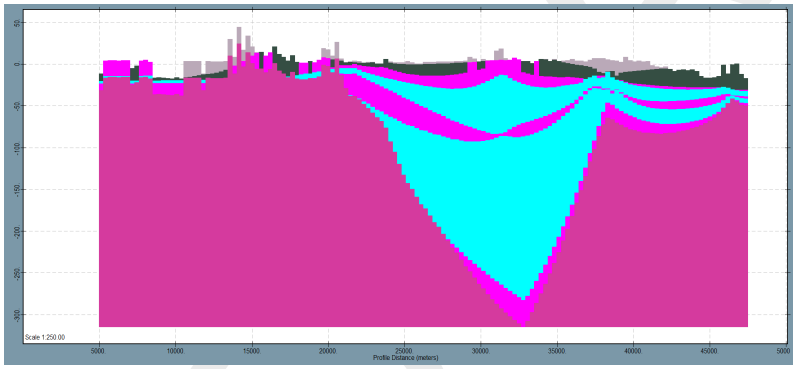
Fix X-axis	Select this checkbox to specify the dimensions of the X-axis.
Minimal/maximal X:	Enter the minimal and maximal values for the X-axis.
Interval:	Enter the interval of the X-axis.
Fix Y-axis	Select this checkbox to specify the dimensions of the Y-axis.
Minimal/maximal Y:	Enter the minimal and maximal values for the Y-axis.
Interval:	Enter the interval of the Y-axis.
Apply Scale Ratio:	Chose the preferred x,z-scale ratio from the drop down-menu or enter a ratio-value in the given field. The ratio is shown on the graph. See figure below for an example of the cross-section window with adjusted ratio.
Fade	Select this option to fade-out the colouring for IPF (points) and/or IFF (lines) whenever they appear at more distant from the line of the cross-section.

Fade, view depth	Enter the distance perpendicular on the line for the cross-section for which points (IPF) and/or lines (IFF) are projected perpendicular on the line of the cross-section. This “area” is displayed as a red rectangle around the drawn line for the cross-section. *Example of the Viewing Depth:* 
Refresh IPF during panning:	Select this option if you have a small borehole IPF dataset. This option forces iMOD to refresh the cross-section view when you move your window extent (=panning). Deselect it whenever the data set is large since this will delay the drawing significantly.
Refresh IFF during panning:	Select this option if you have a small pathline IFF file. This option forces iMOD to refresh the pathline view when you move your window extent (=panning). Deselect it whenever the data set is large since this will delay the drawing significantly.
Legend	Select this option to display a legend on the graph
Survey Cross-section	Select this option to display a 2D map of the location of the cross-section.
BR, TR, BL, TL	Select on of the following to specify the location of the survey, BR is Bottom-Right, TR is TopRight, BL is BottomLeft and TL is TopLeft.
Large, Medium, Small	Select on of the following to specify the size of the location of the survey.
Minimal IPF Column Thickness (0-1):	Select this option to set a minimal Column thickness of the boreholes. This prevent iMOD from hiding a certain borehole when it is at the same location as another borehole in the display window.
Plot values selected column left of borehole:	Select this option to plot the values of the defined column (e.g 26) in the IPF associated file at the left side of the borehole in the Cross-section plotting window.
Plot values selected column right of borehole:	Select this option to plot the values of the defined column (e.g 28) in the IPF associated file at the right side of the borehole in the Cross-section plotting window.

Example of the plotting window with the values of the defined columns (from the associated text file) plotted next to the borehole:

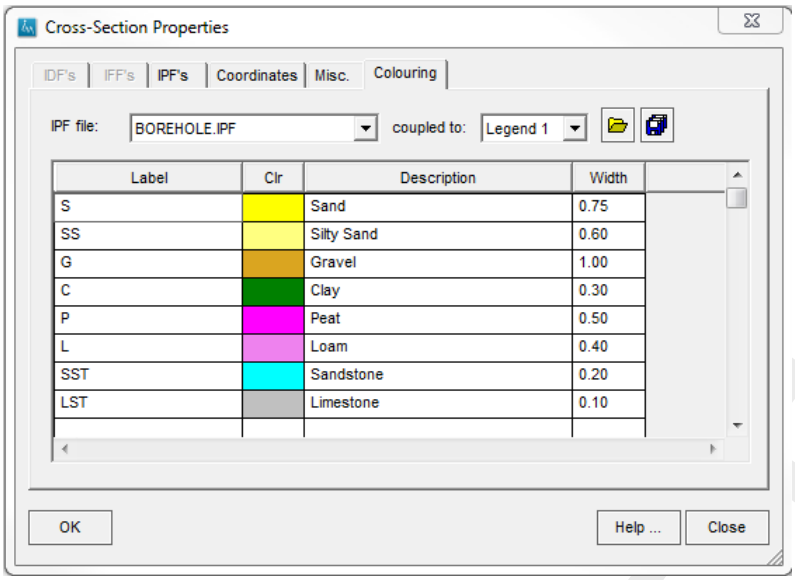


Example of the cross-section window with adjusted ratio:

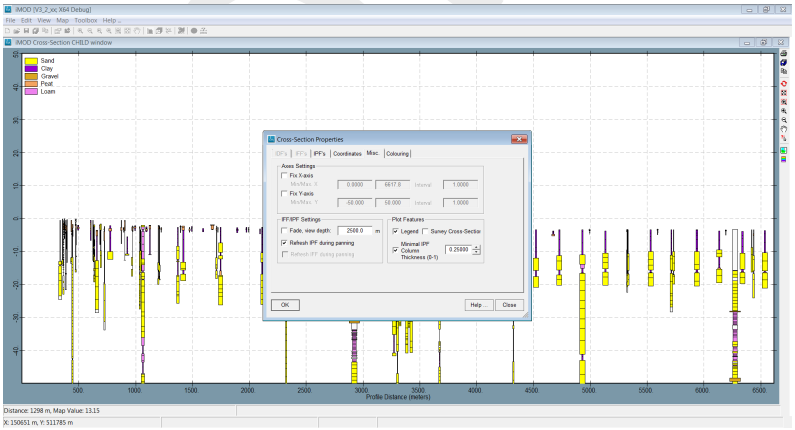


Cross-Section Properties window, Colouring tab:

The table on this tab is used to display the boreholes that might be associated with the selected IPF-files. On default the file: {user}\settings\DRILL.DLF will be read.



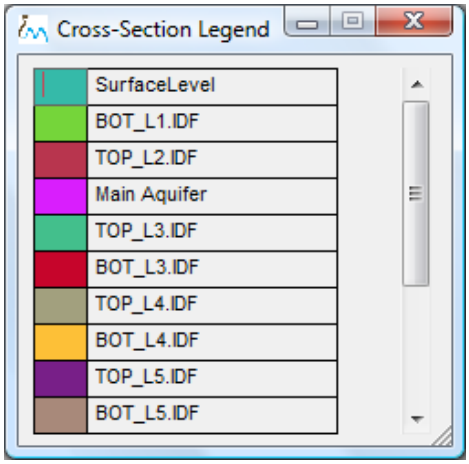
IPF file:	This dropdown menu shows the IPF-files that are currently available in the *IPF Analyse.* For each IPF-file a different legend can be defined by making use of the *Legend:* dropdown menu.
Legend:	This dropdown menu allows you to select and/or define a different legend per available IPF-file.
	*Open* Click this button to open a *.DLF-file
	*Save As* Click this button to save the current legend into a DLF file format
Label	Associated label to the specific Legend attribute.
Clr	Associated colour to the specific Legend attribute.
Description	Associated description to the specific Legend attribute.
Width	Adjustable width of the borehole unit related to the specific legend attribute. If differences in width are defined between the layers this is visible in the cross-section of the boreholes (see figure below).



7.1.2 Profile Legend

Click the option *Cross-Section Legend* on the *Draw Cross-Section* window to open the *Profile Legend* window.

Cross-Section Legend window:

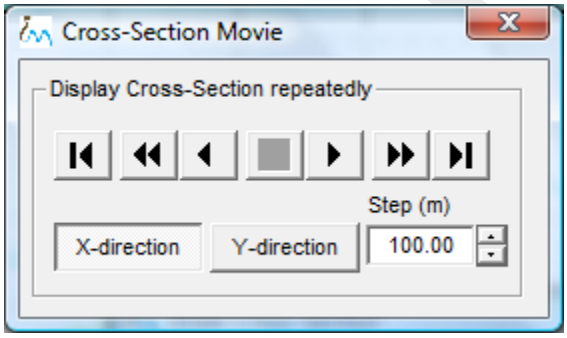


The *Cross-Section Legend* window displays the filenames of the IDF's, however, these can be adjusted in the *Cross-Section Properties* window. Moreover, a legend for each of the items in the Cross-Section are displayed on the *iMOD Cross-Section* window too.

7.1.3 Movie

Click the option *Cross-Section Movie* on the *Draw Cross-Section* window to open the *Cross-Section Movie* window.

Cross-Section Movie window:



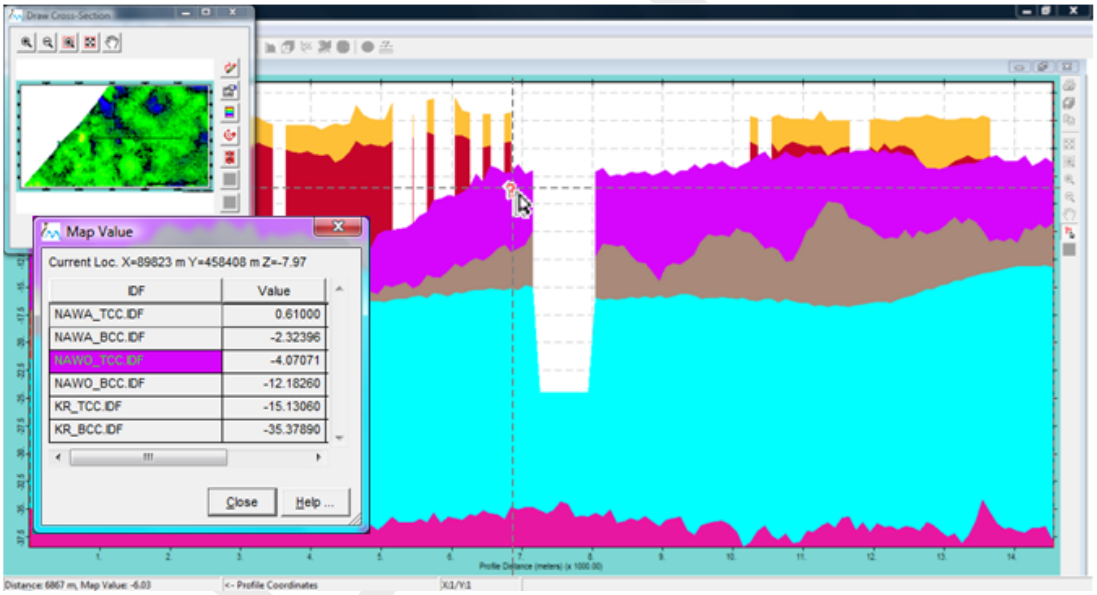
	Complete Backward Click this button to move the Cross-Section line to the utmost left (*X direction*) or utmost top (*Y-direction*) coordinate of the current extent of the graphical window.
	Fast Backward Click this button to move the Cross-Section line against the *X*- or *Y-direction*, repeatedly with the chosen *Step*.
	Single Backward Click this button to move the Cross-Section a single step against the *X*- or *Y-direction*.
	Stop Click this button to stop the actions *Fast Backwards* or *Fast Forwards*.

▶	*Single Forward* Click this button to move the Cross-Section a single step along the *X*- or *Y-direction*.
▶▶	*Fast Forward* Click this button to move the Cross-Section line along the *X*- or *Y-direction*, repeatedly with the chosen *Step*
▶|	*Complete Forward* Click this button to move the Cross-Section line to the utmost right (*X direction*) or utmost bottom (*Y-direction*) of the current extent of the graphical window.
X-direction *Y-direction*	Choose one of the directions in which the line of the cross-section moves.
Step (m)	Insert the interval for which the line of the cross-section moves repeatedly.

7.1.4 Cross-Section Inspector

Click the *Cross-Section Inspector* on the *iMOD Cross-Section* window to identify the values for each IDF at the selected position in the cross-section. You can move the mouse-cursor over the cross-section and the IDF values for each selected IDF will be displayed in the *Map Value* window.

Example of the Cross-Section Inspector option:

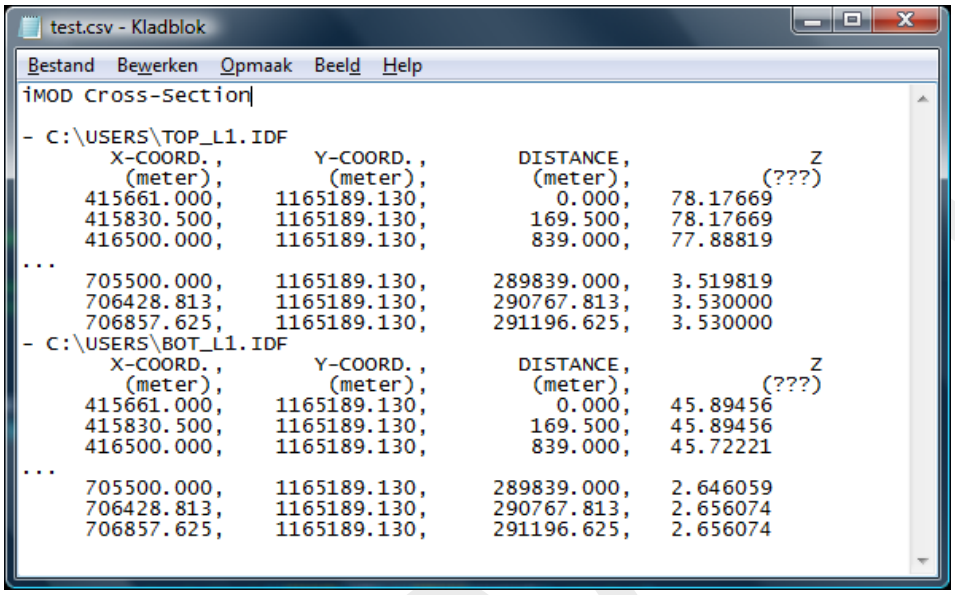


The position of the mouse will be displayed on top of the *Map Value* window (*e.g. Current Loc. X=89823 m Y=458408 m Z=-7.97*). iMOD will colour the name of the IDF in the *Map Value* window nearest to the mouse cursor. If there is any inconsistency in the IDF values (that is, whenever the IDF are not arranged such that they represent values that increase from the first IDF to the last IDF), iMOD will not colour any field, but will present the message *Inconsistency in top/bottom at current location* on the bottom of the *Map Value* window. The *Cross-Section Inspector* can be closed by clicking the left-mouse button or select the *Close* button on the *Map Value* window.

7.1.5 Export

There are two ways to export a cross-section: (1) as an image (BMP): use the *Copy to Clipboard* option from the *iMOD Cross-Section* window and paste it into a third party software application, (2) as data (*.csv): use the *Export* option from the *iMOD Cross-Section* window. For the latter, an example is given of the file format. A new data-block starts for each IDF, since, the points of intersection might differ.

Example of an export of a cross-section:



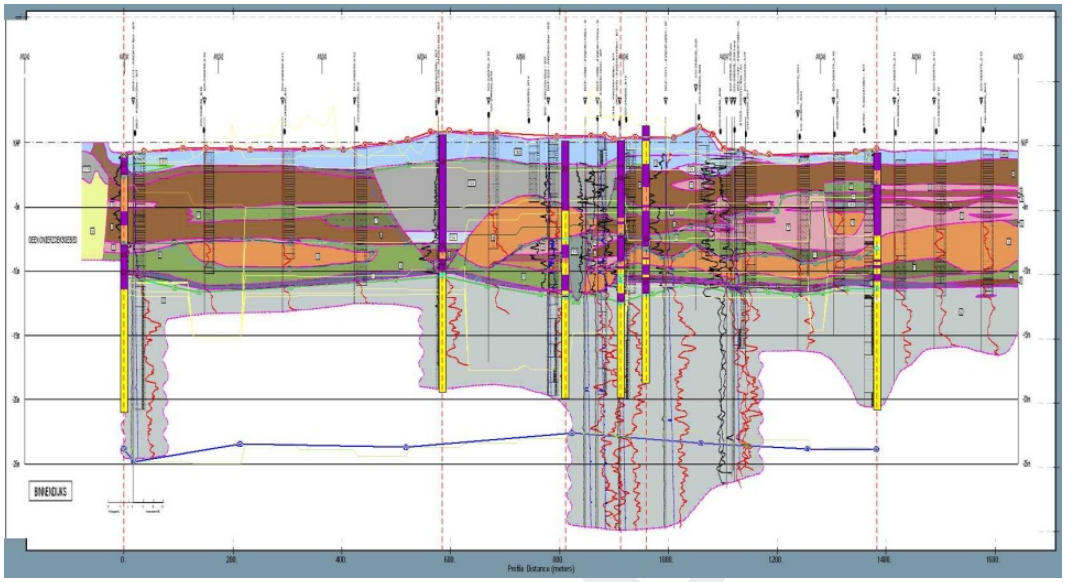
7.1.6 Background Bitmaps

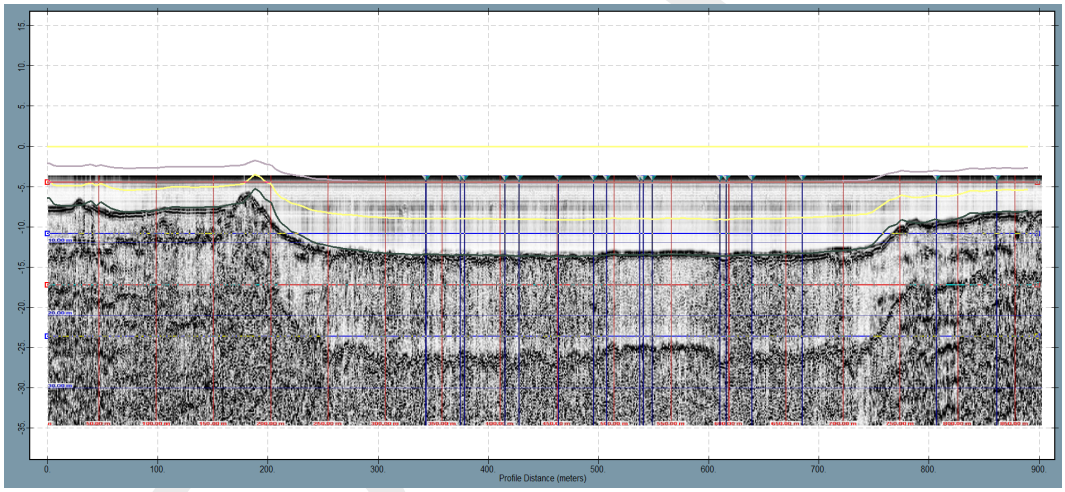
A graphical *.JPG, *.BMP- or *.png-file can be added to the iMOD Cross-Section window to be displayed as background. This could be a profile prepared outside iMOD with other graphical tools.

Click the *Add a bitmap as background* button on the *iMOD Cross-Section* window to open the file manager to select a bitmap. The bitmap will be added to the cross-section. Next the position and size will have to be set in the cross-section using the mouse.

Move your cursor on to the bitmap and you will see it change in:. Click your left mouse button and move the bitmap. Move your mouse to the edge of the bitmap, you will see it change and move the edge. Repeat this until your bitmap fits the iMOD cross-section. The position of those bitmaps will be save in a SPF file (section 9.21) that is used by the Solid Tool (section 7.4).

Examples of an iMOD Cross-Section with borehole information showing sub-surface layers added as background:





7.2 Timeseries Tool

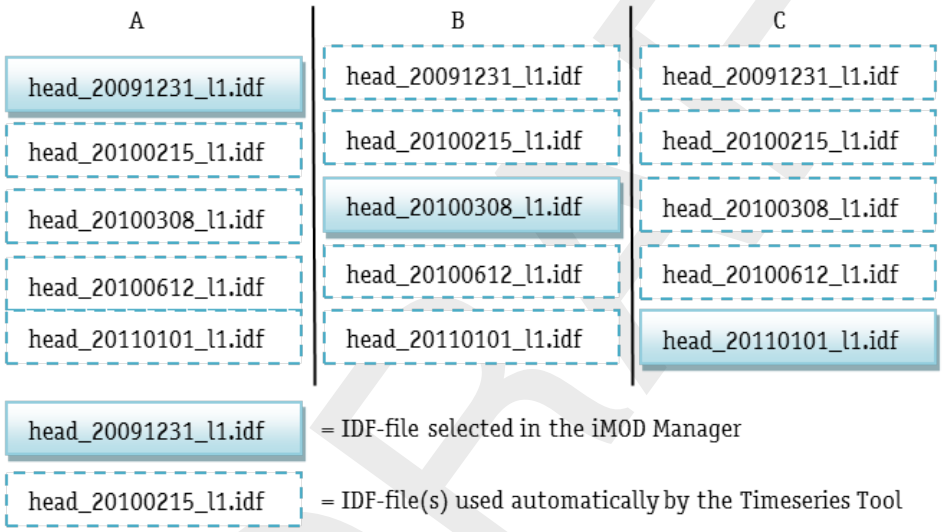
WHY?

The *Timeseries Tool* can be used to view timeseries directly for data stored in different IDF-files. These can be combined with timeseries that are associated to IPF-files.

WHAT?

The *Timeseries Tool* allows you to point at a particular location inside the full extent of an IDF-file with time-dependent data. iMOD will construct a timeseries for that particular location by collecting all data from the other related time-dependent IDF-files. Related time-dependent IDF-files have identical names but have a different date string. A date string is an eight digit continuous number, e.g. 20091231 meaning the 31^{th} of December 2009. It is not necessary to load all related time-dependent IDF-files in the *iMOD Manager.* At least one is sufficient to view the entire timeseries.

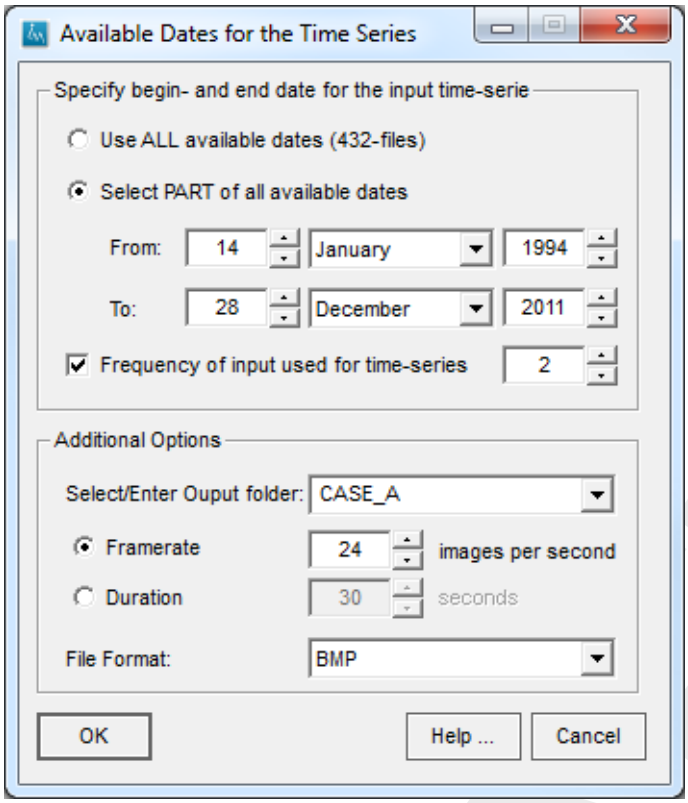
Example of IDF-files (A,B and/or C) available in the iMOD Manager prior to the start of the Timeseries Tool:



HOW?

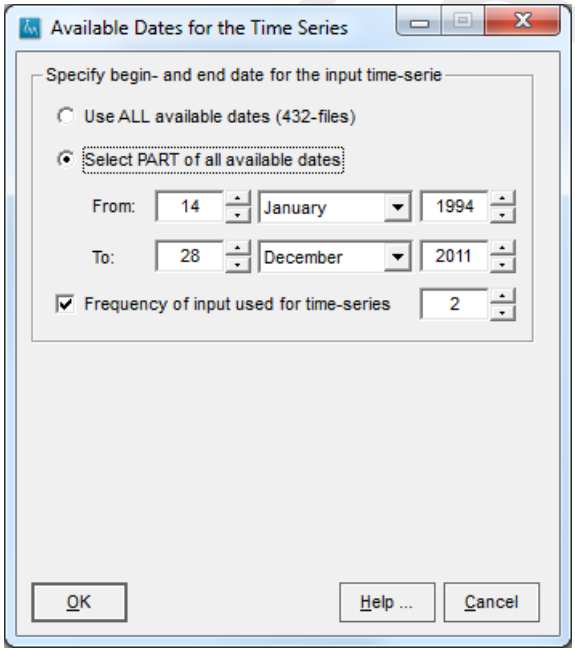
To start the *Timeseries Tool* select *Toolbox* from the main menu, choose *Timeseries Tool*. Alternatively, you can click the *Timeseries* button ( ) at the main toolbar. In both cases, you should select at least one IDF-file in the *iMOD Manager* that has a date notation in its name. This is a continuous number with eight-digits (yyyymmdd), e.g. *20110115.* In this case, it represents the 15^{th} of January, 2011. If iMOD can **not** find such a date notation somewhere in the filename (in at least one of the selected IDF-files), the following window will appear and the *Timeseries Tool* will not start.

Warning window:



If a proper IDF-file(s) has been selected in the*iMOD Manager*, the following window will appear.

Available Dates window:



Use ALL available dates (297-files)	Select this option whenever you want to display timeseries for the entire time window that iMOD found.
Select PART of all available dates	Select this option to specify a different time window. This may gain processing time as less files need to be opened.

From:	Enter the start date of the time window. On default it displays the earliest date of the data.
To:	Enter the end data of the time window. On default it displays the latest date of the data.
Frequency of input used for timeseries	Select this option to decrease the number of dates used, e.g. by entering the value 2 iMOD will skip each second available date of the time series.
OK	Click this button to start the *Timeseries Tool* for the selected time window. The *Available Dates* window will close.
Help . . .	Click this button to start the iMOD Help Functionality.
Cancel	Click this button to close the *Available Dates* window; the *Timeseries Tool* will not start.

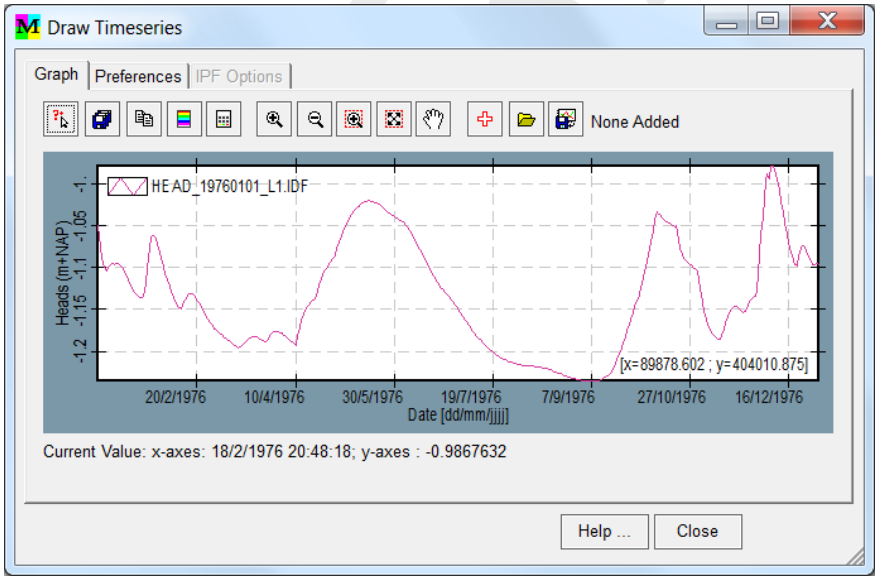
Note: You should select at least one IDF with date information in its filename, other IDF-files that are selected without a date information, will be displayed as time-constant. In this way you can easily make a combination with time-variant information (e.g. heads) and time-invariant information (e.g. surfacelevel).

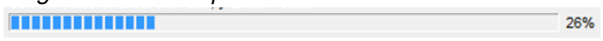
Note: When you select an IPF with associated timeseries, these timeseries will be displayed simultaneously with those obtained from the IDF-file(s). In case you specify one IDF and one IPF, the option to compute differences between them becomes available.

7.2.1 Draw Timeseries

The Draw Timeseries window consists of three tabs. The IPF Options tab is only available when IPF-files are loaded.

Draw Timeseries window, Graph tab:



	*Timeserie Hovering* Click this button to start the hovering of timeseries. Move your mouse over the main graphical window to compute the timeseries. The current location will be highlighted on the map and the corresponding timeserie(s) will be displayed immediately. iMOD will try to read and process the entire timeseries (could be more than one) within one second. If this fails, the progress bar shows the amount of data that could be processed within this time limit. *Progress bar in Graph tab:*  To complete the timeseries, you need to press the left mouse button in the main graphical window. If the progress bar is absent, the timeseries are displayed completely. To stop the *Timeserie Hovering* you need to press the right mouse button.
	*Save As* Click this button to save the current graph to a comma-separated-values file (*.CSV). More detailed information.
	*Copy* Click this button to copy the graph to windows *Clipboard*.
	*Legend* Click this button to open the*Individual Colouring* window.
	*Statistics* Click this button to open the *Time-statistics* window (. . .).
	*Zoom In* Click this button to zoom in at the location of the mouse. Stop by clicking the right mouse button.
	*Zoom Out* Click this button to zoom out at the location of the mouse. Stop by clicking the right mouse button.
	*Zoom Box* Click this button to draw a zoom window in the graph to adjust the zoom level to. Click your left mouse button for the first point and click your left mouse button for the second point.
	*Zoom Full* Click this button to adjust the zoom level to the initial value.
	*Move* Click this button to move the graph. Use your left mouse button to start moving and click your right mouse button to stop moving. The displayed differences, (*Compute Residuals* from the *Preferences* tab), are not affected by any vertical moving.
	*Add Point* Click this button to store the current location internally.
	*Open IPF-file* Click this button to open an IPF-file for which all locations are stored internally, as if you were clicking the *Add Point* button for each location.
	*Save IPF* Click this button to compute and save the timeseries for all locations that are stored internally (*Add Point* and/or *Open IPF File*). This yields an IPF-file that you can name, with associated timeseries.
Added 2 points	Displays the number of points that are added by clicking the *Add Point* button.
Help . . .	Click this button to start the iMOD Help Functionality.
Close	Click this button to close the *Timeseries Tool* window.

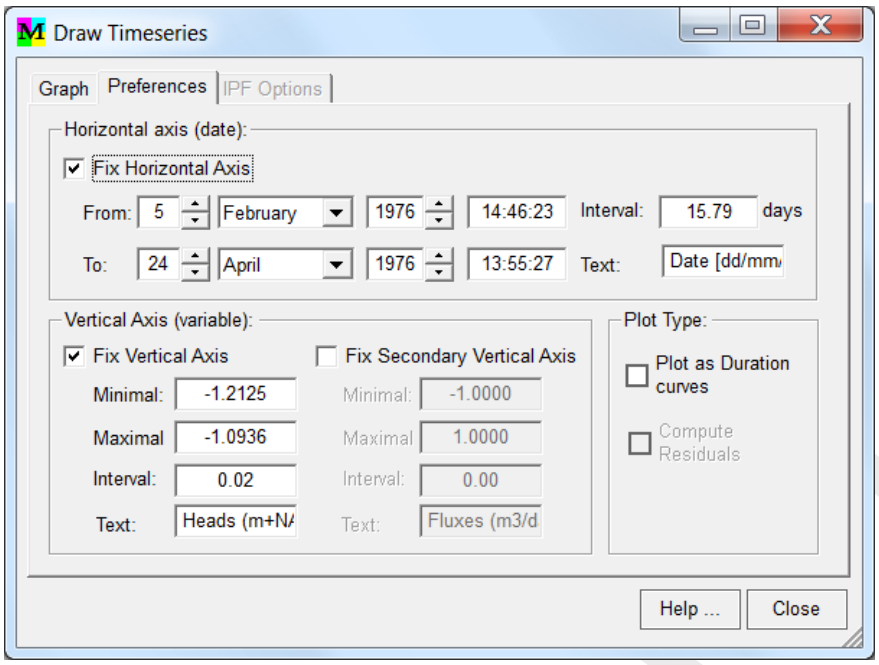
Note: Whenever you move the mouse over the *Timeseries* graphical window, the current coordinates are displayed below the graph. It shows the current date (x-axis) and the corre-

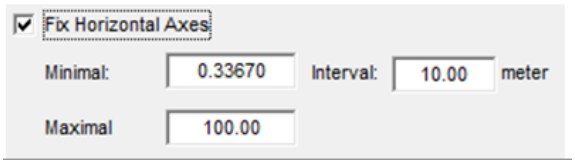
sponding value on the y-axis. It shows *Current Value: x-axis 14/2/1997; y-axis: -1.154*.

Note: Whenever you include an IPF-file with associated timeseries, iMOD will display the timeserie(s) of the point of the IPF that nearest to the current location of the mouse. You can fixate a particular point on the *IPF-options* tab.

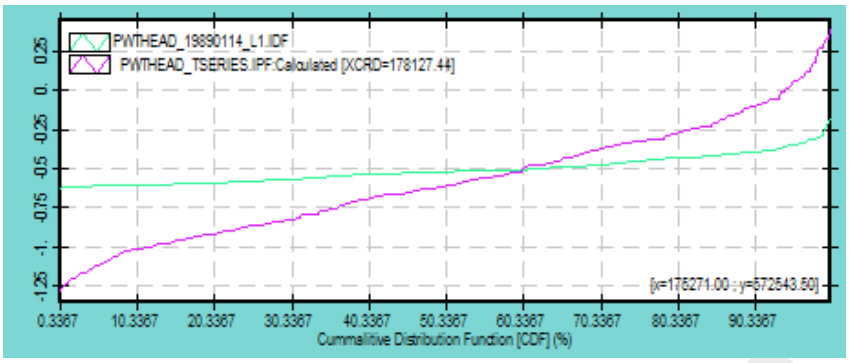
Note: When using the *Timeseries Tool* you will be able to use the functions *ZoomIn, ZoomOut, ZoomBox, ZoomFull, Move* and *DistanceTool* of the main graphical window.

Draw timeseries window, Preferences tab:



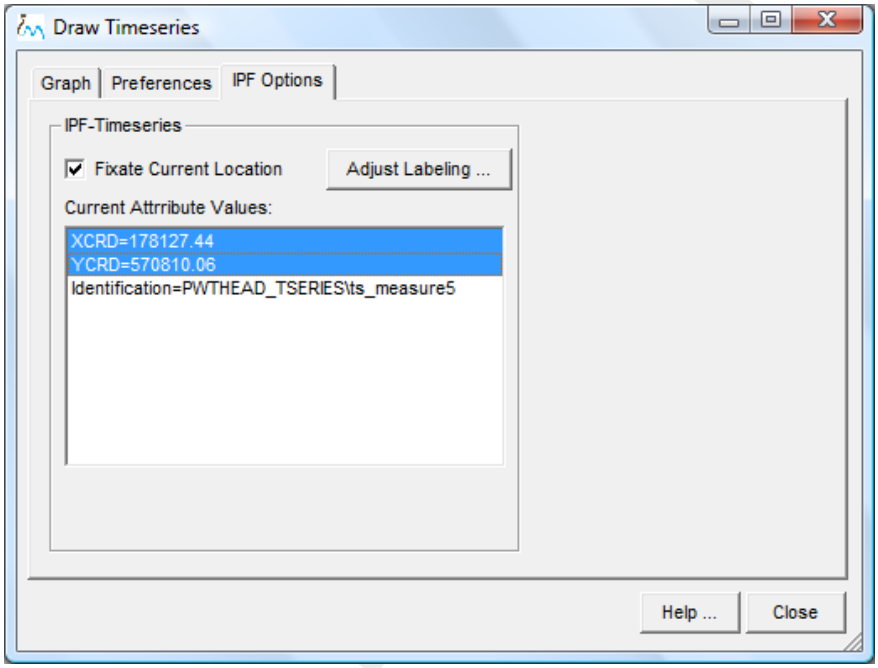
Fix Horizontal Axis (date)	Click this option to specify the horizontal axis manually.
From:	Enter the start date of the horizontal axis, you can specify start time as hh:mm:ss.
To:	Enter the end date of the horizontal axis, you can specify end time as hh:mm:ss.
Interval (days)	Enter the interval of the horizontal axis in days.
Text:	Enter the text to be displayed at the horizontal axis
Fix Vertical Axes (variable)	Click this option to specify the vertical axes manually.
Fix Secondary Vertical Axes	Click this option to specify the second vertical axes manually (appears on the right of the graph). This option becomes available whenever the option *Compute Residuals* is selected.
Minimal	Enter the minimum value for the vertical axes.
Maximal	Enter the maximum value for the vertical axes.
Interval	Enter the interval for the vertical axes.
Text:	Enter the text to be displayed at the vertical axis
Plot as duration curve	Click this option to plot all figures as duration curve (Cumulative Distribution Function). The entry for the option *Fix Horizontal Axis* changes whenever this checkbox is selected. *Fix Horizontal Axis for Duration curves:* 

Example of duration curves:



Compute Residuals	Click this option to compute and display differences between the first and second file. That could be the difference between two IDFs or the difference between an IDF and the associated timeserie of a point in an IPF.

Draw timeseries window, IPF Options tab:

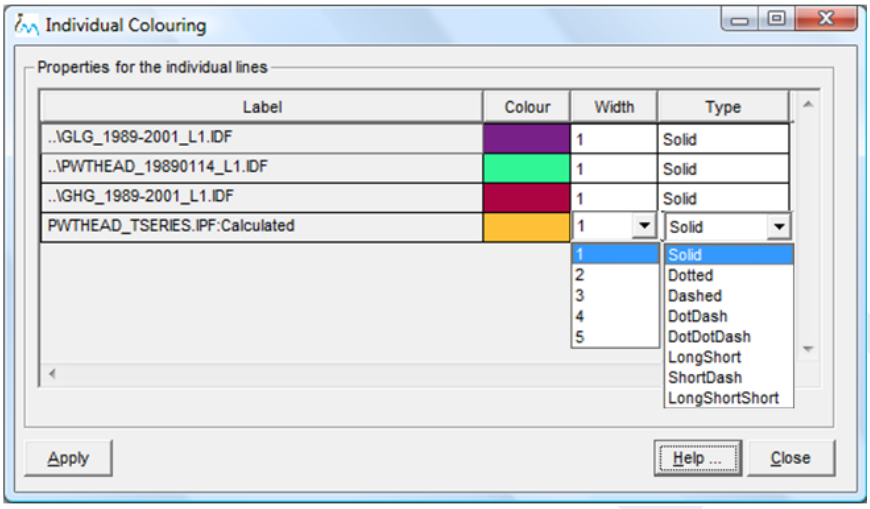


Fixate Current Location	Check this option to fix the current location of the point in the IPF that is used to display the timeserie.
Adjust Labeling	Select this button to display labels for the points in the IPF on the main graphical window.
Current Attribute Values	Select the attributes from the list that will be displayed in the legend section on the graph.

7.2.2 Legends

Click the option *Legend*  on the *Graph* tab to open the *Individual Colouring* window.

Individual Colouring window:



Label	The label of the loaded file (IDFs and/or IPFs)
Colour	The current colour of the line in the graph. Click this field to open the *Colour* window.
Width	Click this field to open a dropdown menu with the width of the lines (1-5)
Type	Click this field to open a dropdown menu with the different linetypes.
LineStyle	Click this field to open a dropdown menu to choose between Continuous or Blocklines
Axes	Click this field to open a dropdown menu to choose the axis
Apply	Click this button to accept the adjustments and close *the Individual Colouring* window.
Help . . .	Click this button to start the iMOD Help Functionality.
Close	Click this button to close *the Individual Colouring* window without any adjustments.

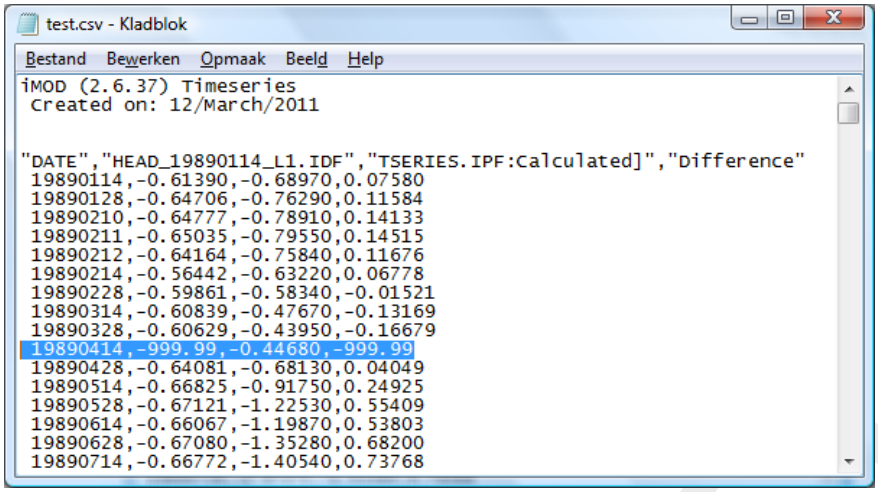
7.2.3 TimeSeries Export

There are two ways to export the timeseries data from the Timeseries Tool to ASCII-files, as comma-separated-file-values (*.CSV) to be read by Excel, or/and as an IPF-file with associated timeseries for direct use in iMOD. When the option *Compute Residuals* on the *Preferences* tab is selected, an extra column will be added with the computed differences. Moreover, whenever the option *Plot as duration curve* on the *Preferences* tab is selected, the export will describe the duration curves instead. Time-invariant IDF–file(s) will be exported with their time constant values.

Comma-Separated-Values File

To export the current timeserie(s), click the *Save As* option from the *Graph* tab to select a file. The export file will be a comma-separated-values file (*.CSV) and can be read directly into Excel. The first column of the export file prints the data (yyyymmdd), the second, third and so on (as many columns as files active in the Timeserie Tool) print the timeserie(s).

Example of a comma-separated-values file by the Timeserie Tool:

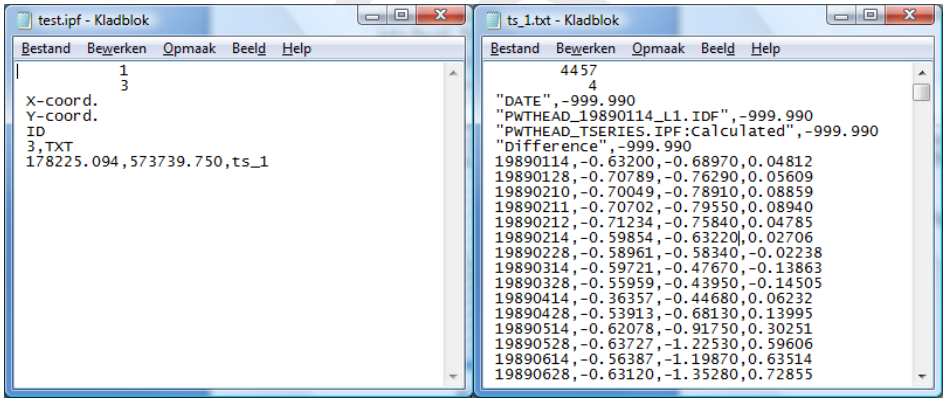


Note: The number of records in the export file depends on the dates that contain data for each column. In case column2 has no data *(NoDataValue=-999.99)* than the computed difference is nodata too. Since, column three has data, the 14th of April, 1989 is exported whatsoever.

IPF-file

To export the current timeserie(s), click the *Save IPF* option from the *Graph* tab to select a file. The export file will be an *.IPF file with an associated timeserie TXT-file. The latter can be read into Excel directly.

Example of an IPF-file (left) and the associated text file (right) exported by the Timeserie Tool:



7.3 3D Tool

WHY?

The 3D Tool can be used to visualize raster data (IDF), point data (IPF, with/without boreholes), polygon data (GEN), flowline data (IFF) in a 3D viewing environment.

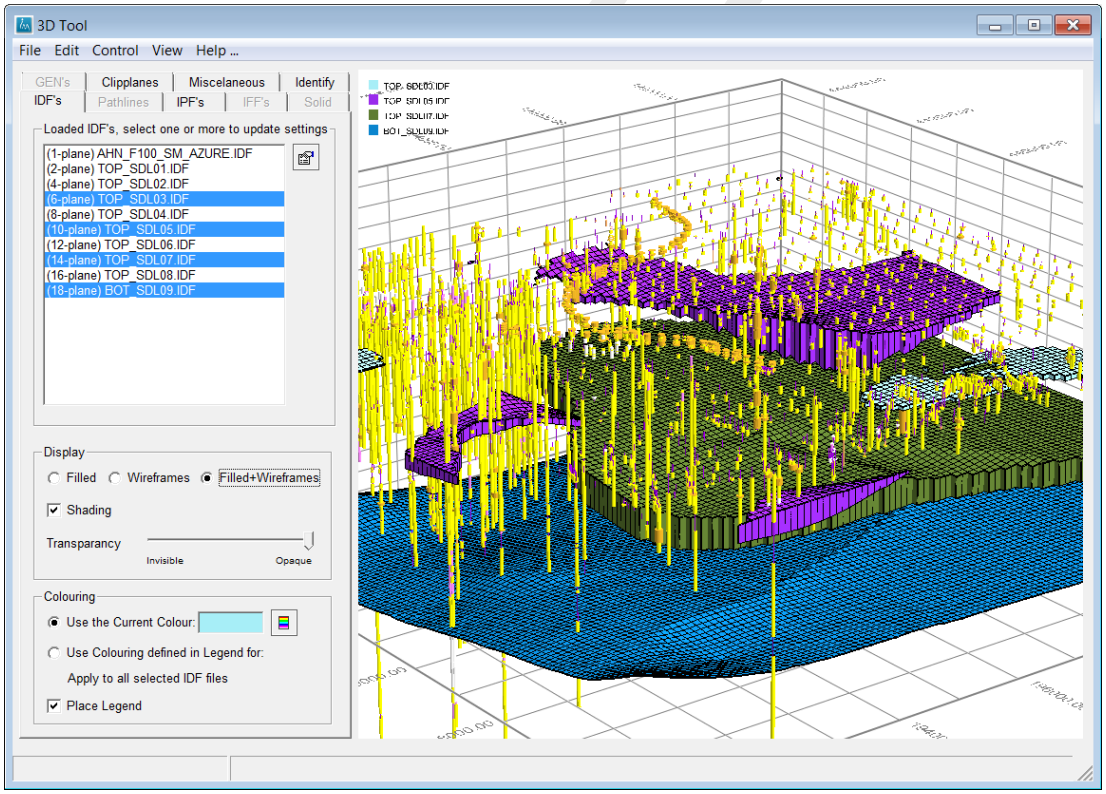
WHAT?

iMOD is equipped with an OpenGL library (Open Graphics Library) that is a standard specification for writing applications that produce 2D and 3D computer graphics. OpenGL was developed by Silicon Graphics Inc. (SGI) in 1992 and is widely used in CAD, virtual reality, scientific visualization, information visualization, flight simulation, and video games.

HOW?

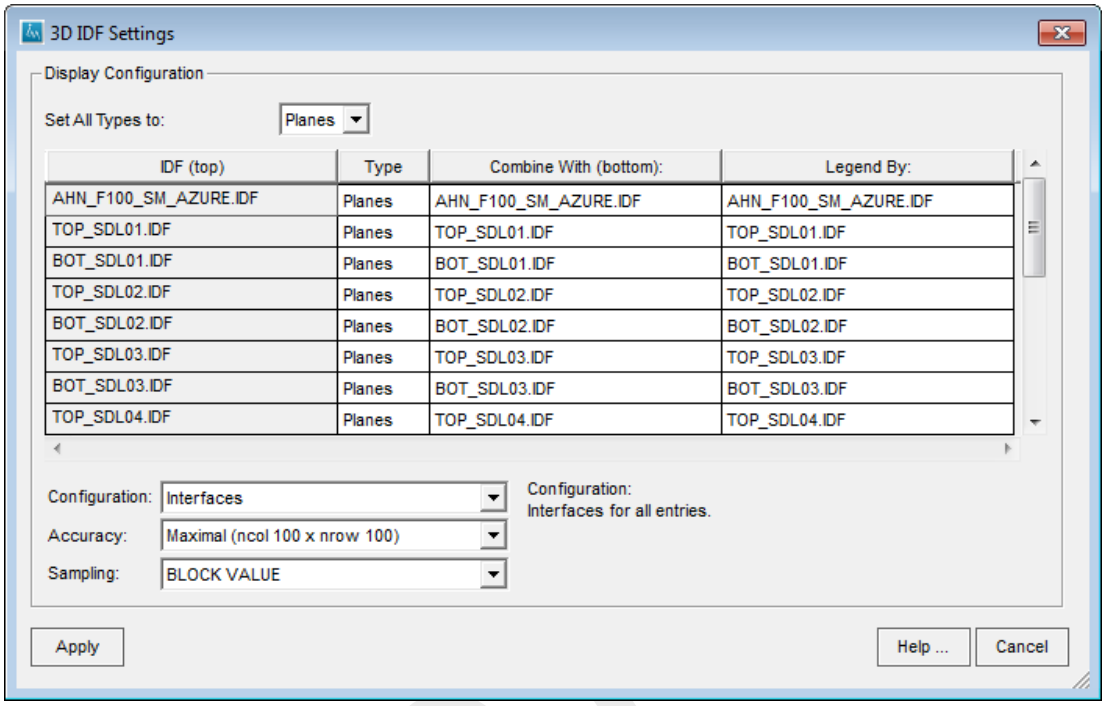
To use the OpenGL functionalities in iMOD, simply select the map(s)(IDF, IPF, IFF, GEN, MDF) from the *iMOD Manager* and select the option *Toolbox* from the main menu and then choose the option *3D Tool* to start the *3D Tool*.

Example of the 3D tool:



7.3.1 Starting the 3D Tool

The *3D Tool* starts by selecting the *3D Tool* option from the main menu option *Toolbox*. Alternatively, the  -button can be selected from the main window. In the situation IDF files are selected in the *iMOD Manager* window, the *3D IDF Settings* window pops up. In this window each IDF can be configured how it is presented in 3D.

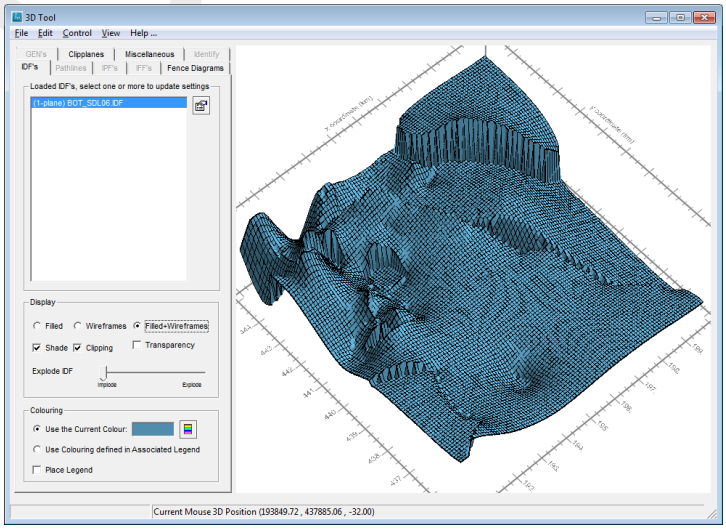


Set all Types to

Select the type for all IDF files, any individual modification can be done in the table. Select one of the following:

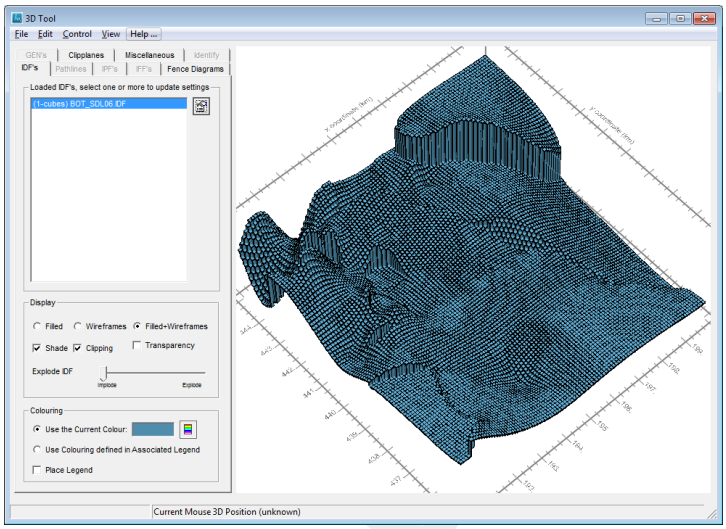
Planes
Select this option to compute the 3D image as quadrilateral between the cell mid of four adjacent cells.

Example of a planes representation.



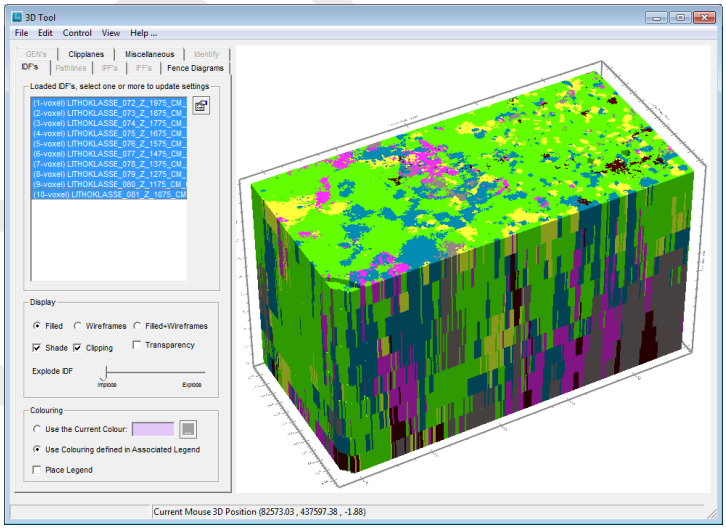
Cubes
Select this option to present the IDF using the individual grid cells values as constant for each individual cell.

Example of a cubic representation.



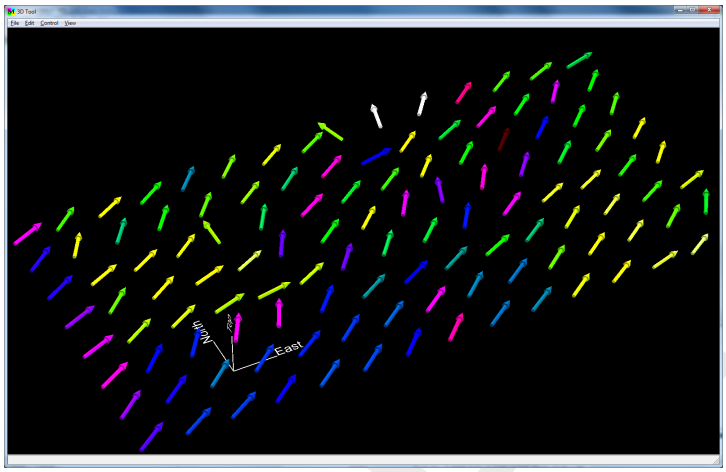
Voxels
This option is selected automatically whenever the IDF-file represents a voxel (see section 9.5 for the syntax of an IDF file and how to switch to a voxel representation for an IDF file).

Example of a voxel representation.



Vectors
This option is selected automatically whenever the IDF represents a vector field. See section 9.5 for more information about vectors. Use the iMOD-Batch function CREATEIVF to create vector IDF's.

Example of a vector representation.



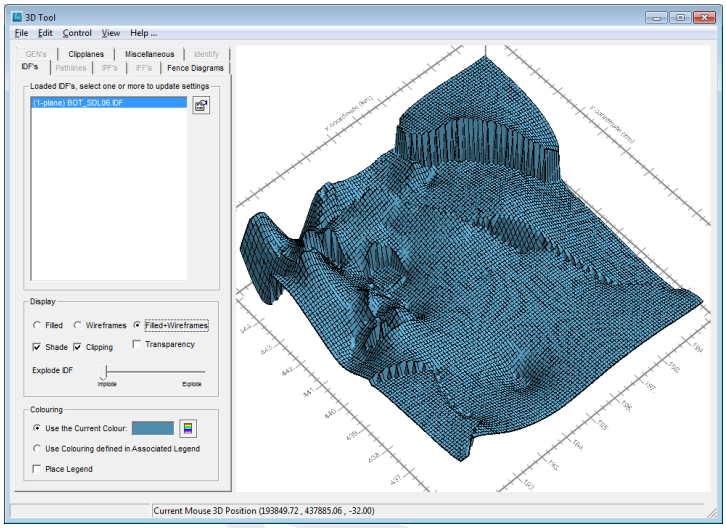
Off
Select this option to turn the IDF off temporarily.

Table: IDF (top)	Displays the current name of the IDF-file, combined with another IDF this would be the top. It is not possible to change the entry.
Table: Type	A type can be selected per individual IDF. For the available types see the *'Set All Types to'*-description above.
Table: Combine With (bottom):	Select the IDF-file from the drop-down menu that needs to be combined with the IDF-file specified in the first column (*IDF (top)*) of this table. In this way the first IDF will be used to identify the top of the solid, the *'Combine With (bottom)'* IDF will be the bottom. It gives a solid representation of these two IDF's.
Table: Legend By:	Select the IDF-file from the drop-down menu for which the associated legend needs to be used to colour the 3D representation of the *IDF (top)* in combination with the entered IDF at *Combine With (bottom)*. Several possibilities arise, you may want to colour an IDF using an associated legend of another IDF and/or whenever two IDF-files are combined as a solid, it could be coloured by an associated legend of an IDF that represent the permeability of the solid. Use the different options at *Configuration* to modify the setting in the table automatically.

Configuration Select an option from the drop down list *Configuration* to display a layer model using a predefined display mode.

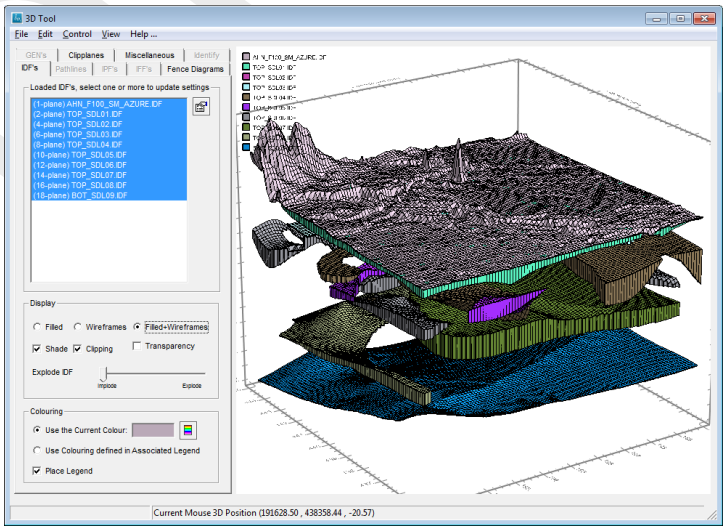
Interfaces
Use this option to represent each IDF separately as an interface.

Example of an Interface visualisation.



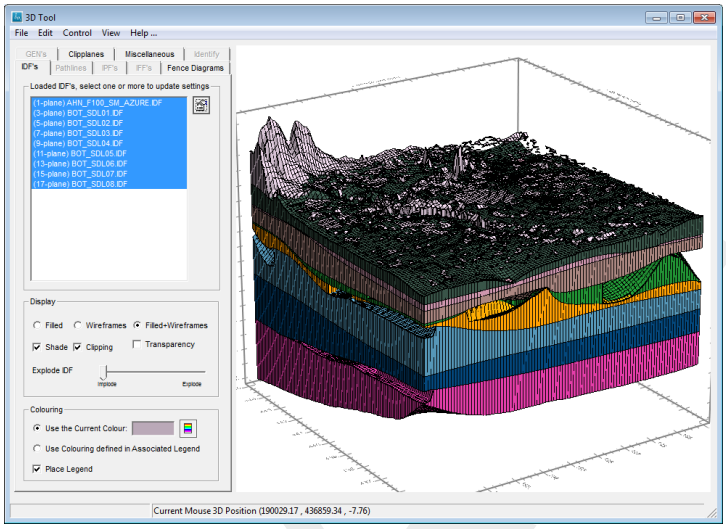
Quasi 3D model (aquitard)
Select this configuration to arrange the *Table* such that the first IDF will be the top surface level and presented as a white fishnet. Further downwards, IDF number 2 will be combined with number 3, number 4 with number 5 and so on. The last IDF will be presented as a normal plane. This option is especially handy whenever a Quasi-3D model needs to be presented showing aquitards as solid bodies.

Example of a Quasi 3D visualisation of aquitards.



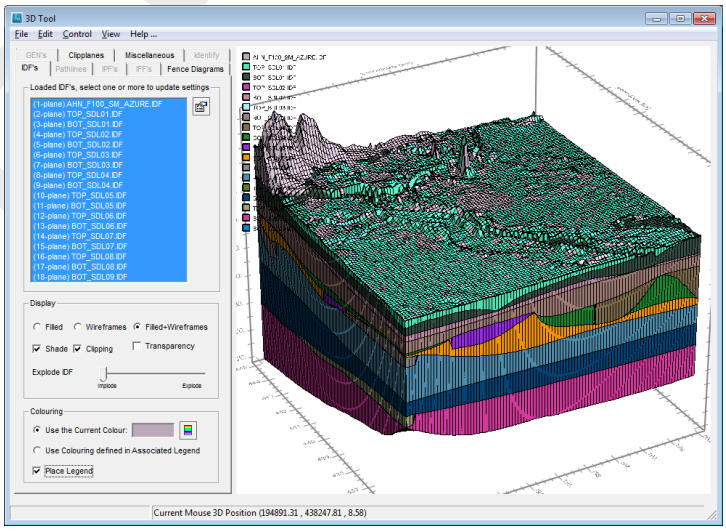
Quasi 3D model (aquifer)
Select this configuration to arrange the *Table* such that the first IDF will be the top aquifer and combined with the second surface; further downwards, IDF number 3 will be combined with number 4, number 5 with number 6 and so on. This option is especially handy when a Quasi-3D model needs to be presented showing aquifers as solid bodies.

Example of a Quasi 3D visualisation of aquifers.



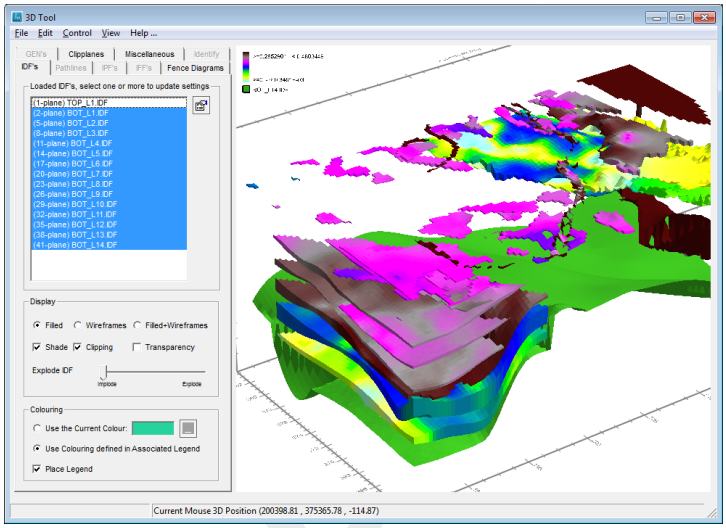
3D Model
Use this configuration to combine each IDF with the following IDF, so IDF number 1 will be combined with IDF number 2 and that will be combined with IDF number 3.

Example of a 3D Model visualisation.



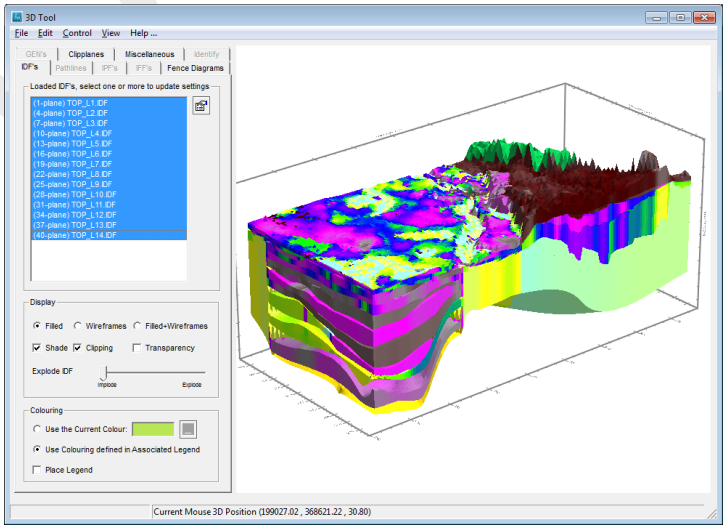
Coloured Quasi 3D Model (aquitard)
Use this configuration to present and colour aquitards by sets of 3 IDF's per aquitard using the associated legend of each second IDF of that set of 3 IDF's to colour the aquitard. The first IDF will be draped as a white fishnet. Then it will combine the second with the fourth IDF to form a solid and colour it by the associated legend of the third IDF. Then it will combine the fifth with the seventh to form a solid and colour it using the legend of the sixth, and so on.

Example of a coloured Quasi 3D (aquitard) visualisation.



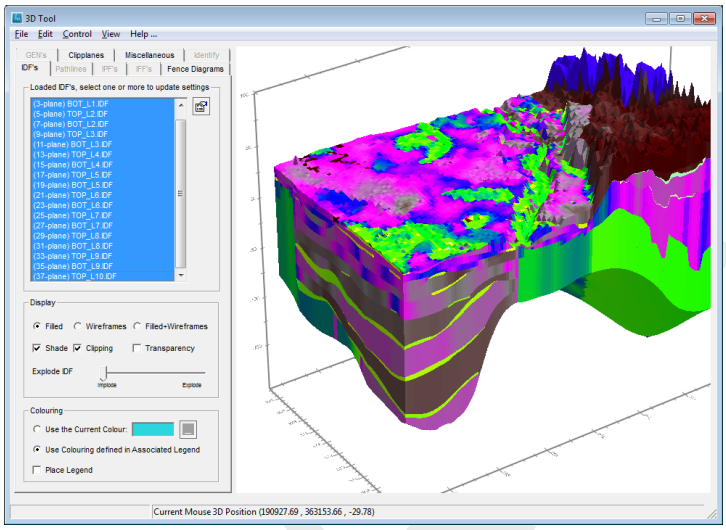
Coloured Quasi 3D Model (aquifer)
Use this configuration to present and colour aquifers by sets of 3 IDF's per aquifer using the associated legend of each second IDF of that set of 3 IDF's to colour the aquifer. It will combine the first with the third IDF to form a solid and colour it by the associated legend of the second IDF. Then it will combine the fourth with the sixth and colour it by using the associated legend of the fifth, and so on.

Example of a coloured Quasi 3D (aquifer) visualisation.



Coloured 3D Model
Use this configuration to present a complete 3D model with the first IDF combined with the third to create a solid and colour it according to the associated legend of the second IDF. Then it will create a solid with the third and fifth IDF and colour it using the associated legend of the fourth IDF and so on.

Example of a coloured 3D Model visualisation.



Accuracy	Select the accuracy of the representation of the loaded IDF-files. To increase the performance select a lower resolution, to increase the accuracy of the representation select a higher resolution. Bear in mind that a high resolution will need more resources from the graphical card. If this fails, then iMOD will crash. Choose from: ◇ **Minimal (100 x 100):** IDF-file will be represented by a maximum 100 x 100 cells; ◇ **Low (max 250 x 250):** IDF-file will be represented by maximum 250 x 250 cells; ◇ **Normal (max 500 x 500):** IDF-file will be represented by maximum 500 x 500 cells; ◇ **High (max 750 x 750):** IDF-file will be represented by maximum 750 x 750 cells; ◇ **Very High (max 1000 x 1000):** IDF-file will be represented by max 1000 x 1000 cells; ◇ **Maximal (max ncol x nrow):** IDF-file will be represented by maximal the size of the IDF, e.g ncol x nrow cells.
Sampling	Select the method of upscaling whenever the chosen *Accuracy* is lower than the dimensions of the original IDF-file. Select from the dropdown list, for more detail about these type of upscaling, go to section 6.7.3.
Apply	Click this button to to close the *3D Plot Settings* window and apply the chosen display configurations and start the *3D Tool* window.
Help . . .	Click this button to start the Help functionality.
Cancel	Click this button to close the *3D Plot Settings* window, the *3D Tool* window will not start.

Note: Bear in mind that iMOD will blank out all elements of an IDF that have a white colour. In this way it is easy to blank out a specific area, such as high permeable areas by giving them

a white colour.

Note: It is possible to select the configuration of the table and paste it into e.g. Excel. From there may it might be easier to detail your configuration and copy and paste it from Excel to the iMOD your configuration-table of the *3D Tool Settings* window.

7.3.2 3D Tool: the Menu bar

The *3D Tool* graphics window has the following pull down menus:

File	*Print . . .* Select this item to print the current view via the Windows Print Manager onto the external printer.
	Save As . . . Select this item to save a bitmap (*.BMP; *.PNG; *.JPG) of the 3D image.
	Save As Demo . . . Save 3D Tool settings as an iMOD-demo in a new IMF-file.
	Quit 3D Tool . . . Select this item to close the 3D Tool.
Edit	*Copy to Clipboard* Click this item to copy the current image to the Windows Clipboard.
Control	*Walk Mode* The image can be moved with the left mouse button as if walking through the 3D display.
	Mouse Left Button; *Mouse Wheel;* *Mouse Right Button;* *Keyboard Cursor Keys* Select these to modify the behaviour of mouse buttons, mouse wheel and keyboard cursor keys. For all choices the following operations can be assigned: ◇ ***Rotate:*** Assign this item to rotate the image; ◇ ***Pan:*** Assign this item to pan/move the image; ◇ ***Zoom:*** Assign this item to zoom in/out the image; ◇ ***Scale X:*** Assign this item to scale the horizontal x axes; ◇ ***Scale Z:*** Assign this item to scale the vertical z axes; ◇ ***Scale Y:*** Assign this item to scale the horizontal y axes; ◇ ***Scale XY:*** Assign this item to scale both horizontal axes simultaneously.
View	*All Solids* Select this item to transform all IDF-files to solids.
	All Wireframes Select this item to transform all IDF-files to wireframes.
	All Solids+Wireframes Select this item to transform all IDF-files to solids and wireframes.
	All Shades Select this item to switch a shade on/off for all IDF-files (only for option *All Solids*).
	Show IDF Legends Select this item to display all legends for selected IDF files.
	All Single Colour Select this item to apply a single colour to the IDF-files.

All Legend Colour
Select this item to apply colours with the associated legend.

Show Axes
Select this item to display axes.

Show Orientation
Select this item to display an orientation box.

Show 3D Coordinates
Select this item to display the position of the mouse in 3D coordinates at the status bar (always showing this) of the *3D Tool* window as well as a graphical display of the location via a straight red line. The position of the mouse can only be determined when the mouse is positioned on a visible IDF/IPF/IFF file. Here the use of a transparency-option does hamper to yield a correct mouse position.

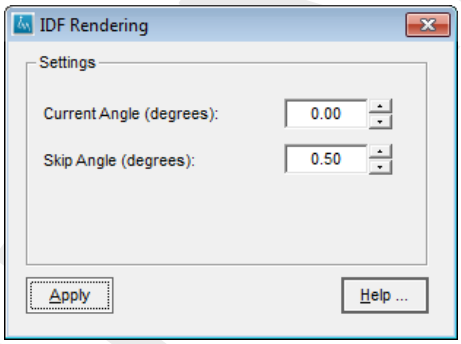
Anaglyph
Switch to an anaglyph representation of the 3D image. Use this option to gain a 3D experience using a cyan-red coloured glasses.

Orthographic Projection
Switch to an orthographic projection without perspective.

Reset View Angle
Select this item to reset the viewing angle to the initial view.

Scenic Path . . .
Select this item to specify the dimensions of a circle around the current viewpoint. The following window pops up:

Scenic Path window.



Current Angle (degrees)
Enter the angle ...

Skip Angle (degrees)
Enter the angle to be skipped in sequential rotations, e.g. a value of 5 will skip 5 degrees during a sequential rotation.

Apply
Select this button to apply the modified entries.

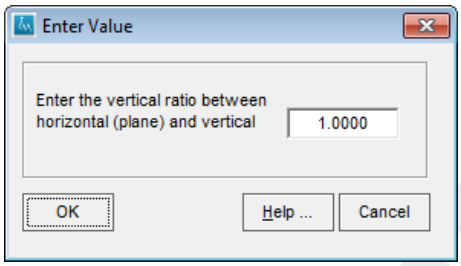
Help . . .
Select this button to start the Help functionality.

Whenever the spacebar is invoked, iMOD will render the image following the *scenic path* automatically and repeatedly.

Horizontal / Vertical Ratio
Select one of the following 5 predefined ratios *Scale 1:*$\{x\}$ from the horizontal and vertical dimensions. iMOD computes these automatically whereby the maximal scale generates a 3D box, that has the same height as the maximum between the size in x- or y-direction. In linear steps this is decreased, e.g. *Scale 1:25*; *Scale 1:20 Scale 1:15 Scale 1:10 Scale 1:5*.

Specify . . .
Select this option to enter a user-specified ratio via the following window that pops up.

Enter Value window



Enter the vertical ratio between horizontal (plane) and vertical
Enter a value to express the ratio between the horizontal plane and the vertical, e.g. 2.0 means that the vertical is exaggerated twice compared to the horizontal.

OK
Select this button to apply the entered vertical scale ratio. After this the *Enter Value* window will be closed and the entered vertical scale ratio is added to the list of available ratio in the pop up menu at the option *Horizontal / Vertical Ratio*.

Help . . .
Select this button to start the Help functionality.

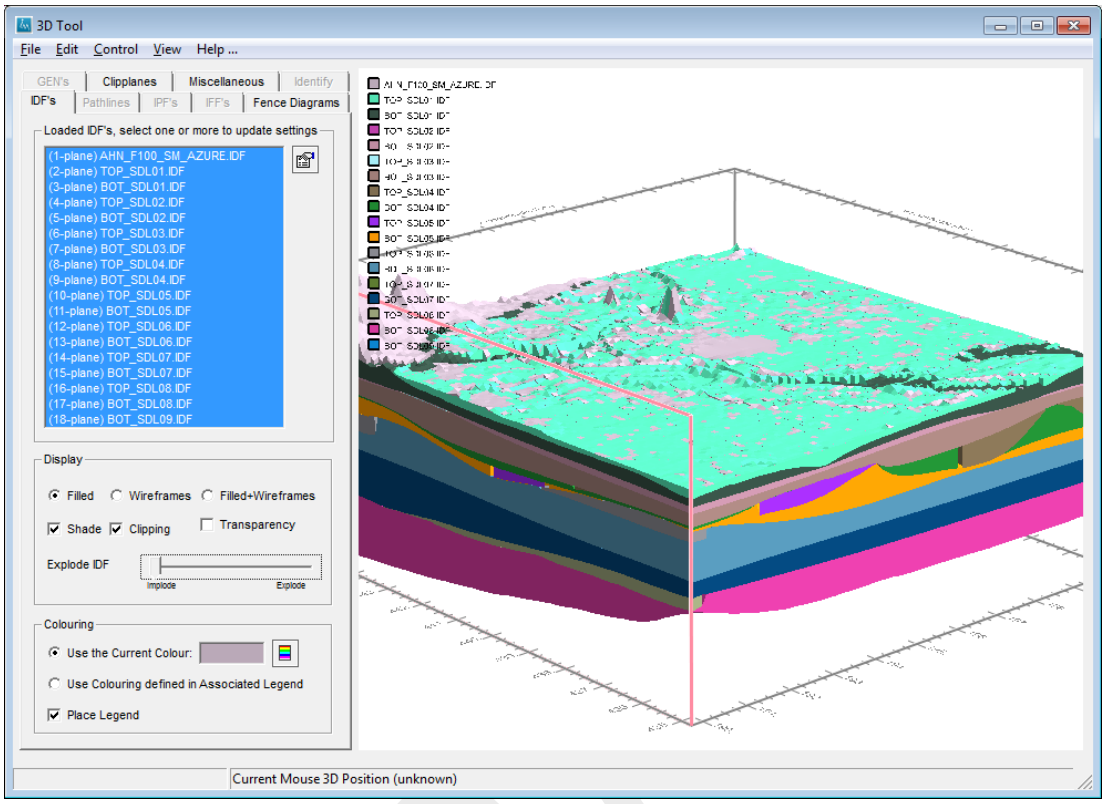
Cancel
Select this button to close the *Enter Value* window without modifying the current vertical scale ratio.

Click one of these buttons to minimize, maximize and/or close the *3D Tool*.

The *3D Tool* window has separate tabs to configure the display for different types of data, e.g. there is a tab to configure IDF files and a tab to configure IPF files. There is a tab for general settings as well. In the following sections, those tabs will be explained in detail.

7.3.3 3D Tool: the IDF-settings tab

Here, the configuration can be applied for the display of IDF files. This is done on the *IDFs* tab on the *3D Plot* window.

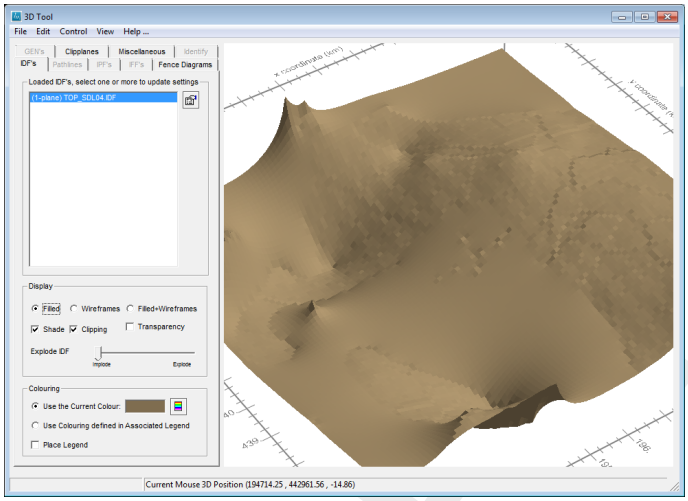


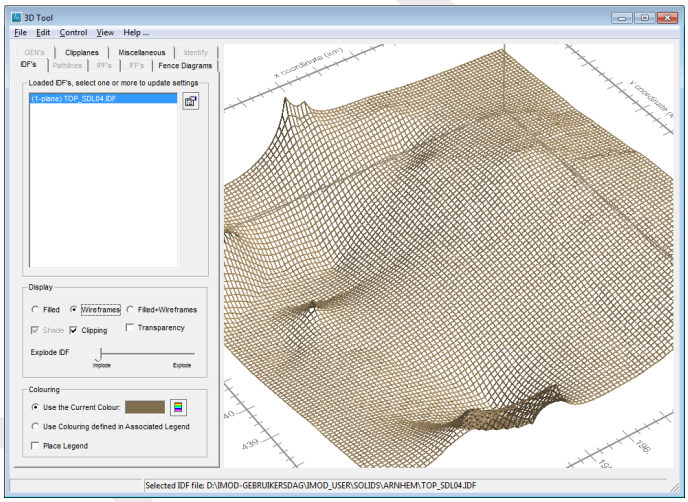
Loaded IDF's, select one or more to update settings	Select one or more IDF-files from the list to activate them in the 3D visualization. The 3D display will update each time an IDF is (de)selected. Click the option *Recompute vertical axes instantaneously* in the tab *Miscellaneous* to recompute the vertical axes instantaneously each time a different (set of) IDF-files is selected.
	*Properties* Click this button to change the appearance of IDF-files. The *3D IDF Settings* window will appear. This is the same window which appears when starting the 3D tool, see section section 7.3.1.

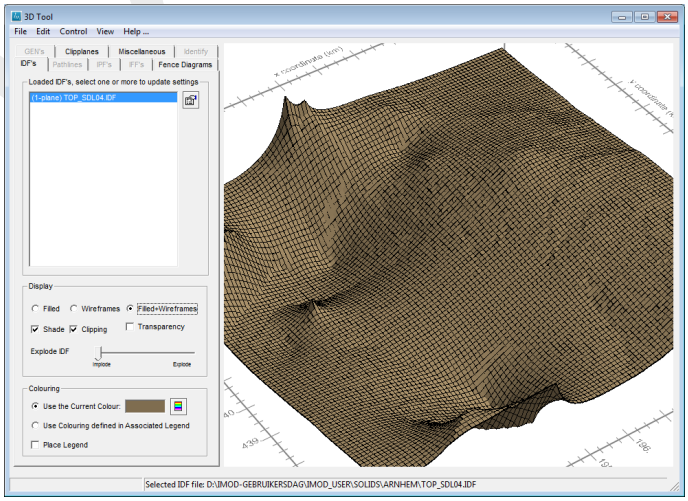
Filled; Wireframes; Filled+Wireframes

The following options configure the appearance of the IDF files. Choose one of these to display the selected IDF-files.

Examples when selecting 'Filled' (top figure), 'Wireframes' (middle figure) and 'Filled+Wireframes' (lower figure) in the IDF-Display sub-window:



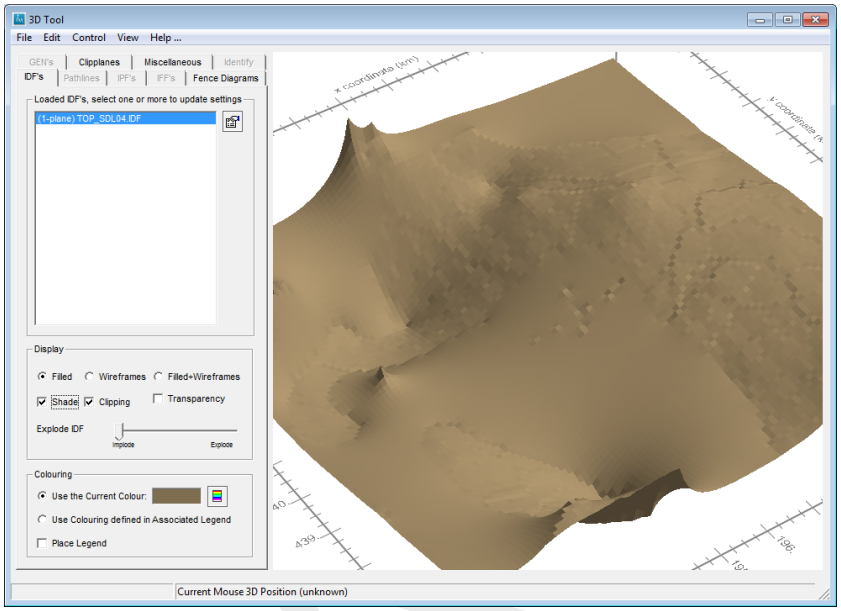


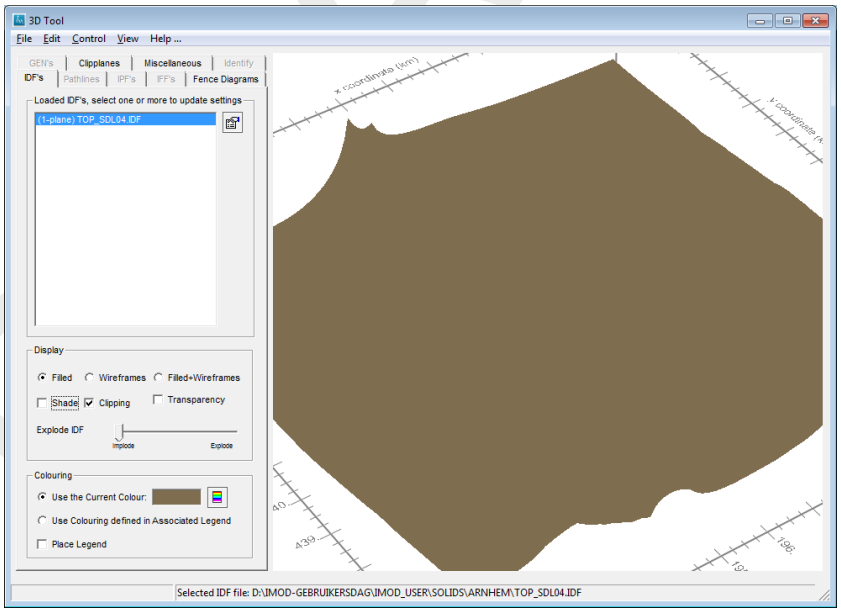


Shade

Select this checkbox to apply shades directly to the selected IDF-files. If no shade is applied, the surfaces appear flat. This can be specified per IDF file separately or as a group.

Examples when turning Display-option 'Shade' on (upper figure) and off (lower figure):

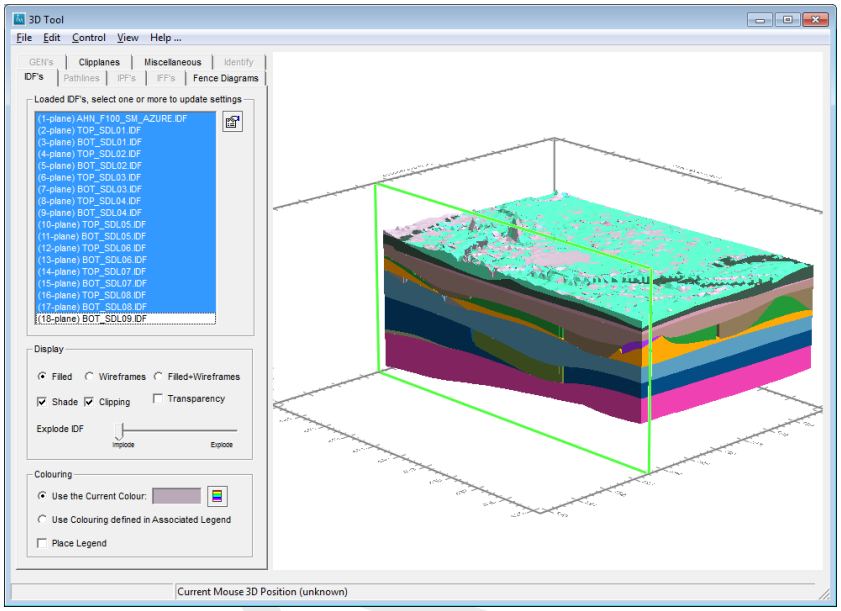


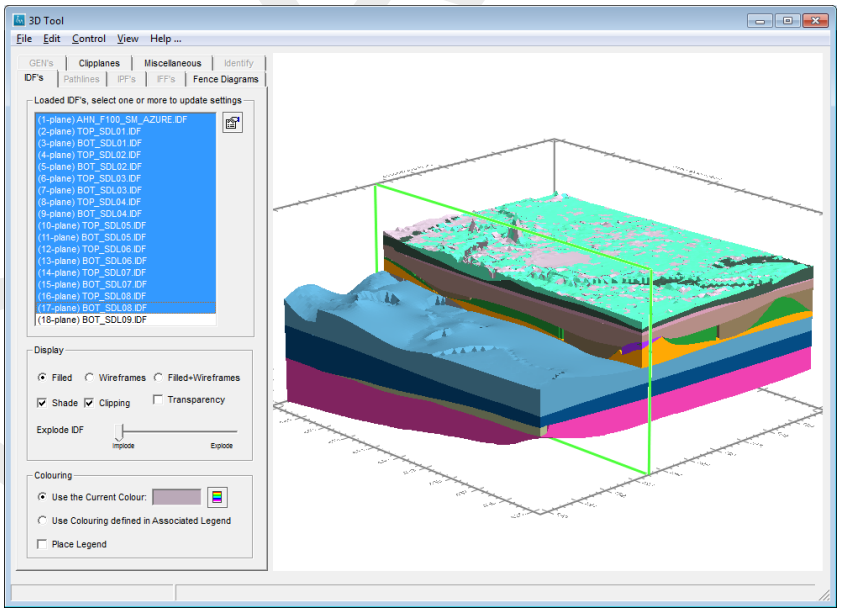


Clipping

Select this checkbox to clip a selected set of IDF file. By default all IDF files are clipped, but it is possible to set the clipping per IDF or for a group of IDF files. The actual clipping is done via the *Clipping* tab in the *3D Tool* window.

Example of clipping for all IDF files (upper figure) and clipping a selected set of IDF files (lower figure):

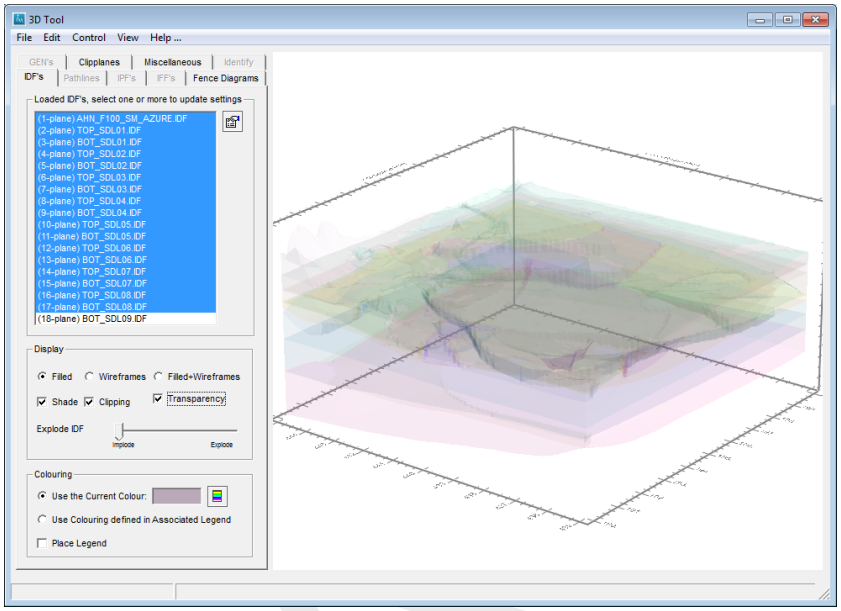


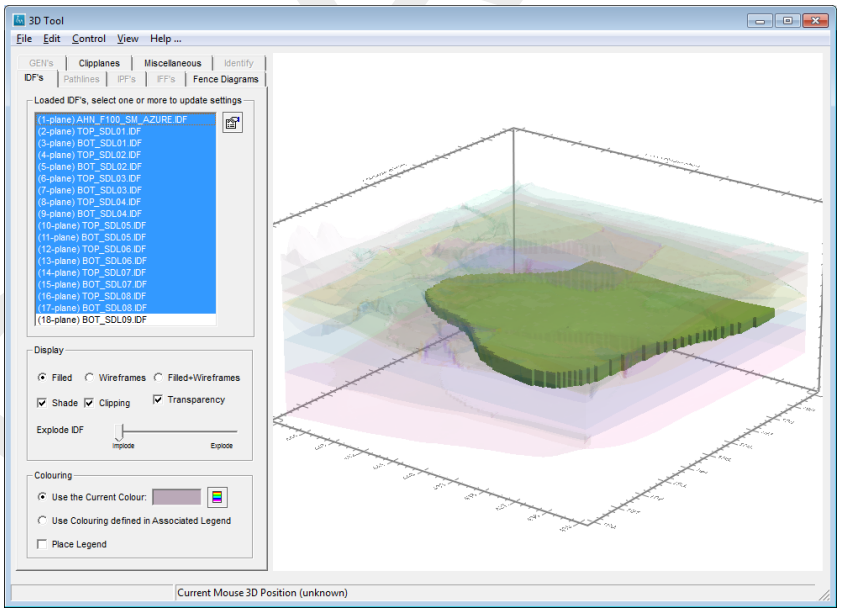


Transparency

Select this checkbox to apply transparency to the selected IDF files. Each selected IDF is assigned the same amount of transparency. It is also possible to specify transparency per IDF file.

Example of transparency for all IDF files (upper figure) and excluding a single IDF file from being displayed transparent (lower figure):

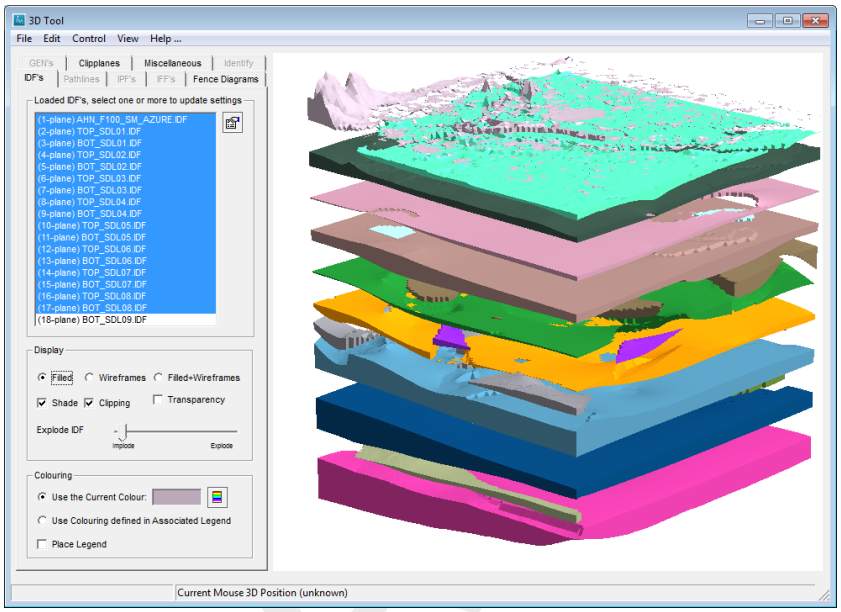


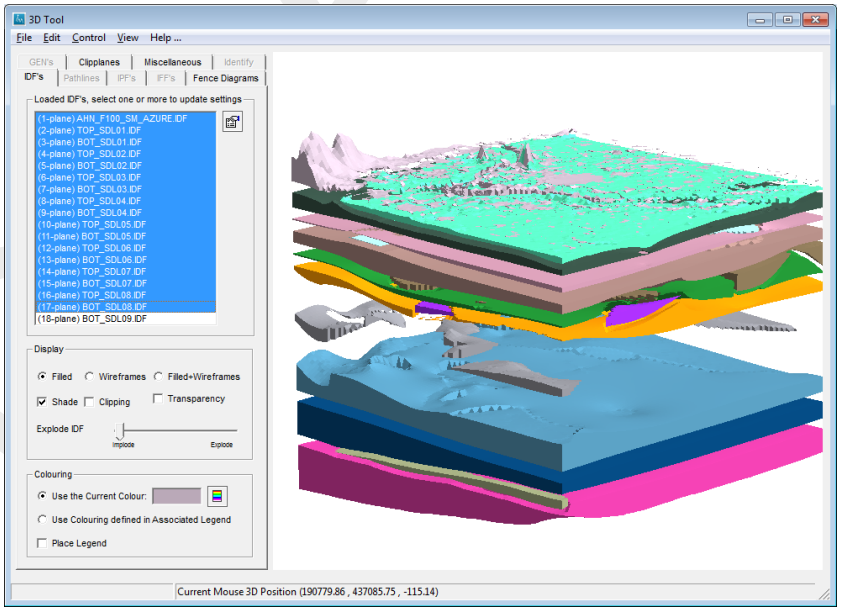


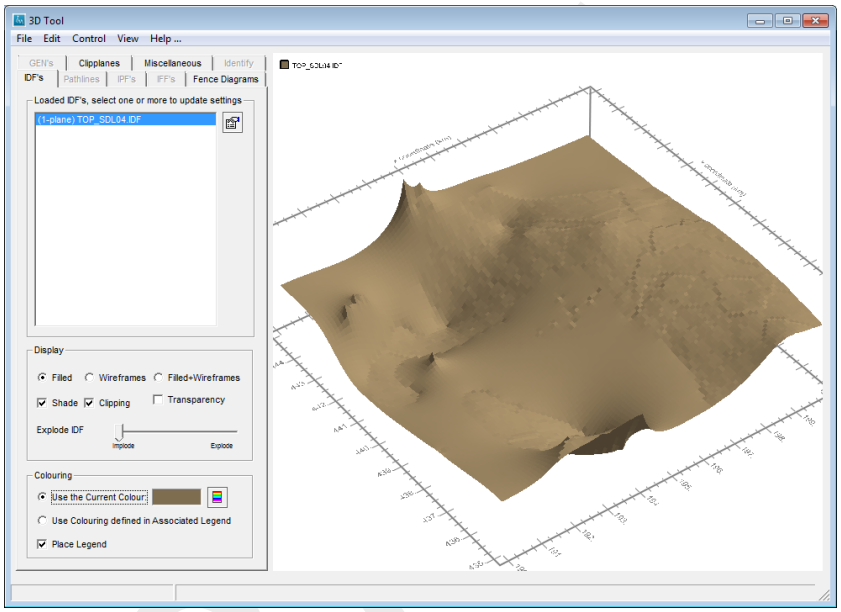
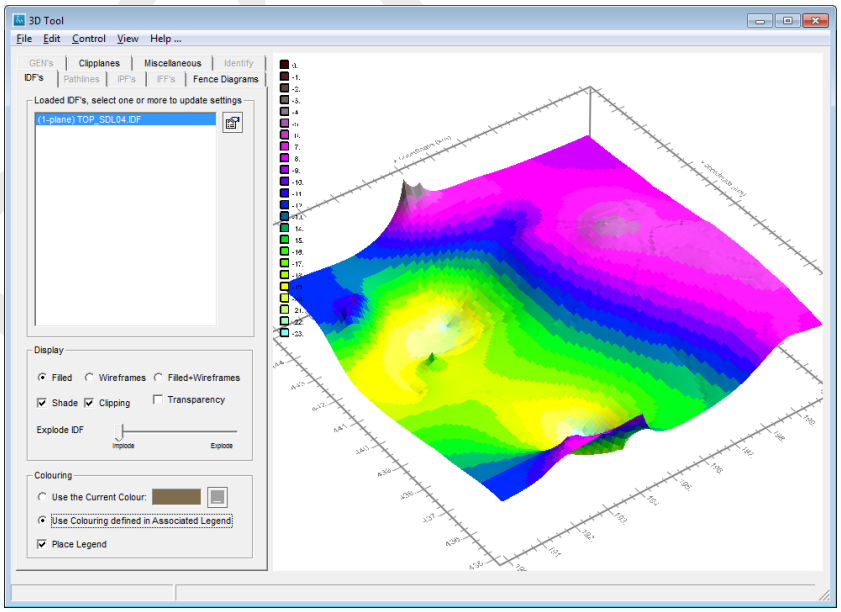
Explode IDF

Use the 'Explode IDF'-slider to increase (explode) or decrease (implode) additional vertical distance between the selected individual IDF's. Each IDF file can be configured to explode with its own offset, simple set the slide for the selected IDF files separately.

Example of exploding all IDF files equally (top figure) or explode some IDF files more than others (bottom figure).

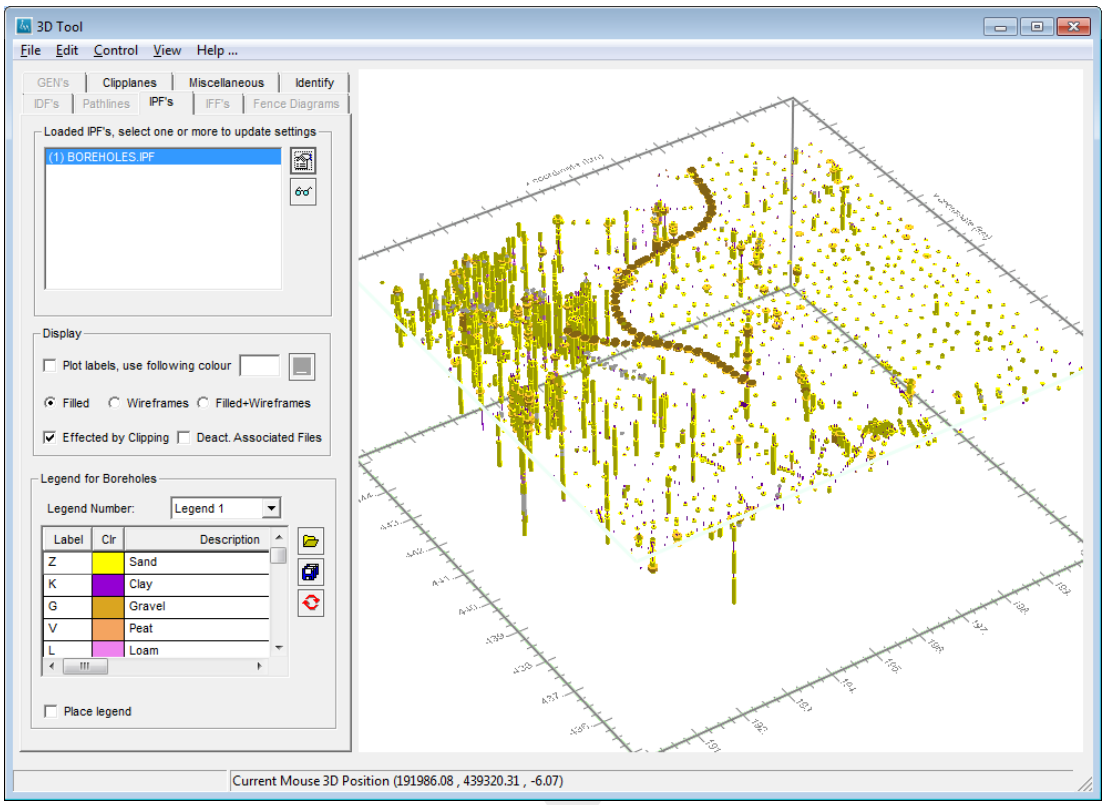




Use the current colour	Select this option to display the selected IDF-file(s) by a single colour as displayed to the right of this option.
	*Legend* Click this button to start the default Colour window to select a colour for the first of the selected IDF-file(s).
Use colouring defined in Associated Legend	Select this option to colour all selected IDF-files by their associated legend definitions, i.e. the colour legend they have. *Example of single colouring (upper figure) and legend colouring (lower figure)*  
Place Legend	Check the box to show the legend. This can be set for each IDF file individually.

7.3.4 3D Tool: the IPF-settings tab

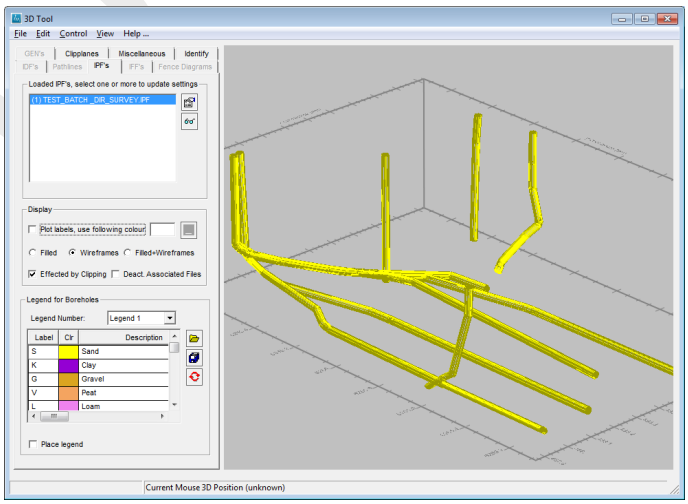
The *IPF's* tab of the *3D Plot* window allows configuring the display of IPF files.



Loaded IPF's, select one of more to update settings

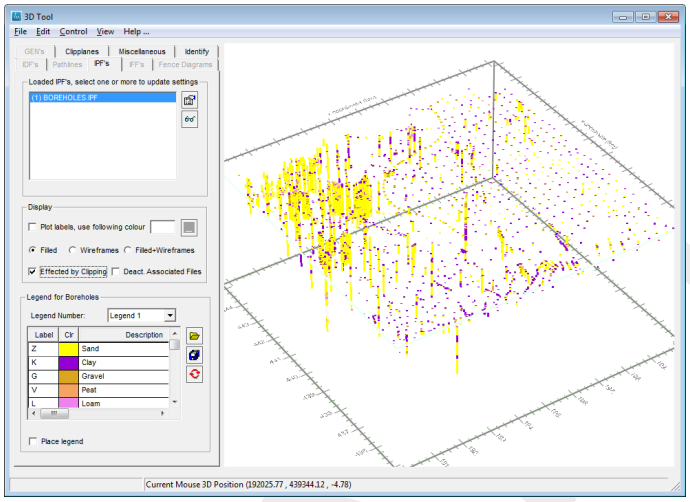
Select one or more IPF files from the list to activate them in the 3D visualization. The 3D display will update each time an IPF is (de)selected. It depends on the configuration of the IPF(s), how the points (and associated files) will be displayed.

Example of an IPF file showing deviated wells.



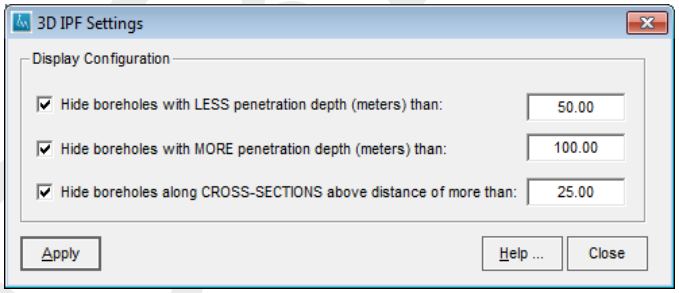
Properties
Click this button to define labels to be plotted. The presentation of boreholes can configured in this window, see section section 6.8.2 for more details. When less than 1000 points are read iMOD will display boreholes by *fancy* tubes, when more are loaded the boreholes are represented as single (vertical) lines instead.

Example of a simple boreholes representation:



Select
Click this button to change the selection of boreholes using their length or distance to a cross-section, it will open the 3D IPF Settings window:

3D IPF Settings window:



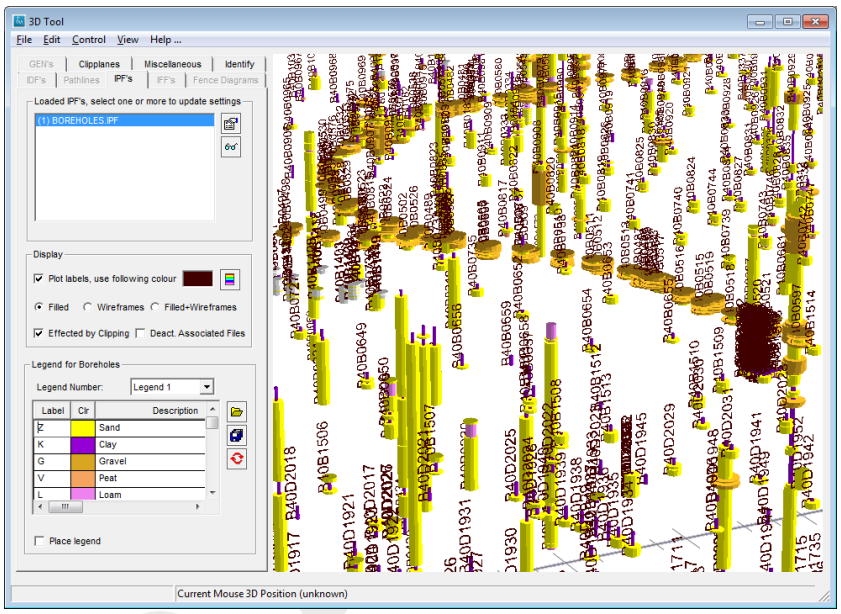
Hide boreholes with LESS penetration depth (meters) than:
Enter the maximal depth for which boreholes will be hidden, e.g. a value of 50 will hide all boreholes with less than 50 meter depth.

Hide boreholes with MORE penetration depth (meters) than:
Enter the minimal depth for which boreholes will be hidden, e.g. a value of 100 will hide all boreholes with more than 100 meter depth.

Hide boreholes with larger distance to CROSS-SECTIONS than:
Enter the distance perpendicular to any selected cross-sections/fence diagram (see 'Fade, view depth' in the 'Misc.' tab of the 'Cross-Section Properties' window in section section 7.1.1) for boreholes to be hidden, e.g. a value of 25 will hide boreholes on both sides of a selected cross-section/fence diagram with more than 25 meter distance.

Apply
Select this button to apply the entered display configuration.

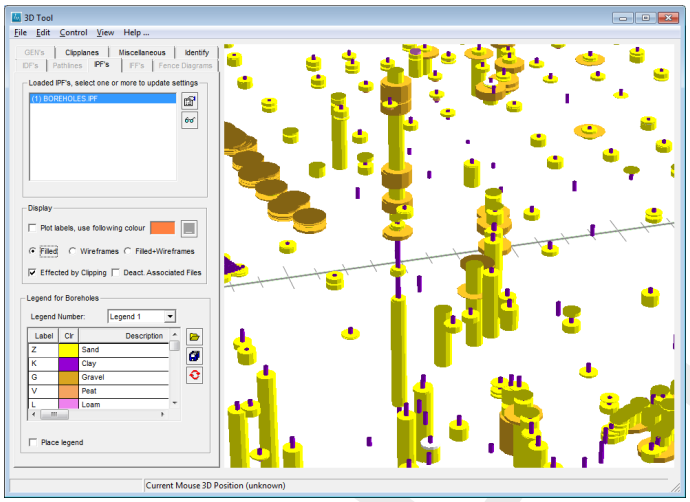
Help . . .
Select this button to start the Help functionality.

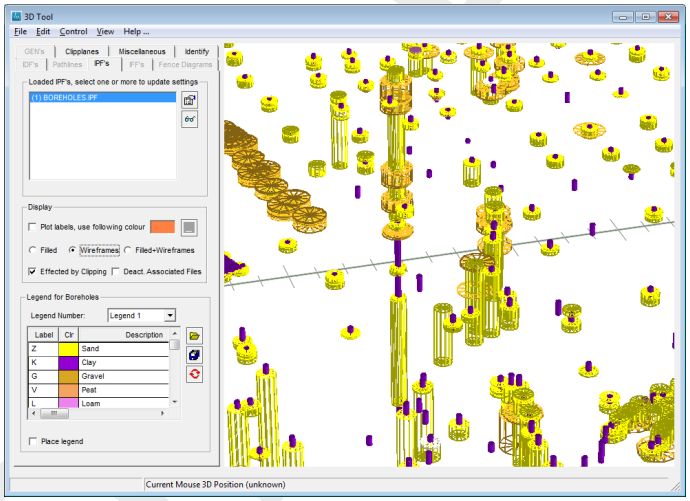
	Close Select this button to close the *3D IPF Settings* window without modifying existing entries.
Plot labels, use following colour	Check this option to display labels as defined by the options entered via *Properties*. After the option is selected it is possible to enter a different label colour (default is white) via the button  . This will start the default Colour window in which a colour can be picked for the selected IPF file(s). *Example of labelling a borehole* 

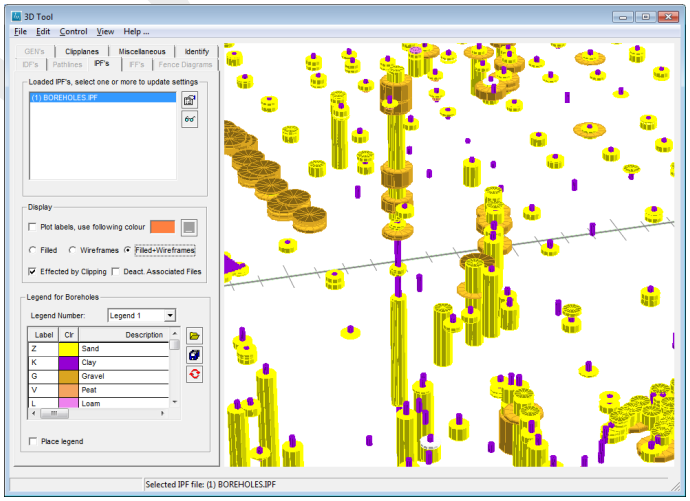
Filled, Wireframe, Filled+Wireframe

Select of the options to display the boreholes of the selected IPF files.

Example of Filled (top figure), Wireframes (middle figure) and Filled+Wireframe (lower figure) configurations:



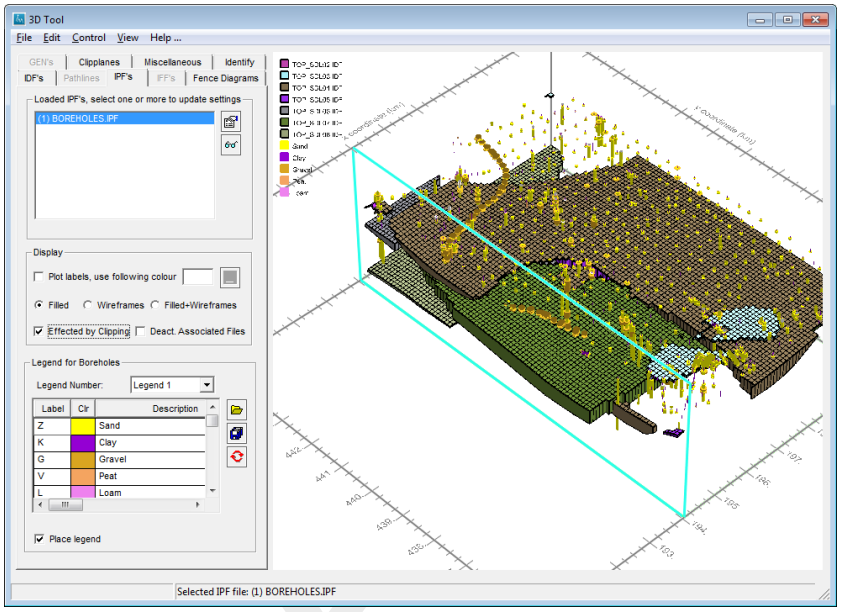


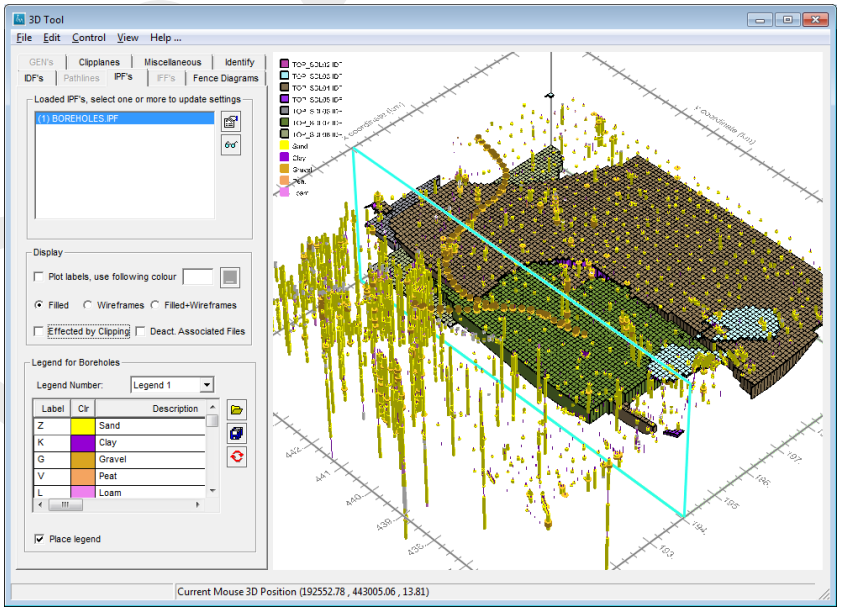


Effected by clipping

Select this option to activate clipping for each of the IPF file(s) separately, by default all IPF files are effected by clipping. If all IPF files are selected, any modification of this checkbox effect all, but it is possible to (de)select this option per IPF. This option is handy whenever a surface model needs to be clipped away and the boreholes should not.

Example of clipping a surface model (top figure) and deactivate clipping (lower figure) for boreholes

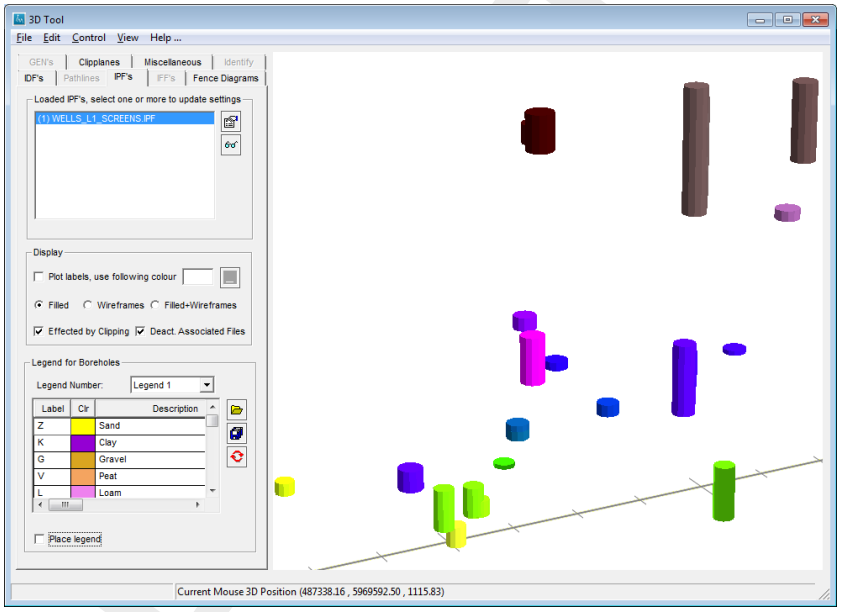


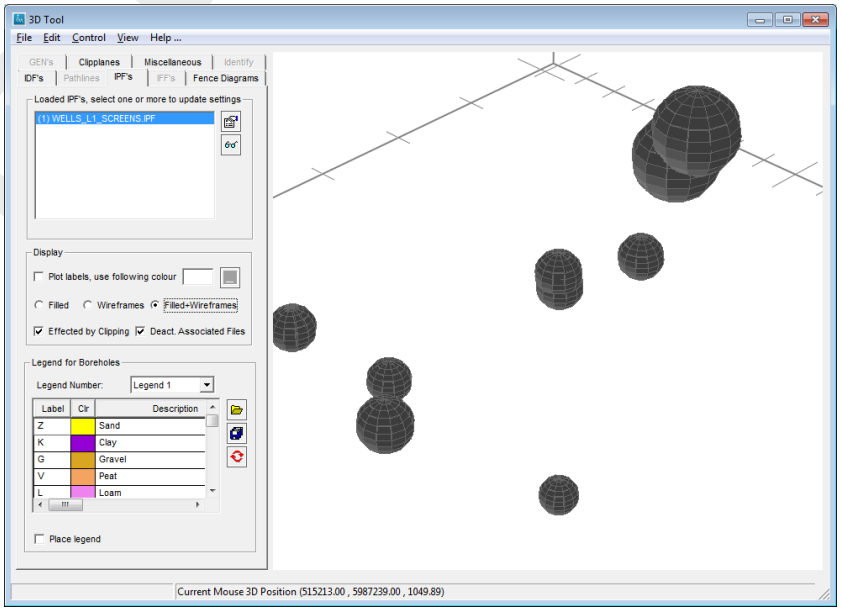


Deact. Associated Files

Click this checkbox to ignore the associated files that are attached to the selected IPF file. This checkbox will be greyed out whenever no associated files are available and/or automatically deselected for associated files that represent time series as they can not be displayed in a 3D environment. For IPF files with boreholes this checkbox will deselected by default. When this option is selected, iMOD will use the columns from the IPF file that are assigned to the *Z-coordinate* and if applicable the *Sec. Z-Crd.*, see section section 6.8.1. In this way iMOD will create spheres (only *Z-coordinate* is assigned) or vertical tubes when both attributes are assigned. It is also possible to colour the tubes/spheres according to the attribute *Apply Colouring To* that is assigned for colouring.

Example of a tube (top figure) and sphere (bottom figure) representation of an IPF file with deactivated/no associated txt files.

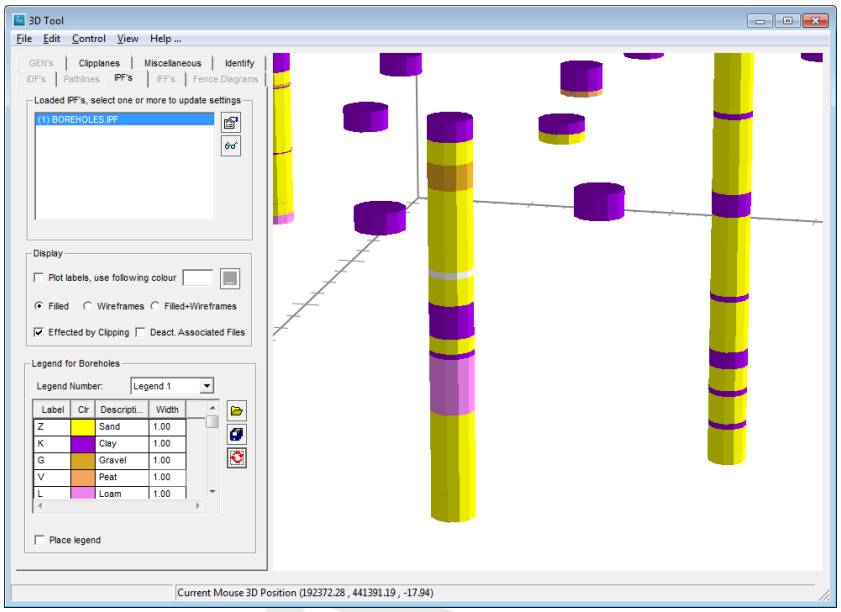


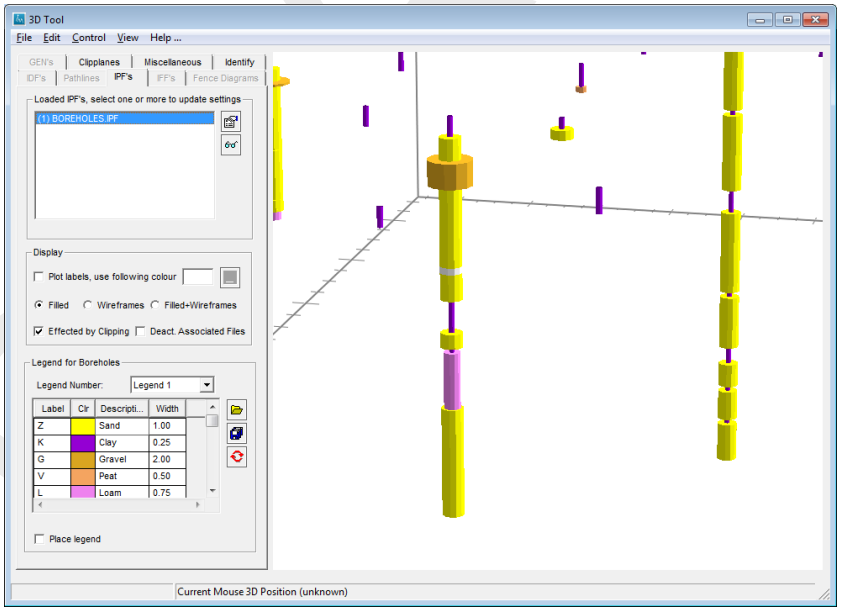


Legend Number:	Select one of the ten predefined legends from the drop down menu. Each time another item is selected, the current legend from the table below will be saved in that predefined legend number. In this way a multiply set of legends can be managed efficiently.
Legend for boreholes	Table of the legend used for colouring the boreholes associated to the selected IPF file(s).
	Label The first column will be used to represent the label that need to be matched (case insensitive) with a particular column in the associated files (*Cylinder Class Column*, see section section 6.8.2). This is the number of the column in the associated files that need to be used for the size of the cylinders plotted for the boreholes.
	Clr The second column will be used to represent the colour that will be used for those items that match the entered label in the column *Label*.
	Description The third column is for descriptive purposes only. This will be the actual text that will be plotted in the legend on the 3D window.

Width:
The fourth column can be used to specify a particular width to the individual parts of the borehole.

Example of Borehole with constant width (top figure) and variable width (bottom figure)

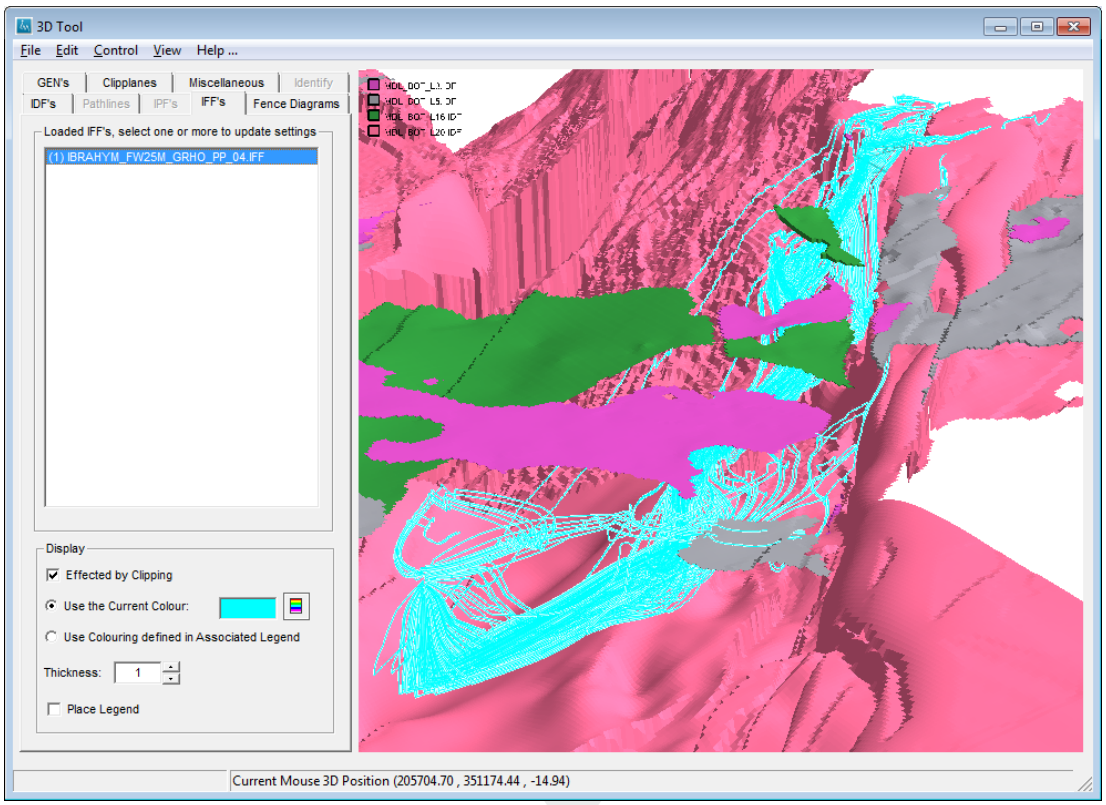


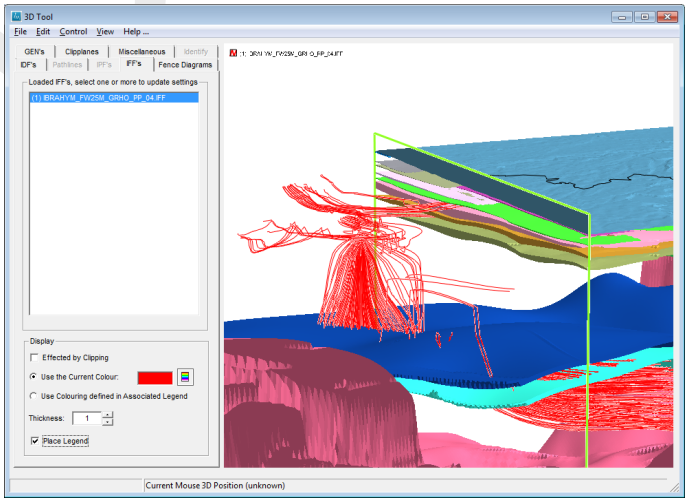


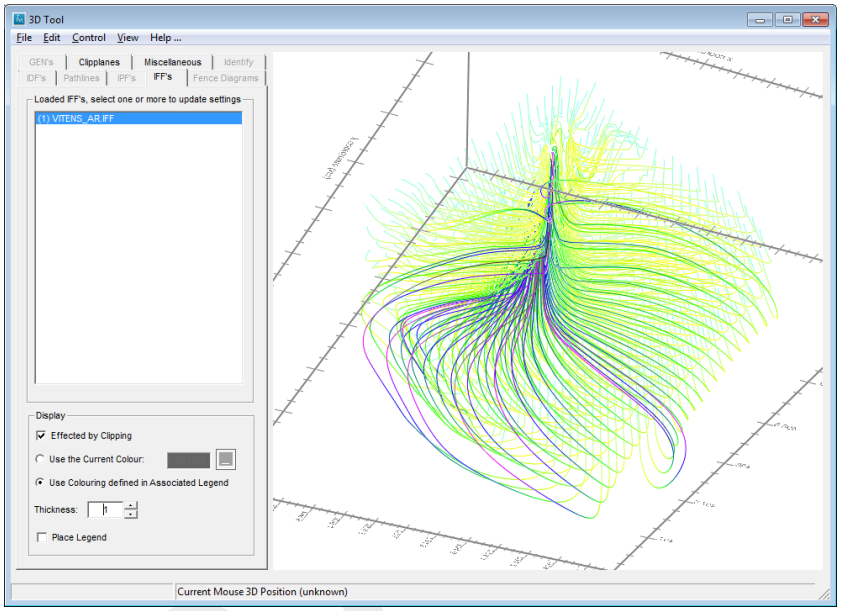
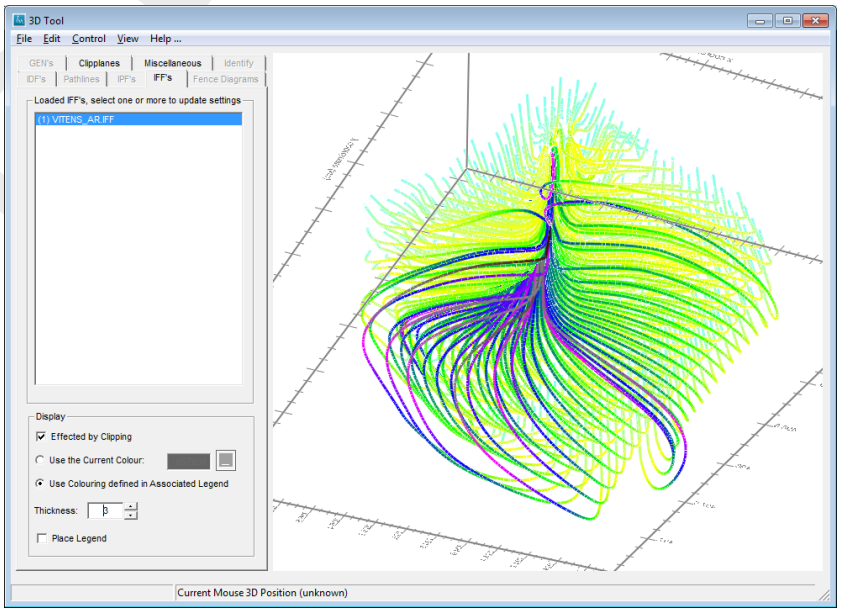
	Open DLF-file Select this button to open a DLF file (see section section 9.17) that will be used to colour the boreholes associated to the IPF file.
	*Save As DLF-file* Select this button to save the current legend to a DLF file (see section section 9.17).
	*Redraw* Select this button to redraw the IPF file with the adjusted legend specified in the table. Any adjustment in legend colour and/or width will be applied.
Place legend	Select the check box to show the legend on the graphical canvas. This will affect the selected IPF files only.

7.3.5 3D Tool: the IFF-settings tab

The *IFF's* tab of the *3D Plot* window allows configuring the display of IFF files.

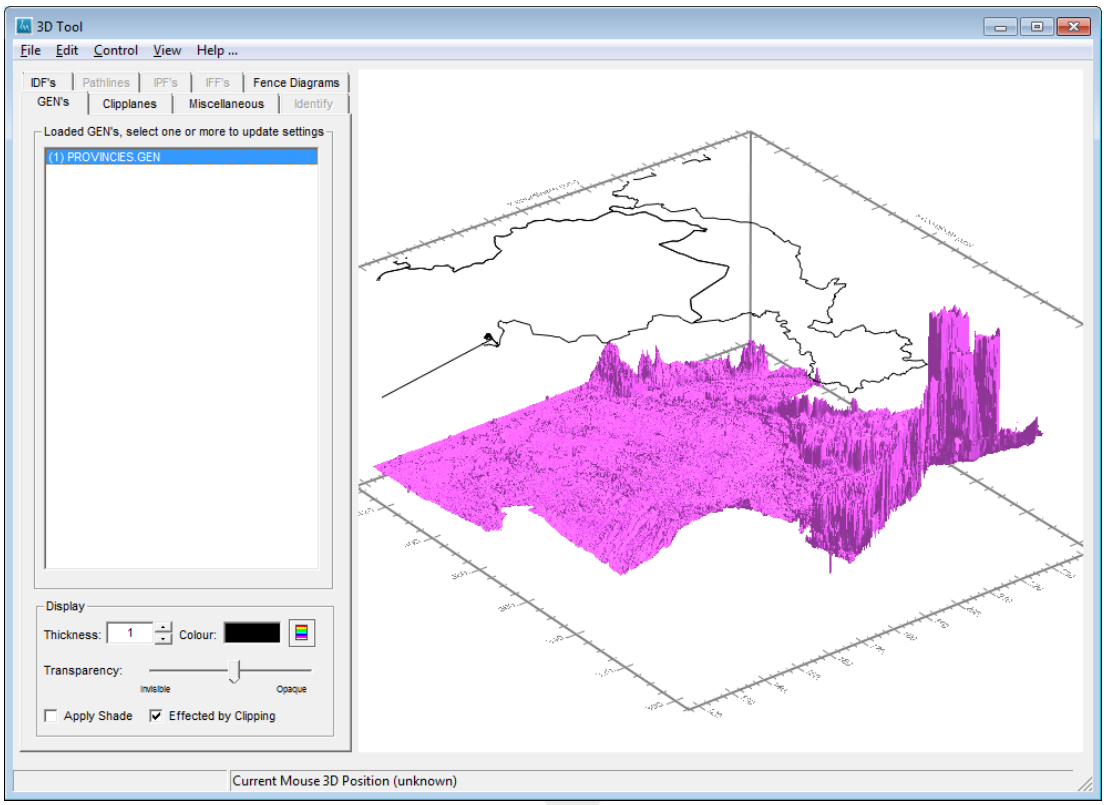


Loaded IFF's, select one or more to update settings	Select one or more IFF files from the list to activate them in the 3D visualization. The 3D display will update each time an IFF is (de)selected.
Effected by Clipping	Select this option to effect the IFF file for clipping, this is the default. It can be illustrative to ignore clipping for an IFF in combination with a subsurface model that will be clipped. *Example of a subsurface clip and non clipping an IFF file.* 

Use the current colour	Click this option to display the selected IFF file by a single colour as displayed to the right of the option.
	*Legend* Click this button to start the default Colour window to select a colour for the first of the selected IFF file(s).
Use Colouring defined in Associated Legend	Select this option to present the all selected IFF files by their associated legend definition. *Example of an IFF file coloured by age.* 
Thickness	Enter the thickness of the lines used to display the flowlines. The higher the number, the thicker the line appears on the graphical display. *Example of an IFF file coloured by age with a line thickness of 3.* 
Place legend	Check the box to show the legend.

7.3.6 3D Tool: the GEN-settings tab

Use the GEN Settings tab to configure the display of GEN files.



Loaded GENs, select one of more to update settings

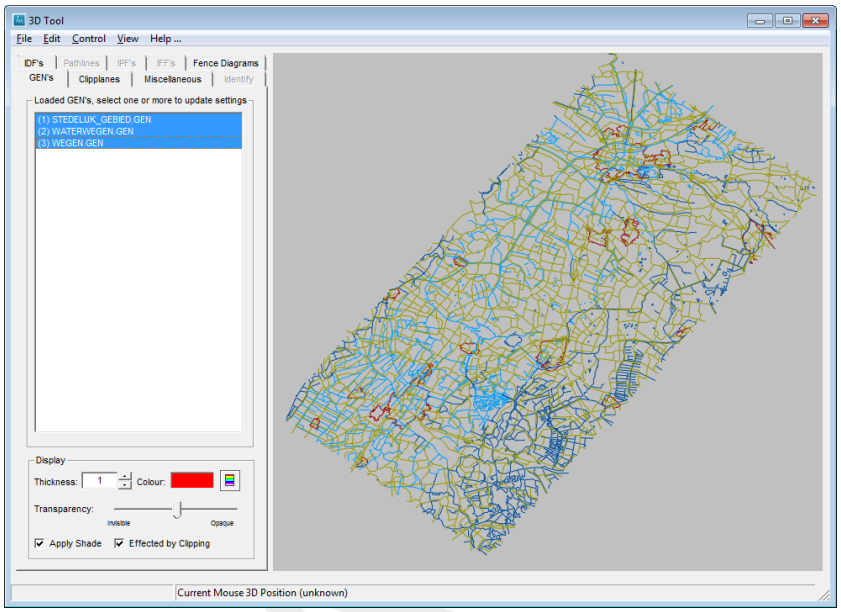
Select one or more GEN-files from the list to activate them in the 3D visualization. The 3D display will update each time a GEN is (de)selected. GEN-files can be loaded in the **Maps tab** or the **Overlays tab** in the *iMOD Manager*; the behaviour is different and illustrated below.

Overlay tab
This is the most common way to load GEN files into iMOD. Each selected file from the *Map* tab on the *iMOD Manager* will be presented as a separate entry for which all setting described below can be set. There are two options for a GEN file (see section section 9.10 for more detailed information):

2D GENs

These are the most common files and used for topographical information, such as e.g. road and borders.

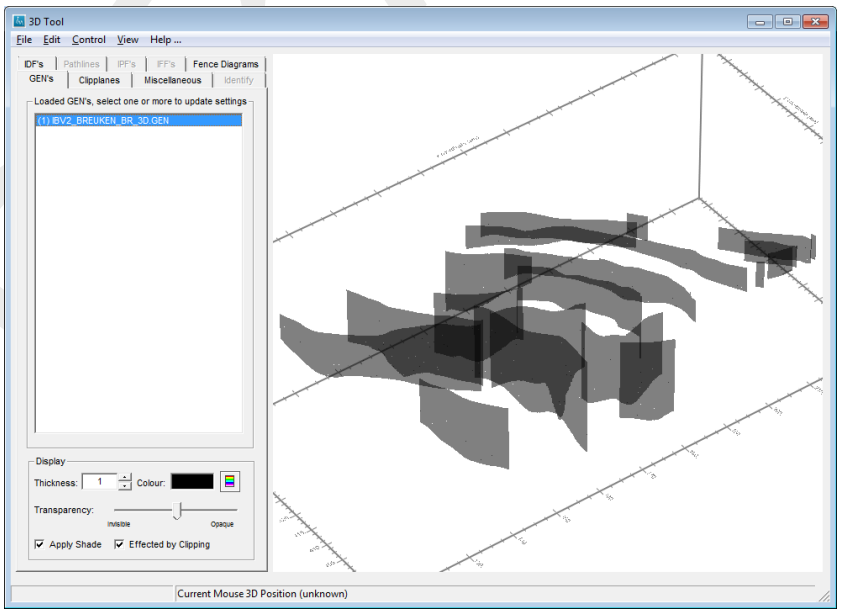
Example of a display of 2D GEN showing, rivers (blue), urban area (red) and roads (green).



3D GENs

These can be used to display 3D spatial information, such as faults in 3D.

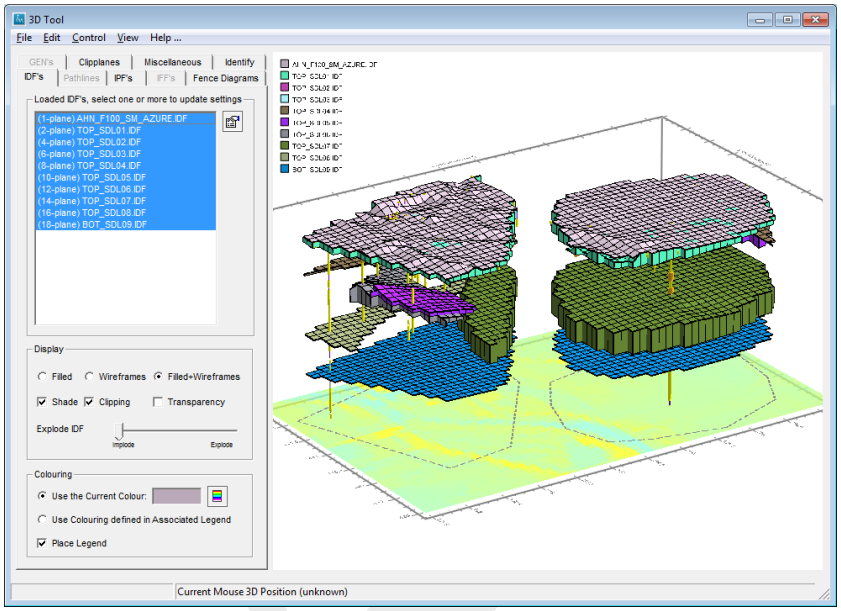
Example of a display of 3D GEN showing a fault outline.



Map tab

Whenever a GEN file is selected in the *Map* tab on the *iMOD Manager* window, the GEN files acts as a cookie-cutter. In this way it is possible to cut-out e.g. solids using irregular shaped polygons and display these in 3D.

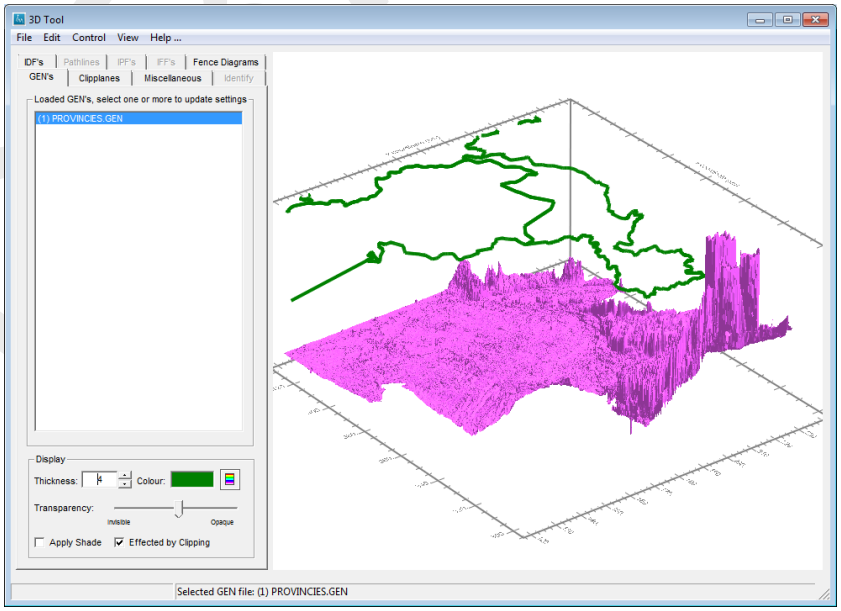
Example of an irregular shaped GEN file acting as a cookie-cutter in the 3D.



Thickness

Enter the thickness of the lines used to display the GEN-files.

Example showing the effect of a thick green line for a GEN file.



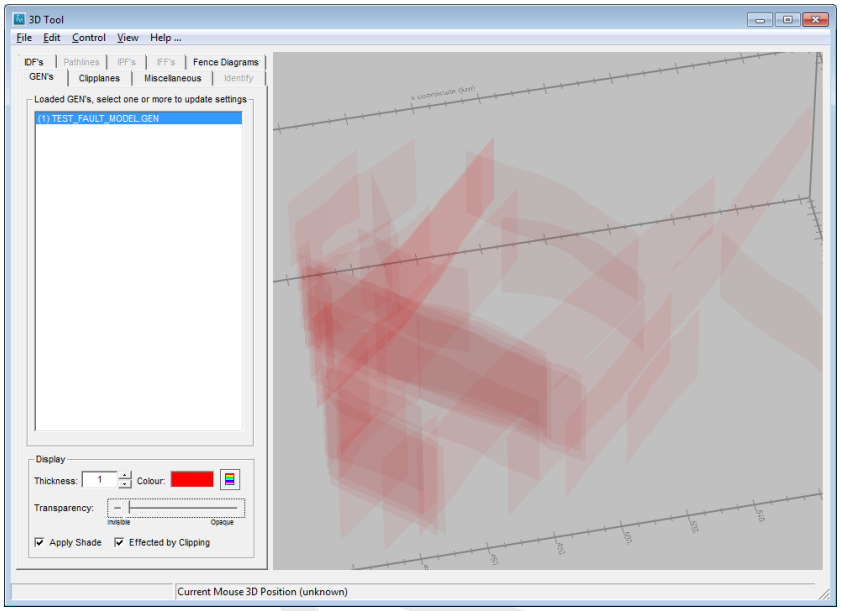
Legend

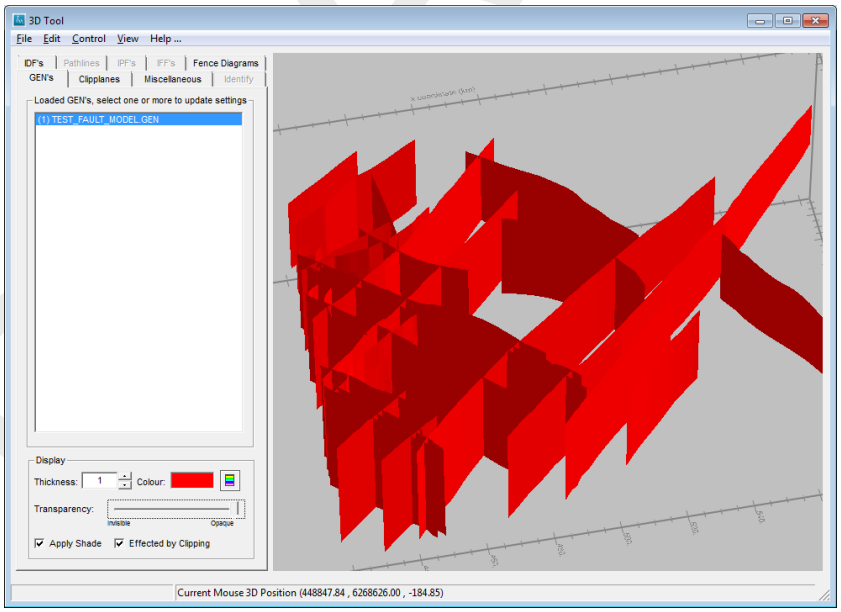
Click this button to start the default Colour window to select a colour for the selected GEN-file(s).

Transparency

Move the 'Transparency-slider' to change the transparency within the range 'Opaque' (not transparent) and 'Invisible' (full transparent). This is especially nice for 3D GEN file, see section section 9.10.

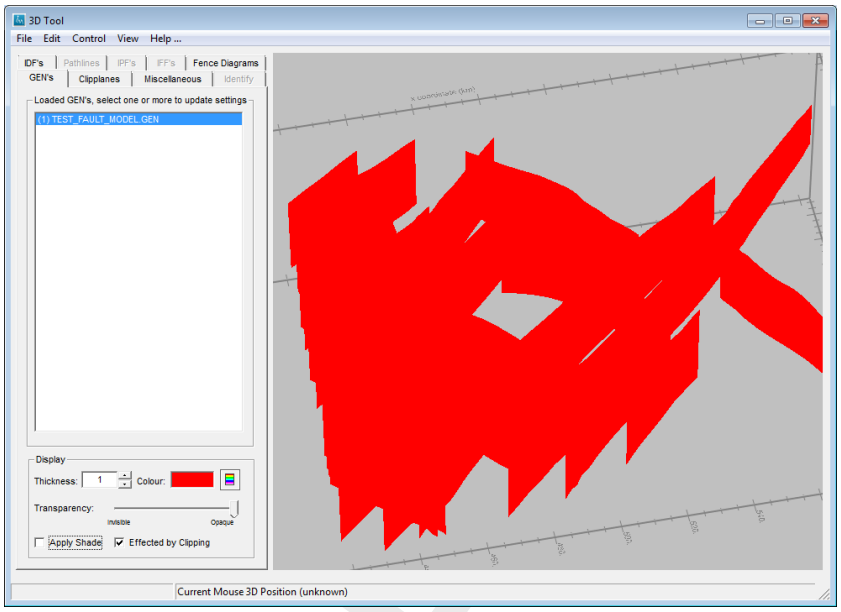
Example of a transparent 3D GEN (top figure) and an opaque display (lower figure).



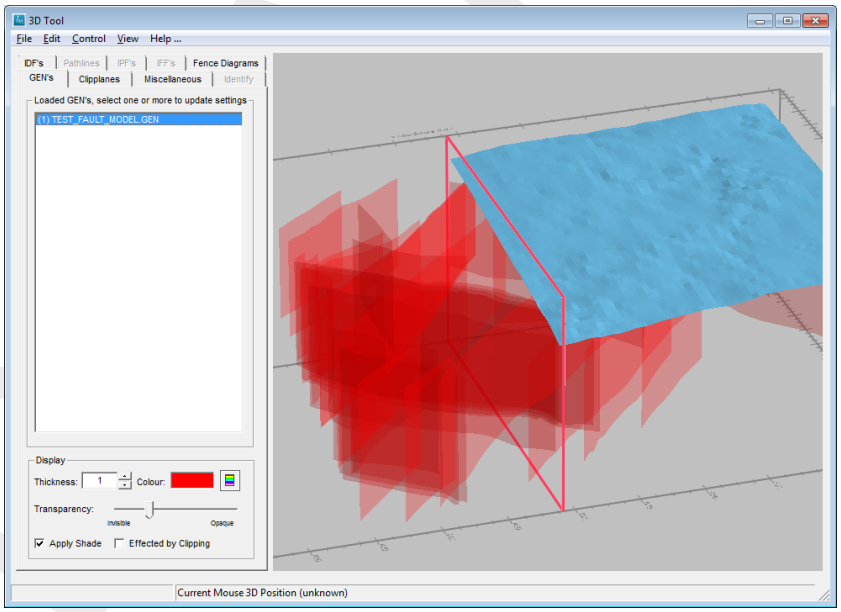


Apply Shade

Example of showing a 3D GEN without shade.

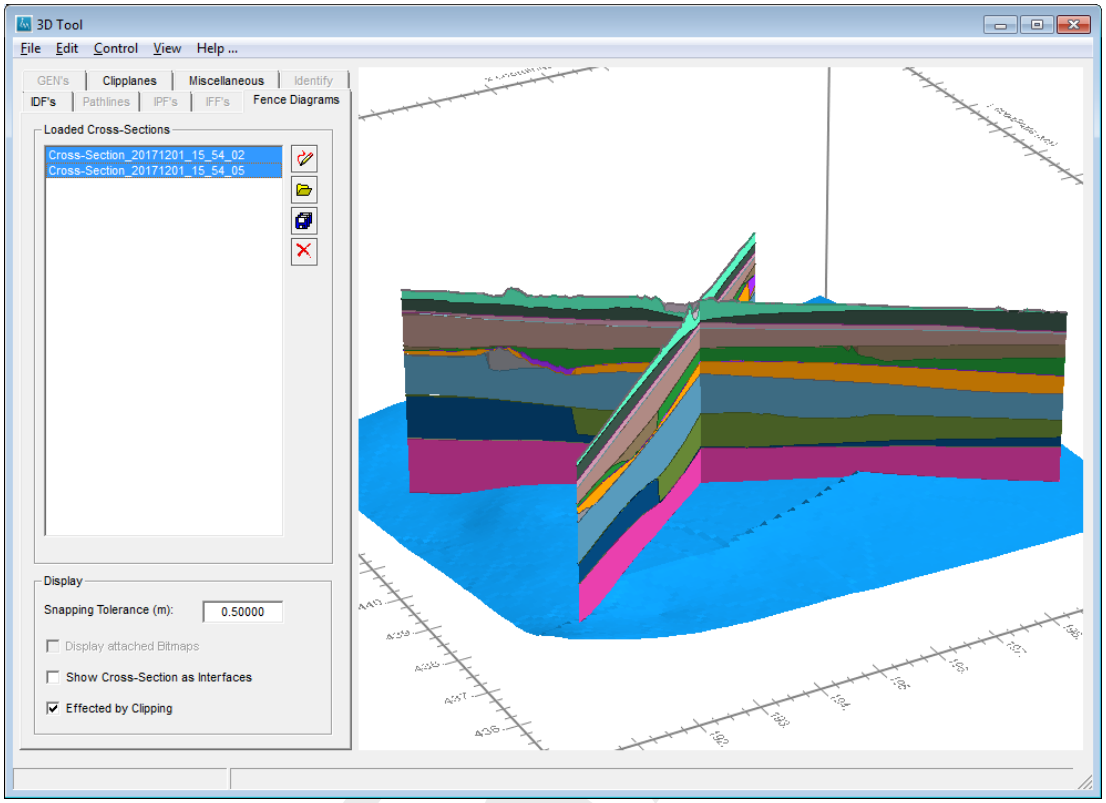


Effected by Clipping



7.3.7 3D Tool: the Fence Diagrams-tab

The functionality of this tab *Fence Diagram* is twofold. It will be used to display cross-sections as fence diagrams from the SOLID Tool (see section section 7.4.3 and secondly whenever the regular 3D Tool is started, this tab allows to interactively draw the location of a fence-diagram, iMOD will convert this to a fence diagram and display it on the graphical display.

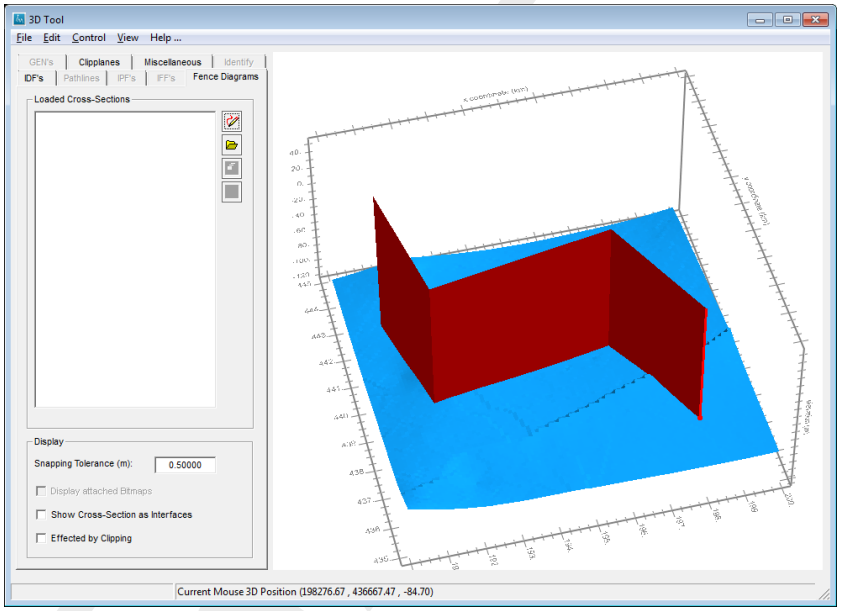


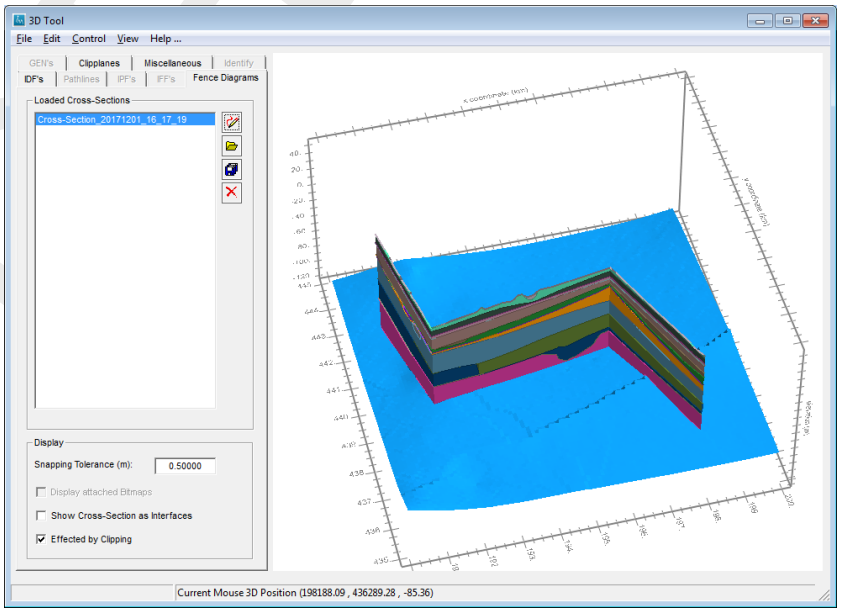
Loaded Cross-Sections	Select one or more cross-section files from the list to activate them in the 3D visualization. The 3D display will update each time a cross-section is (de)selected.

Draw

Select this button to start drawing a cross-section in the 3D graphical canvas. The line is started by clicking the left-mouse button and terminated with the right mouse button. After that, iMOD generates a fence-diagram and adds it as a separate entry in the menu list; the naming convention is {Cross-Section_yyyymmdd_hh_mm_ss}. Subsequent left-mouse clicking will insert additional nodes to the same cross-sections. Be aware that it is necessary to select at least one IDF file, as it is only possible to draw a cross-section over an IDF that is visible; do not use transparency!

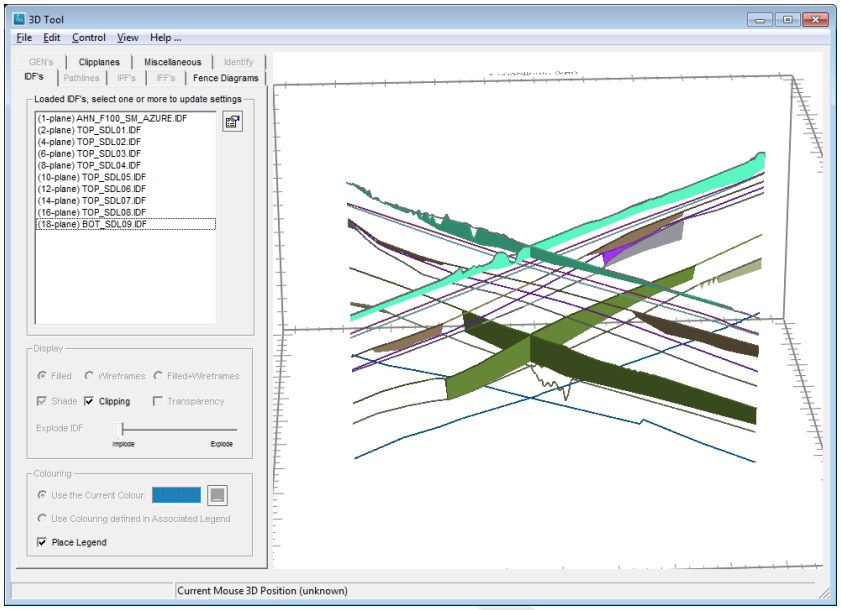
Example of a cross-sections drawn on lower most IDF file (top figure) and the yielding fence diagram (bottom figure).

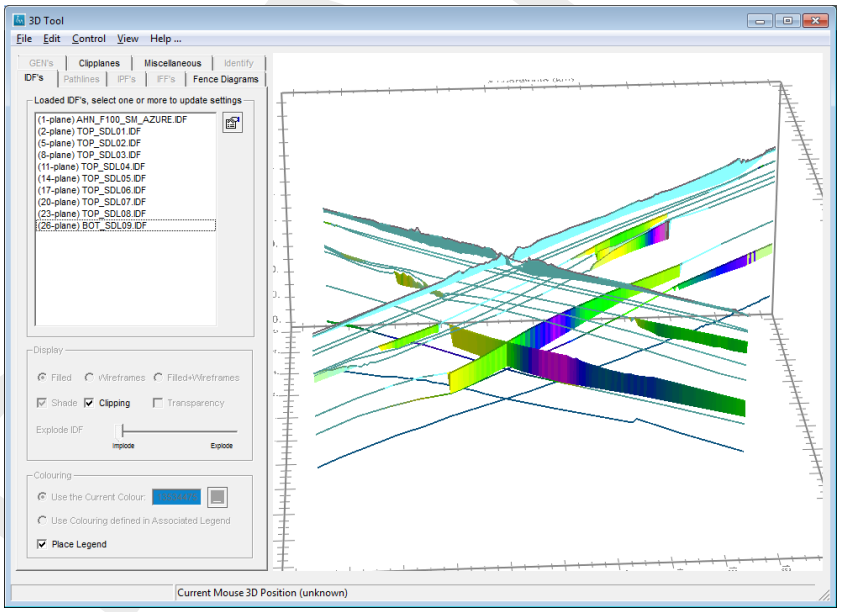




Note: It is important to note that the appearance of the fence-diagram depends on the configuration of the individual interfaces. So whenever you pick the *Quasi 3D Model (aquitard)* configuration, the fence-diagram appears as filling the aquitard only. Moreover, if you select the *Coloured Quasi 3D Model (aquitard)* configuration, the fence-diagram is filled in with the property that is selected.

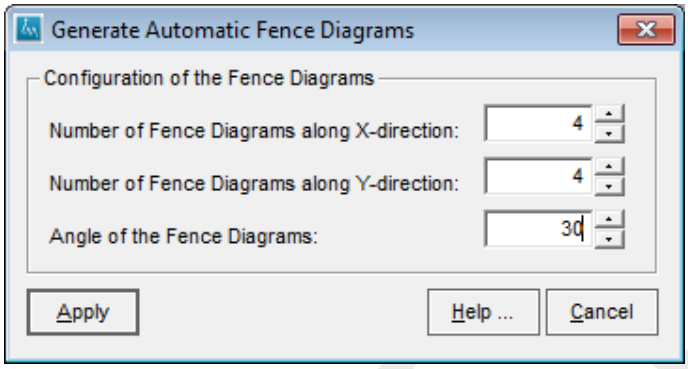
Example of a cross-sections drawn for aquitards only (top figure) or filled in by the selected property (bottom figure).





Automatic Fence Diagrams
Click this button to open the *Generate Automatic Fence Diagrams* window. With this window it is possible to generate a regular pattern of fence diagrams.

Generate Automatic Fence Diagrams window.



Number of Fence Diagrams along X-direction
Enter the number of fence diagrams along the X-direction; when specifying 0 no fence diagrams will be generated in the X-direction.

Number of Fence Diagrams along Y-direction
Enter the number of fence diagrams along the Y-direction, when specifying 0 no fence diagrams will be generated in the Y-direction.

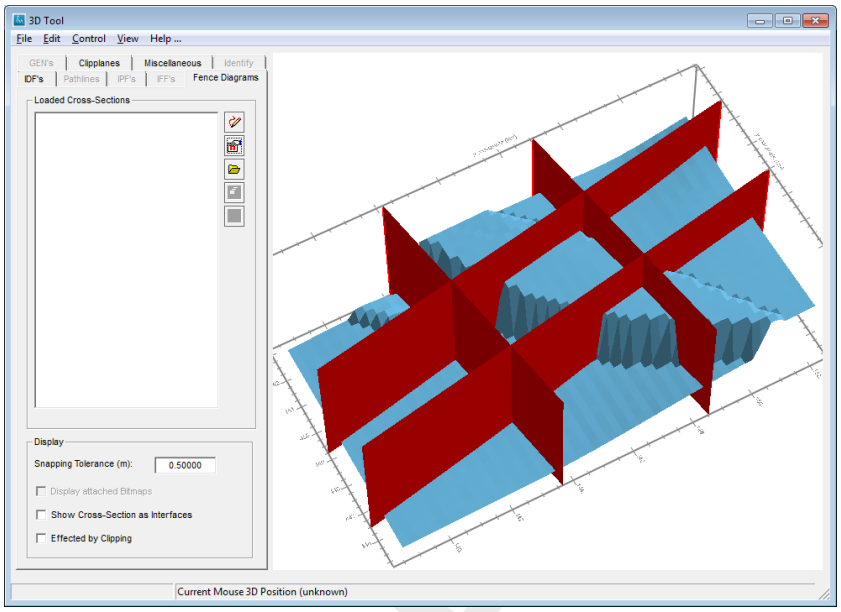
Angle of the Fence Diagrams
Enter the angle for which all fence diagram need to be rotated, specify 0.0 to avoid any rotation.

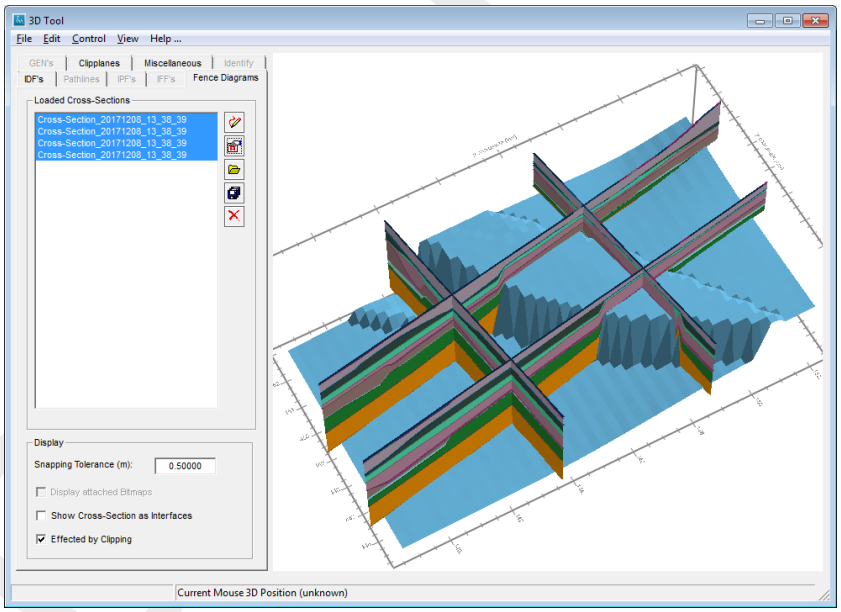
Apply
Click this button to generate the fence diagrams and close the *Generate Automatic Fence Diagrams* window.

Help . . .
Click this button to start the HELP functionality.

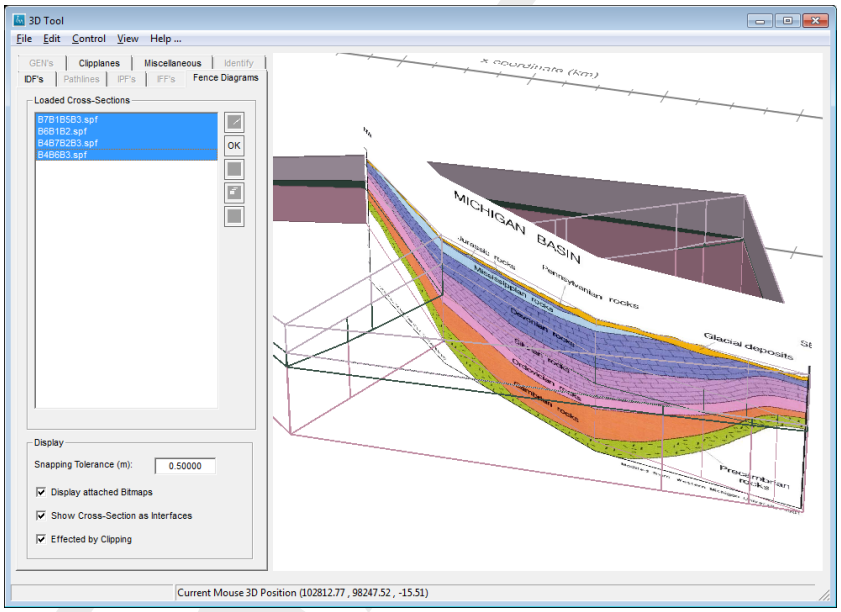
Cancel
Click this button to close the *Generate Automatic Fence Diagrams* window without adding any fence diagrams.

Example of configuring 2 x 2 fence diagram rotated 10 degrees (top figure) and the yielding fence diagrams (bottom figure).





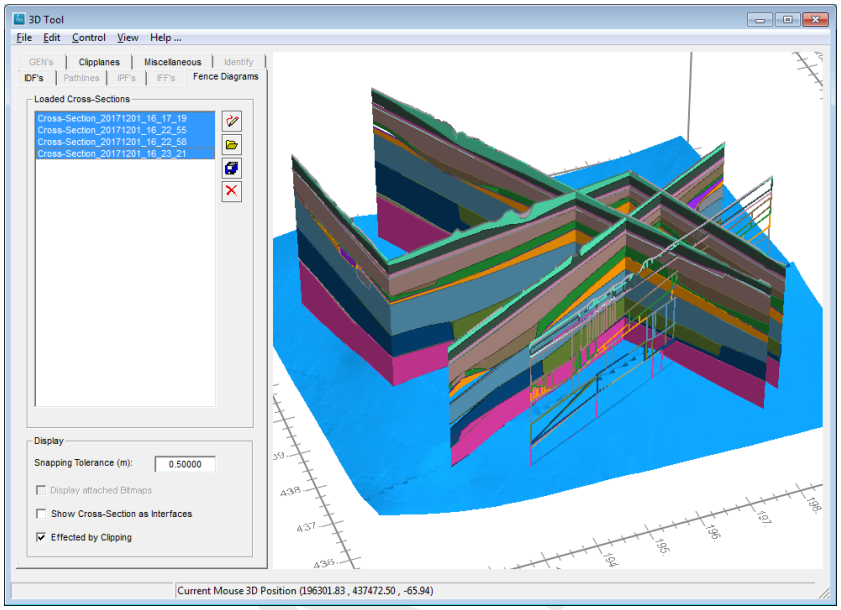
	*Open SPF File* Click this button to open an *.SPF file (see section 9.21). This file described the entire cross-sections per interface and can be used in the SOLID Tool (see section 7.4).
	*Save As SPF File* Click this button to save the selected cross-sections to an *.SPF-file (see section 9.21). This type of file can be used in the SOLID Tool (see section 7.4). The SPF file is saved with the name of the cross-sections in the iMOD TMP folder at the user folder associated to the PRF-keyword USER (see section section 9.1), e.g. IMOD_USER \TMP *.SPF.
	*Delete* Click this button to delete the selected cross-section(s) from the list. This action can not be undone, however, you will be asked first whether you are sure to delete the cross-section(s).

Snapping Tolerance	Enter a value for the snapping tolerance, the smaller the more accurate the computed fence-diagram becomes. This value denotes that acceptable error between the estimated fence-diagram and the true IDF value at that location. The higher the snapping tolerance, the more this is allowed to differ but, generates a fence-diagram with less points.
Display Attached Bitmaps	Check this box to display the attached bitmap on the cross-section. Those bitmaps need to be added to the cross-section by the Profile Tool (section 7.1) that has been started by the Solid Tool (section 7.4) and need to be available in the SPF file (section 9.21). *Example showing an image of a geological profile attached on the cross-section.* 

Show Cross-Section as Interfaces

Select this checkbox to show the cross-sections a interfaces only, so the individual polygons will not be filled in.

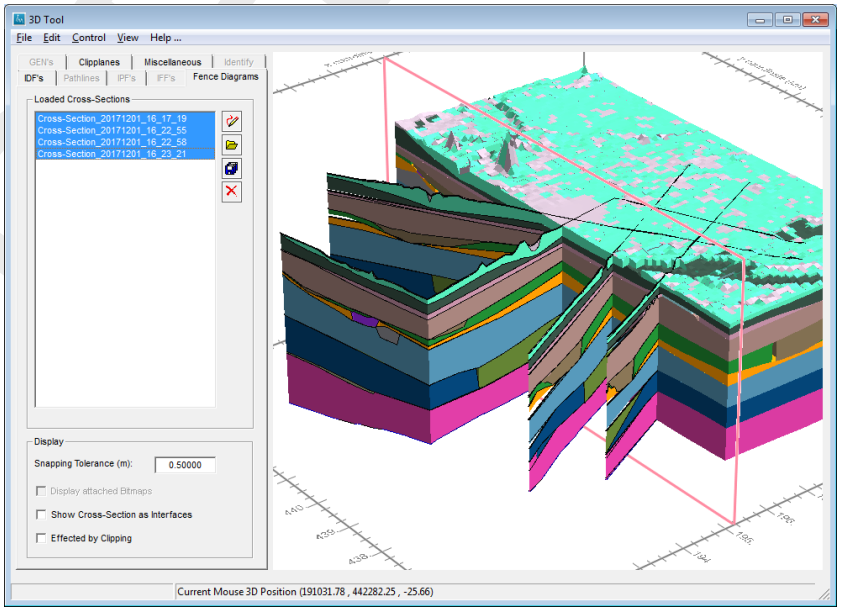
Example of multiple cross-sections for which some of them are drawn as an interface.



Effected by Clipping

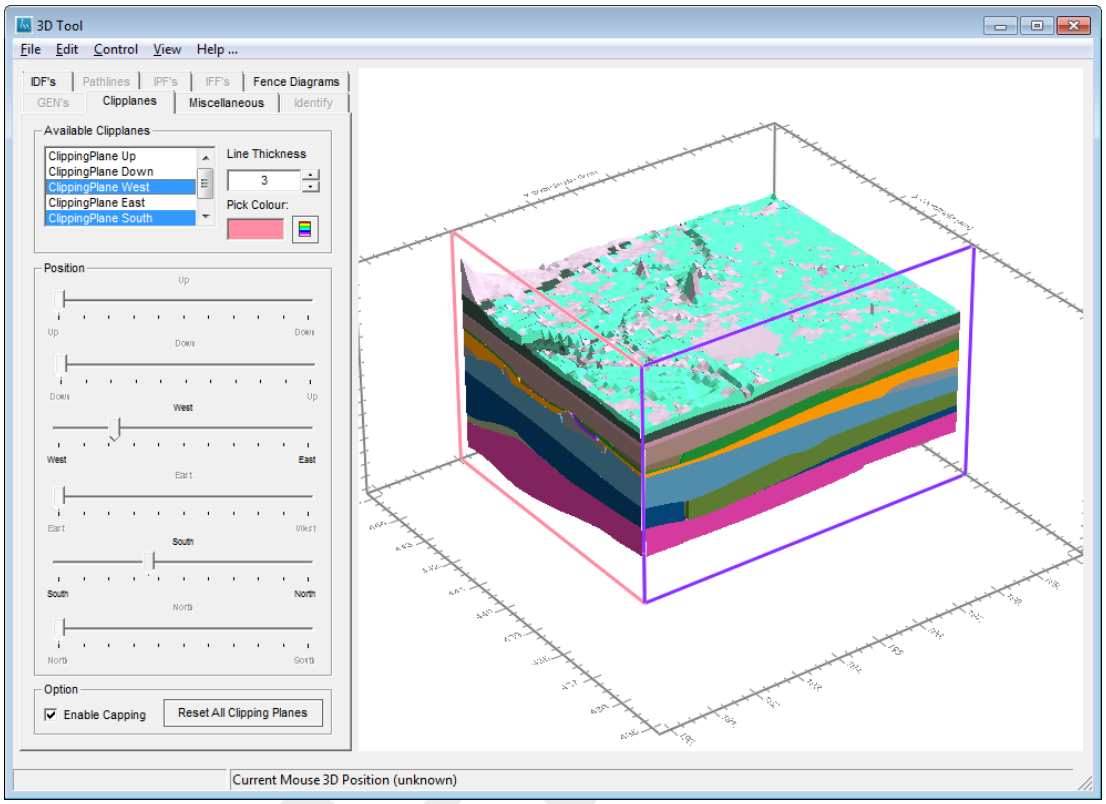
Select this checkbox to effect clipping of the cross-sections. You might select different choices per cross-section, which allows to clip some cross-sections, and leave others unclipped.

Example of multiple cross-sections which are not clipped in combination with a subsurface model that is clipped.



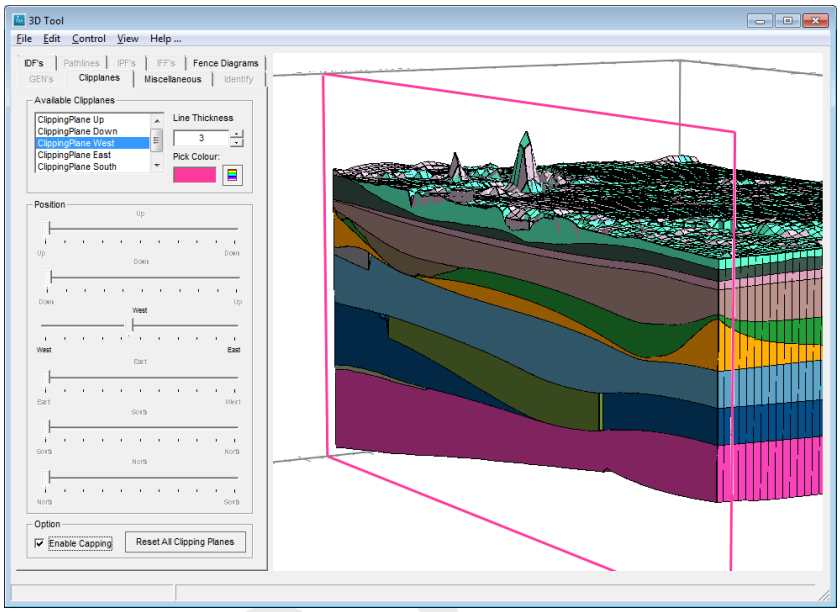
7.3.8 3D Tool: the Clipplanes-tab

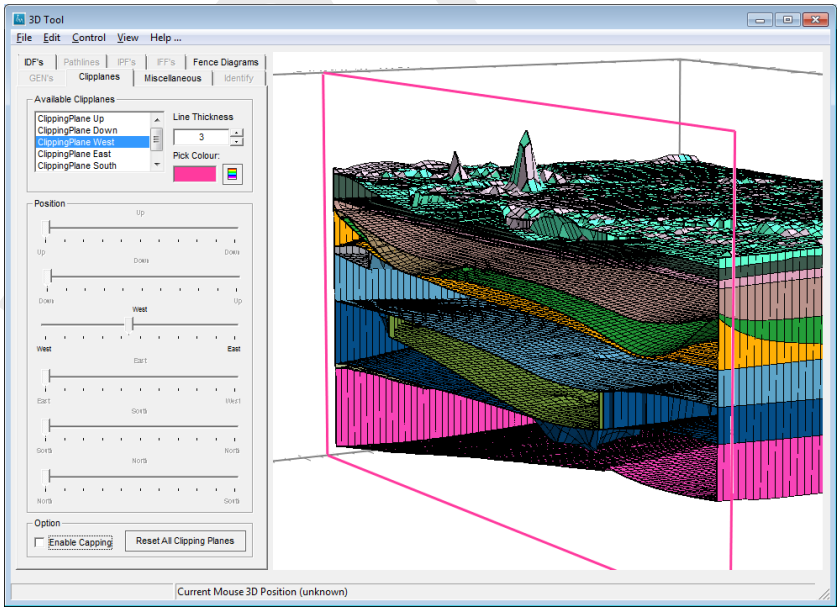
The 'Clipplanes-tab' of the 3D Tool allows configuring the clipping planes. With clipping planes it is possible to clip part of a 3D image and observe what is behind the clipping plane. In the *IDF*-tab (see section section 7.3.3), the *IPF*-tab (see section section 7.3.4), and the *IFF*-tab (see section section 7.3.5) it is possible to activate the clipping planes. It is possible to e.g. clip IDF files but leave IPF files intact.



Available clipplanes	Select one or more *clip planes* to activate: ◇ **ClippingPlane Up** Defines a clip plane from the top of the image and moves downwards; ◇ **ClippingPlane Down** Defines a clip plane from the bottom of the image and moves upwards; ◇ **ClippingPlane West** Defines a clip plane from the west of the image and moves to the east; ◇ **ClippingPlane East** Defines a clip plane from the east of the image and moves to the west; ◇ **ClippingPlane South** Defines a clip plane from the south of the image and moves to the north; ◇ **ClippingPlane North** Defines a clip plane from the north of the image and moves to the south. By selecting a particular clip plane, the corresponding slider becomes active.
Line Thickness	Enter a value to specify the thickness of the outline of the selected clip plane. Enter a value of zero to hide the outline.
	*Pick Colour* Use this button to start the default *Colour Picking* window to assign a particular colour to the outline of the selected clip plane(s).

Position	For each selected clip plane its corresponding slider becomes active. For example when the clip planes *ClippingPlane West* and *ClippingPlane South* are selected both the corresponding sliders are available and can be dragged to move the corresponding clipping plane.
Enable Capping	Capping is a technique of putting a solid colour on the interface of a 3D image that has been clipped away. *Example of cross-sections drawn with capping (top figure) and without capping (bottom figure).*

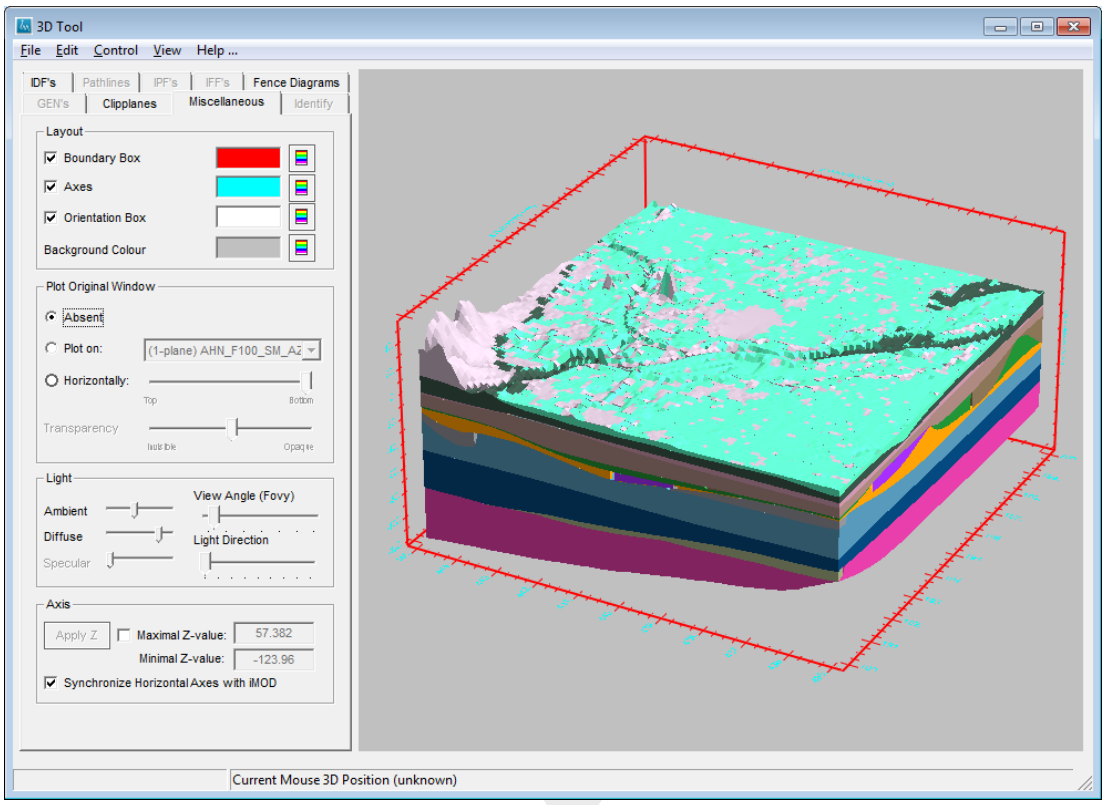




Reset All Clipping Planes	Click this button to reset all clipping planes to their default values. iMOD will ask the user to confirm before resetting all clipping planes.

7.3.9 3D Tool: the Miscellaneous-tab

The Miscellaneous tab provides several layout functions.

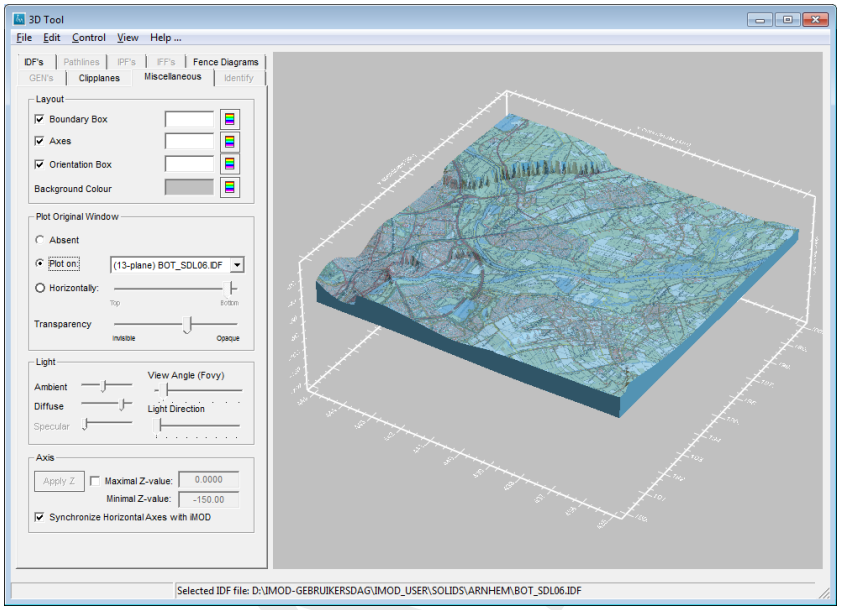


Boundary Box	Click this checkbox to turn the boundary box on or off. The colour of the Boundary Box can be changed by clicking the button (see section 2.6.4 for use of this *Colour Picking* window.).
Axes	Select this checkbox to turn on axes around the 3D image. The colour of the axes can be modified by selecting the button.
Orientation Box	Click this option to plot a simple orientation box with directions to North, East and West. The colour of this orientation box can be modified via the button.
Background Colour	The background colour can be modified by selecting the button. Sometimes, it gives an improved image if the background colour is not white.
Absent	Select this option to remove the image as currently displayed in the 2D graphical canvas.

Plot on

Select this option to drape the currently displayed image of the 2D graphical canvas on the selected IDF file from the drop-down menu.

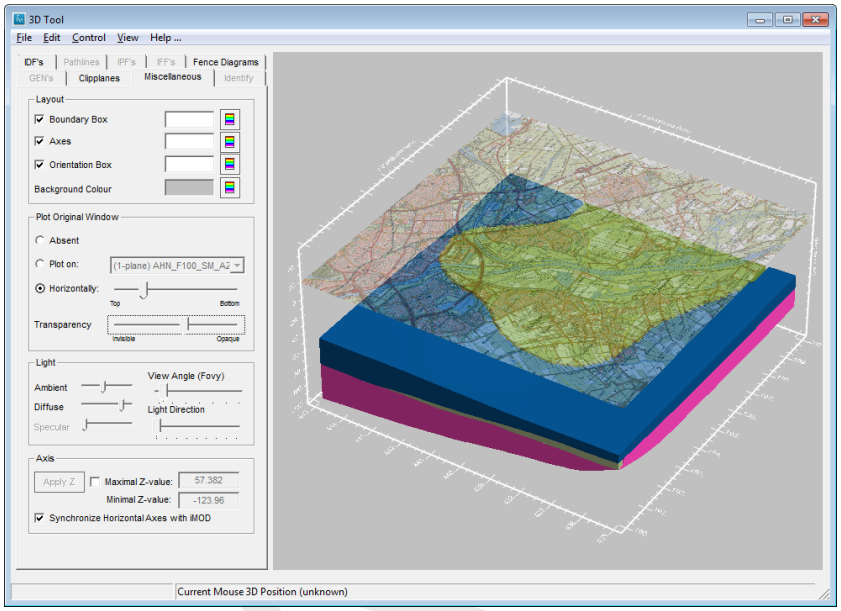
Example of draping the active image of the 2D graphical canvas on a selected IDF file.

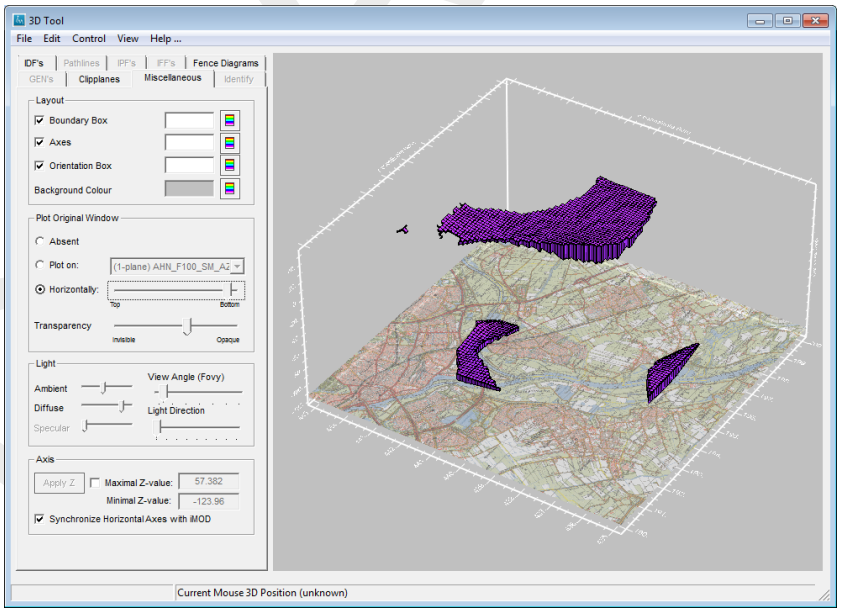


Horizontally

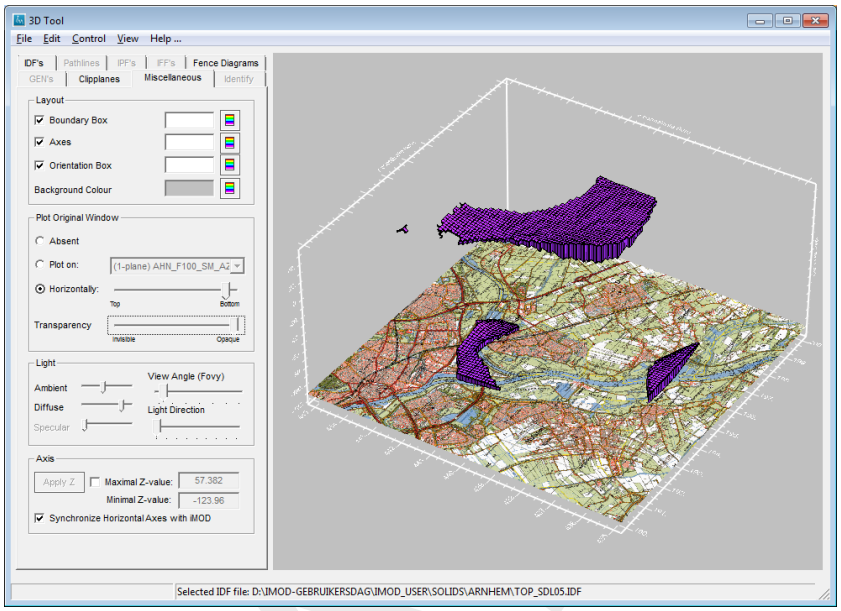
Select this option to display the active image of the 2D graphical canvas as a horizontal plane; use the slider (*Top* up to *Bottom*) to position the plane vertically between the top and bottom of the 3D image.

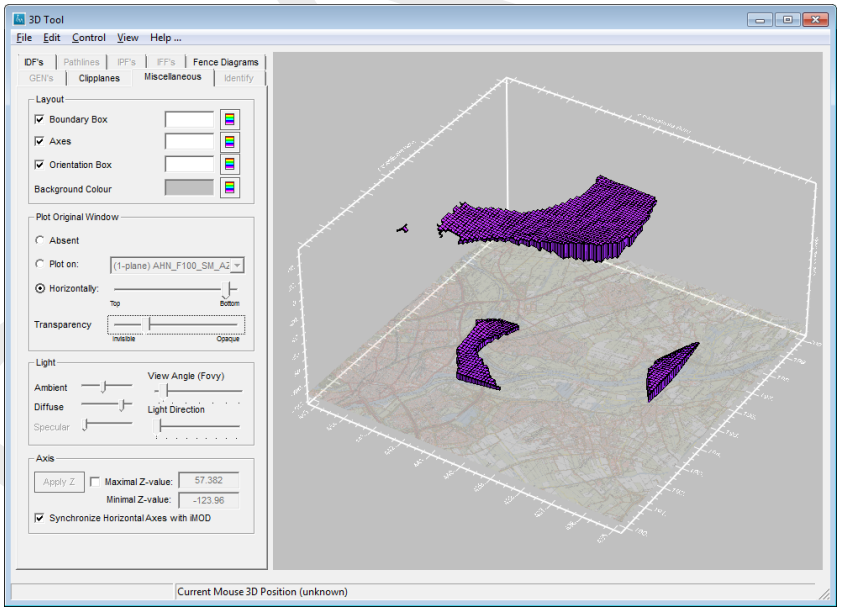
Example of a horizontal plot of the active image of the 2D graphical canvas.





Transparency	Drag the slider between fully transparent (*Invisible* on the left) and fully opaque (*Opaque* to the right). *Example of using transparency: fully opaque (top figure) and almost invisible (bottom figure).*



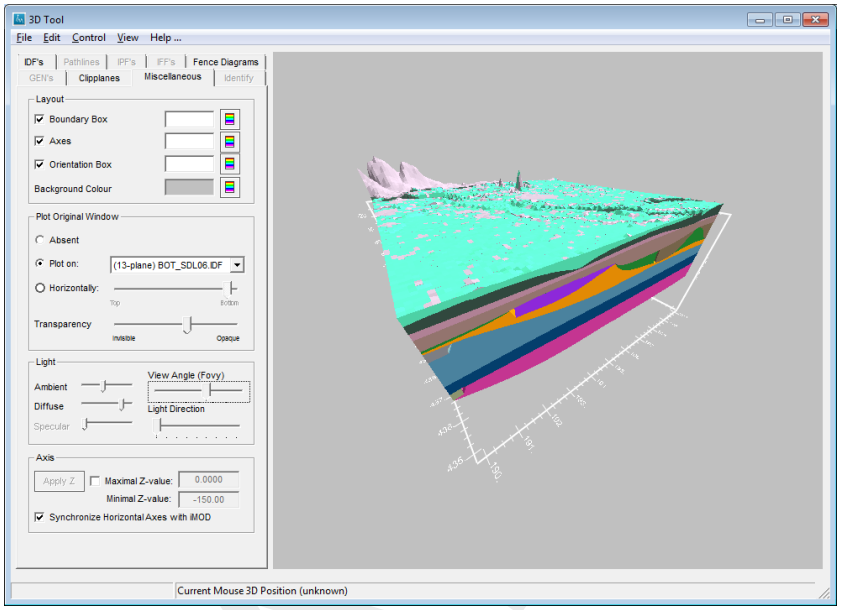


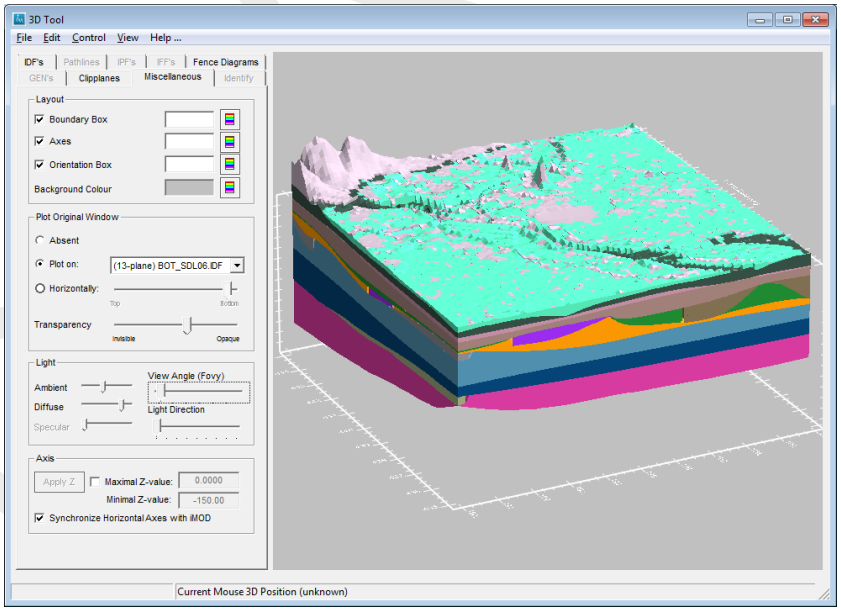
Ambient	Use the slider to increase the ambient light component (directional light component, generated shades).
Diffuse	Use the slider to increase the diffuse light component (background light).
Specular	Use the slider to increase the diffuse light component (shininess).

View Angle (Fovy)

Use the slider to change the view angle from a fish-eye (180 degrees) to 1 degrees (very narrow). A normal view-angle is 10% as a wider view angle gives a more flattened view, e.g. 60%.

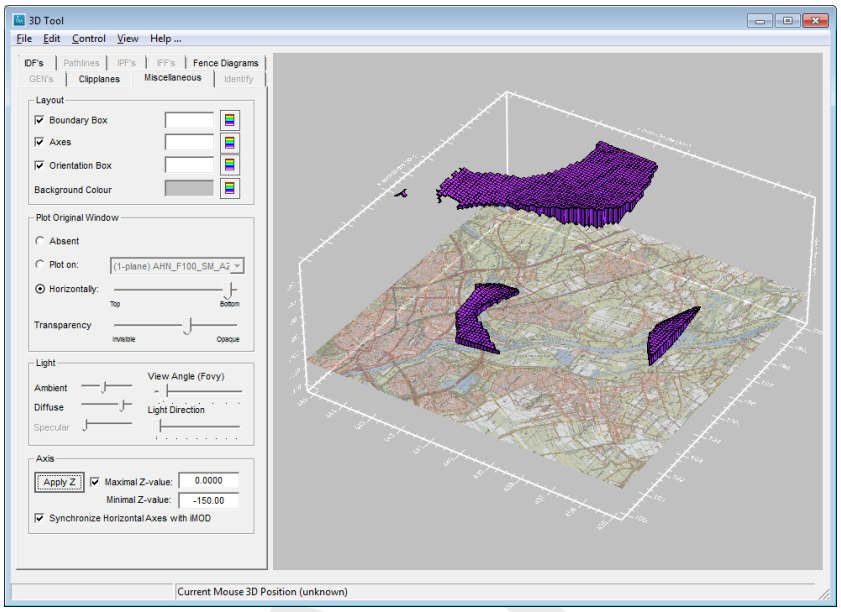
Example of a Fisheye view and a normal view.





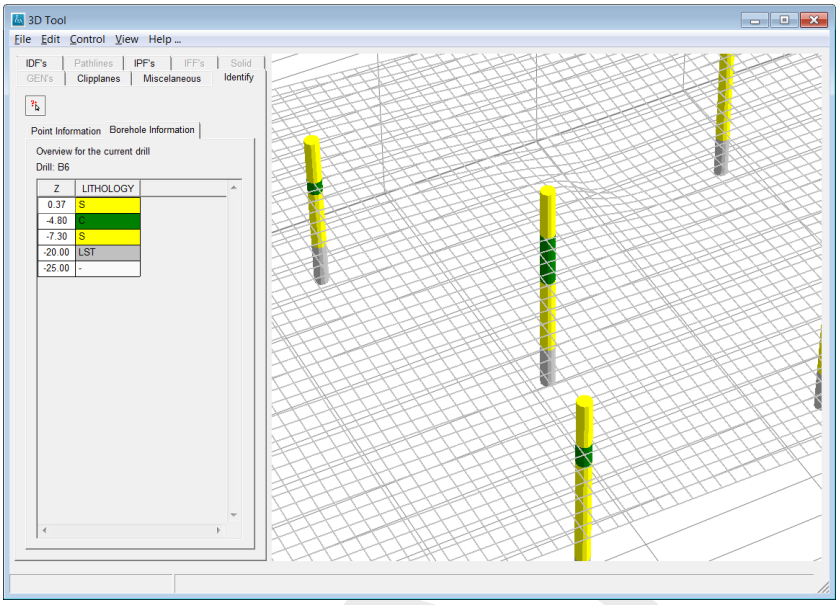
Light direction

Use the slider to change the light direction and shading.

Apply Z:	Click this button to reset the 3D image with the entered values for minimum and maximum values on the Z-scale. *Example of a different entry for the maximal- and minimal Z-values.* 
Maximal Z-value	Select this option to enter a different value for the maximum Z-value in the 3D image. The option *Minimal Z-value* becomes available as well.
Minimal Z-value	Enter a different value for the minimum Z-value in the 3D image.
Synchronize Horizontal Axes with iMOD	This option is checked to allow the 3D Tool to be synchronized with the extent of the 2D view of iMOD. Whenever this extent is changed (zoomed or panned) the 3D view will be adjusted accordingly. Uncheck this option to allow to change the extent in the 2-D window, without updating the 3-D view.

7.3.10 3D Tool: the 3D Identify-tab

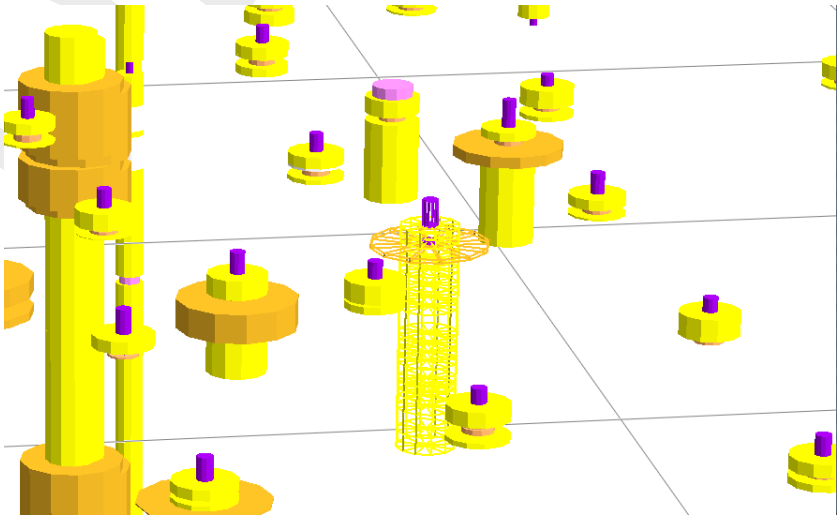
Use the 3D Tool *Identity* functionality to identify individual boreholes for inspection purposes. Select the *Identify* tab from the *3*D Tool window, this tab becomes available in case IPF's are selected for display.



The options of the *Identity* tab are described below:

Select
Click this button to (re)start selecting a borehole on the graphical canvas by hoovering the mouse pointer over a borehole. Clicking the left or right mouse button once on a borehole will stop the hoovering. Once a borehole has been selected its appearance will change to a wireframe representation and the table will be filled with the values of the IPF file of the selected borehole.

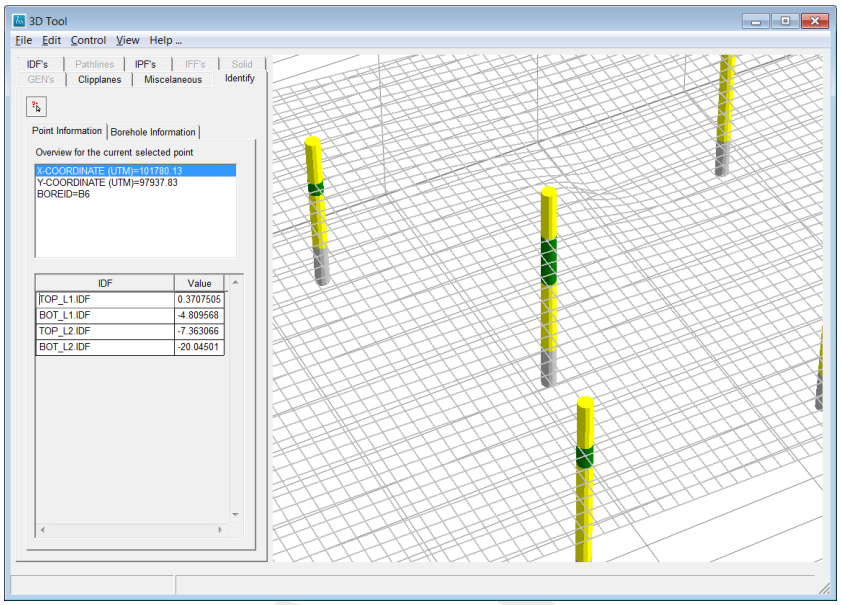
Example of a selected borehole with a wireframe appearance:



Point Information tab

This table shows the content of IPF file for the selected borehole and all values for the IDF files at the location of the particular point.

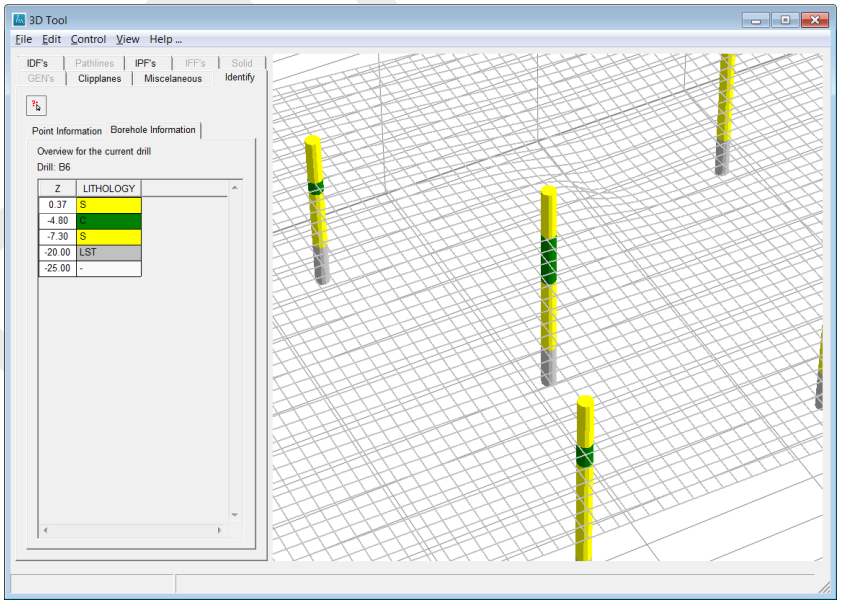
Point Information table



Borehole Information tab

This table shows the content of the associated file for the selected borehole; information on the actual borehole.

Borehole Information table



7.4 Solid Tool

WHY?

The *Solid Tool* is an instrument that can be used to create a 3D representation of the subsurface in which modellayers with different composition can be distinguished.

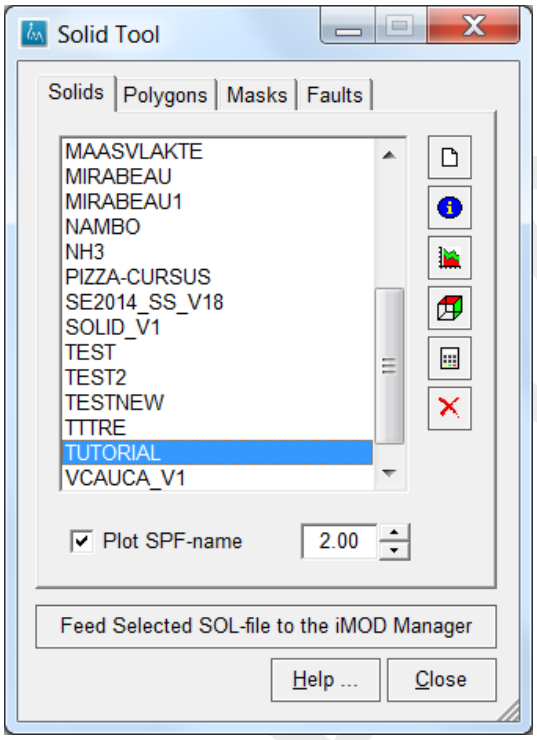
WHAT?

A solid is a collection of IDF-files that describes the different geohydrological modellayers of the subsurface. Each modellayer is represented by a top and bottom elevation and these are stored as IDF-files (TOP_L{i}.IDF and BOT_L{i}.IDF) in the folder {USER}\SOLIDS. The IDF-files are created or updated with the *Solid Tool* by interpolation of interface depths derived from cross-sections that describe the elevation of all modellayer interfaces.

HOW?

Select the option *Toolbox* from the main menu and then choose the option *Solid Tool* to start the *Solid Tool* window.

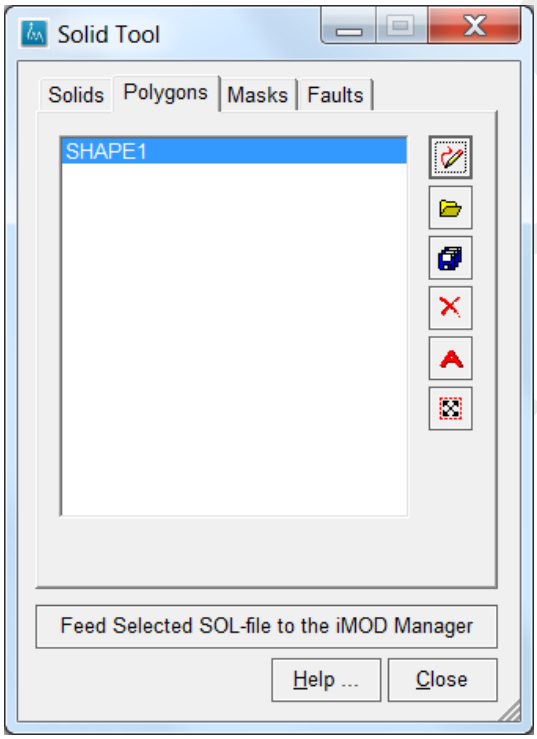
Solid Tool window*, Solids* tab:



List	The *Solid Tool* window shows the solids saved in the folder {USER}\{SOLIDS}. Select a solid from the list by clicking on the name.
	*New* Click this button to create a new solid (see section 7.4.1).
	*Info* Click this button to open the SOL file for the selected solid. Borehole logs can be included in the cross-section by a definition added to the solid file (*.SOL), see section section 9.20 for more detailed information about a SOL file.
	*Cross-Section Tool* Click this button to start the *Cross-Section Tool* (see section 7.1) in combination with the selected Solid to create and/or edit *Solid Cross-Section Files* (SPF) for more detailed information about SPF files.

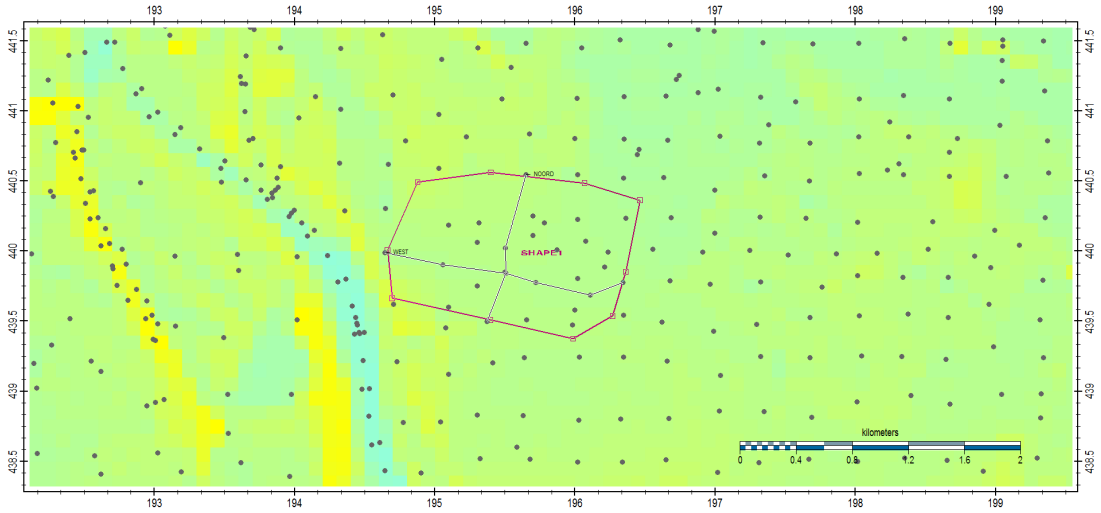
	*3D Tool* Click this button to start the *3D Tool* (see section 7.3) in combination with the selected Solid.
	*Compute* Click this button to compute the elevation of all modellayers in the selected solid based on the cross-sections described in the SPF files (see section section 9.21) and mentioned in the SOL file. See section section 7.4.4 for more information.
	*Delete* Click this button to delete the selected solid, iMOD will remove the folder {USER}\{SOLIDS} and its content.
Feed Selected SOL-file to the iMOD Manager	Click this item to let iMOD read automatically the solid properties and to add the solid IDF-files to the iMOD Manager when clicking the name of the solid in the list.
Plot SPF name	Select this option to display the name of each SPF in the 2-D plot.
2.00	Enter the letter size at which the SPF names are plotted.
Help ...	Click this button to start the Help functionality.
Close	Click this button to close the *Solid Tool* window.

Solid Tool window*, Polygons* tab:



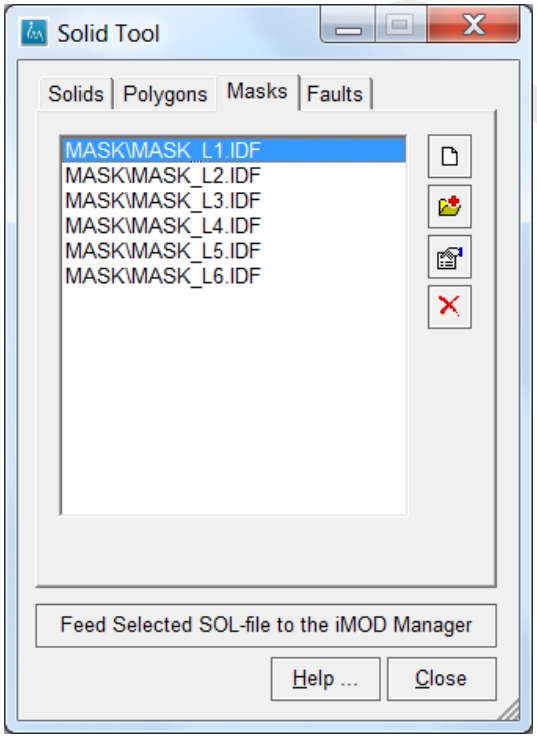
The interpolation of the interfaces of the solid may be executed within the limits of one or more defined polygons solely. In this manner it is possible to adjust any geological model locally. Each polygon will act similar for to modellayers. If different areas need to be applied for different model layers, the use of the Masks window (see *AquitardsMasks* tab on the *Solid Tool* is more appropriate. The polygons are defined in the *Solid Tool* window, *Polygons* tab and the functions of those buttons are described in detail in section section 4.2.

Example of a polygon (SHAPE1) that defines the area for which interfaces of the solid will be adjusted by the two drawn cross-sections.



The defined polygon is saved to the solid when tabbing towards to the *Solids* tab of the *Solid Tool* window. The defined polygon is not saved to the solid when clicking the *Close* button on the *Solid Tool* window.

Solid Tool window*, AquitardsMasks* tab:

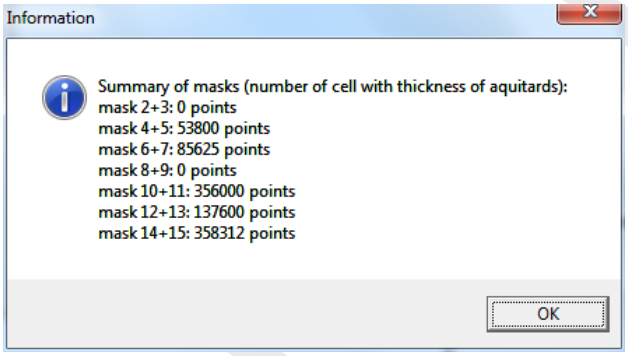


The extent of the aquitards located between the bottom and top interfaces of subsequent layers can be defined using masks. In this manner the extent can be formed by values in the mask files. Those mask files are IDF files that can be created using the *New* button (see

below) and/or created differently as long as the dimensions of the Mask-IDF is identical to the IDF files listed in the SOL-file. Eventually, any Mask IDF file will be saved in the selected SOL file whenever the tab *Solids* on the *Solid Tool* window is selected. The values in the Mask IDF behave as follows:

-2	iMOD applies a value of -2 internally to fixate locations that are effected by cross-sections.
-1	Use this value to specify areas that do not need to be computed. iMOD will use the original values instead.
0	Use this value to specify areas that are excluded.
1	Use this value to specify areas that need to be compute.
2	Use this value to specify areas that need to be equal to the values of the upper layer. It will act as if its value is -1 but uses the results from the upper layer as fixated value. The value 2 is recommended to extent aquitards, specify a value of $+2$ outside the extent of an existing aquitard or equivalent in order to define the boundaries of these aquitard.

New
Click this button to create new masks for all aquitards. iMOD will compute a Mask Value of 1 for the bottom elevation of modellayer i where there is a positive difference with the underlying top of modellayer $i + 1$. Outside those area, the Mask Value is 1 and for the top of the underlying modellayer 2. An information window appears specifying the created masks.

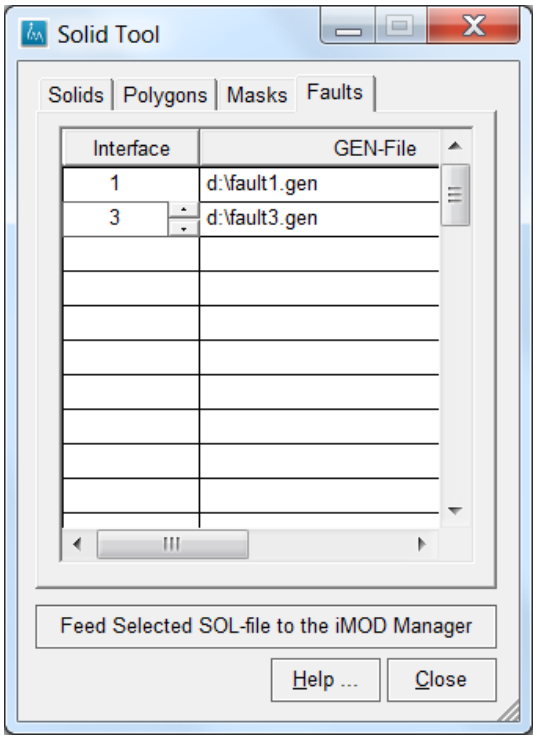


Open Map
Click this button to open an IDF-file to be used as mask file.

Properties
Click this button to open the properties of the mask.

Delete
Click this button to remove the mask file from the list.

Solid Tool window*, Faults* tab:



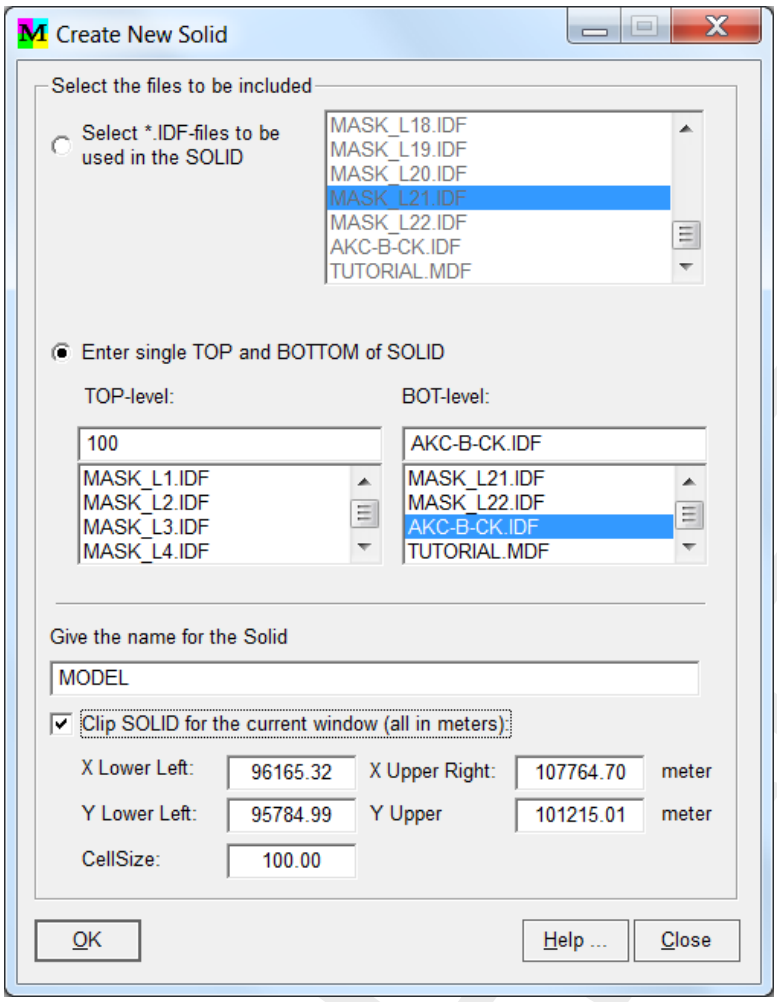
The interpolation can be obstructed by the shape of faults, or breaklines. Those need to be entered as GEN-files (see section section 9.10).

Interface	Enter the number of the interface for which the faults or breaklines need to be applied.
GEN-file	Enter the name of the GEN file which contains the breaklines. More than one breakline may be present in each GEN file.

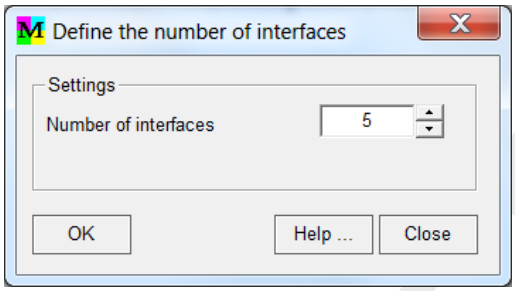
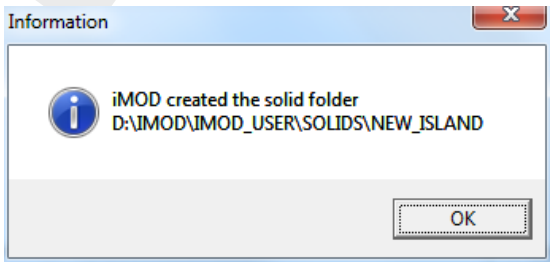
7.4.1 Create a Solid

A solid is a collection of IDF-files that describe the top and bottom elevations of geohydrological interfaces in the subsurface. The solid is created by selecting the relevant IDF-files in *the iMOD Manager and by* clicking the *New* button on the *Solid Tool* window to start the *Create New Solid* window.

Create New Solid window:



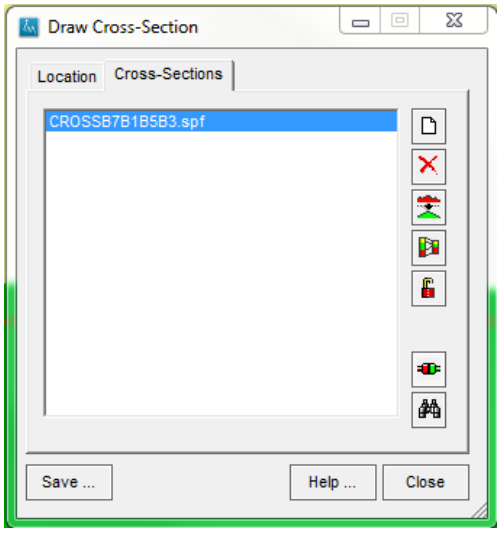
*Select *.IDF files to be used in the SOLID*	Select this option to specify IDF files from the list that need to be used as individual interfaces in the SOLID. iMOD will copy or clip the selected IDF files and save them in the specified folder in which all SOLIDS are saved (i.e. {USER}\{SOLIDS}*.IDF.

Enter single TOP and BOTTOM of SOLID	Select this option to specify a constant value and/or IDF file for the definition of a TOP- and BOTTOM level of the SOLID. iMOD will copy or create these files (in case of constant values) to the SOLID folder and rename them to INT_L{i}.IDF and INT_L{n}.IDF, identical to the order of the selected files. The actual number of interfaces can be entered in the following appearing menu *Define the number of interfaces*window:  *Number of interfaces* Select the number of interfaces a separate IDF file is generated. iMOD will divide the distance between the entered TOP and BOTTOM values into equally distances. All these files are located in the SOLIDS subdirectory.
Give the name for the Solid	Enter the name of the solid, e.g. ISLAND. iMOD will create a folder called {user}\solids\ISLAND and saves the SOL file: ISLAND.SOL for more detailed information about SOL files.
CellSize:	Enter the cell size of the IDF files for the interfaces. Bear in mind that those cell sizes can be modified easily in the *Compute Interfaces* window. This item is compulsory whenever no IDF files are entered whenever the option *Enter single TOP and BOTTOM of SOLID* is selected, otherwise the IDF dimensions are use of the specified IDF files.
Clip SOLID for the current window (all in meters)	Check the checkbox in case you want to enter an extent different from the area of the selected IDFs. Enter the coordinates for the lower left and upper right corner of the solid. Make sure that these coordinates are within the extent of the selected IDFs.
OK	Click this button to create: ◇ A solid folder in {USER}\{SOLIDS}\{SOLIDNAME}; ◇ A solid file (*.SOL) inside the SOLIDS folder; ◇ A collection of INT_L{i}.IDF and INT_L{n}.IDF-files inside the SOLIDS folder where n represents the number of interfaces. *Information* window after a successful completion of the creation process. 
Help . . .	Click this button to start the Help functionality.
Close	Click this button to close the *Create New Solid* window and return to the *Solid Tool* window.

7.4.2 Solid Editing using Cross-Sections

Click the *Cross-Section Tool* button on the *Solid Tool* window to start the *Draw Cross-Section* window and the *Cross-Sections* window. The *Draw Cross-Section* window has the same functions as described for the Cross-Section Tool in section 7.1. The cross-section will be displayed in the *iMOD Cross-Section CHILD* window. The interfaces of the model layers in the solid can be edited manually using the *Cross-Sections* window.

Cross-Sections window:

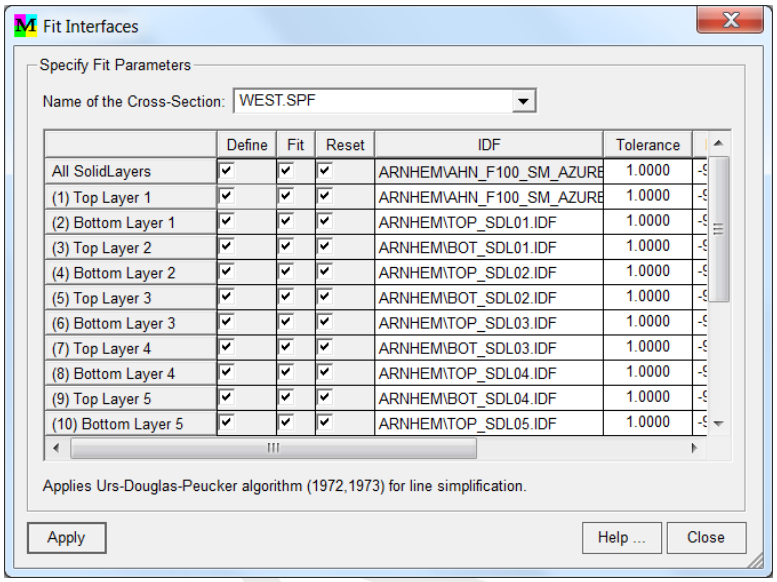


List of Available Cross-sections	The cross-sections defined for the selected solid are shown here.
	New Click this button to create a new cross-section. Make sure that you draw a new cross-section first using the *Draw* button on the *Draw Cross-Section* window. If not, a copy will be made of the selected cross-section in the list. The *Fit Interfaces* window will be started to define the name of the cross-section and enter initial settings to fit the interfaces.
	Delete Click this button *to delete the selected cross-section. The cross-section will be deleted from the list in the Cross-Section* window. *The cross-section will be deleted from the solid once* you close the *Cross-Section* window and confirms the Question to save the (adjusted/added) cross-sections. However the cross-section SPF-file is not removed and remains available for later use.

Fit Interfaces

Click this button to open the *Fit Interfaces* window. This window offers the possibility to start an initial guess for the interfaces in the cross-section by fitting the interfaces along the cross-section on the values read from the corresponding IDFs as mentioned and assigned to in the selected SOL-file.

Example of the Fit Interfaces window:



Name of the Cross-Section:	Specify the name of the selected cross-section; the name can not be modified once cross-sections have been defined.
Define	Check the box to define an interface line that can be modified.
Fit	Check the box to allow fitting of the interface of the IDF.
Reset	Check the box to initialize an interface line; the result will be a horizontal interface.
IDF	Select the IDF for each interface from the dropdown list.
Tolerance	Enter the accuracy in meters for which an interface is fitted.
Exclude	Enter the value of the IDF not used in the fitting; this is usually (by default) the *NoDataValue* of the IDF files.
Apply	Click this button to start to fit each interface line to the corresponding IDF-files.
Help ...	Click this button to start the Help functionality.
Close	Click this button to close the *Fit Cross-Section* window and return to the *Solid Tool* window.

Create interfaces from borehole logs

Click this button to create interfaces based on the interfaces defined in the borehole logs which are part of the solid definition.

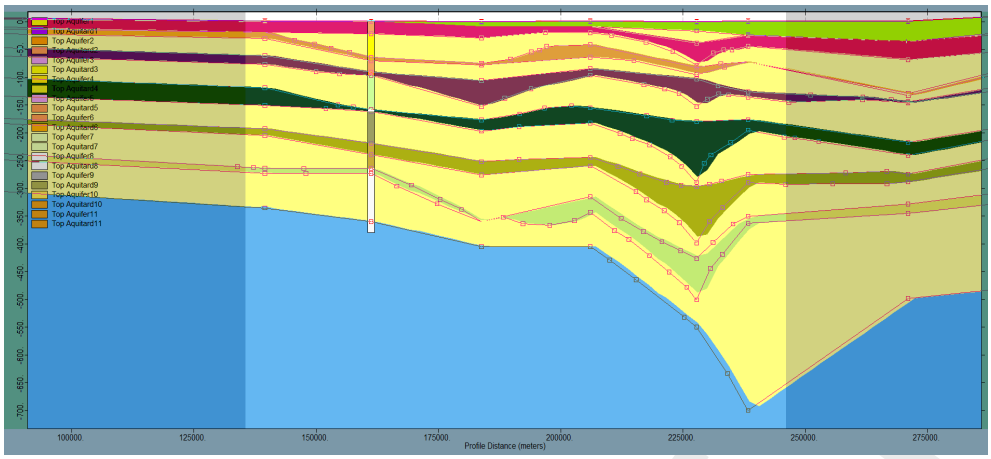
Lock

Click this button to lock the behaviour of each interfaces in between upper- and lower interfaces. Whenever they might cross due to a movement, they will be locked in order to avoid that they cross. Whenever two nodes are exactly on each other (which is allowed), this can prevent a movement of the node. Uncheck this button in that case to allow full editing of the interfaces.

View Editable Area

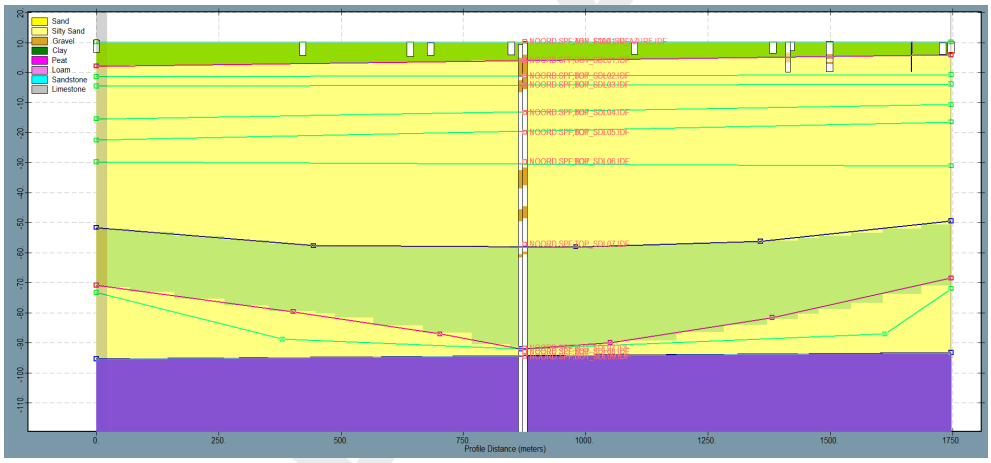
Click this button to view the editable area based on the extent of the entered polygons as specified on the *Solid Tool* window, *Polygons* tab. Though, interfaces can be changed outside the editable areas (grey areas), they will not be used in the interpolation of new interfaces.

Example of a cross-section showing the editable area and non-editable areas (grey)

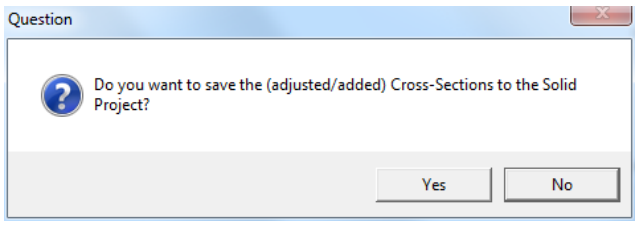


	*Snap* Select this option to snap the selected node on the interface i to the nearest interface $i-1$ and $i+1$. Whenever the snap does not work, it is probably caused by the fact that the interface to be snapped at, is not directly above- of beneath the interface considered.
	*View* Click this button to show the name of the cross-section and the name of the interfaces that crosses the current cross-section as shown on the *Cross-Section CHILD* window.

Example of a cross-section showing the interfaces of a crossing cross-section.

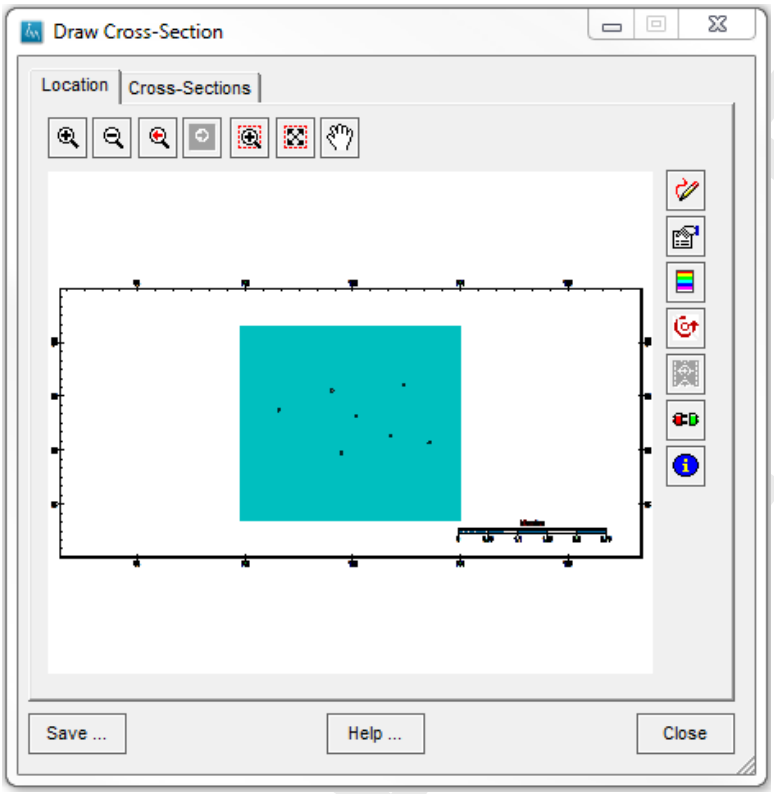


Help . . .	Click this button to start the Help functionality.
Close	Click this button to close the *Solid Tool* window (and therefore the *Cross-Section Tool* window) and to return to the *Solid Tool* window. The cross-section(s) will be saved into separate SPF-files to the solid folder when confirming the Question to save the (adjusted/added) cross-sections.



The *iMOD Cross-Section CHILD* window provides the opportunity to edit the interfaces manually. When you move the cursor in the neighborhood of a (red, blue or green) line it changes in a red arrow and you can click the left mouse button and drag the line to another position. When the cursor becomes a black arrow you can modify the existing node of the line. This editing mode is similar to modifying polygons, see section 4.4. Be aware that there is no possibility to undo move actions.

Example of iMOD Cross-Section CHILD window:

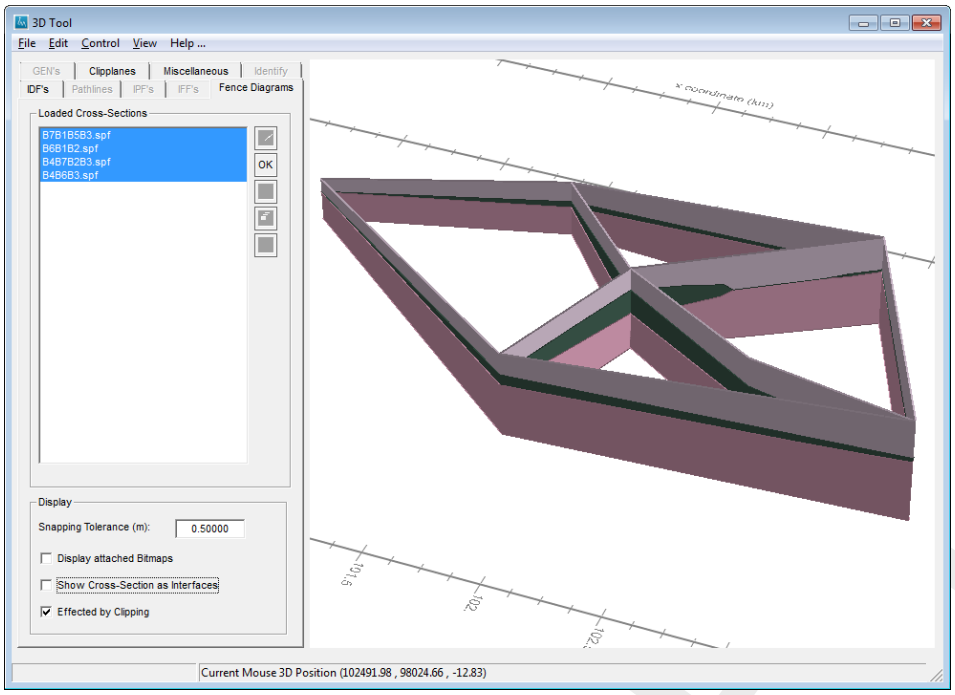


The *iMOD Cross-Section CHILD* window shows the interfaces of the model layers in three colors: red, blue and green. The red colour is used for the top interface of a modellayer, the blue colour is used for the bottom interface of a modellayer. The green colour is used when bottom and top interfaces of subsequent model layers overlap.

7.4.3 Solid Analysing using the 3D Tool

Click the *3D Tool* button on the *Solid Tool* window to start the*3D Tool* graphical window. The *3D IDF Settings* window will appear first whenever IDFs are selected in the *iMOD Manager* window. This window has the same functions as described for the 3D Tool in section 7.3. After that, all cross-sections are listed in the tab *Fence Diagrams* on the *3D-Tool* window, see section section 7.3.7 for further explanation.

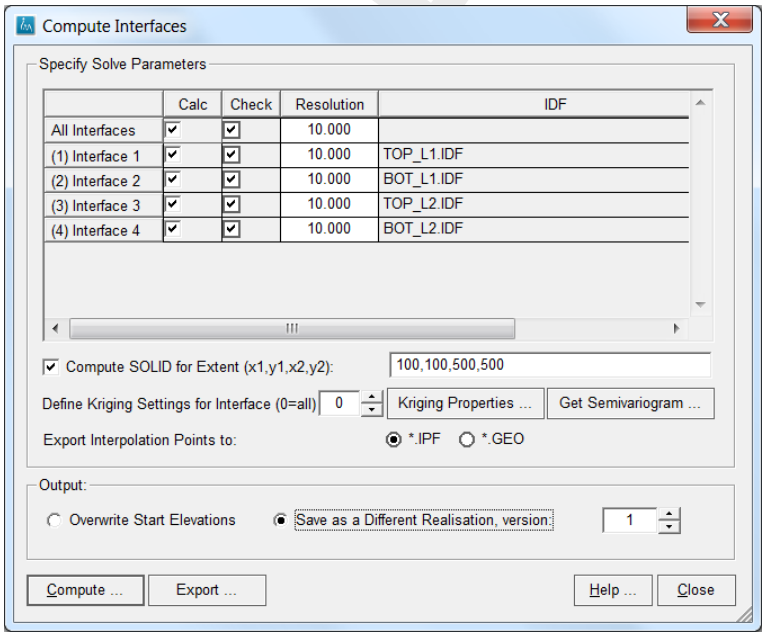
Example of a 3D image of the possible outline of cross-sections of a solid.

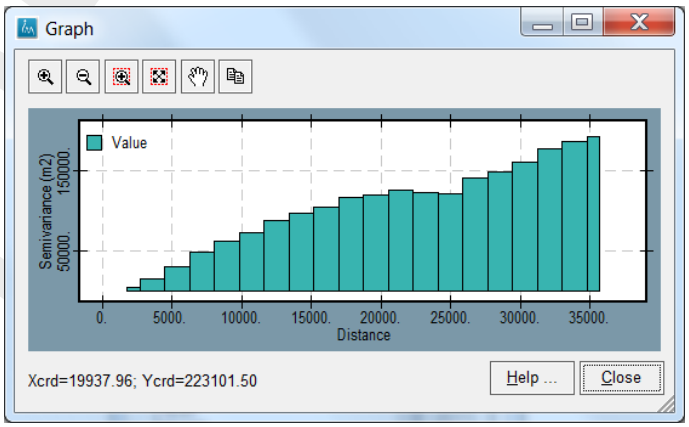


7.4.4 Compute Interfaces

iMOD facilitates two methodologies to convert the cross-section into a 3D representation of the subsoil, named a SOLID. Basically, iMOD uses (a) a linear interpolation of the entire interface in each cross-section yielding an accurate representation of the interfaces or (b) performs a Kriging interpolation using only the knick points in the interfaces, yielding a more smooth interpolation. Select the option *Compute* from the *Solid Tool* window top start the *Compute Interfaces* window.

Compute Interfaces window



Calc:	Check this option to select the interfaces to be recomputed, if not selected, those will remain unchanged, unless the option *Check* is activated.
Check:	Check this option to perform a consistency check upon the interfaces, this applied whether it will be computed or not. iMOD uses the rule that interface i must be $<=$ interface $i-1$, if not interface i becomes equal to the value of interface $i-1$.
Resolution:	Enter a resolution of the IDF to be interpolated for the individual interfaces. It has the advantage to start at a coarse scale (e.g. 100m) to have a quick results of the interpolation and whenever the SOLID improves, the final interpolation can be carried out on a finer scale. Bear in mind, that the resolution should be at least as fine as the detail of the cross-sections. At the end, the specified cell size in the *runfile* will smoothen the interpolated interfaces furthermore if desired.
IDF:	List of the used and written IDF file names for the interfaces. The SOLID tool always uses those IDF files at the *root* of the SOLID folder, results can be written in different version folder, see *Output* settings.
Compute SOLID for Extent (x1,y1,x2,y2):	Enter the extent or which the SOLID need to be computed. In this manner the existing SOLID files can be enlarged or reduced.
Kriging Proporties ...	Click this button to show the settings of the Kriging interpolation, see Table 4.1 for more detailed information. Whenever 0 is given in the integer field to the left, the Kriging settings apply for all interfaces. Alternatively, whenever a value of e.g. 4 is entered, the specific Kriging settings apply for interface 4 solely. Bear in mind, that the most important parameter for Kriging is the range over which the semivariogram extends. Changing that parameter does probably have the largest impact on the results of the interpolation.
Get Semivariogram ...	Click this button to compute the semivariogram for the selected layers at *Calc* in the table. For each of the interfaces the semivariogram will be displayed in a graph. By clicking the *Cancel* button the next interface will be computed. Before the semivariogram will be computed, it is necessary to confirm this action since, it might can take some while to compute. *Example of a computed Semivariogram* 
Export Interpolation Points to	Select the option *.IPF or *.GEO to export the knickpoints on each interface to an IPF or GEO file. A separate IPF or GEO file will be created and stored in {USER}\SOLIDS\{SOLIDNAME}\{VERSION}\EXPORT.
Overwrite Start Elevations:	Overwrite the original interfaces in the folder {USER}\SOLIDS\{SOLIDNAME}. However, it is not recommendable to do this.

Save as a different Realisation, version:	Save the result of the interpolation as a new version with the specified version number e.g. {USER}\SOLIDS\{SOLIDNAME}\{VERSION}. This is the default option.
Compute:	Start the interpolation for each interface.
Export:	Start the export of the knickpoints for each interface.
Help:	Click this button to start the Help functionality
Close:	Click this button to close the *Compute Interfaces* window and return to the *Solid Tool* window.

7.5 Movie Tool

WHY?

The *Movie Tool* is an instrument that can be used to create an playable movie out of a sequence of images created from 2D plot, e.g. contour maps of draw downs in time can be collected into a single *.AVI file to be displayed in a movie player. These movies can illustrate in a dynamical way how groundwater responds to measures.

WHAT?

Any combination of images, such as contour lines, grid and overlay can be used to create a movie file. These movie files are *.AVI and can be played with an existing and installed movie player.

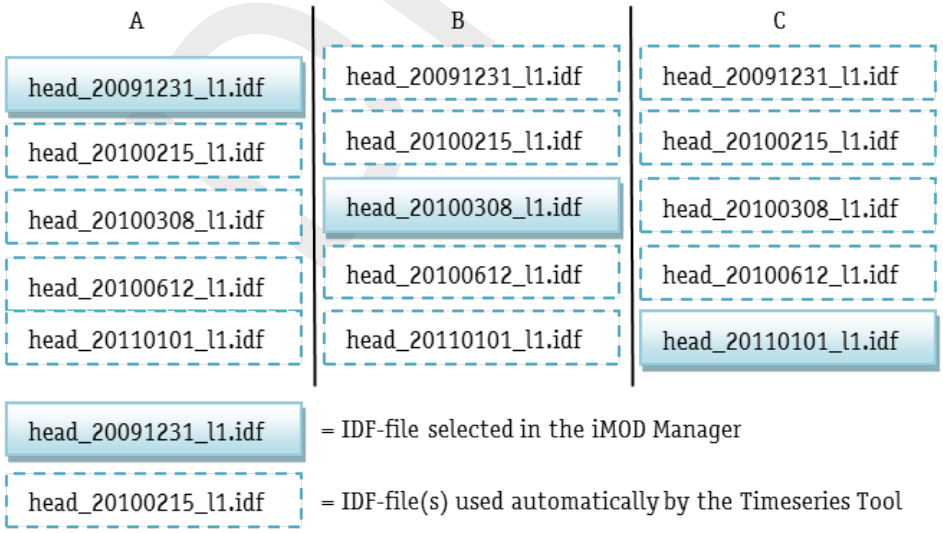
HOW?

Select the option *Toolbox* from the main menu and then choose the option *Movie Tool*, from there a option exists to create a movie via the option *Create a New Movie . . .* and to display and start the movie via the option *Play an Existing Movie . . .*.

7.5.1 Create a New Movie

The *Create a New Movie Tool* allows you to create an AVI from a selected set of plots with time independent and time dependent IDF files. iMOD will construct these files by collecting all data from the other related time-dependent IDF-files. Related time-dependent IDF-files have identical names but have a different date string. A date string is an eight digit continuous number, e.g. 20091231 meaning the 31^{th} of December 2009. It is not necessary to load all related time-dependent IDF-files in the *iMOD Manager*. At least one is sufficient to construct the entire time series. If a multiply set of IDF files is selected, each of them is used to load all time-dependent files. In the end all unique time steps will be used to generate images and IDF files will be re-used whenever they do not coincide with the current time step.

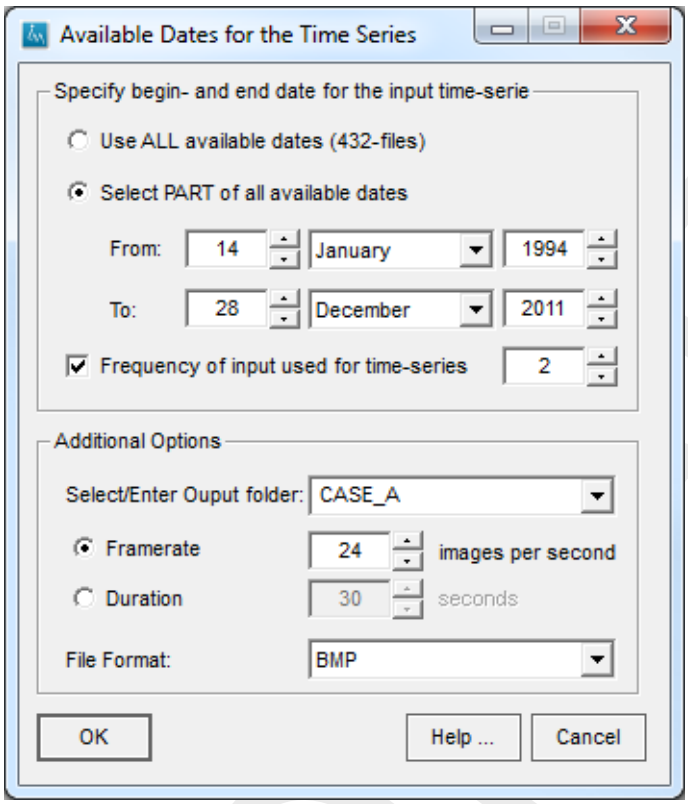
Example of IDF-files (A,B and/or C) available in the iMOD Manager prior to the start of the Create a New Movie Tool:



To start the *Create a New Movie* window, select the option *Toolbox* from the main menu, choose *Movie Tool* and then choose *Create a New Movie . . .*. Alternatively, you can click the

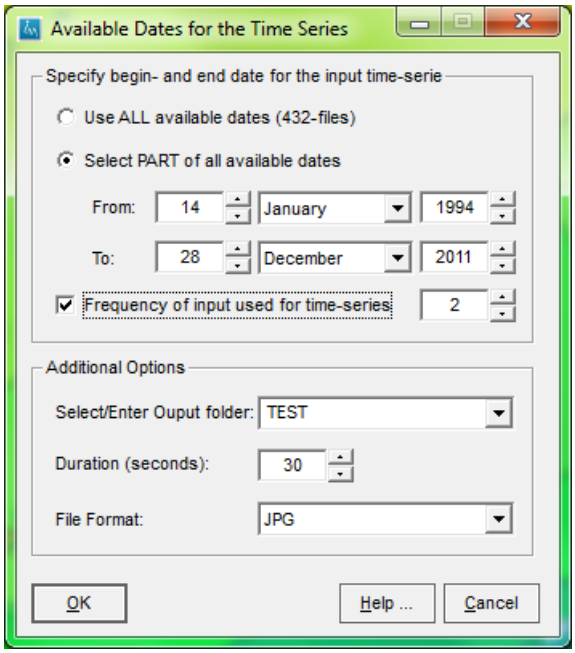
Create a New Movie button ( ) at the main toolbar. In both cases, you should select at least one IDF-file in the *iMOD Manager* that has a date notation in its name. This is a continuous number with eight-digits (yyyymmdd), e.g. *19940114* or fourteen-digits (yyyymmddhhmmss), e.g. 20121228123010. In the first case, it represents the 14$^{\text{th}}$ of January, 1994, for the second case is represents the 28$^{\text{th}}$ of December 2012 at 12 hours, 30 minutes and 10 seconds. If iMOD can **not** find such a date notation somewhere in the filename (in at least one of the selected IDF-files), the following window will appear and the *Create a New Movie Tool* will not start.

Warning window:



If a proper IDF-file(s) has been selected in the*iMOD Manager*, the following window will appear.

Available Dates window:



Use ALL available dates (432-files)	Select this option whenever you want to create an movie file for the entire time window that iMOD found.
Select PART of all available dates	Select this option to specify a different time window. This may gain processing time as less files need to be opened.
From:	Enter the start date of the time window. On default it displays the earliest date of the data.
To:	Enter the end data of the time window. On default it displays the latest date of the data.
Frequency of input used for time series	Select this option to decrease the number of dates used, e.g. by entering the value 2 iMOD will skip each second available date of the time series.
Select/Enter Output folder:	Select an existing output folder in which the individual images will be saved that are used to create a movie file (AVI) out of it. iMOD lists all the available folders in the user folder {IMOD_USER \MOVIE}. If an existing folder is selected, iMOD will ask for confirmation to delete its entire content before proceeding. It is also possible to enter a non-existing folder by typing in the name of the new folder.
Framerate	Enter the framerate of the movie in frames per seconds. Suppose iMOD generates a movie for 48 images (frames) and the framerate of the movie is 24 frame per seconds, the *Duration* of the movie will be 2 seconds.
Duration	Enter the duration of the movie in seconds. Suppose iMOD generates a movie for 48 images (frames) and the duration of the movie need to be 4 seconds, the *framerate* will be 12 frames per seconds. The higher the framerate the faster the consecutive images will appear.

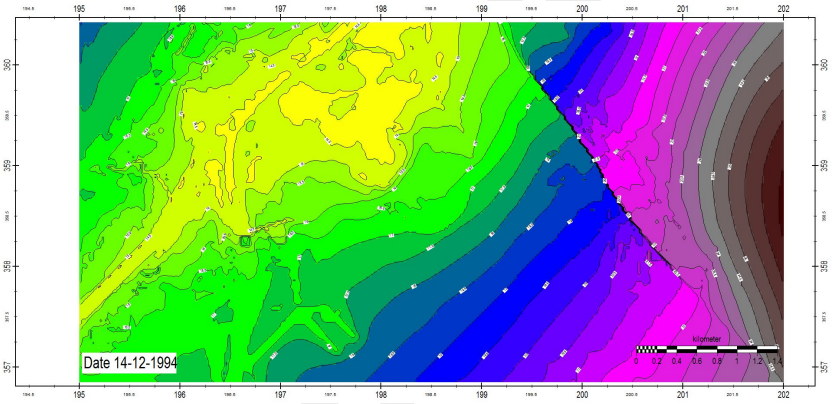
File Format

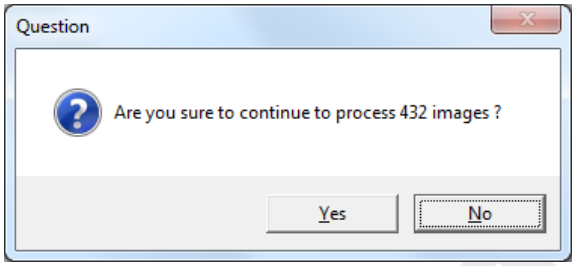
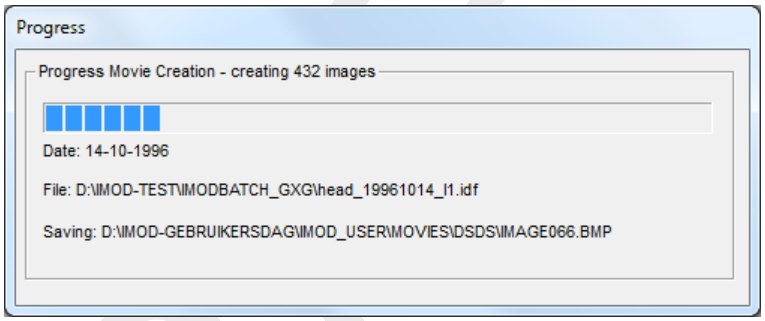
Choose one of the following file formats:

- ◇ Bitmap (*.BMP);
- ◇ Portable Network Graphics (*.PNG);
- ◇ ZSoft PC Paintbrush (*.PCX);
- ◇ JPEG image (*.JPG).

Be aware that the BMP file format is not compressed and gives a more detailed quality, it consumes more disc-space per image, approximately 7.0Mb. Instead, PCX or even better JPG file formats can be used that consume significantly less disc-space per image, approximately 0.1Mb. For each image, a separate file is create in the selected *Output Folder*, e.g. IMAGE001.JPG. Each file has a time stamp as well.

Example of one of the consecutive images



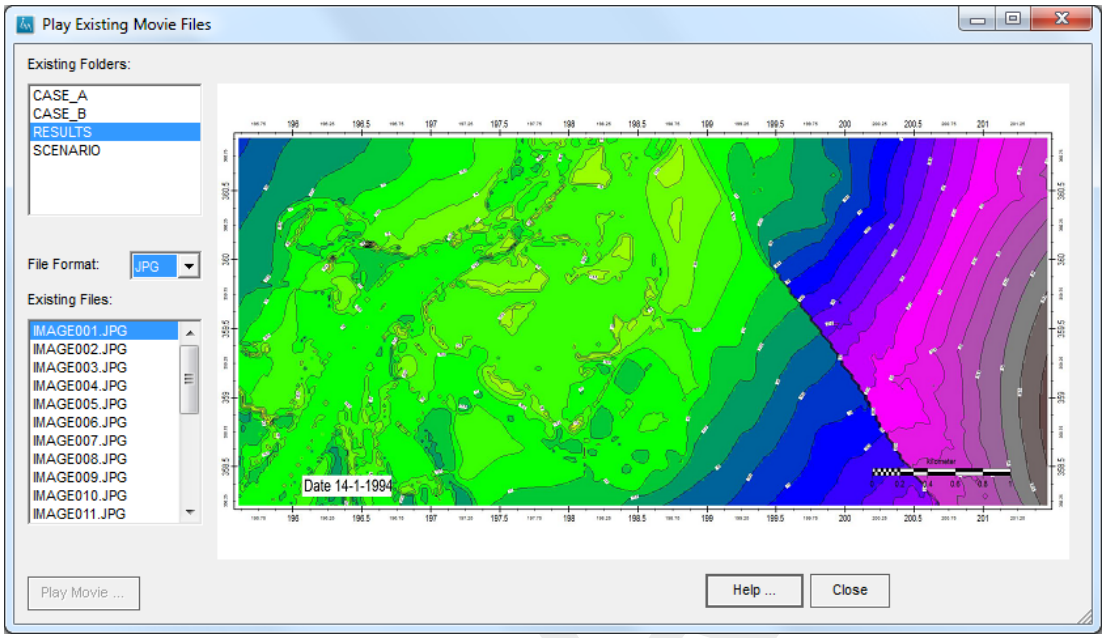
OK	Click this button to start the *Create a New Movie Tool* for the selected time window. The *Available Dates* window will close. iMOD will generate a question first to confirm the generation of the consecutive images. *Question for confirmation of the generation of consecutive images*  Enduring the process of generating these consecutive images, the progress is displayed in the following window. *Progress window in the process of generating consecutive images*  The process cannot be stopped, but will if there might be an error while saving an image to disc, e.g. whenever the disc-space runs out. After the process, iMOD asks to start the *Play an Existing Movie* Tool to display the images, see section 7.5.2.
Help . . .	Click this button to start the iMOD Help Functionality.
Cancel	Click this button to close the *Available Dates* window; the *Timeseries Tool* will not start.

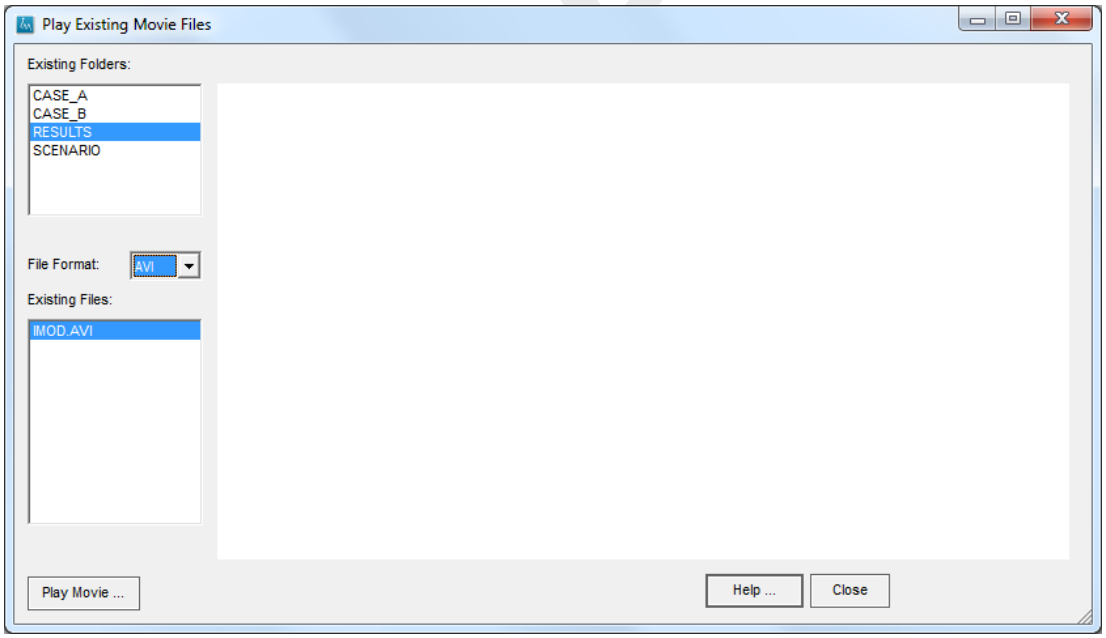
Note: You should select at least one IDF with date information in its filename, other IDF-files that are selected without a date information, will be displayed as time-constant. In this way you can easily make a combination with time-variant information (e.g. drawdown) and time-invariant information (e.g. landuse).

7.5.2 Play an Existing Movie

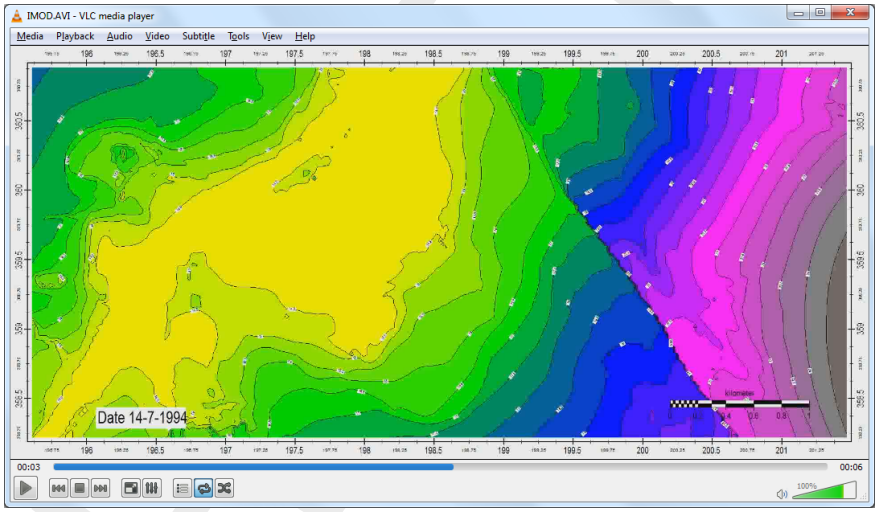
Start the *Play Existing Movie Files* window by selecting the option *Toolbox* from the main menu, choose *Movie Tool* and then choose *Play Existing Movie Files . . . .*

Play Existing Movie Files window:





Existing Folders:	Select one of the existing folders in the {IMOD_USER \MOVIE} folder.

File Format:	Select one of the existing file formats for display purposes. Choose one of the following file formats: ◇ Bitmap (*.BMP); ◇ Portable Network Graphics (*.PNG); ◇ ZSoft PC Paintbrush (*.PCX); ◇ JPEG image (*.JPG); ◇ AVI Audio Video Interleave (*.AVI).
Existing Files:	Select one of the existing files to display them in the graphical frame on the right. This is operational as long as one of the following file formates are selected: BMP, PNG, PCX, JPG are selected.
Play Movie . . .	Click this button to start a movie player, such as FFMPLAY or VLCPLAYER. This option is available whenever one of this players are referred to in the PRF-file, see section 9.1 and whenever the AVI file format is selected. *Example of the VLCPLAYER* 
Help. . .	Click this button to start the HELP functionality
Close	Click this button to close the *Play Existing Movie Files* window

7.6 GeoConnect Tool

WHY?

The *GeoConnect Tool* is an instrument that can be used to aggregate geological formations to model layers and vice versa.

WHAT?

With the *GeoConnect Tool* it is possible to aggregate geological formations to a dataset with different model layers containing combined formation information. It is now possible to go from model results with multiple geological formations per layer to a geological formations dataset. Besides, with this tool it is possible to use geological formations to come up with a new model parameterization. This functionality is also available as a batch-function (see section 8.7.7).

HOW?

Before starting the GeoConnect Tool, the user needs to create a textfile with the general settings to be used in the tool. This file needs to be placed in the "IMOD_USER\SETTINGS\" subfolder with the name "GeoConnect.txt".

The textfile with general settings needs to contain the following keywords:

ACTLAYERS= (optional)	Enter a string of values to include or exclude a specific model layer from the computation; 0=inactive, 1=active, on default all layers are used in de computation (similar to e.g.: ACTLAYERS=1111111111). E.g. in case of the amount of model layers is 10 and it is preferred to only take the first 6 layers into account: ACTLAYERS=1111110000.
REGISFOLDER=	Give the directory and name of the folder where all REGIS-files are stored. **Note:** subdirectories are not allowed and the filenames need to be of the following format: abbreviation formation name-t/b/ks/kv-ck/sk.idf ('t' and 'b' need to be combined with 'ck', and 'ks' and 'kv' with 'sk'), e.g. d:\Model\REGIS\bez1-b-ck.idf.
TOPFOLDER=	Give the directory and name of the folder of the model TOP-files, e.g. d:\Model\TOP\TOP.
BOTFOLDER=	Give the directory and name of the folder of the model BOT-files, e.g. d:\Model\BOT\BOT.
NLAY=	Enter the amount of model layers, e.g. NLAY=10.

These settings are automatically placed on the Settings tab on the GeoConnect window whenever the tool is started. The aggregation process is described in the example below.

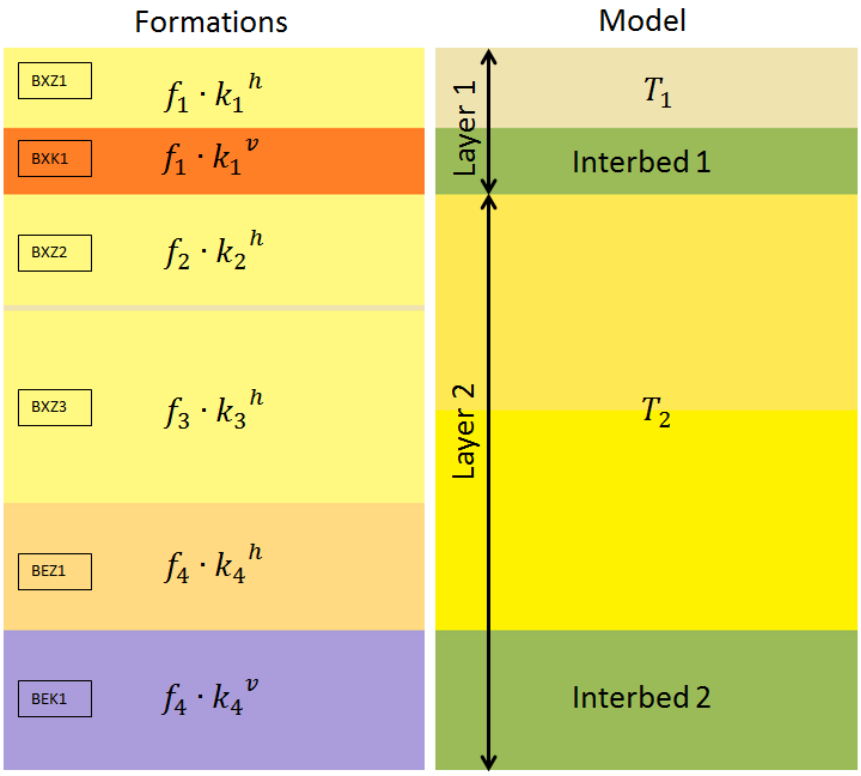
Example 1
ACTLAYERS=1111110000000000000
REGISFOLDER=d:\Model_Ibrahym\REGIS\
TOPFOLDER=d:\Model_Ibrahym\TOP\VERSION_1\
BOTFOLDER=d:\Model_Ibrahym\BOT\VERSION_1\
NLAY=19

Geostratigraphy textfile
The *geostratigraphy.txt* file contains the order of the formations (from surface layer to base) and needs to be located in the Settings-folder in the iMOD user environment (in case of using the GUI), contains all formation abbrevations as used in the geological dataset and corresponding factors. This file can be loaded into the *Postprocessing*-tab on the *GeoConnect* window in the iMOD-GUI.

Example of the first 11 and last 4 lines of a geostratigraphy.txt:
HLC

BXSCK1
BXZ1
BXLMK1
BXK1
BXZ2
BXK2
BXZ3
BEROK1
BEZ1
BEK1
...
MTQ
GUQ
VAC
AKC

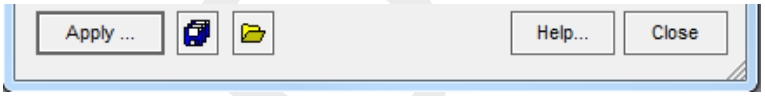
Calculation/Aggregation of formations:
This example shows the mathematical theory behind the aggregation of formation characteristics within one model layer. The formulas in the table are used to calculate the new transmissivity, groundwater head vertical resistance and flux. The choice of the formula depends on the variable that needs to be aggregated. In case, it is preferred to calculate the transmissivity (see figure) of the first and second model layer based on the formations given in the formation dataset, the first formula in the table is used to achieve this. In example:
$T1 = (d_1^f \times k_1^h \times f_1)$
$T2 = (d_2^f \times k_2^h \times f_2) + (d_3^f \times k_3^h \times f_3) + (d_4^f \times k_4^h \times f_4)$



Aggregation Formulas	Explanatory list	
$T_j = \sum_{i=1}^{n} {d_i}^f \cdot {k_i}^h \cdot {f_i}^p$	${f_i}^p$	Pest factor for i'th formation
	k_{fi}	Horizontal Permeability for the i'th formation (m/d)
	${k_i}^v$	Vertical Permeability for i'th model layer
$H_j = \frac{1}{\sum_{i=1}^{n} {d_i}^f} \cdot \sum_{i=1}^{n} {d_i}^f \cdot H_j$	Q_j	Flux (m^3/d)
	H_j	Groundwater head for the j'th model layer (m)
	${d_i}^f$	Layer thickness for j'th formation (m)
$Q_j = \sum_{i=1}^{n} \frac{{d_i}^f}{{d_j}^m} \cdot Q_j$	${d_j}^m$	Layer thickness for i'th formation (m)
	n	Number of formations to aggregate
	i	Formations layer number
$C_j = \sum_{i=1}^{n} \frac{{d_i}^f}{{k_i}^v \cdot {f_i}^p}$	j	Model layer number
	T_j	Transmissivity (m^2/d)
	C_j	Vertical resistance (d)

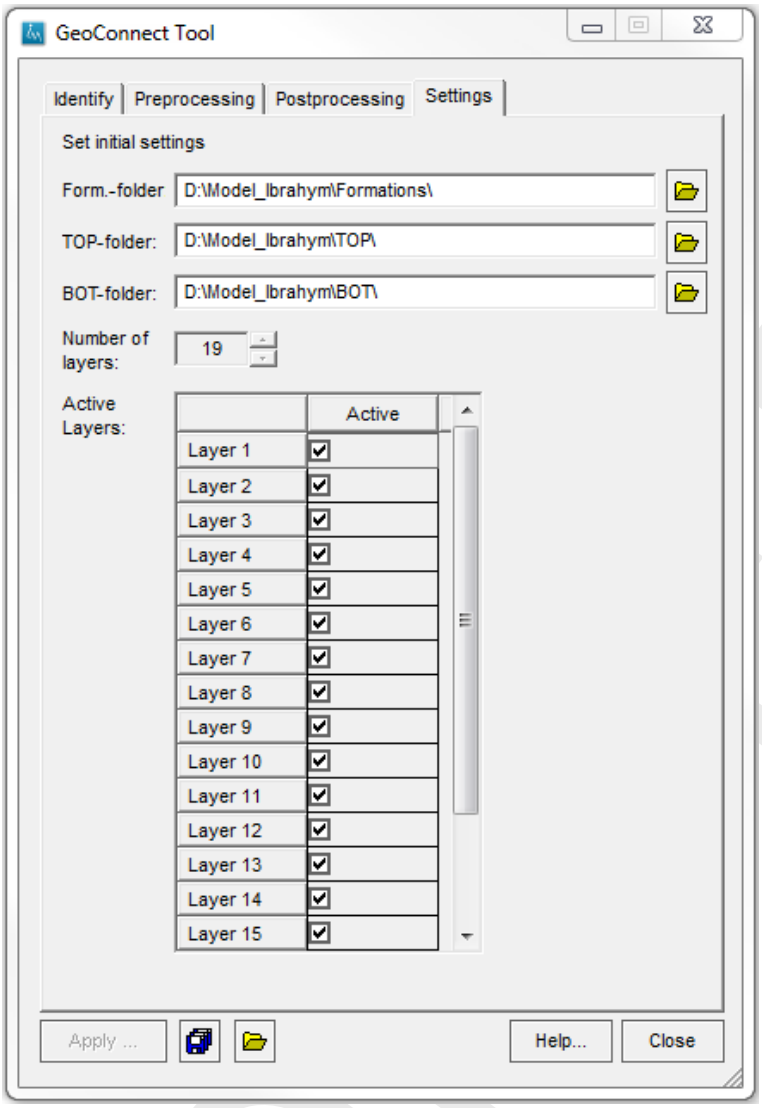
Start the Tool

To start the *GeoConnect Tool* select *Toolbox* from the main menu, choose *GeoConnect Tool*. This tool allows the user to continue working in the same iMOD session and using the GeoConnect tool at the same time. Included are a preprocessing, a postprocessing and an identify functionality. The tool opens with the Settings tab up front. The general settings on the toolwindow are:



Apply...	Click this button to start the preprocessing or postprocessing computation.
	*Save as...* Click this button to save the settings in a GeoConnect.txt file in case of *Settings tab*, or alternatively in an iMODBATCH *.ini file in case the *Preprocessing* tab or *Postprocessing* tab is selected.
	*Open* Click this button to get the settings from a GeoConnect.txt file in case of *Settings tab*, or alternatively from an iMODBATCH *.ini file in case the *Preprocessing* tab or *Postprocessing* tab is selected.
Help...	Click this button to start the Help functionality.
Close	Click this button to close the window.

Example of the GeoConnect window; Settings tab



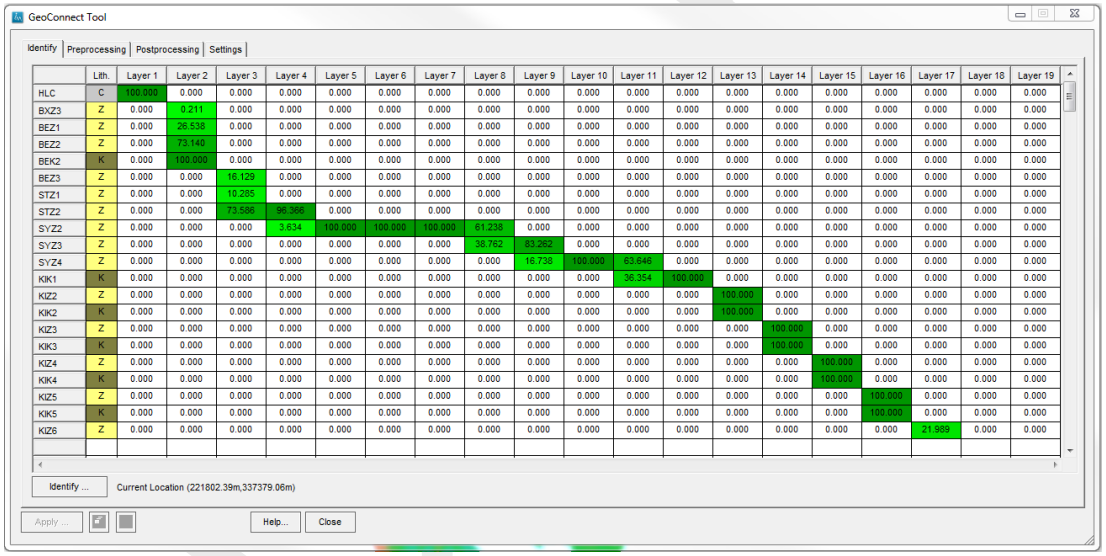
Form.-folder:	Give the name of the folder where all geological formation files are stored. **Note:** subdirectories are not allowed and the filenames need to be of the following format: abbreviation formation name-t/b/ks/kv-ck/sk.idf ('t' and 'b' need to be combined with 'ck', and 'ks' and 'kv' with 'sk'), e.g. d:\MODEL_Ibrahym\Formations\bez1-b-ck.idf.
	*Open* Click on this button to search for the name of the folder containing geological formation data.
Top-folder:	Give the name of the folder of the model TOP-files; e.g. d:\MODEL_Ibrahym\TOP. The tool searches for "d:\MODEL_Ibrahym\TOP\TOP_L1.IDF" in the given folder. This is repeated for each layer.
	*Open* Click on this button to search for the name of the folder containing the Top elevation of the model.
Bot-folder:	Give the name of the folder of the model BOT-files, e.g. d:\MODEL_Ibrahym\BOT. The tool searches for "d:\MODEL_Ibrahym\BOT\BOT_L1.IDF" in the given folder. This is repeated for each layer.

	*Open* Click on this button to search for the name of the folder containing the Bottom elevation of the model.
Number of layers	Represents the amount of model layers as defined in settings file that you can read into the GUI on the *Postprocessing*-tab, e.g. NLAY=10.
Active layers	Check a checkbox for a particular layer to include a specific model layer in the Identify routine (see **Identify**-tab).

Identify

The Identify option can be found on the first tab of the *GeoConnect* window. With this option it is possible to analyze, "on-the-fly", the model composition per individual location by making use of the *Identify* button. The functionality shows the content of the geological formations in the model at mouse position. Aquitard layers (e.g. clayey formations) are identified as connected to the Aquifer layers (sandy formations), the clayey layers are so called "interbed" layers. When a model layer contains a clayey layer the fraction shown in the *Identify* table is always 100%. See for example Layer 2 in the figure below (*Identify*-tab). This layer consists for 0.211% of BXZ3, 26.538% of BEZ1, 73.140% of BEZ2 and for 100% of BEK2.

Example of the GeoConnect window; Identify tab

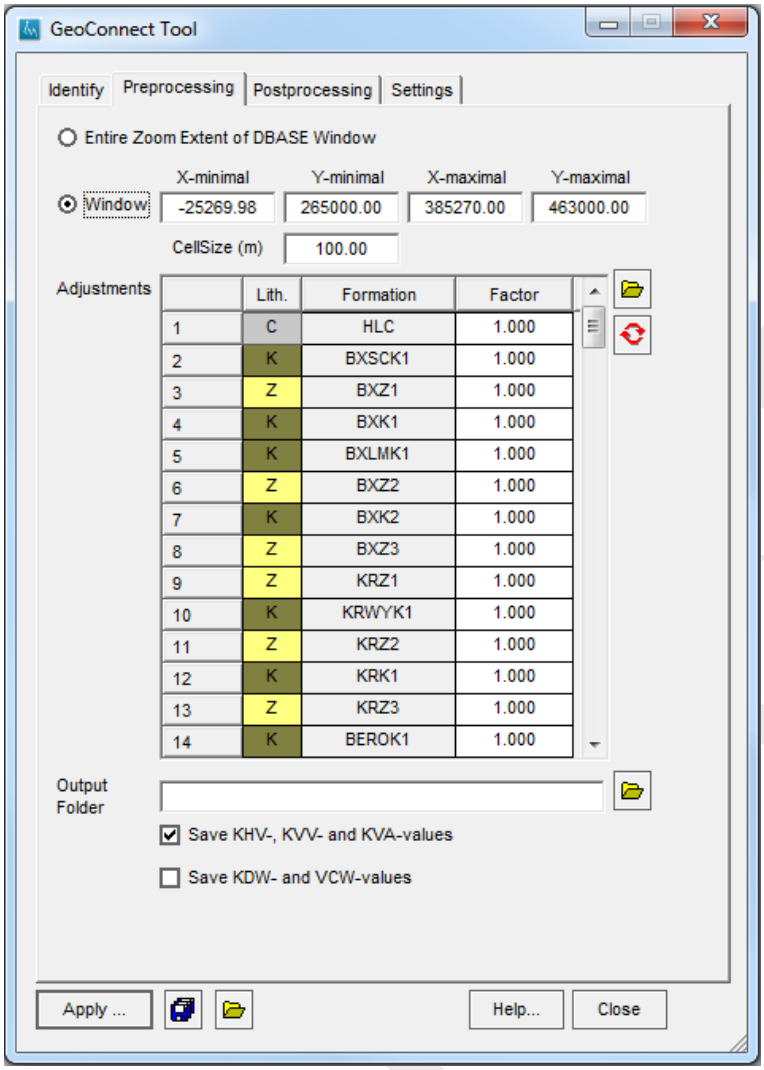


Identify. . .	Click this button to be able to hover over the map in the main iMOD window.	
Table description	*Lith.*:	Lithology class of the specific formation shown in the first column of the table.
	Layer}{i}:	i'th model layer
	"100.0":	Fraction of specific geological formation in given model layer, e.g. Layer 2, row 3: "26.538" means that layer 2 consists of 26.538% BEZ1 formation. The darker the color green the higher the fraction.
Current location (. . . m, . . . m)	Gives the current location (in meters) of the model composition at the mouse position.	

Preprocessing

The preprocessing functionality in the GeoConnect tool can be used to compute a renewed model parameterization out of the geological formation related top-, bot- and k-values.

Example of the GeoConnect window; preprocessing tab

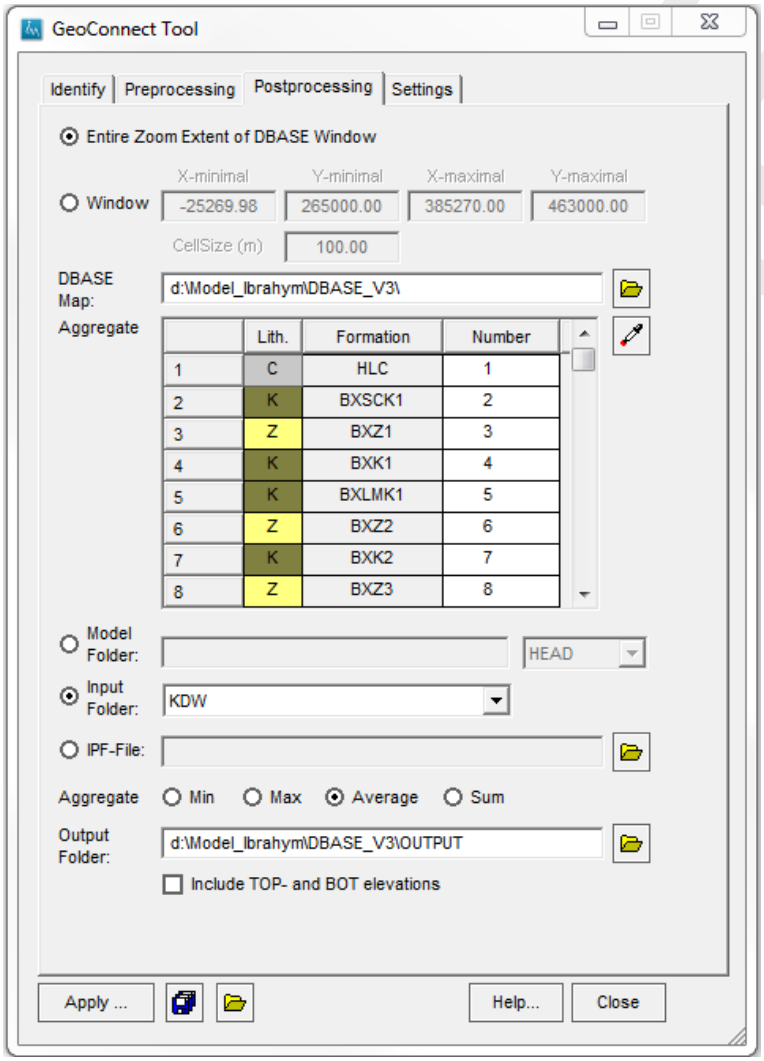


Entire Zoom Extent. . .	Select this option to use the complete model extent as described by the IDF-files in the TOP-folder (as defined on the *Settings* tab).	
Window	Select this option to compute only for the given window extent. Click on "Window" to refresh the extent accordingly to the current extent in the main iMOD window.	
X-minimal	Give a minimum X-coordinate value in meters.	
Y-minimal	Give a minimum Y-coordinate value in meters.	
X-maximal	Give a maximum X-coordinate value in meters.	
Y-maximal	Give a maximum Y-coordinate value in meters.	
CellSize (m)	Give the cell size in meters of the new model to be computed.	
Adjustments	Adjustable grid with factors sorted per formation.	
Table description:	*Lith.*:	In this column the lithology class is shown related to a specific geological formation, e.g. "K" means Clay, "Z" means Sand. The cells are colored accordingly to the individual lithology class.
	Formation:	In this column the geological formations are displayed.
	Factor:	In this column the factors per formation are given and can be adjusted, either by loading in an file with factors (on default factors.txt is read from the IMOD_USER\SETTINGS folder, see for an example section 8.7.7) or by changing the factors manually.
	*Open* Click this button to read a file with factors into the *Adjustments* grid.	

Refresh	Click this button to refresh the factors in the grid to 1.0.
Output Folder	Give the directory and name of the folder to store the results of the preprocessing computation.
	*Open* Click on this button to search for the name of the folder to store the results of the preprocessing computation.
Save KHV-,. . .	Check this option to calculate new parameter files for KHV, KVV and KVA.
Save KDW-,. . .	Check this option to calculate new parameter files for KDW and VCW.

Postprocessing
The postprocessing functionality in the GeoConnect tool can be used to perform model layer aggregation for variables to be chosen based on the formation settings.

Example of the GeoConnect window; postprocessing tab



Current Zoom Extent. . .	Select this option to use the complete model extent as described by the IDF-files in the TOP-folder (as defined on the *Settings* tab).
Window	Select this option to compute only for the given window extent. Click on "Window" to refresh the extent accordingly to the current extent in the main iMOD window.

DBASE map:	Give the directory and name of the folder containing DBASE model information.	
	*Open* Click this button to search for the specific DBASE model folder.	
	*Pipet* Click this button to fill the aggregated grid with all available geological formations at the current window extent.	
Aggregate	Grid with geological formations, arranged by the order given in GEOSTRATIGRAPHY.TXT (see section 8.7.7). Changing the numbers in the 4th column makes aggregation of geological layers possible, e.g. to aggregate all the BX* formations give them all the same number for example 2 or 5.	
Table description:	*Lith.*:	In this column the lithology class is shown related to the specific geological formation, e.g. "K" means Clay, "Z" means Sand. The cells are colored accordingly to the individual lithology class.
	Formation:	In this column the geological formations are displayed.
	Number:	In this column each unique number represents one geological formation. To combine different geological formations change the number, e.g. if you want to consider the Boxtel Sands/Clays (*BX**) as one model layer, enter the same number (e.g. 2) for each Boxtel formation related geological layer in this column.
Model Folder:	Give the directory and name of the folder containing the preferred model output information. Choose the favored variable in the related dropdown menu to apply the aggregation to. Options in dropdown menu are: "HEAD", "BDGWEL", "BDGRIV", "BDGDRN".	
Input Folder:	Choose the preferred variable from the dropdown menu to apply the aggregation to. Options are: "KDW", "VCW", "KHV", "KVV".	
IPF-File:	Give the name of the IPF-file to select points in the IPF that coincides with the aggregated formations.	
	*Open* Click on this button to search for the directory and name of the IPF-file to apply the aggregation to.	
Aggregate	Select the aggregation method to be used. The aggregation can be based on the values minimum (=Min), maximum (=Max), Average or Sum.	
Output Folder:	Enter the name of the folder to store the results of the postprocessing computation, e.g. (OUTPUTFOLDER)\BEZ1_BEZ2_BEZ_KDW.IDF, (OUTPUTFOLDER)\BEZ1_BEZ2_BEZ_TOP.IDF, (OUTPUTFOLDER)\BEZ1_BEZ2_BEZ_THK.IDF.	
	*Open* Click on this button to search for the name of the folder to store the results of the postprocessing computation.	
Include TOP-,...	Select this option to include TOP- and BOT elevation and thickness IDF-files in the aggregation and save these files in the given Output folder.	

7.7 Plugin Tool

WHY?

The *Plugin Tool* is an instrument that can be used to run external executables (from complicated to simple) or batchfiles within iMOD.

WHAT?

The Plugin tool makes it possible to include advanced plugins that come close to fully software integration as well as plugins that simply run an executable file. This allows you to start a specific program within an iMOD-session without stop working in this session. A number of pre-settings and couplings are necessary before you can launch a plugin. iMOD needs information about the plugin and vice versa.

HOW?

Before starting the iMOD Plugin Tool, you need to walk through a couple of preprocessing steps before a plugin can be used within the iMOD gui.

1 Predefine `PLUGIN1=` and/or `PLUGIN2=` in the iMOD preference file (**IMOD_INIT.PRF**). With these two keywords you can set the specific file-directory where iMOD can find the plugin(s). You are able to use two different plugin-directories inside iMOD. The iMOD preference file might look like this (See section 9.1 for a more extended description of the preference file):

```
PLUGIN1 "C:\PLUGIN1"
PLUGIN2 "E:\PLUGIN2"
```

2 Create a subfolder with the plugin-name in which you can store all the needed "coupling"-files, including the executable file itself. **Note:** this subfolder should be located in de `PLUGIN1` or `PLUGIN2` folder.

3 An initialization file called **PLUG-IN.INI**, needs to be created that should be placed within the plugin-subfolder. This file can contain the following keywords:

Keyword	Description
TXT=	This keyword is linked to the filename of the text file that contains a short description of the plugin.
CMD=	With this keyword the specific plugin-executable or batch-file is called. You have to make an additional choice whether you prefer to run the plugin in "WAIT" modus or in the background ("NOWAIT") of other iMOD-processes, e.g. CMD=DEMO.EXE NOWAIT
MENU=	This keyword is optional. MENU refers to a file that describes an optional menu to be displayed, before the actual plugin will start, e.g. MENU=D:\PLUGIN1\PLUGIN.MENU. The file contains the visual-settings of the specific plugin window. See description in section section 7.7.1 for the **PLUG-IN.MENU** setup.
HELP=	This keyword is optional and refers to the helpfile related to the plugin itself (PDF or HTML). In case the helpfile is in PDF-format, the keyword *ACROBATREADER* is needed to be defined in the preference file to be able to read the PDF-file.
BACK=	With this keyword you specify a filename that iMOD will check repeatedly (minute time interval), e.g. BACK=PLUG-IN.OUT, see section section 7.7.1 for more detailed information about the content of this file.

Example1

TXT= WATERBALANCE.TXT
CMD= WATERBALANCE.EXE WAIT

Example2

TXT= RASTERCONVERSION.TXT
MENU= RASTERCONVERSION.MENU
HELP= RASTERCONVERSION_MANUAL.PDF
CMD= RASTERCONVERSION.EXE NOWAIT
BACK= RASTERCONVERSION.OUT

The files related to the keywords are described in section section 7.7.1.

4 Now the *Plugin Tool* can be used. For the steps to be taken see section 7.7.2

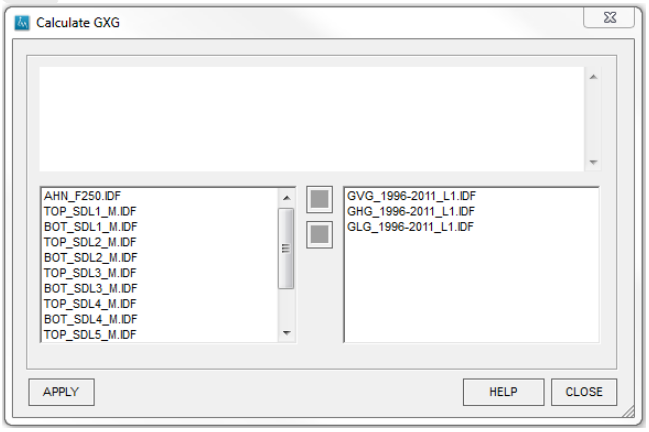
Note: You are able to save the activated plugins for later use within the IMF-file. You can do this by saving the current iMOD project session via the *Save*-button or via *File->Save as...* in the main-menu. Also the chosen time interval (see TMO.PITools.execution how to set this interval) on which iMOD tests whether a plugin is running or not, is saved in the IMF-file.

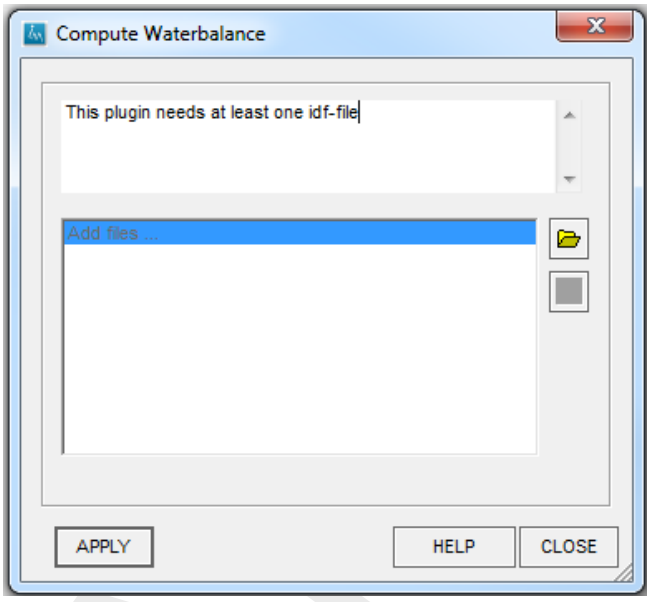
7.7.1 Plugin file description

In the **PLUG-IN.INI** different initialization files are listed behind the above explained keywords. Some of these files also contain a list of keywords. We will explain them here.

7.7.1.1 Plugin MENU file

All keywords are optional.

Keyword	Description
TITLE=	Optional keyword that sets the title of the menu window, e.g. TITLE="This plugin computes residuals".
BUTTON1=	Optional keyword that sets the text on the *Close*-button, e.g. BUTTON1=Afbreken.
BUTTON2=	Optional keyword that sets the text on the *Help*-button, e.g. BUTTON=Hilfe.
BUTTON3=	Optional keyword that sets the text on the *Apply*-button, e.g. BUTTON3=Go-for-it
LIST=	Optional keyword that loads all available files from a certain place or directory. If not defined, iMOD shows a diaolog in which you need to select files yourself. The following options are available: IMODMANAGER Whenever LIST=IMODMANAGER, iMOD will display the selected files from the *iMOD Manager* in a separate window. Be aware that iMOD places a '+'-sign in front of the directory name in case the related file is selected in the *iMOD Manager* and a '-'-sign in case the file is not selected! To be able to read the file properly the plugin needs to handle this. *Example*  Use the buttons *left* and *right* to move the selected items from left to right or the other way around

*.IDF / *.*	Whenever LIST=*.IDF iMOD will display a dialog in which files can be selected, e.g. IDF-files. Any combination of wildcards can be uses, e.g. LIST=*.*, LIST=*.RUN, or whatsoever the plugin need as an input. *Example*  Use the "open"-button () to add files to the dialog.
TEXT=	Refers to an addition text file with explanatory text with requirements for using the chosen plugin, e.g. TEXT=EXPLAINATION.TXT.

The list of files defined with keyword LIST or by making use of the plugin menu-dialog, is stored in a file called "PLUG-IN.IN" (see section 7.7.1.2).

Example1

TITLE= Compute Waterbalance
BUTTON1= Quit
BUTTON2= Info
BUTTON3= Run
LIST= IMODMANAGER
TEXT= EXPLAIN.TXT

Example2

TITLE= Convert Raster settings
LIST= *.IDF
TEXT= EXPLAIN.TXT

7.7.1.2 Plugin IN file

The list of files that is used by the plugin is saved in this file "PLUG-IN.IN", and can serve as a checkup for the user of the plugin. In case LIST="IMODMANAGER" is given in the MENU-file, iMOD puts a '+'- or a '-'-sign in front of each individual file listed in PLUG-IN.IN. The '+'-sign is written when a file is selected in the iMOD manager, else a '-'-sign appears.

7.7.1.3 Plugin OUT file

This file needs to be written by the plugin executable after or during its execution. It needs to be prepared within the code of the plugin. To allow iMOD to interact with the plugin, it is required to include at least one of the following keywords in the OUT file.

Keyword	Description	
WINDOW=	With this keyword the plotting window extent (decimal coordinates) in iMOD can be set. Once this is read, iMOD will adapt the current window extent to the extent mentioned by WINDOW. Use the format: XMIN,YMIN,XMAX,YMAX, e.g. WINDOW=10000.0,250000.0,150000.0,300000.0.	
NFILE=	Number of files to be read into the *iMOD Manager*.	
	FILE{i}=	The i^{th} file to be read into *iMOD Manager*, e.g. FILE1=FLUX.IDF. Given file needs to be available in the specific plugin-(sub)folder. Only name of the (sub-folder+)file+extension are needed to be given. As mentioned the use of subfolders is allowed.
MESSAGE_{...}=	Contains some text lines to be given at a certain moment in de executable process.	
	{INFO}	Contains general information to be given in a popup window at a certain moment during de executable process.
	{ERROR}	Contains error-information to be given in a popup window at a certain moment during de executable process.
	{PROGRESS}	Contains progress-information to be given at a certain moment during de executable process. This will be visible in the information-bar at the bottom of the iMOD main-window.

Note: It is the responsibility of the plugin owner to process the message(s) of the plugin in the right way and at the right moment. After reading the OUT-file, iMOD will delete the file from disc to avoid any double interpretation of the file.

Example1

WINDOW= 140000.0,445000.0,150000.0,452000.0
NFILE= 2
FILE1 = WBAL_2014_SUMMER.IDF
FILE2 = WBAL_2014_WINTER.IDF
MESSAGE_INFO= "The waterbalance is succesfully calculated."

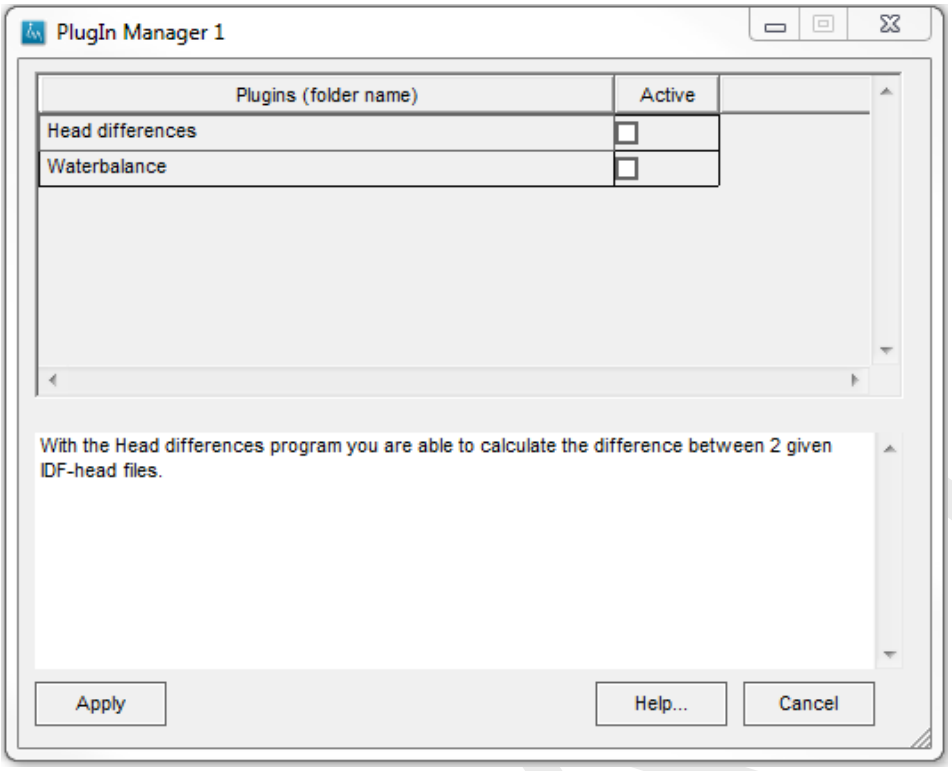
Example2

MESSAGE_PROGRESS= "Plugin is still busy ..."

7.7.2 Using the Plugin

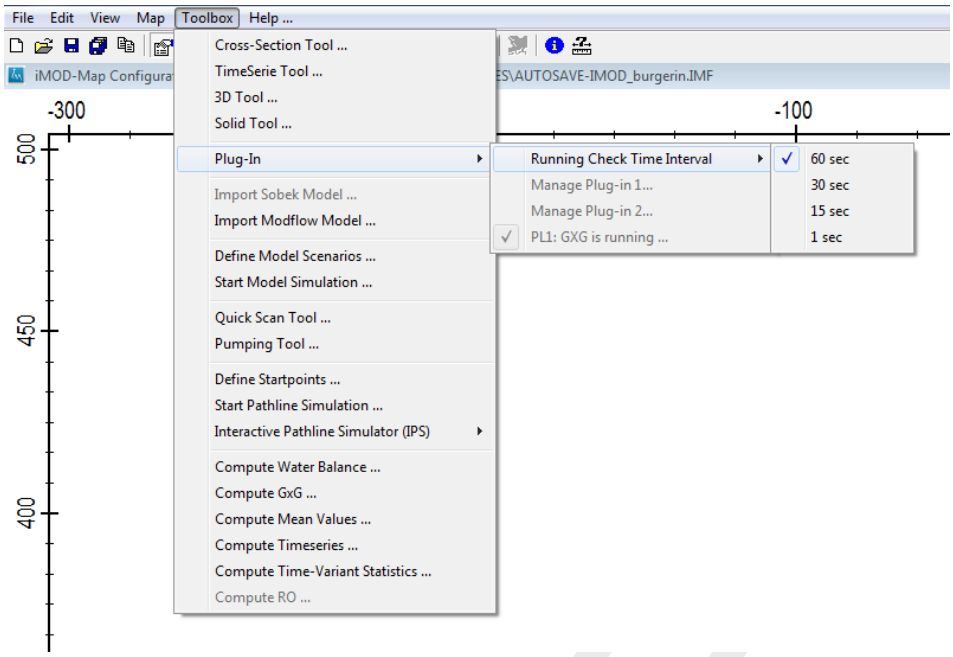
To start the *Plugin Tool* from the main menu, choose *Plug-in*. A sub-menu will appear. If you did define `PLUGIN1` and/or `PLUGIN2` in the iMOD preference file, you can choose *Manage Plug-in 1 ...* and/or *Manage Plug-in 2 ...*. To start the *Plugin Manager* window, click on either of the two.

Example of Plugin Manager window



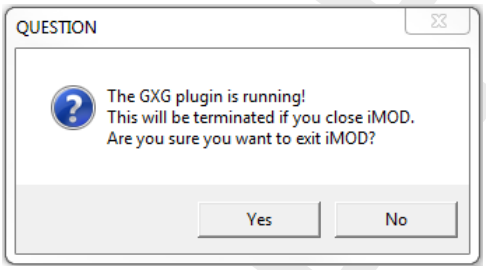
Plugins (folder name)	The list displays all sub-folders in the {*PLUGIN1*}\folder. Click on the sub-folder. If a {PLUGIN}.txt is available in the sub-folder an explanatory text of the plugin will appear in the sub-window below.
Active	Select the check box to (de)activate a specific plugin. When you activate a plugin, the plugin appears in the main sub-menu *Tools* and *Plugin*.
Apply	Click this button to apply your plugin-settings and to close the window.
Cancel	Click this button to close the window and not applying your modified plugin-settings.
Help. . .	Click this button to start the Help functionality.

After clicking on *Apply* button, the activated plugins are added to the Plugin menu. At maximum, 10 plugins can be activated at once into this menu list. The Plugin-menu might look like this:



It depends on the configuration of the Plugin what will happen whenever the plugin is selected from the menu. During the whole running process iMOD checks at predefined fixed moments (intervals of 60, 30, 15 or 1 seconds) if there are plugins running. The running check time interval can be changed by selecting the similar called option in the Plug-In menu (see figure below). On default this interval is set on 60 seconds. As long as a single plugin or multiply plugins are running, the plugin manager(s) cannot be reached (Menu options are grayed-out) for making changes. Though you are able to use already selected plugins from the plugin-menu.

As shown in the example above, only the plugins that are running are grayed-out. If there are still plugins running when trying to close iMOD, iMOD asks the following question for every plugin that is running:



Note: When using a batch file as a plugin to call an executable, this plugin cannot be terminated by iMOD. The plugin continues running after iMOD is closed.

7.8 Import Tools

7.8.1 Import SOBEK Models

WHY?

SOBEK models are surface water models made with the SOBEK suite software available from Deltares. SOBEK models may be imported in iMOD to be used with iMODFLOW.

WHAT?

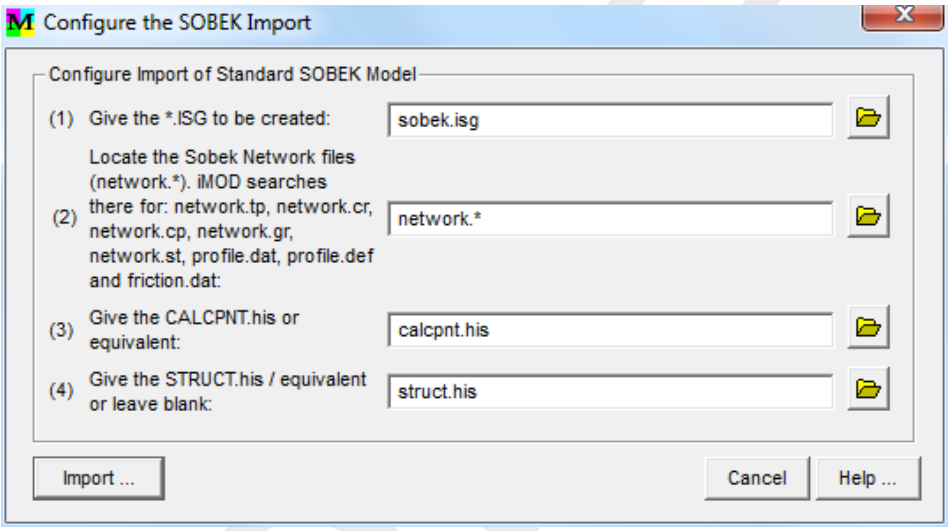
The available SOBEK files are converted to iMOD format and saved as ISG-file. All SOBEK Network, Profile and His-files are converted

Note: This function is only available in the X32-bits version of iMOD, in the X64-bits version of iMOD this functionality is greyed-out. This functionality is supported also by iMOD Batch.

HOW?

Select the option *Toolbox* from the main menu and then choose the option *Import Sobek Model* to start the *Configure the SOBEK Import* window.

Example of Configure the SOBEK Import window:



(1)	Give the *.ISG to be created Enter the name of the ISG-file to be created. All information from the SOBEK files will be saved in one ISG-file. The location of the ISG-file can be chosen with the save button.
(2)	Locate the Sobek Network files iMOD searches for network.tp, network.cr, network.cp, network.gr, network.st, profile.dat, profile.def and friction.dat. Select the name of the network file and iMOD will search all other files in the same folder.
(3)	Give the CALCPNT.his or equivalent Select the HIS file that contains the computed waterlevels at the SOBEK calculation points.
(4)	Give the STRUCT.his / equivalent or leave blank Select the HIS file that contains the computed waterlevels at the SOBEK structures points. Leave this blank when not required or available.
	Open Click this button to select the required file.
Import	Click this button to import the SOBEK configuration into the ISG-file.
Cancel	Click this button to close the window
Help. . .	Click this button to start the Help functionality.

7.8.2 Import Modflow Models

WHY?

MODFLOW *models made outside iMOD may be imported in iMOD to be used with iMODFLOW.*

WHAT?

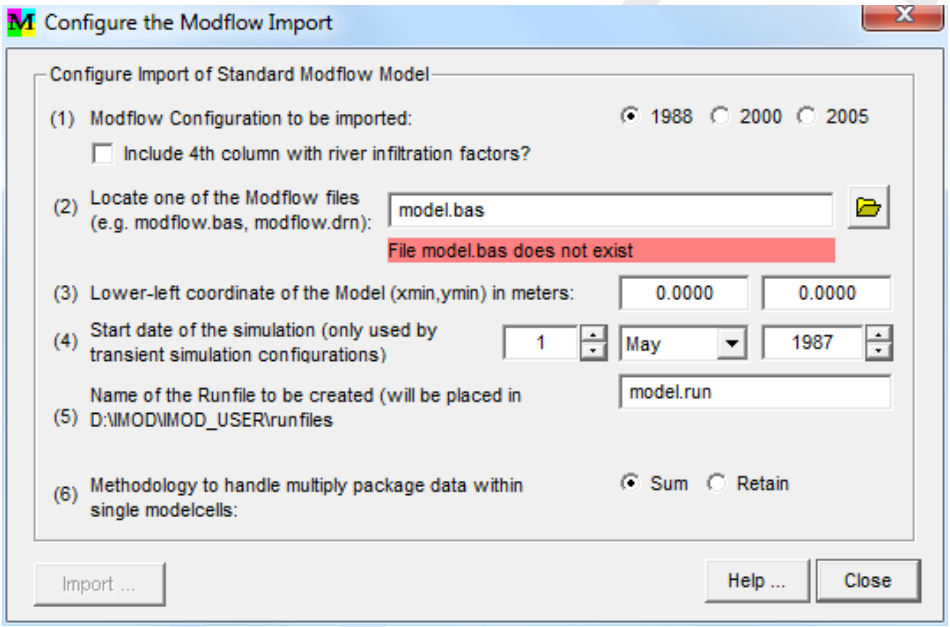
An existing standard MODFLOW configuration is converted into iMOD files (e.g. IDFs, IPFs and GENs). The conversion works for three MODFLOW versions: 1988, 2000 and 2005. iMOD will convert the MODFLOW packages once the location of one of the packages is defined. The conversion will stop in case no BAS-file (1988 version) or NAM-file (2000 and 2005 version) is found.

This functionality is supported also by iMOD Batch.

HOW?

Select the option *Toolbox* from the main menu and then choose the option *Import Modflow Model* to start the *Configure the Modflow Import* window.

Configure the Modflow Import window:



(1)	*Modflow configuration to be imported* Select the year of the Modflow version (1988, 2000, 2005) Check the box at *Include* 4^{th} *column with river infiltration factors* in case the RIV-package has a 4^{th} column defining the ratio in conductance between infiltration and drainage
(2)	*Locate one of the Modflow files (e.g. modflow.bas, modflow.drn)* Select the name of one of the Modflow input files and iMOD will search all other files in the same folder. iMOD will look for a BAS-file for the 1988 version and will look for a NAM-file for the 2000 and 2005 versions. A remark is shown in case the BAS-package or NAM-package input file is not found.
	*Open* Click this button to select the required file as defined in option (1).
(3)	*Lower-left coordinate of the Model (xmin, ymin) in meters:* Enter the X- and Y-coordinate of the lower left corner of the model.
(4)	Start date of the simulation (only used by transient simulation configurations) Enter the date of the first time step by selecting the appropriate day, month and year. This is required for transient models only.

(5)	*Name of the runfile to be created (will be placed in . . . \runfiles)* Enter a name for the runfile to be created in the RUNFILES folder of the iMOD_USER folder). For more information on the runfile see section 7.9.
(6)	*Methodology to handle multiple package data within single modelcells:* Select the appropriate option: ◇ Sum: all existing package information is summed into a single modelcell. This is the default. Whenever more elements occur in a single modelcell, they will be lumped together to form one value. ◇ Retain: all existing package information in a single modelcell is extracted and stored in individual iMOD files.
Import	Import the MODFLOW configuration into iMOD The iMOD model files will be stored in the iMOD_USER\MODELS folder.
Close	Click this button to close the window.
Help. . .	Click this button to start the Help functionality.

7.9 Start Model Simulation

WHY?

A runfile is required for a groundwater flow model that is configured with the iMOD concept. Such a runfile (see chapter 10) consists of a header with general information about the location, horizontal resolution, number of modellayers and solver settings. This type of information can be easily modified in the *Start Model Simulation Tool* described in this section. A runfile can be constructed using the *Project Manager*, see section 5.5. Alternatively, a simulation can be initiated using the *Project Manager* and/or using the iMOD Batch Function **RUNFILE**, see section 8.6.5.

WHAT?

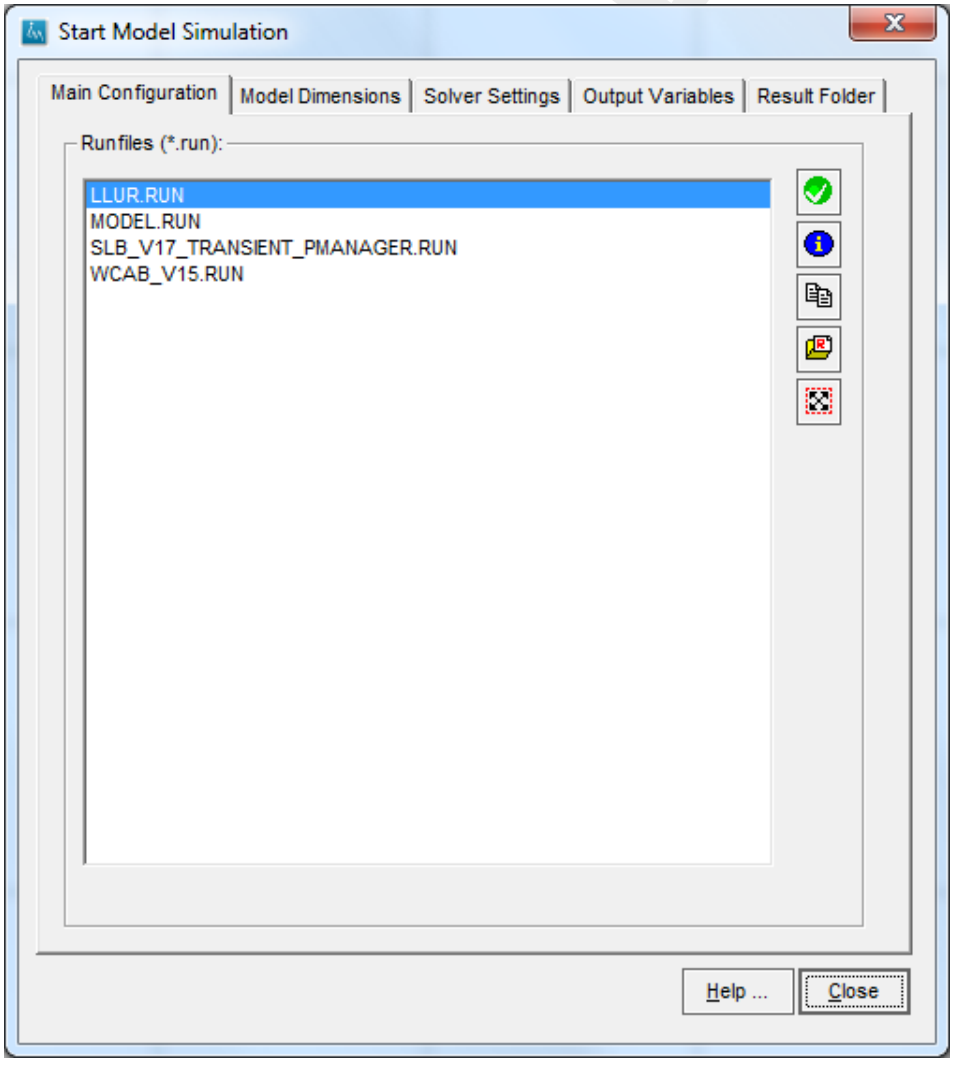
It takes two requirements to start a model simulation within iMOD:

- ◇ Include the keyword MODFLOW in the preference file (section 9.1) and add the appropriate iMOD-FLOW executable;
- ◇ Include at least one runfile in the {user}\runfiles folder.

HOW?

Select the option *Toolbox* from the main menu and then choose the option *Start Model Simulation* to display the *Start Model Simulation* window.

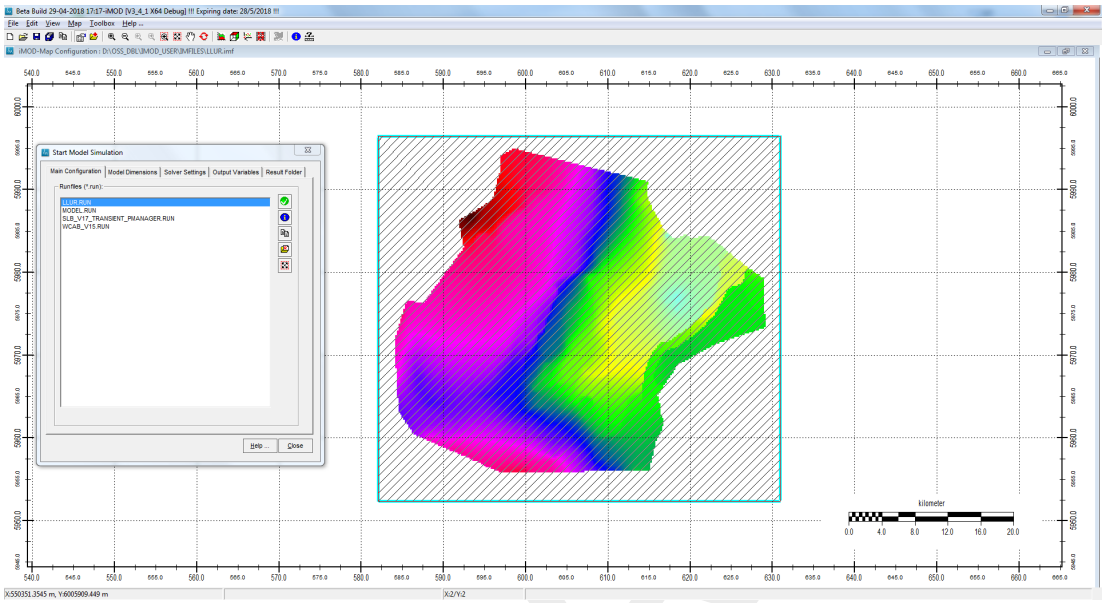
Start Model Simulation window, Main Configuration tab:



Runfiles (.run)*

This list displays the existing runfiles (*.run) in the folder {user}\runfiles (see for more detailed information about runfiles *Vermeulen*, 2011). Whenever a runfile is selected, iMOD will try to read the header information and if no errors are found, the extent of the model (as described by the BNDFILE in the runfile) will be displayed on the graphical display (hatched area).

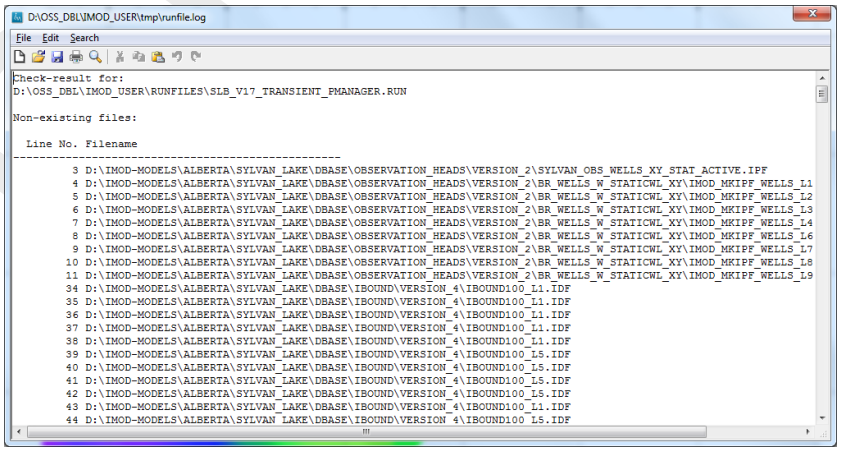
Example of the graphical display:



CheckRun

Click this button to check the existence of all files in the selected runfile. iMOD will check files with the extensions *.IDF, *.IPF, *.ISG and *.GEN as these are valid to be used in a runfile. A list of all missing files are recorded in a file: {user}\tmp\runfile.log.

Example of a runfile.log:



```
D:\OSS_DBL\IMOD_USER\tmp\runfile.log
File  Edit  Search
Check-result for:
D:\OSS_DBL\IMOD_USER\RUNFILES\SLB_V17_TRANSIENT_PMANAGER.RUN

Non-existing files:

  Line No. Filename
-------------------------------------------------
         3 D:\IMOD-MODELS\ALBERTA\SYLVAN_LAKE\DBASE\OBSERVATION_HEADS\VERSION_2\SYLVAN_OBS_WELLS_XY_STAT_ACTIVE.IPF
         4 D:\IMOD-MODELS\ALBERTA\SYLVAN_LAKE\DBASE\OBSERVATION_HEADS\VERSION_2\BR_WELLS_W_STATICWL_XY\IMOD_MKIPF_WELLS_L1
         5 D:\IMOD-MODELS\ALBERTA\SYLVAN_LAKE\DBASE\OBSERVATION_HEADS\VERSION_2\BR_WELLS_W_STATICWL_XY\IMOD_MKIPF_WELLS_L2
         6 D:\IMOD-MODELS\ALBERTA\SYLVAN_LAKE\DBASE\OBSERVATION_HEADS\VERSION_2\BR_WELLS_W_STATICWL_XY\IMOD_MKIPF_WELLS_L3
         7 D:\IMOD-MODELS\ALBERTA\SYLVAN_LAKE\DBASE\OBSERVATION_HEADS\VERSION_2\BR_WELLS_W_STATICWL_XY\IMOD_MKIPF_WELLS_L4
         8 D:\IMOD-MODELS\ALBERTA\SYLVAN_LAKE\DBASE\OBSERVATION_HEADS\VERSION_2\BR_WELLS_W_STATICWL_XY\IMOD_MKIPF_WELLS_L6
         9 D:\IMOD-MODELS\ALBERTA\SYLVAN_LAKE\DBASE\OBSERVATION_HEADS\VERSION_2\BR_WELLS_W_STATICWL_XY\IMOD_MKIPF_WELLS_L7
        10 D:\IMOD-MODELS\ALBERTA\SYLVAN_LAKE\DBASE\OBSERVATION_HEADS\VERSION_2\BR_WELLS_W_STATICWL_XY\IMOD_MKIPF_WELLS_L8
        11 D:\IMOD-MODELS\ALBERTA\SYLVAN_LAKE\DBASE\OBSERVATION_HEADS\VERSION_2\BR_WELLS_W_STATICWL_XY\IMOD_MKIPF_WELLS_L9
        34 D:\IMOD-MODELS\ALBERTA\SYLVAN_LAKE\DBASE\IBOUND\VERSION_4\IBOUND100_L1.IDF
        35 D:\IMOD-MODELS\ALBERTA\SYLVAN_LAKE\DBASE\IBOUND\VERSION_4\IBOUND100_L1.IDF
        36 D:\IMOD-MODELS\ALBERTA\SYLVAN_LAKE\DBASE\IBOUND\VERSION_4\IBOUND100_L1.IDF
        37 D:\IMOD-MODELS\ALBERTA\SYLVAN_LAKE\DBASE\IBOUND\VERSION_4\IBOUND100_L1.IDF
        38 D:\IMOD-MODELS\ALBERTA\SYLVAN_LAKE\DBASE\IBOUND\VERSION_4\IBOUND100_L1.IDF
        39 D:\IMOD-MODELS\ALBERTA\SYLVAN_LAKE\DBASE\IBOUND\VERSION_4\IBOUND100_L5.IDF
        40 D:\IMOD-MODELS\ALBERTA\SYLVAN_LAKE\DBASE\IBOUND\VERSION_4\IBOUND100_L5.IDF
        41 D:\IMOD-MODELS\ALBERTA\SYLVAN_LAKE\DBASE\IBOUND\VERSION_4\IBOUND100_L5.IDF
        42 D:\IMOD-MODELS\ALBERTA\SYLVAN_LAKE\DBASE\IBOUND\VERSION_4\IBOUND100_L5.IDF
        43 D:\IMOD-MODELS\ALBERTA\SYLVAN_LAKE\DBASE\IBOUND\VERSION_4\IBOUND100_L1.IDF
        44 D:\IMOD-MODELS\ALBERTA\SYLVAN_LAKE\DBASE\IBOUND\VERSION_4\IBOUND100_L5.IDF
```

Info

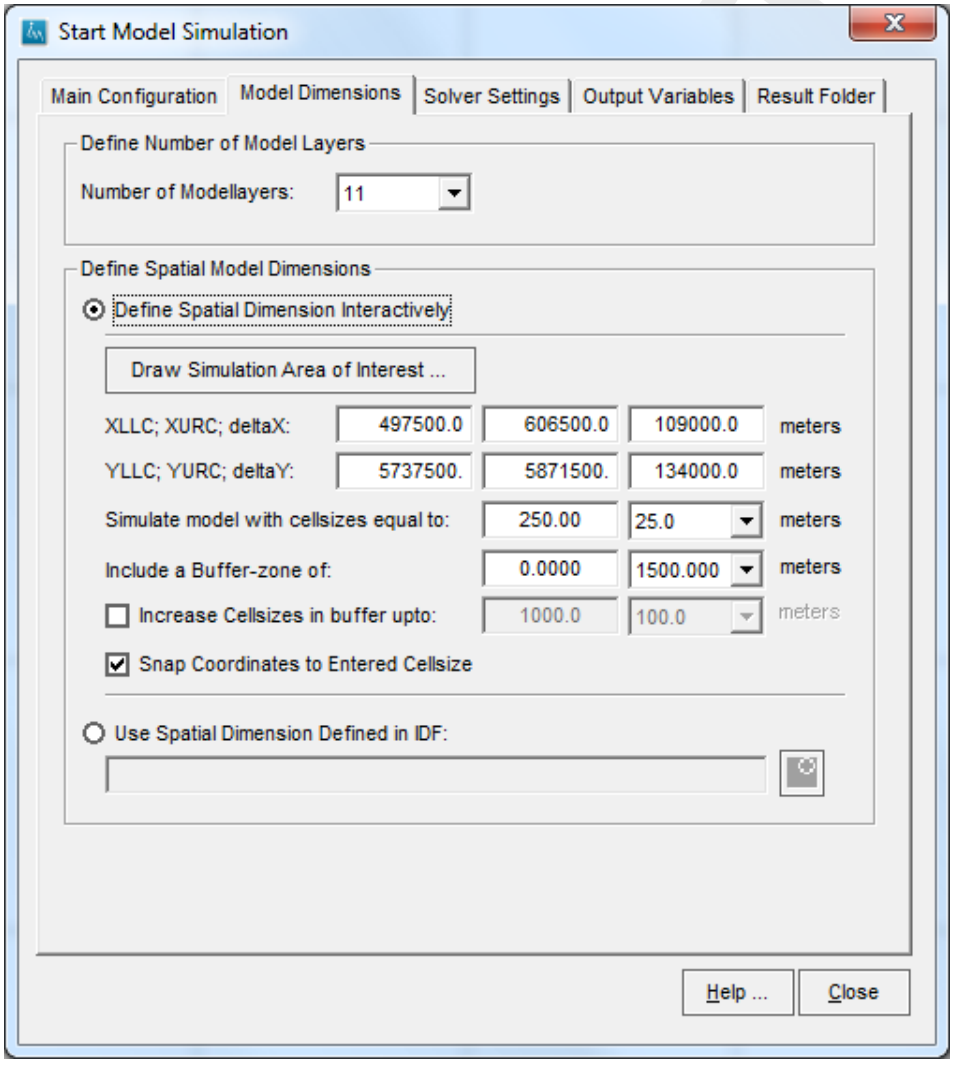
Click this button to display the content of the selected runfile (*.RUN) in a texteditor.

RunfileCopy

Click this button to make a complete copy of the content of the selected runfile for the current window. All IDF and IPF files that can be found in the runfile will be clipped to this window.

	*Project Manager* Click this button to display the *Project Manager* (section 5.5) and read in the selected runfile. This might be handy as different runfiles can be generated from the *Project Manager* as well as steering a runfile from the iMOD Batch Function **RUNFILE** (section 8.6.5) is more efficient for simulations that need to be carried repeatedly.
	*ZoomFull* Click this icon to zoom to the entire extent of the model.
Help . . .	Click this button to start the HELP functionality.
Close	Click this button to close the *Start Model Simulation* window.

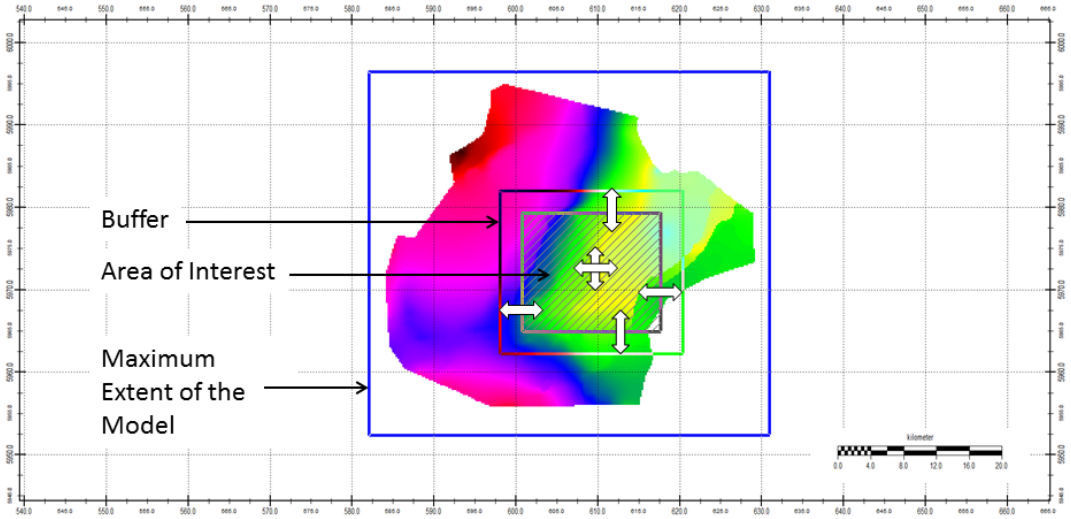
Start Model Simulation window, Model Dimensions tab:



Number of Modellayers	Select the number of modellayers to be used in the current model simulation. Whenever the *Number of Modellayers* is less than the MXNLAY variable in the run-file, iMOD will display a message at the right of the dropdown menu, indicating that the boundary condition of the lowest modellayer will be a Constant Head boundary condition.
Define Spatial Dimensions Interactively	Select this option to determine the dimension and size of the rastercells (computational nodes) interactively.

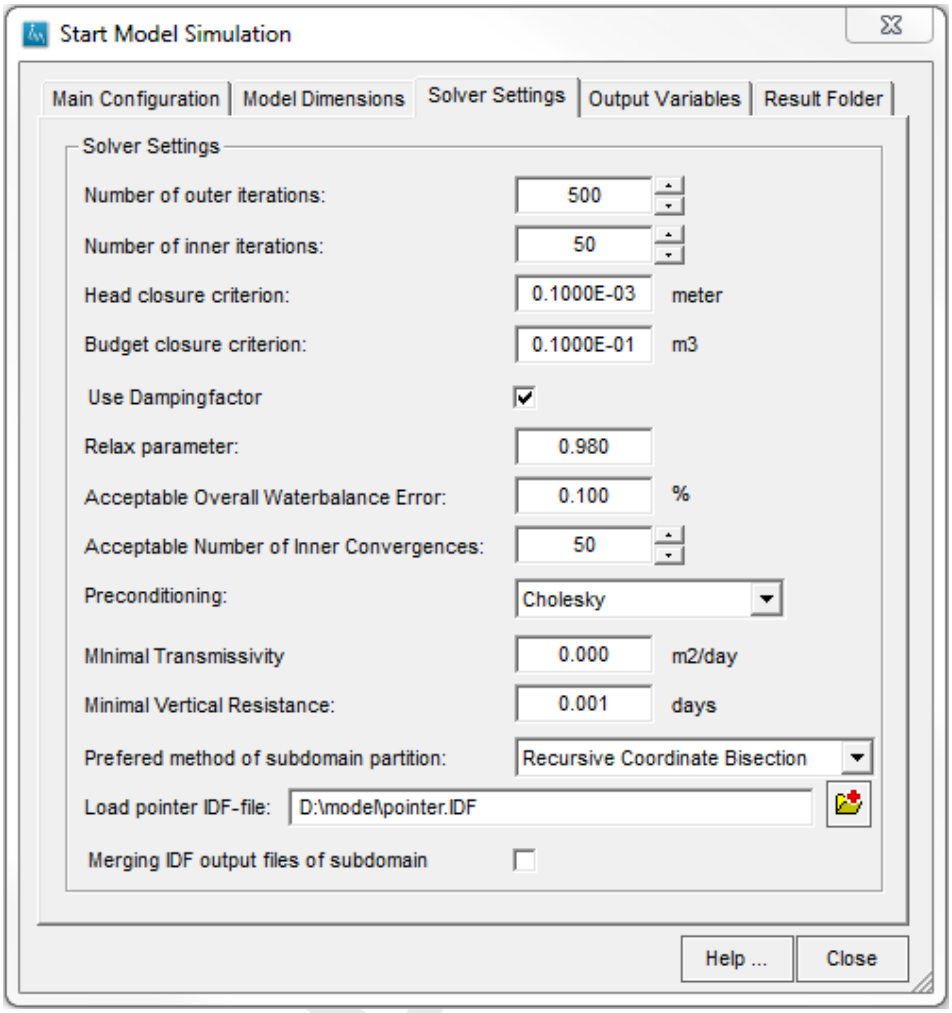
Draw Simulation Area of Interest	Click this button to start drawing a rectangle on the graphical display to indicate the location of the simulation area (hatched area). Use your left-mouse button to position the first points of the rectangle, use the left/right-mouse button to identify the opposite border. Whenever you move the mouse cursor inside the hatched area, a cross-arrow appears indicating that the entire hatched area can be moved while clicking the left-mouse button. Similar the borders can be moved whenever the horizontal/vertical arrows appear.

Example of a model extent after drawn interactively on the graphical display:



XULC,XURC, DeltaX	Enter the X-coordinate for the lower-left-corner *(XULC)* and the upper-right-corner *(XURC)*. The difference between them will be computed automatically (DeltaX). All variables need to be entered in meters, moreover, they will be filled in automatically whenever the simulation area is adjusted interactively on the graphical display.
YULC,YURC, DeltaY	Enter the Y-coordinate for the lower-left-corner *(YULC)* and the upper-right-corner *(YURC)*. The difference between them will be computed automatically (DeltaY). All variables need to be entered in meters, moreover, they will be filled in automatically whenever the simulation area is adjusted interactively on the graphical display.
Simulate model with cellsizes equal to	Select one of the cellsize from the dropdown menu or enter a value in the input field next to it.
Include a Buffer-zone of:	Select one of the buffersizes from the dropdown menu or enter a value in the input field next to it. The *Buffer-zone* is an extra “ring” of modelcells around the chosen simulation area and indicated by a green rectangle.
Increase Cellsize in buffer upto:	Select this option to increase the cellsizes in the *Buffer-zone* upto a cellsize that can be selected from the dropdown menu or entered in the input field next to it.
Snap Coordinates to Entered Cellsize	Select this option to snap the coordinates of the model domain to the entered cellsizes. In this manner, the coordinates of the model will have ”nice-and-round” coordinates.
Use Spatial Dimensions Defined in IDF:	Select this option whenever an (ir)regular network needs to be used that is described in the header information of an IDF that can be entered in the corresponding input field.
	*Open IDF* Select this button to select an IDF-file from the system.

Start Model Simulation window, Solver Settings tab:

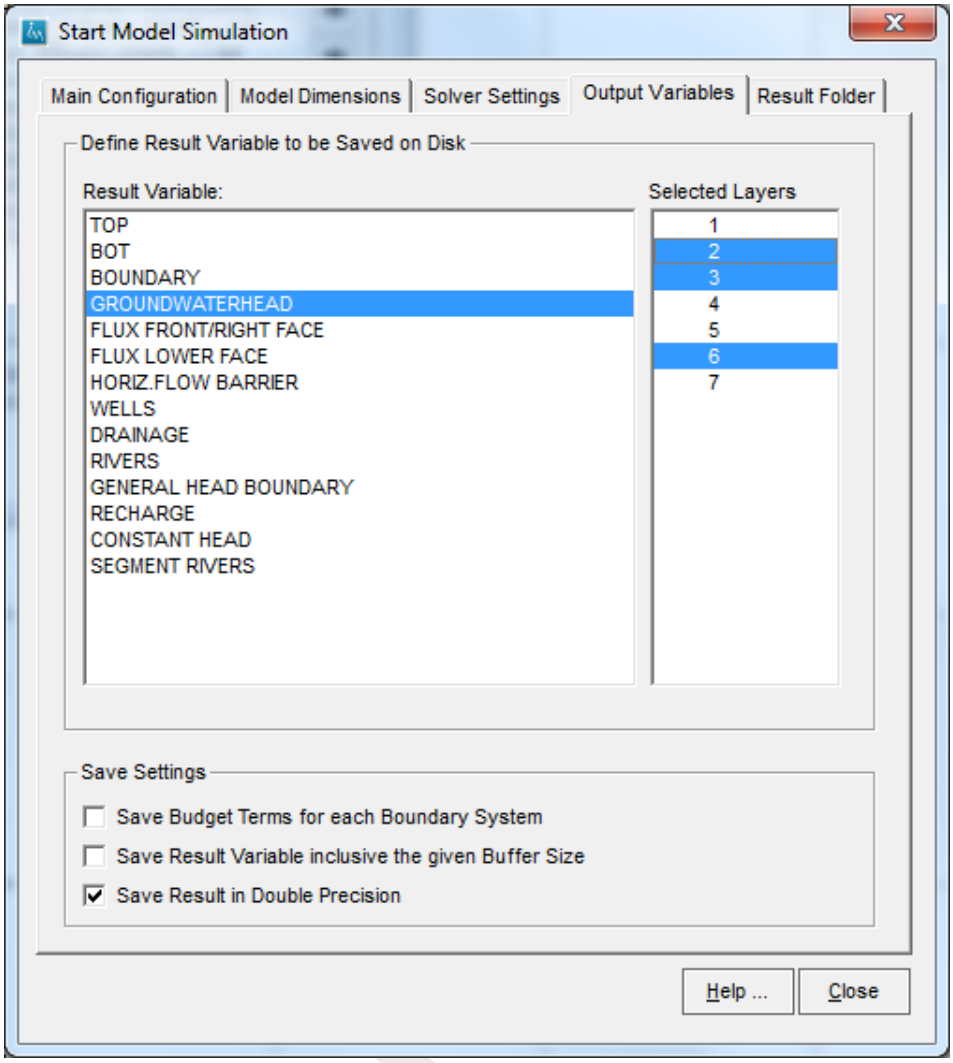


Number of outer iterations:	Enter the maximum number of outer iterations.
Number of inner iterations:	Enter the maximum number of inner iterations.
Head closure criterion:	Enter the maximum head residual in meters.
Budget closure criterion:	Enter the maximum budget residual in cubic meters.
Use Dampingfactor	Check this option to include adaptive damping. The relaxation factor (RELAX) will be adjusted according to Cooley's method with Huyakorn's modification. This will increase convergence for a nonlinear model.
Relax parameter:	Enter the relaxation parameter ($0.0 < relax < 1.0$)
Acceptable Overall Waterbalance Error	Enter the overall acceptable waterbalance error in percentage of the water-balance error (Q_{in}-Q_{out}) divided by the mean of the absolute total of both: 0.5*(Q_{in}+Q_{out}). Whenever the simulation can not find a solution within the number of *outer* * *inner* iterations, it can still continue whenever it meets the *Acceptable Overall Waterbalance Error*.
Acceptable Number of Inner Convergences	Enter the number of sequential inner convergences that forces the solver to stop further iteration. The simulation will continue whenever the result passes the *Acceptable Overall Waterbalance Error* too.

Preconditioning:	Choose the preferred pre-conditioning method (only possible when using the PCG-solver): 1. Modified Incomplete Cholesky (for use on scalar computers), 2. Polynomial matrix conditioning method (for use on vector computers or to conserve computer memory). If the Preconditioning Method is set to Cholesky, the Relaxation parameter can be set. Although the default is 1, in some cases a value of 0.97-0.99 may reduce the number of iterations required for convergence.
Minimal Transmissivity:	Enter a value for the minimal transmissivity that may occur in the model. In case a grid value is found below this threshold this will be corrected before the modelsimulation is started.
Minimal Vertical Resistance:	Enter a value for the minimal vertical resisitance that may occur in the model. In case a grid value is found below this threshold this will be corrected before the modelsimulation is started
Preferred method of subdomain partition:	Choose the preferred subdomain partition option. There are two methods supported: 1. Uniform subdomain partitioning (default) for uniform partitioning in lateral x and y-direction, 2. Recursive Coordinate Bisection (RCB) subdomain partitioning that computes the subdomain dimensions according to be specified pointer IDF grid (selecting a pointer IDF-file is required with this option). By selecting *None* the PKS package will not be used during the simulation. **Note:** Each subdomain always includes all model layers.
Load pointer IDF-file:	In case hte RCB option is chosen, the filename of the needed pointer IDF grid needs to be filled in here.
	*Open* Click this button to find and select an IDF-file with the needed PKS pointer information.
Merging IDF output files of subdomain	Check this option if merging of parallel subdomain IDF output files is needed (see for more information section 10.6). **Note:** Enabling this option could slow down overall parallel computations.

Note: Consult scientific literature regarding the Solver Settings as described above in order to avoid any unwise input.

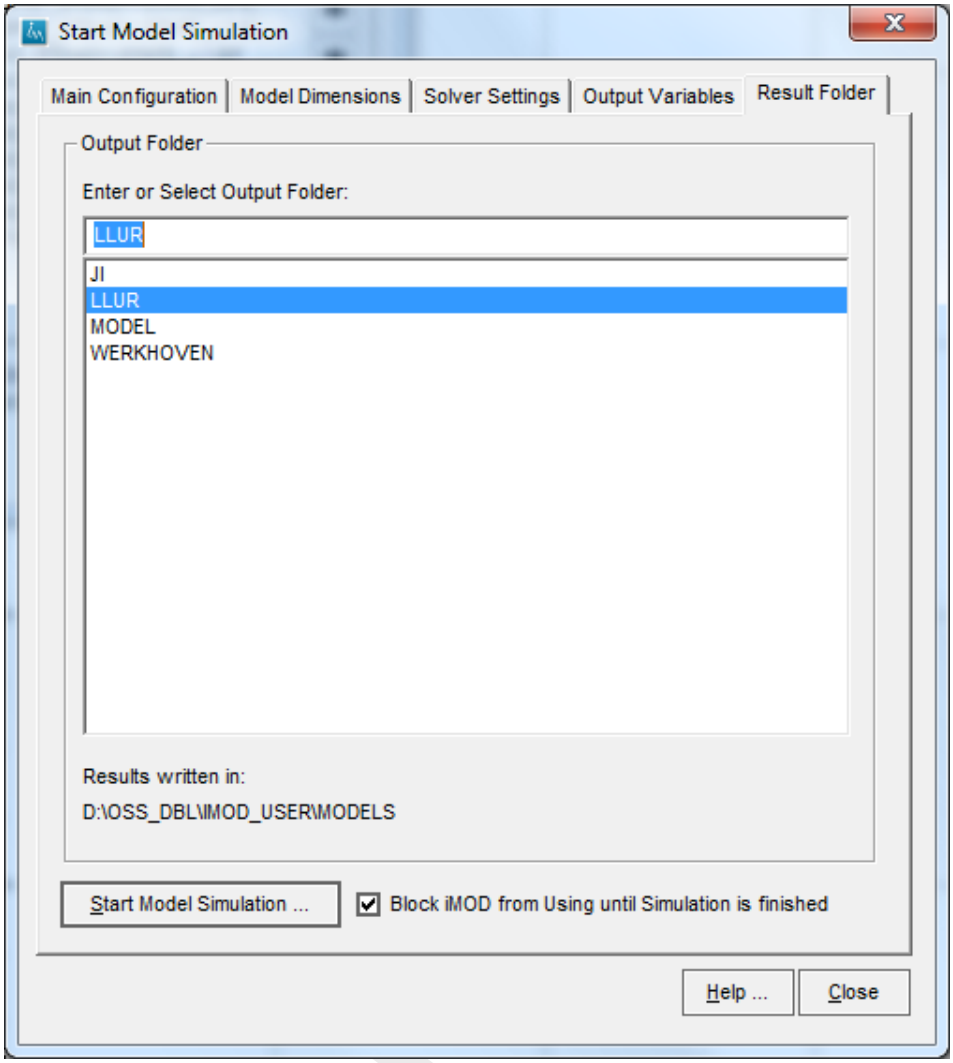
Start Model Simulation window, Output Variables tab:



Result Variable	This list will display the available output variables that are within the selected runfile from the Main Configuration tab. Select one of the following:	
File (*.idf)	***Variable***	***Description***
	SIMGRO	Flux in/out Simgro elements
bdgbnd	BOUNDARY	Flux in/out constant head boundaries
head	GROUNDWATERHEAD	Groundwater head
bdgfff/bdgfrf	FLUX FRONT/RIGHT FACE	Flux in/out front/right cell faces
bdgflf	FLUX LOWER FACE	Flux in/out bottom cell face
bdgsto	STORAGE	Flux in/out storage
	PURGED WATER TABLE	Absent
	ANISOTROPY	Absent
	HORIZ.FLOW BARRIER	Absent
	TOP	Absent
	BOT	Absent
	CONCENTRATION	Absent
	HORIZ.K VALUE	Absent
	VERT.K VALUE	Absent
bdgwel	WELLS	Flux in/out well systems
bdgdrn	DRAINAGE	Flux out drainage systems
bdgriv	RIVERS	Flux in/out river systems
bdgevt	EVAPOTRANSPIRATION	Flux out evapotranspiration

bdgghb	GENERAL HEAD BOUNDARY	Flux in/out general head boundaries
bdgrch	RECHARGE	Flux in recharge
bdgolf	OVERLAND FLOW	Flux out overland flow
bdgbnd	CONSTANT HEAD	Flux in/out constant head boundaries (identical to BOUNDARY)
bdgisg	SEGMENT RIVERS	Flux in/out river systems
bdgibs	INTERBED STORAGE	Flux in/out interbeds
Selected Layers	Select the modellayers for which the current selected variable need to be saved. The number of modellayers to choose from is determined by the Number of Modellayers selected in the *Model Dimensions* tab.	
SaveBudget Terms for each Boundary System	Select this item to save budget terms for each of the defined sub-systems in the selected runfile. Each subsystem will be added to the filename, e.g. bdgriv_steady-state_l1_sys1.idf.	
Saved Result Variable inclusive the given Buffer Size	Select this item to save the results within the specified buffer size entered in the *Include a Buffer-zone of* field on the *Model Dimensions* tab.	
Save Results in Double Precision	Select this option to save all results in double precision accuracy instead of single precision (which is the default). Bear in mind that all files will be doubled in size as well as the option for double precision is selected. Also, result files saved in double precision cannot be read in iMOD version older than v3.4.	

Start Model Simulation window, Result Folder tab:



Enter or Select Output Folder	The selected variables, as specified in the Output Variables tab, will be saved in the folder entered/selected here. Each variable will be saved within a separate folder, e.g. {outputfolder}\bdgrch\bdgrch_steady-state_l1.idf. Whenever NO scenario is included on the Main Configuration tab, the results will be saved in the {user}\models folder. Otherwise, in case a scenario is included, the results will be saved in {user}\scenarios\{scenarioname}. In this latter case, the output folder is determined by the selected scenario and can not be changed.
Start Model Simulation . . .	Click this button to start a model simulation. iMOD will ask you to confirm, before the actual simulation starts. Whenever the model dimensions (number of rows * columns * modellayers) is more than 30.000.000 nodes, this button will be inactivated, since a normal 32-bits operating system can not carry out simulations with higher dimensions.

7.10 Quick Scan Tool

WHY?

The effects of changes in hydrogeological conditions in an area can be visualized once a groundwater model has been created. This can be done by changing the model input (as described in section 7.9) or by using scenarios (currently not available!). However much run time may be needed in case of complicated large groundwater models while many variations of the model input are required to select the desired changes in the model input. The Quick Scan Tool is designed to reduce significantly this run time. The Quick Scan Tool will provide an approximated result indicating the effect of the model variation and enabling to choose the desired model input. After selecting the model input a single final model run is needed to obtain the detailed model output.

WHAT?

The Quick Scan Tool works with a database (the Quick Scan database or QS-database, in previous iMOD versions called the Impulse-Response database or IR-database) which contains the model output of many model runs made by varying combinations of model input. The result of a change in model input can be presented very quickly just by querying this QS-database.

7.10.1 Initial Settings

The Quick Scan Tool needs an input file (*.INI) with initial settings. The input file is defined in the *.PRF file using the keyword QSTOOL (not yet implemented) or IRDBASE.

The Quick Scan Tool input file (*.INI) needs to contain the following parameters:

NIDF	Number of IDF-files used as reference file.
IDFNAME#	Name used in the QS Tool for IDF number #
IDFFILE#	Name of the IDF file used as IDF number #
RESDIR	Name of the folder with the results of the QS Tool
NIR	Number of QS databases used in the QS Tool
DIRIR#	Name of the folder with QS database number #
NAMEIR#	Name used in the QS Tool for QS database number #
IMPIR#	Name of the Impulse used in the QS database
MINIR#	Minimal impulse value
MAXIR#	Maximal impulse value
IDFIR#	IDF with the link to the REFIR#
REFIR#	Name of the file linking the IDFIR# value with the Response data

7.10.2 Start Quick Scan Tool

HOW?

Select the *Toolbox* option from the main menu and then choose *Quick Scan Tool* to start the *Quick Scan Tool* window.

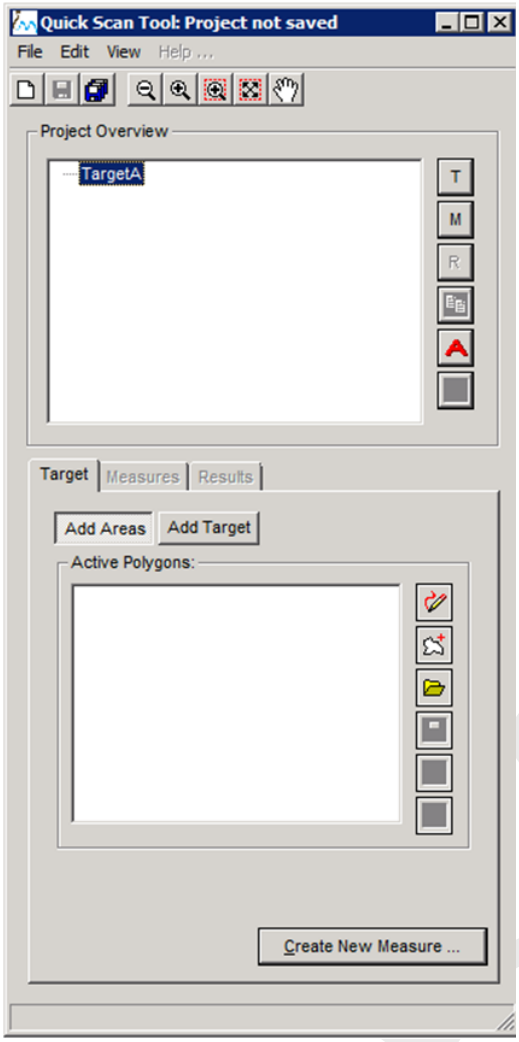
Define Name of Current Target window:

Enter a name for the target and click the *Continue* button. The *Quick Scan Tool* window will appear with the name of the target in the *Project Overview* list.

The *Project Overview* list contains all defined *Targets (T)*, *Measures (M)* and *Results (R)*.

The *Active Polygons* list contains all defined polygons. The targets and measures are defined in the tabs in the bottom half of the window by defining the area and the target or measure values.

Quick Scan Tool window:



A target (Target tab) and a measure (Measure tab) is defined by:

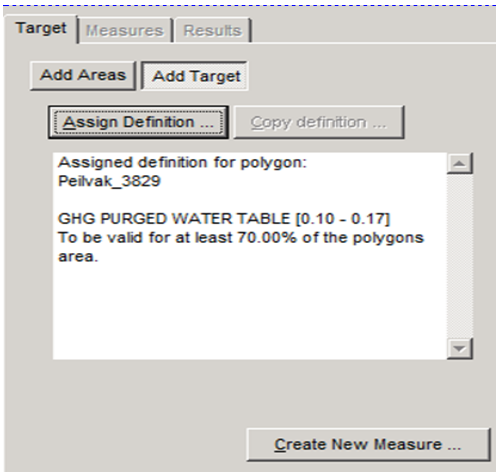
Click on the *Add Areas* button:

	*Draw* Draw a polygon to define the area where the target/measure is valid.
	*Select* Select a surface water level area from the map with surface water level areas. This map is shown in the background.

Click on the *Add Target* or*Add Measure* button: a target/measure is linked to the area defined. Information on the targets/measures linked to a polygon is shown in the bottom half of the *Quick Scan Tool* window when selecting the polygon on the map.

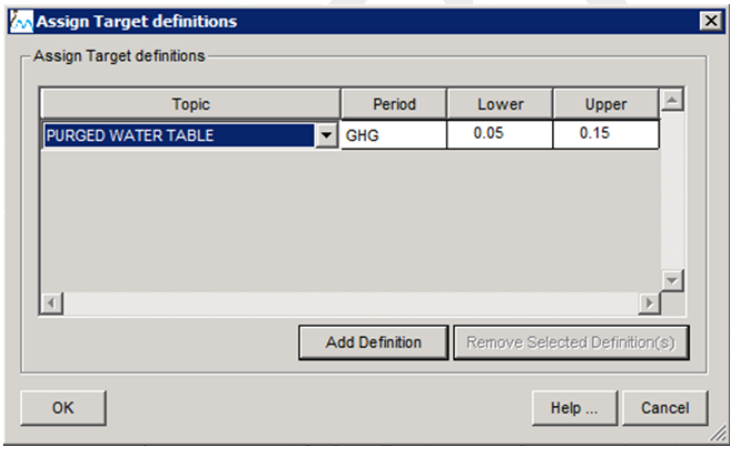
Quick Scan Tool window, bottom half for selected polygon:

for Targets tab:



Assign Definition	Click on the *Assign Definition* button to add the target definition. The *Assign Target Definition* window appears.
Copy Definition	Click on the *Copy Definition* button to copy target definitions from one polygon to other polygons.
Create New Measure	Click on the *Create New Measure* button to make a new measure definition linked to the target.

Assign Target Definition window:

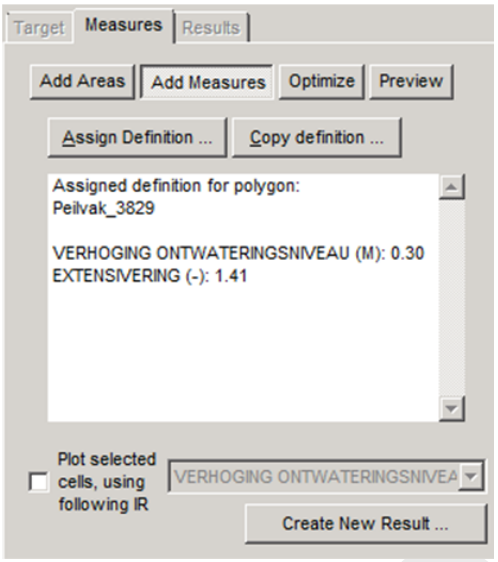


Topic	List of variables for which a target may be defined: ◇ Purged water table: level of the purged groundwater above the confining layer; ◇ Groundwater table: level of the groundwater under the confining layer; ◇ Seepage fluxes: seepage-/infiltration flux from modellayer 1 to modellayer 2; ◇ River fluxes: recharge/discharge of surface water; ◇ Drainage fluxes: discharge by drainage tubes and ditches.
Period	Period or condition for which the target is valid: GHG (average highest groundwater level) or GLG (average lowest groundwater level)

Lower	Lower limit of the target
Upper	Upper limit of the target
Add Definition	Add target definition to the list of targets
Remove Selected Definition(s)	Remove the selected target definitions from the list of targets

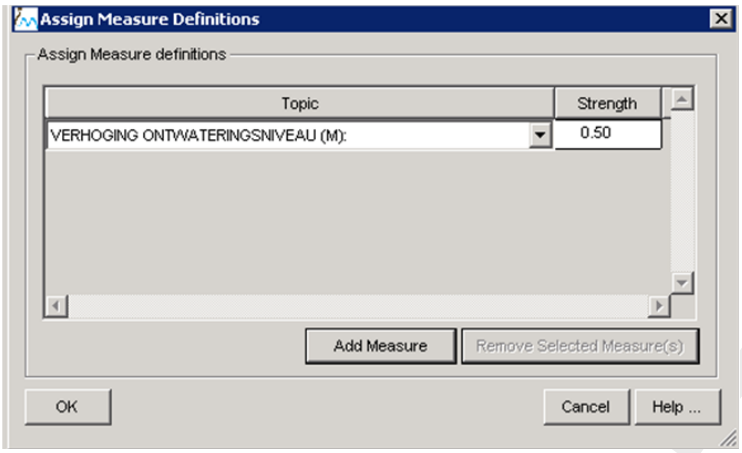
Quick Scan Tool window, bottom half for selected polygon:

for Measures tab, Add Measures button:



Assign Definition	Click on the *Assign Definition* button to add the measure definition. The *Assign Measure Definition* window appears.
Copy Definition	Click on the *Copy Definition* button to copy measure definitions from one polygon to other polygons.
Plot selected cells, using following IR	Check the box to show the QS(=IR) units (model cell clusters) for which the effects of measures are calculated. The measure should be selected in the listbox because each measure may be linked to different units.
Create New Result	Click on the Create New Result button to save the set of measures to a result.

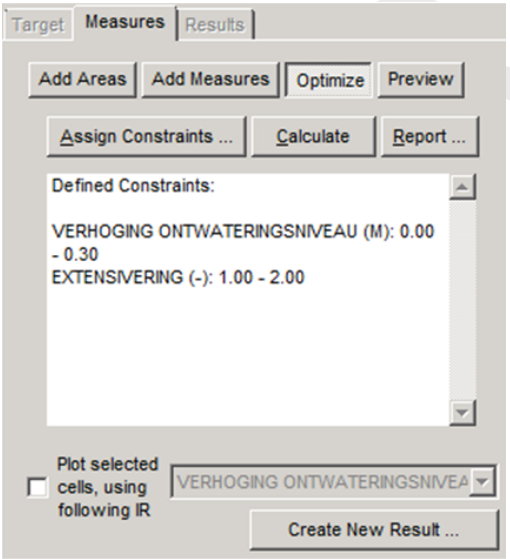
Assign Measure Definition window:



Topic	Name of the measure
Strength	Strength of the measure
Add Measure	Add measure definition to the list of measures
Remove Selected Measure(s)	Remove the selected measure definitions from the list of measure

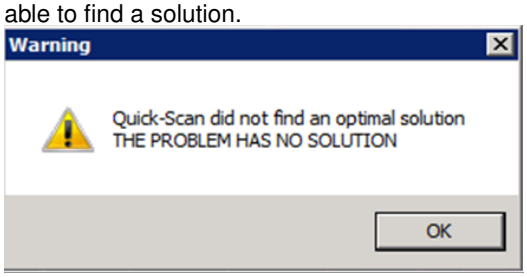
Quick Scan Tool window, bottom half for selected polygon:

for Measures tab, Optimize button:

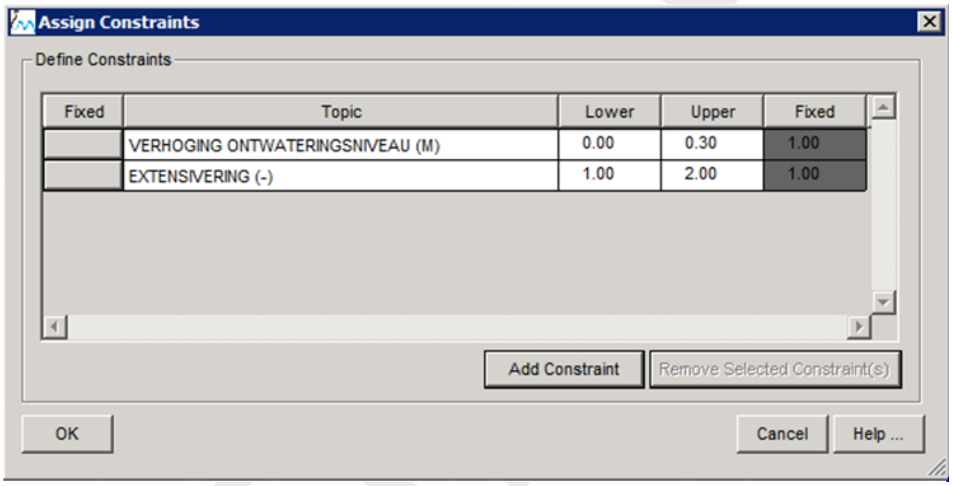


The *Optimize* function gives the opportunity to search in the QS-database for the optimal set of measures. These are the measures which give the desired result with the minimum number of measures.

Assign Constraints	Click on the *Assign Constraints* button to open the *Assign Constraints* window.

Calculate	Click on the *Calculate* button to search for the optimal set of measures using the assigned constraints. A report (see below) is generated in case the QS-database finds a solution. A warning is shown in case the QS-database is not able to find a solution. 
Report	Show the report generated by the *Calculate* function
Plot selected cells, using following IR	Check the box to show the QS(=IR) units (model cell clusters) for which the effects of measures are calculated. The measure should be selected in the listbox because each measure may be linked to different units.
Create New Result	Click on the*Create New Result* buttonto save the set of measures to a result.

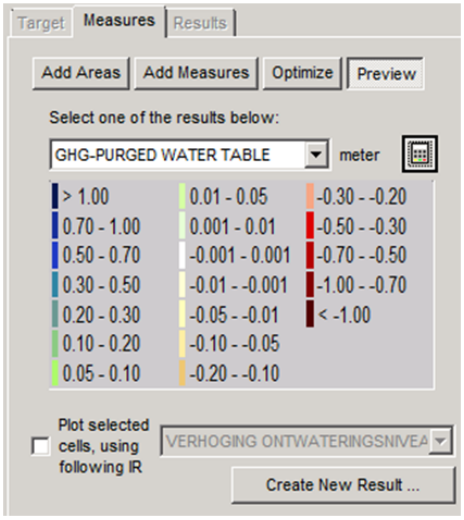
Assign Constraints window:



Fixed	Click on the button to set a fixed constraint. The value of the measure is fixed and can not be changed.
Topic	Name of the measure
Lower	Lower limit of the constraint
Upper	Upper limit of the constraint
Fixed	Fixed value of the constraint (usable when button on first column is clicked)
Add Constraint	Click on the *Add Constraint* button to add a constraint to the list of measures.
Remove Selected Constraint(s)	Remove the selected constraints from the list of constraints
OK	The constraint values are checked to be within the limits of the QS-database and the window closes.

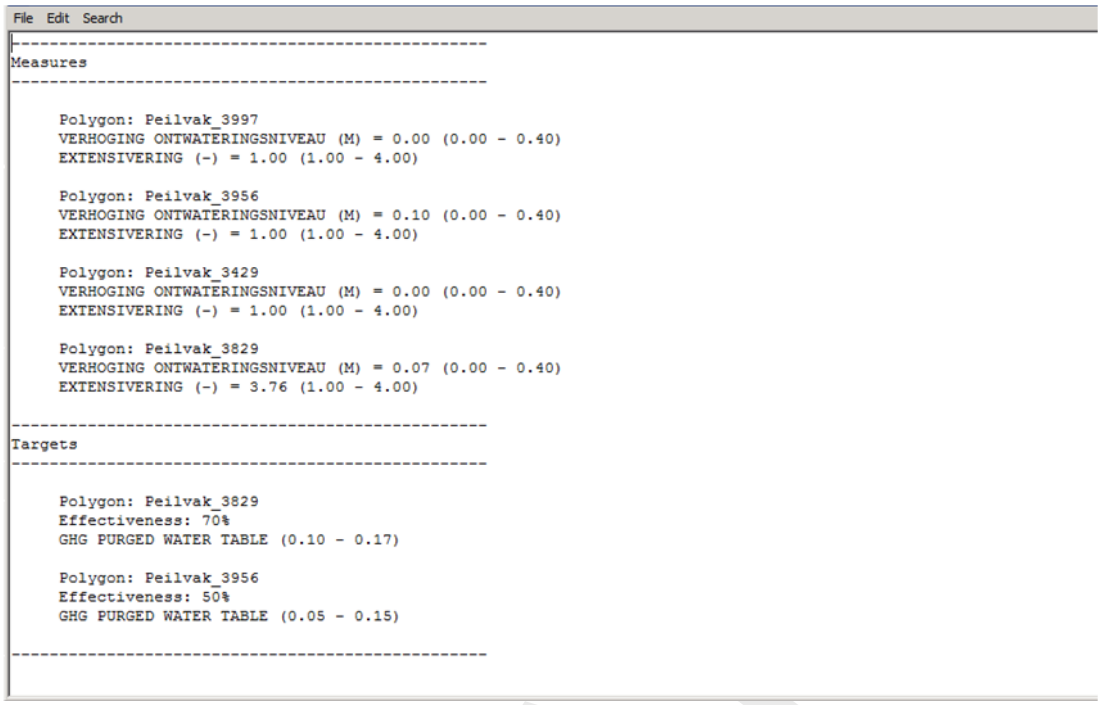
Quick Scan Tool window, bottom half for selected polygon:

for Measures tab, Preview button:



Select one of the results below:	Select the result from the list box. The effect of the measure is shown for the area selected under *Add Areas*.
	*Show the result* Click on the button to show the result on the map. The legend is shown below the list box.
Plot selected cells, using following IR	Check the box to show the QS(=IR) units (model cell clusters) for which the effects of measures are calculated. The measure should be selected in the listbox because each measure may be linked to different units.
Create New Result	Click on the*Create New Result* buttonto save the set of measures to a result.

Example of report generated by Calculate on the Measures tab, Optimize button:



```
File  Edit  Search
------------------------------------------------
Measures
------------------------------------------------

     Polygon: Peilvak_3997
     VERHOGING ONTWATERINGSNIVEAU (M) = 0.00 (0.00 - 0.40)
     EXTENSIVERING (-) = 1.00 (1.00 - 4.00)

     Polygon: Peilvak_3956
     VERHOGING ONTWATERINGSNIVEAU (M) = 0.10 (0.00 - 0.40)
     EXTENSIVERING (-) = 1.00 (1.00 - 4.00)

     Polygon: Peilvak_3429
     VERHOGING ONTWATERINGSNIVEAU (M) = 0.00 (0.00 - 0.40)
     EXTENSIVERING (-) = 1.00 (1.00 - 4.00)

     Polygon: Peilvak_3829
     VERHOGING ONTWATERINGSNIVEAU (M) = 0.07 (0.00 - 0.40)
     EXTENSIVERING (-) = 3.76 (1.00 - 4.00)

------------------------------------------------
Targets
------------------------------------------------

     Polygon: Peilvak_3829
     Effectiveness: 70%
     GHG PURGED WATER TABLE (0.10 - 0.17)

     Polygon: Peilvak_3956
     Effectiveness: 50%
     GHG PURGED WATER TABLE (0.05 - 0.15)

------------------------------------------------
```

Measures	The proposed measures are listed by polygon. The constraints are shown between brackets.
Targets	The desired targets are listed by polygon. The upper and lower limits of the target are shown between brackets.
Exit	The report can be closed with the Window Close button or by [File / Exit].

7.11 Pumping Tool

Note: The update of this functionality is still in progress.

WHY?
iMOD offers the possibility to manipulate existing model features, such as well strength, waterlevels, etc., by means of the functionalities offered by the scenario definitions. However, the scenario definitions are limited to existing features and new elements can not be added easily without changing the runfile manually. The Pumping Tool can be applied to configure new elements to an existing model configuration (runfile) and simulate and analyse the results easily. The Pumping Tool is developed specifically to simulate the effect of pumping, Aquifer-Storage-Recovery systems and/or Thermal-Heat-Storage systems.

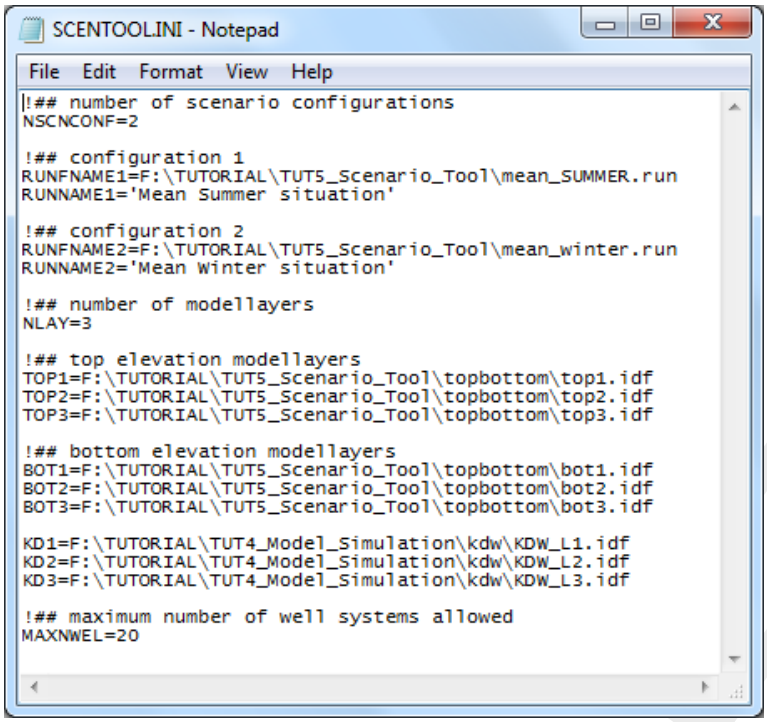
WHAT?
Any pumping system can be outlined by defining its location, vertical screen depths and corresponding well abstraction rates. The Pumping Tool includes these properties to any existing model configuration (runfile) prior to the simulation. The Pumping Tool allocates the well strength to the appropriate model-layer(s) and extends the model simulation time to include the given well regime. After the simulation the Pumping Tool offers a quick tool to analyse the results.

7.11.1 Initial Settings

The Pumping Tool needs an input file (*.INI) with initial settings. These settings are needed by the Pumping Tool to incorporate and allocate the well systems to the runfile(s). In the preference file the keyword SCENTOOL needs to be included that directs to that file, e.g. SCENTOOL {install-folder}\SCENTOOL.INI. The file SCENTOOL.INI needs to contain the following parameters:

NSCNCONF	Number of Scenario Configurations. This will indicate the number of runfiles that will be used to incorporate the well systems, e.g. NSCNCONF=2
RUNFNAME{i}	For each of the NSCNCONF runfiles specify the full filename of the runfile, e.g. RUNFNAME1=c:\scentool\scen_summer.run
RUNNAME{i}	For each of the NSCNCONF runfiles specify an alias for the runfile as described by RUNFNAME{i}, e.g. RUNNAME1='Mean Summer Situation'. This alias will be displayed in the Pumping Tool.
NLAY	Specify the number of modellayers of the model configurations specified by RUNFNAME{i}, e.g. NLAY=8.
TOP{i}	Specify an IDF-file that represents the top elevation of the i^{th} modellayer, e.g. TOP1=c:\scentool\idf\top1.idf.
BOT{i}	Specify an IDF-file that represents the bottom elevation of the i^{th} modellayer, e.g. BOT1=c:\scentool\idf\bot1.idf.
KD{i}	Specify an IDF-file that represents the transmissivity (kD) of the i^{th} modellayer, e.g. KD1=c:\scentool\idf\kd1.idf.
MAXNWEL*	Specify the maximum number of Well Systems in one configuration, e.g. MAXNWEL=20 (default value=10).
MAXNCUT*	Specify the maximum number of Cut-Out Areas in one configuration, e.g. MAXNCUT=20 (default value=10).
MAXNOBS*	Specify the maximum number of Observation Wells in one configuration, e.g. MAXNOBS=20 (default value=10).
MAXNMON*	Specify the maximum number of Monitoring Wells in one configuration, e.g. MAXNMON=20 (default value=10).
MAXNRES*	Specify the maximum number of results in one configuration, e.g. MAXNRES=20 (default value=10).
* optional	

Example of Pumping Tool SCENTOOL.INI file:



7.11.2 Start Pumping Tool

HOW?

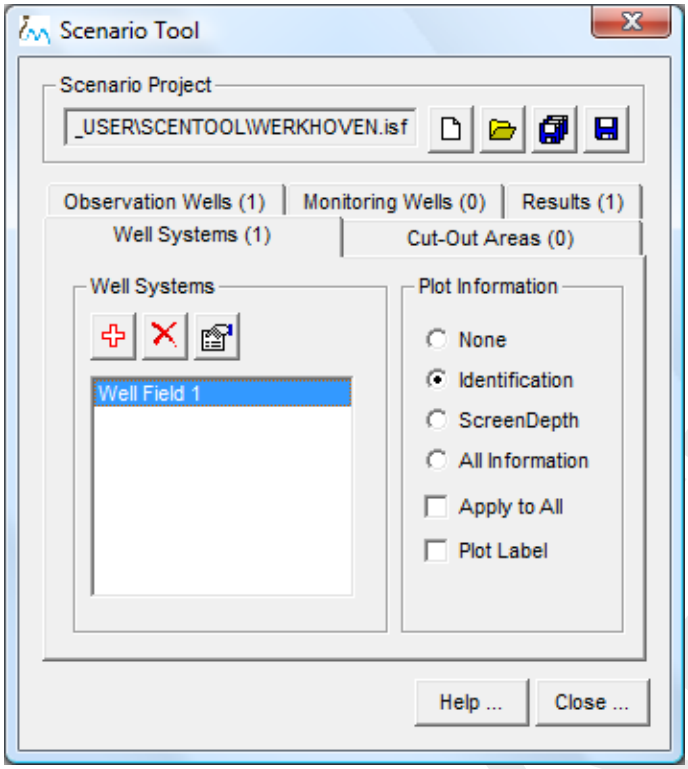
Select the *Toolbox* option from the main menu and then choose *Pumping Tool* to start the *Pumping Tool* window.

The *Pumping Tool* window manages the scenario configuration which is saved in the iMOD Scenario File (*.ISF).

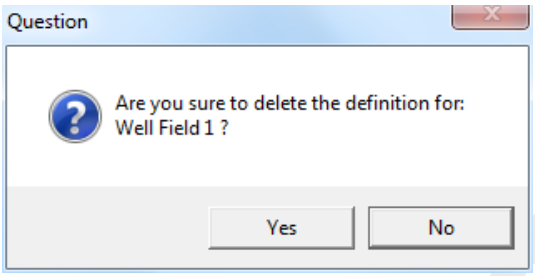
The *Pumping Tool* window has five tabs:

- Well Systems: to define the locations of the wells and the extraction rates
- Cut-Out Areas: to define the location of the cut-out or excavation areas
- Observation Wells: to define the locations of observation screens and the observed groundwater heads
- Monitoring Wells: to define the location of existing monitoring wells with groundwater head time-series
- Results: to define the model simulation configuration and to start the model calculation

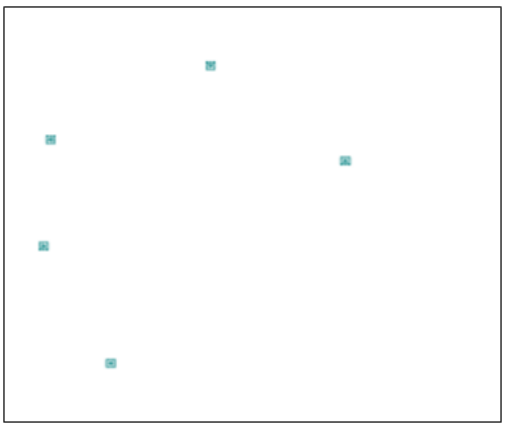
Pumping Tool window, Well Systems (..) tab:



Scenario Project	This displays the iMOD Scenario File (*.ISF) under which the current configuration is saved.
	New Click this button to start a new/empty scenario configuration.
	Open Click this button to open an existing iMOD Scenario File (*.ISF).
	Save As Click this button to save the current scenario configuration to a new iMOD Scenario File (*.ISF).
	Save Click this button to save the current scenario configuration to the current iMOD Scenario File (*.ISF).
Well Systems	Click one of the well systems in the list to activate the *Delete* and *Properties* buttons. The *Plot Information* setting will be applied to the selected well system too.
	Add Click this button to add a new well system.

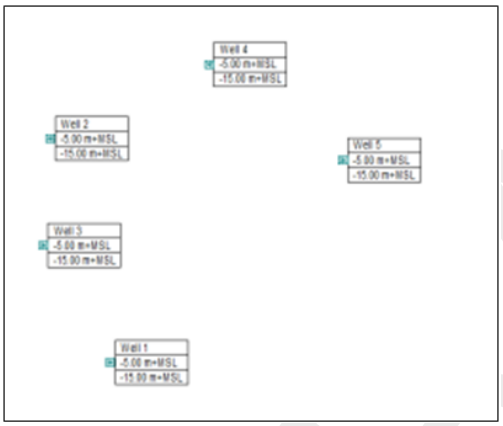
	*Delete* Click this button to remove the selected well system from the scenario configuration. *Question window:* 
	*Properties* Click this button to adjust the parameters for the selected well system.
Plot Information	Select one of the following options to specify the method to display information for the individual pumping locations for the selected well system (see below). ◇ *None* only the specified symbol will be drawn. The symbol can be changed in the *Display* tab of the *Well Systems* window, see section 7.11.3. ◇ *Identification*: displays the identifications; ◇ *Screendepth:* displays all screen depths; ◇ *All Information:* displays all information available.
Apply to All	Select this option to apply the display settings to all available well systems at once.
Plot Label	Select this option to add the labels to the different parameters.
Help ...	Click this button to start the Help functionality.
Close ...	Click this button to stop the *Pumping Tool*. You will be asked to save your work before the *Pumping Tool* need to be closed.

Example of the Plot Information: (1) None, (2) Identification, (3) ScreenDepth, (4) All Information without and (5) with labels

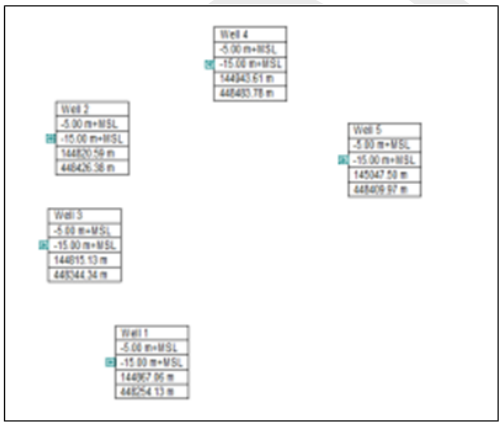


(1)

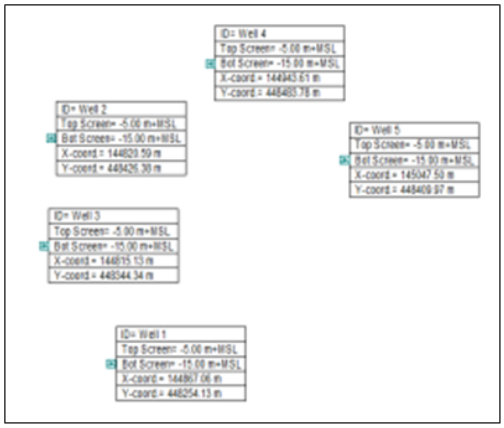
(2)



(3)



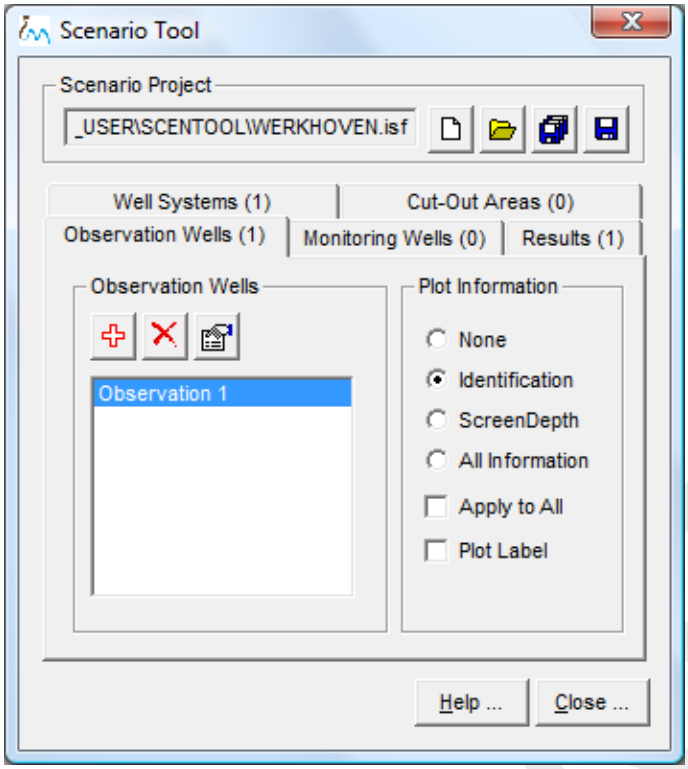
(4)



(5)

Note: All other iMOD functionalities remain active whenever the *Pumping Tool* is loaded. This means that map operations can be carried out, legends can be changed, but the *Profile Tool* (see section 7.1) and/or *3D Tool* can not be used without leaving the *Pumping Tool*.

Pumping Tool window, Observation Wells (..) tab:

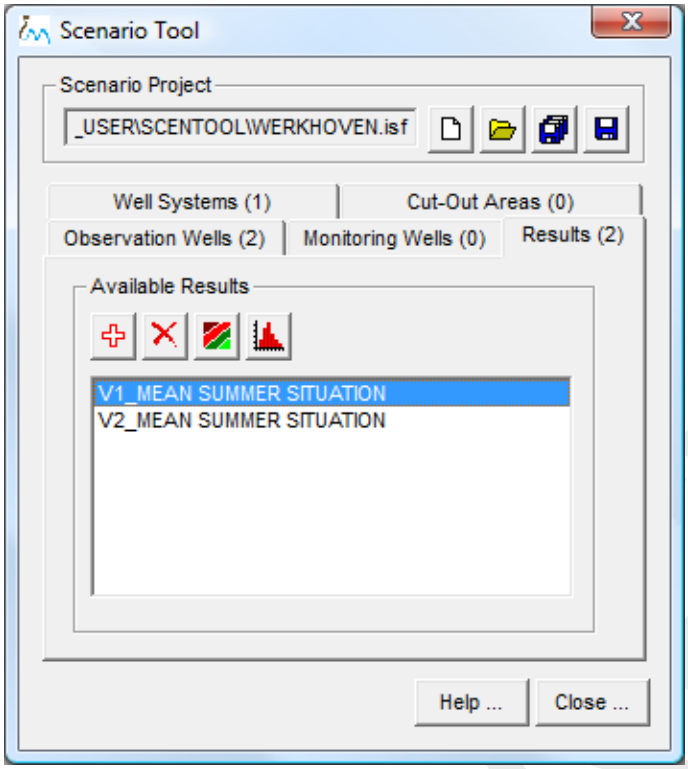


The observation wells are used to define screens where the calculated groundwater head can be compared with the observed groundwater head.

All functionalities of the *Observation Wells* tab behave similar to those described for the *Wells Systems* tab and for the *Wells Systems* window.

	*Add/Adjust* Click these buttons to add/adjust a new/existing observation well, see section 7.11.4.

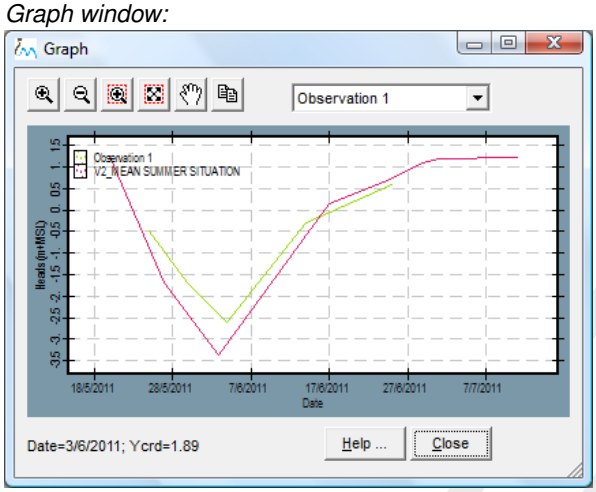
Pumping Tool window, Results (..) tab:



Available Results	Display of all available result folders in the scenario project folder, e.g. folder C:\..\USER\SCENTOOL\WERKHOVEN* for the C:\..\USER\SCENTOOL\WERKHOVEN.ISF file. The list will be refreshed whenever the tab is toggled with other tabs on the *Pumping Tool* window.
	*Add* Click this button to start a new simulation.
	*Delete* Click this button to delete the selected result in the *Available Results* list. You will be asked to confirm this action, since all files for the selected result will be deleted from disk.
	*Contour* Click this button to start the *Quick Open* window and specify the results that need to be displayed.

Graph
Click this button to display the selected result in combination with observation wells. Therefore, this button is only available whenever Observation Wells are defined.

Graph window:

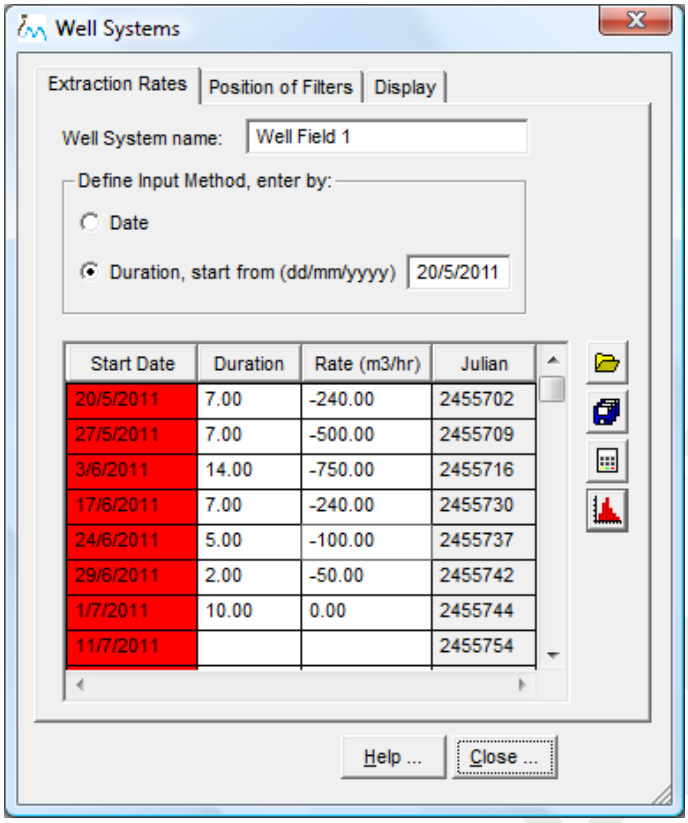


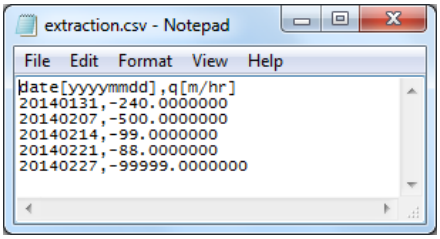
See section 6.6.1 for more information on the use of the available functions of this window. In this graph window a drop-down menu is added. The graph is updated whenever another item (Observation Well) is selected from the dropdown menu.

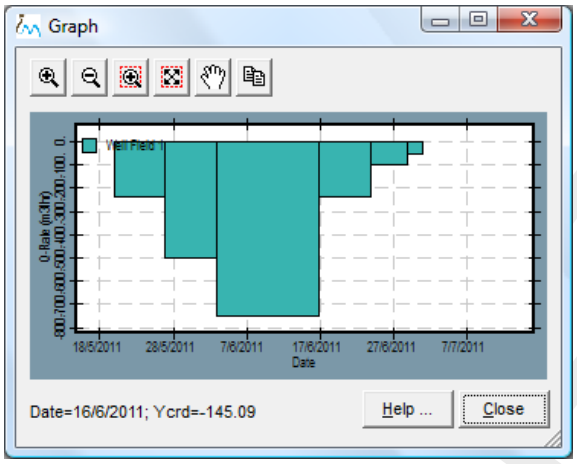
7.11.3 Well Systems

Select the *Add* or *Properties* button on the *Well Systems* tab on the *Pumping Tool* window, to display the *Well Systems* window.

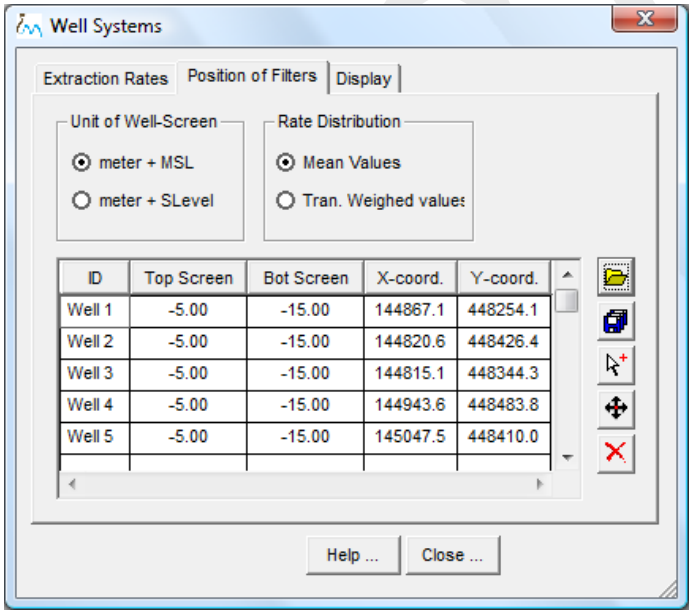
Wells Systems window, Extraction Rates tab:



Well System name	Enter the name of the well system, maximum 24 characters.
Date	Select this option to enter the extraction rates by their (start)date, the *Start Date* column in the table will become enabled to enter values. The values within the *Duration* column will be computed, automatically.
Duration, start from (dd/mm/yyyy)	Select this option to enter the extraction rates by specifying an initial date, e.g. 20/5/2011. The column *Duration* becomes enabled to enter the duration for each extraction rate, e.g. 7.0, 7.0, 14.0 and 7.0 days. The date within the *Start Date* column will be computed automatically whenever the *Calculate* button is clicked or when closing the window.
Table	Each row in the table expresses an extraction rate for the entire well system. The red coloured column can not be edited and is computed whenever the *Calculate* button is clicked. Enter the extraction rates (strength) in the third column in m^3/hr (during the actual modeling, those values will be translated on the background into m^3/day). The last column (*Julian*) is not editable and is use by iMOD internally.
	*Open* Click this button to open a plain text file that contains date and extraction rates. *Example of a text file with extraction data* 
	*Save As* Click this button to save a plain text file that contains date and extraction rates.

	*Calculate* Click this button to compute the *Start Date* or *Duration* column in the table.
	*Graph* Click this button to display a bar diagram of the extraction rates (see section 6.6.1 for more information on the use of the available functions of this window). *Graph window:* 
Help . . .	Click this button to start the Help functionality.
Close . . .	Click this button to close the *Well Systems* window. You will be asked to save any adjustments or you can cancel closure.

Well Systems window, Position of Filters tab:

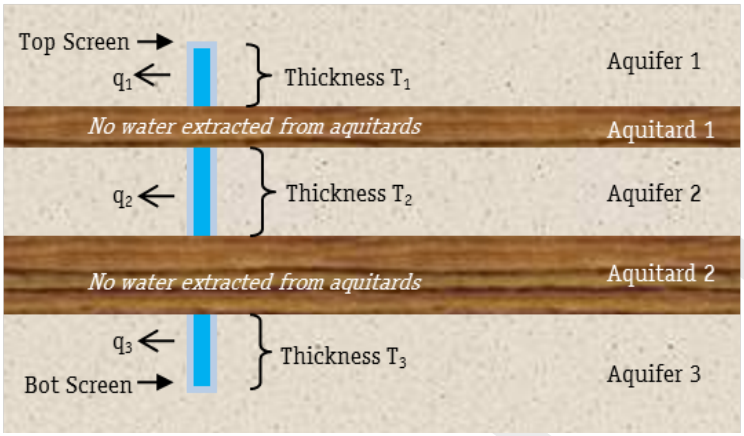


Unit of Well-Screen	Select one the following options to determine the treatment of well screens: Meter+MSL: enter the screen depths in meter+Mean-Sea-Level (MSL) Meter+SLevel: enter the screen depths in meter+Surface Level (SLevel). The surface level will be extracted from the {TOP1}.idf as defined in the initialization file (SCEN-TOOL.INI).

Rate Distribution

Select one of the following options to determine the distribution of extraction rates among all individual well screens.
Mean Values: distribute the extraction rate evenly among all filter screens using the thickness of each screen in penetrating aquifers, e.g. $q_1=T_1/(T_1+T_2+T_3)*Q$
Tran. Weighed values: distribute the extraction rate weighed by their thicknesses of each screen penetrating aquifers and their corresponding transmissivity values, e.g. $q_1=T_1*Tran_1/(T_1*Tran_1+T_2*Tran_2+T_3*Tran_3)*Q$

Extraction well (Q) and rate allocation to the different screens ($q_1+q_2+q_3$):



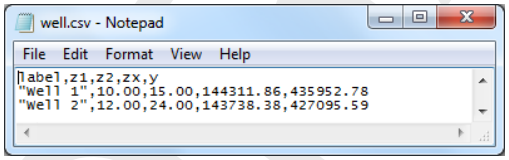
Table

Each row in the tables represents a well location and corresponding well screen depths. It is allowed to edit the table directly (screen depths). It is recommended to change lateral positions (the coordinates) by means of the *Move* option, however.

Open

Click this button to open a plain text file that contains date and extraction rates.

Example of a text file with well location and screen depth



Save As

Click this button to save a text file that contains date and extraction rates.

Add

Click this button to add a new extraction well by clicking your left mouse button at the location of the well. Each time you click the left mouse button a new well will be added. Click your right mouse button to stop.

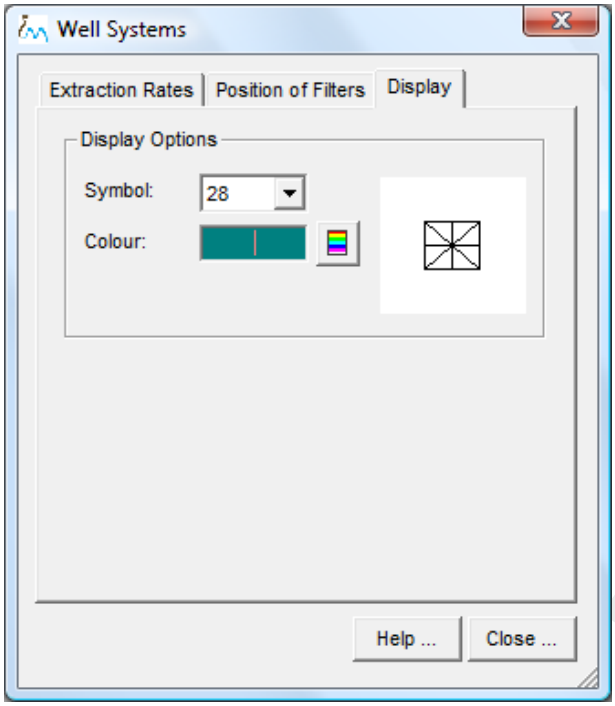
Move

Click this button to move an existing well. Move your mouse cursor in the neighbourhood of a well and observe that the mouse cursor changes to  and the corresponding row in the table changes to red. Hold your left mouse button down and move the well by moving the mouse cursor. Release the left mouse button to start moving another well. Click your right mouse button to stop.

Delete

Click this button to delete an existing well. Move your mouse cursor in the neighbourhood of a well and observe that the mouse cursor changes to  and the corresponding row in the table changes to red. Click your left mouse button to delete the well. Click another well to delete it, or click your right mouse button to stop.

Well Systems window, Display tab:

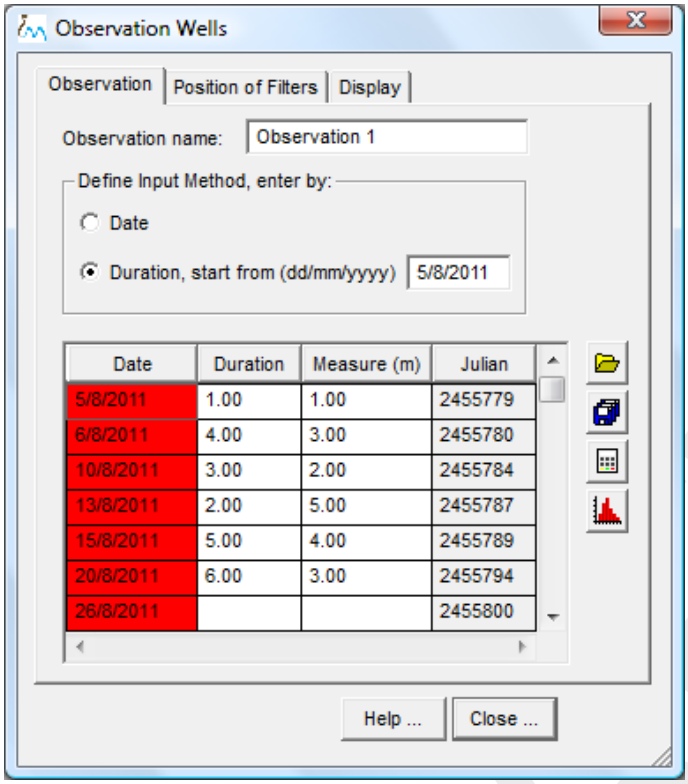


Symbol	Select a symbol number (see section 5.7 for available symbols). The current symbol is displayed on the right.
Colour	Displays the current symbol colour (dark green in this example).
	*Colour* Click this button to change the colour by means of the default Windows Colour window.

7.11.4 Observation Wells

Select the *Add* or *Properties* button on the *Observation Wells* tab on the *Pumping Tool* window, to display the *Observation Wells* window.

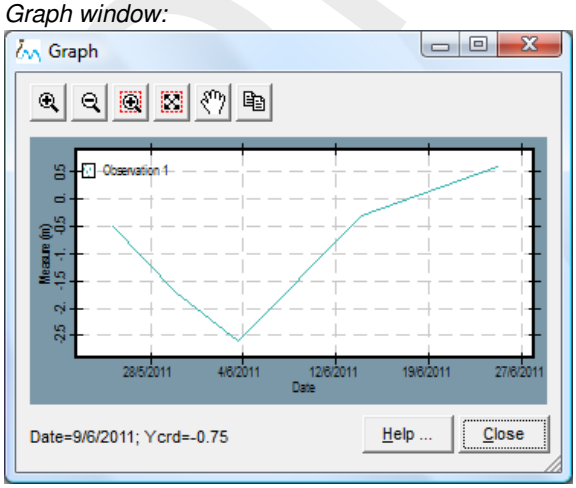
Observation Well window, Observation tab:



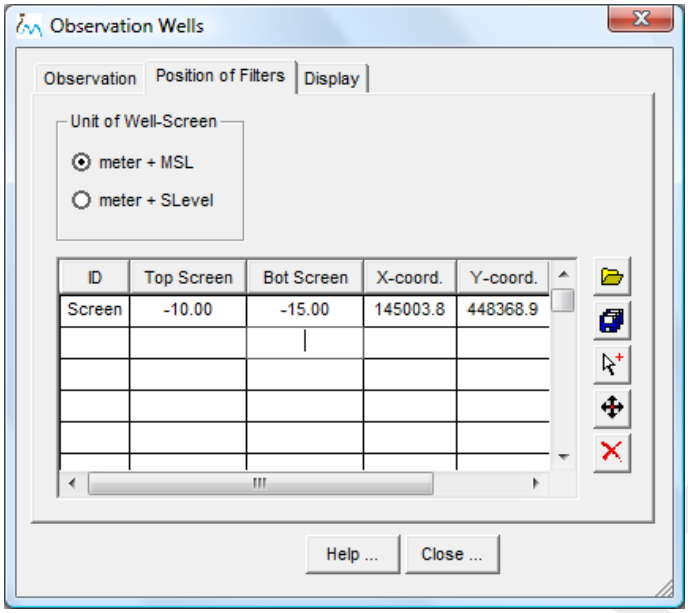
All functionalities of the *Observation* tab behave similar to those described for the Extraction Rates tab on the *Wells Systemswindow*, however, a few remarks are needed.

Graph
Click these buttons to display a graph for the observations. The graph will represent the observations by plotting straight lines between the given dates, ignoring the last (final) date without any given observation (26/8/2011).

Graph window:



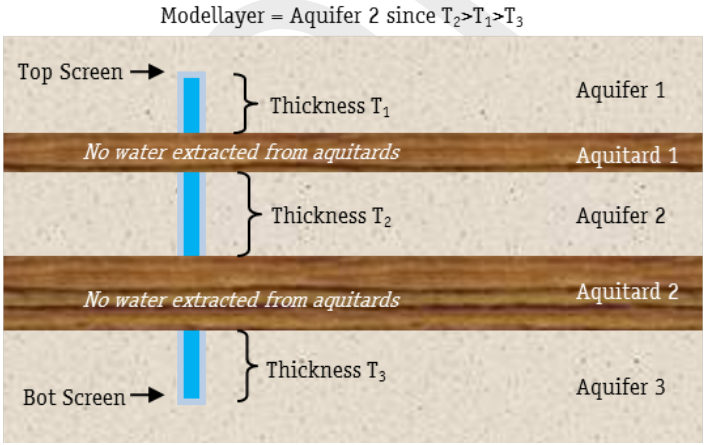
Observation Wells window, Position of Filters tab:



All functionalities of the *Position of Filters* tab behave similar to those described for the same tab on the *Wells Systems* window, however, a few remarks are needed.

Table	Each row in the table represents a single observation and corresponding well screen depth. Only one location is sustained for each observation. It is allowed to edit the table directly (screen depths). It is recommended to change lateral positions by means of the *Move* option, however. Observed heads will be used to compare with model results and the modellayer that will correspond to each observation depends on the screen depths. iMOD will assign a modellayer that is occupied by the largest part of the entire screen.

Allocation of screens of observation wells to modellayers:

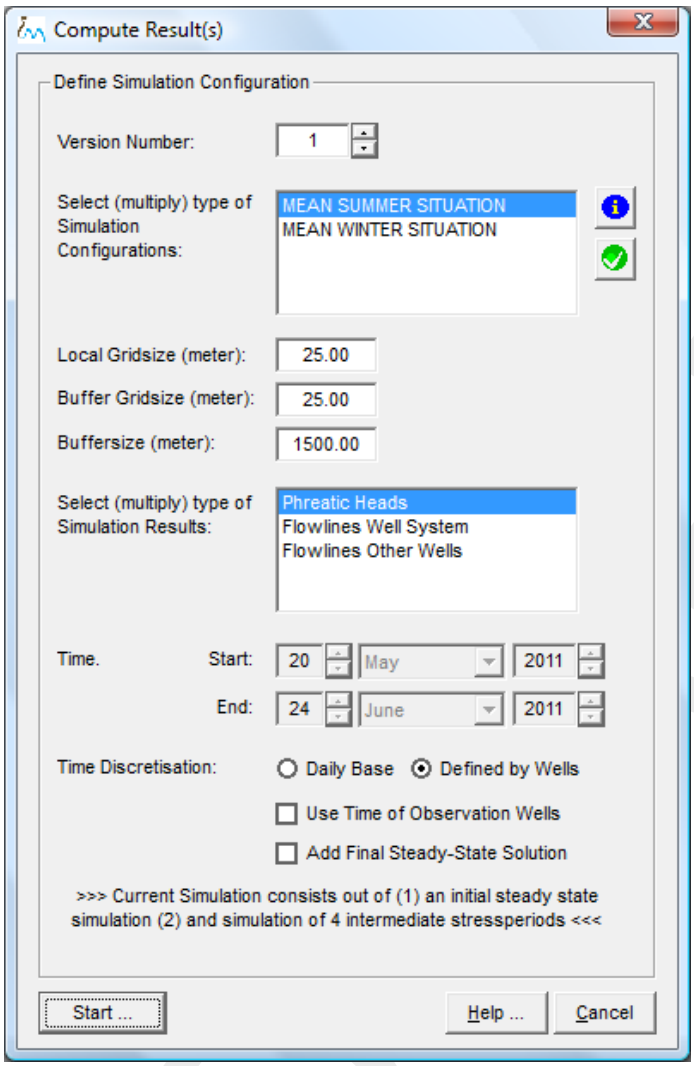


Observation window, Display tab:

This window behaves and is identical to the window described before for the *Wells Systems* tab.

7.11.5 Results

Select the *Add* button  on the *Results* tab on the *Pumping Tool* window, to display the *Compute Result(s)* window.

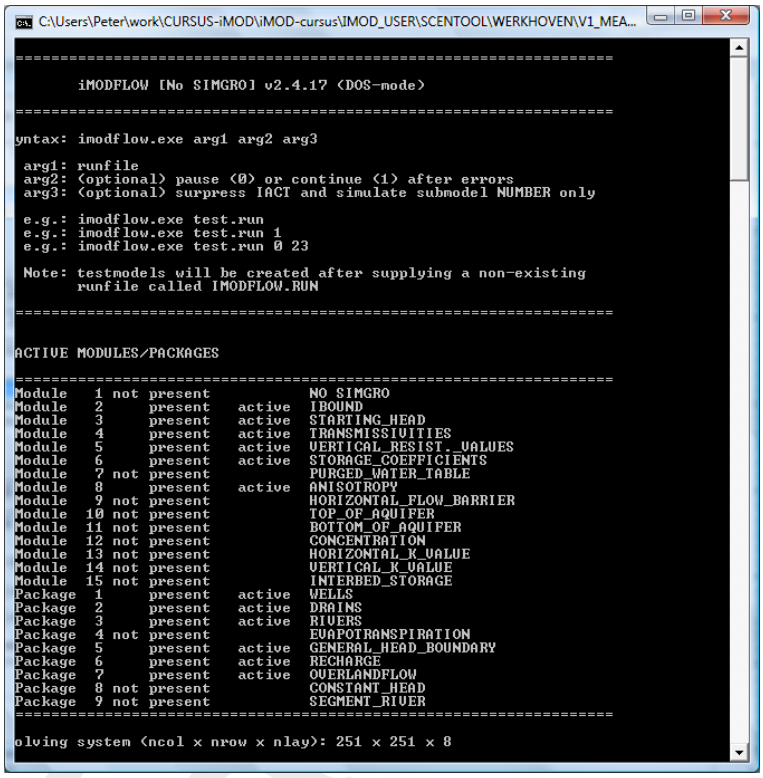


Version Number	Enter a version number. The version number will be added to the result folder that is formed by the name of the model configuration, e.g. V1_{modelconfiguration}.
Select (multiple) type of Simulation Configurations	Select one or more model configurations from the list. The list is reflecting the content of the parameter RUNNAME{i} in the initialization file for the *Pumping Tool*. It is an alias for the runfile described by the keyword RUNFNAME{i].
	Info Click this button to edit the i^{th} selected runfile (RUNFNAME{i}) that is associated to the alias (RUNNAME{i}) from the list.
	CheckRun Click this button to check the content of the i^{th} selected runfile (RUNFNAME{i}) that is associated to the alias (RUNNAME{i}) from the list. More information about this kind of runfile check.
Local GridSize (m)	Enter the gridsize inside the area of interest, e.g. 25.0 meter. The area of interest will be computed automatically based on the layout of the scenario, i.e. the lateral position of the extraction wells, cut-out areas and observation wells.
Buffer GridSize (m)	Enter the gridsize to be used inside the buffer defined by the *Buffersize (m)*, e.g. 100.0 meter.

Buffersize (m)	Enter the size of the buffer in meters, e.g. 1500 meter.
Select (multiple) type of Simulation Results	Select one or more of the supplied result options. ◇ Phreatic Heads – choose this option to compute Phreatic Heads (and piezometric heads as well). ◇ Drawdown ◇ Flowlines Well Systems *Not yet implemented* ◇ Flowlines All Wells *Not yet implemented*
Time	Display of the *Start* and *End* dates (not editable). These are determined automatically by the entered extraction rates. The simulation duration is primarily determined by the sequence of extraction.

Time Discretisation	Choose out of the following:
	◇ Daily base – choose this option to simulate between the *Start* and *End* date on a daily base. A summary is given below:
	>>> Current Simulation consists out of (1) an initial steady state simulation (2) and simulation of 35 intermediate stressperiods <<<
	◇ Defined by wells – choose this option to simulate between the *Start* and *End* with stressperiod lengths that depend on the entered moment of extraction. A summary is given below:
	>>> Current Simulation consists out of (1) an initial steady state simulation (2) and simulation of 4 intermediate stressperiods <<<

Use Time of Observation wells	Select this option to include (extra) intermediate stressperiods to take the moment of observations into account. This ensures that any comparison between observations and computed heads are measured/computed at identical moments. Bear in mind that computational times are linearly related to the number of stressperiods.
Add Final Steady-state solution	Select this option to add a final stress-period that simulates a steady-state of the last entered model input (i.e. extraction rates). A summary is given below: >>> Current Simulation consists out of (1) an initial steady state simulation (2) and simulation of 3 intermediate stressperiods and (3) a final steady state simulation <<<

Start . . .	Click this button to start the simulation(s). You will be asked to confirm this. iMOD will add the appropriate information regarding extraction wells to the selected runfile(s) and start iMODFLOW as defined by the keyword MODFLOW in the active preference file. iMODFLOW will start in a separate commandtool window. *Scenario Simulation Command window:*  After a successful simulation, both the *Scenario Simulation Command* window and the *Compute Results* window will be closed.
Help . . .	Click this button to start the Help functionality.
Cancel	Click this button to cancel and close the *Compute Results* window.

Note: A simulation will block any use of iMOD. A simulation can be terminated by pressing the (red in Windows Vista) closing window button on the upper-right corner of the *Scenario Simulation Command* window.

7.12 RO-tool

WHY?

The iMOD RO-tool is based on the ArcGIS WaterNOOD-application ("WaterNOOD"approach: "Watersysteemgericht NOrmeren, Ontwerpen en Dimensioneren") by Stichting Toegepast Onderzoek Waterbeheer (STOWA).

The WaterNOOD-application provides water managers with information on hydrological requirements of farm crops and vegetation types, and helps them to predict the effects of water management measures on crop-yields and vegetation. The application is used in the water management of nature conservation areas and surrounding agricultural or urban areas.

The iMOD RO-tool (like the ArcGIS WaterNOOD-application) incorporates two knowledge/database/model-based tools, that are widely used in the Netherlands to determine productivity of farm-crops and potential for development of groundwater dependent vegetation types under various abiotic conditions (soil type and soil-hydrology):

- "HELP2005-tabellen"for calculating farm crop productivity. (see also: http://help200x.alterra.nl/HELP2005.pdf and http://help200x.alterra.nl/)
- "WATERNOOD - Hydrologische randvoorwaarden natuur"for calculating potential for development of groundwater dependent vegetation types.(see also: http://www.synbiosys.alterra.nl/waternood/ and http://waterwijzer.stowa.nl/upload/publicatie2014/STOWA%202015%2022_Webversie%20LR.pdf)

Note: Because this is a Dutch spatial planning tool, based on a Dutch policy-makers'approach for optimizing groundwater regimes for agriculture and nature restoration, Dutch remarks are included.

WHAT?

Abiotic conditions determine productivity of farm-crops and potential for development of groundwater dependent vegetation types. The tool calculates productivity of farm-crops and potential for development of groundwater dependent vegetation types, as a function of abiotic conditions (soil type, seasonal groundwatertables).

Agriculture: actual crop-yield, compared to crop-yield under optimal conditions. The RO-tool calculates the actual crop-yield (relative to crop-yield under optimal abiotic conditions) and decrease of crop-yield due to moisture-stress ("natschade landbouw") or respiration-stress ("droogteschade landbouw"), given the actual, site specific abiotic conditions (soil type, seasonal groundwatertables). Decrease of crop-yield is calculated in % and/or Euros. doelrealisaties (= 1 –[opbrengstreductiepercentage t.o.v. optimale condities]/100)

Natural vegetation: potential for development of groundwater dependent vegetation types. The RO-tool calculates the potential for development of pre-defined groundwater dependent vegetation types. The potential is expressed as a percentage, relative to the potential for development under optimal abiotic conditions. doelrealisaties (= 1 –[actuele ontwikkeling vegetatie/ontwikkeling vegetaties onder optimale condities]

Urban area: suitability for urban development, as a function of groundwatertable-depth. The RO-tool calculates the suitability for urban development, based on a range of drainage-level, defined in a lookup-table. Groundwatertable above the upper range results in a 0% suitability for building, groundwatertable between upper and lower range results in a 50% suitability for building, and groundwatertable below the lower range results in a 100% suitability for building.

HOW?

The RO-tool is a post-processing scenario-tool. The user can use pre-defined maps of soil types, landuse (crop types) and vegetation types and pre-defined tables of hydrological requirements for vegetation and/or habitat types. All these maps and tables can also be modified by the user, to meet special interests. Input for the hydrological conditions are the Dutch groundwater-table-depth statistics maps ("GHG, GLG, GVG"). They are the result of a groundwate rmodel simulation. The user can run the RO-tool for different groundwater model scenarios.

Explanation of the Dutch groundwatertable-depth statistics ("GHG, GLG, GVG"): The groundwater-table-depth statistics (Mean Highest, Mean Lowest or Mean Spring Watertable depth). The Mean Highest Watertable (MHW or GHG) is defined as the mean value of the three shallowest groundwater depths measured in one year (in meter beneath soil surface), averaged over a period of 8 years with bi-weekly measurements or simulation-results. The Mean Lowest Watertable (MLW or GLG) is defined likewise, with the deepest groundwater depths. The groundwater depths measured at three dates nearest to April 1, are used for calculation of the Mean Spring Watertable (MSW or GVG). The period to calculate MHW is defined between October 1st and April 1st.

Important Note: Despite the fact that the iMOD RO-tool is based on version 2 of the "WaterNOOD"application, it is possible to implement tables with hydrological requirements for user defined vegetation/habitat-types, created with "WaterNOOD"version 3. The tables can be created with and exported from the "WaterNOOD"application, version 3. To download "WaterNOOD"version 3 go to: http://www.synbiosys.alterra.nl/waternood/ and follow the instructions in the manual to export the user created tables for later use in iMOD RO-tool.

Before starting the RO-tool, the desired settings need to be defined in the Preference file (see section 9.1). The preference tab at the RO-tool window is filled with these settings. The keywords in the preference file are linked to the required input files of the RO-tool. section 7.12.2 contains a list and description of these files. In section 7.12.3 the operational setup of the RO-tool is explained and in section 7.12.4 a list of output files and a short explanation can be found. To start the RO-tool window, select the *Toolbox* option from the main menu and then choose *RO-tool* from the bottom of the menu-list.

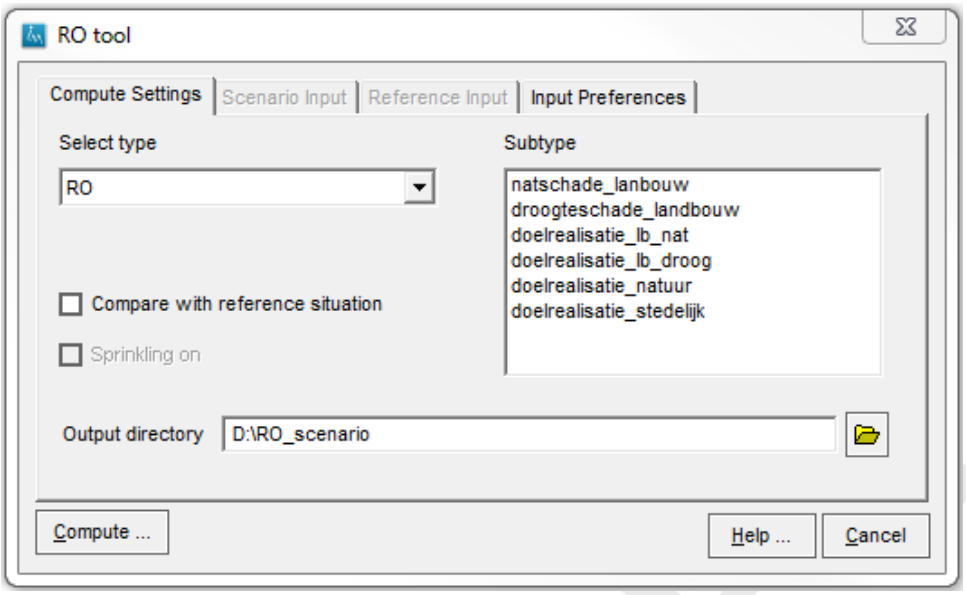
7.12.1 RO-tool window

In this section an explanation is given about each tab on the RO-tool window. General settings on the RO-tool window are:

Compute...	Click on this button to start the scenario based map calculation with the given settings.
Help...	Click on this button to search for relevant information in the iMOD-manual if the directory of the file is defined in the PRF-file (see section 9.1).
Cancel	Click on this button to Quit the RO tool without saving or computing anything.

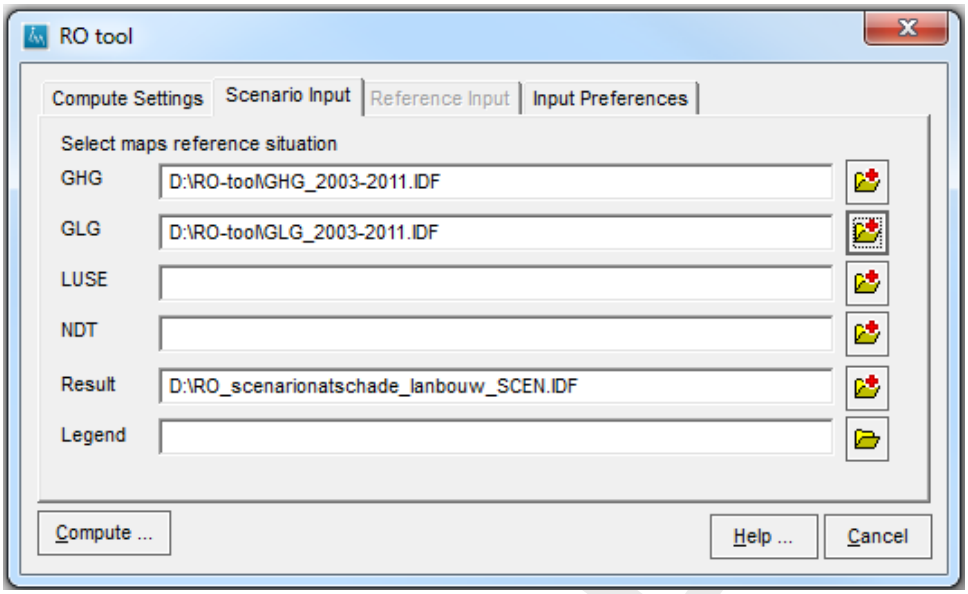
After all settings are selected the user can click on the *Compute...* button to start the scenario computation. Directly after starting the calculation iMOD checks continuously which input files and lookup-tables are needed. The computation is performed per selected subtype. In case one scenario is chosen in combination with a reference situation, iMOD creates, additionally to the reference/scenario related result maps, a map with the difference between reference results and scenario results. Result-files are immediately loaded in the iMOD Manager.

Compute Settings tab:



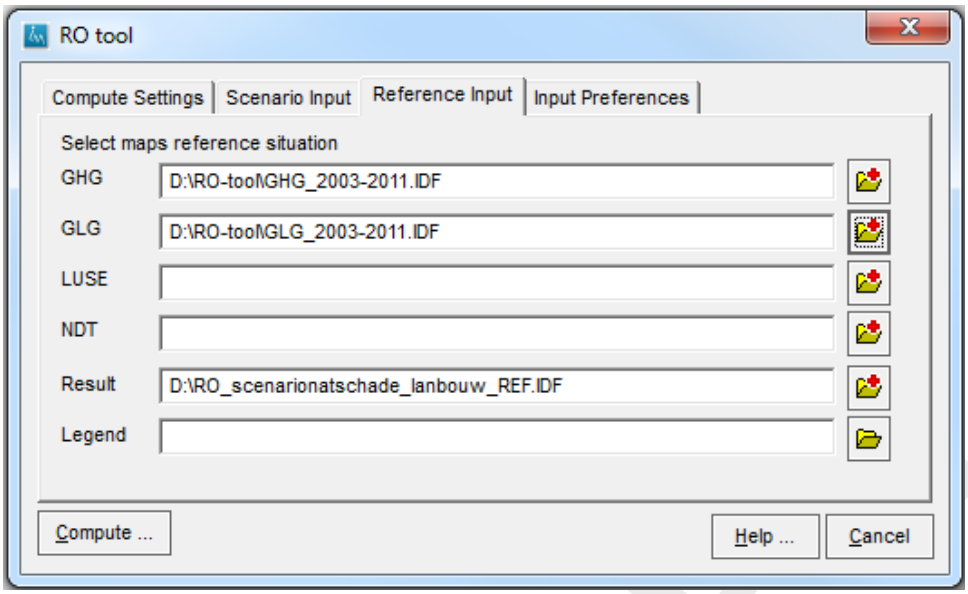
Select type	Choose the preferred group of investigation questions. (**Note:** at this moment only the RO (flood related) type is available.)
Subtype	Depending on the selected type a list of scenario related subtypes appears. **Selecting 1 or more subtypes enables the Scenario Input tab.**
Compare with reference situation	Check this option if it is preferred to compare the selected scenario with a reference (or other) situation. **Checking this option enables the Reference Input tab.**
Sprinkling on	**This option is only available in drought related scenarios.** Check this option if re-wetting through sprinkling during a period of drought is wished. By applying this option, respiration-stress of farm crops is eliminated.
Output directory	Enter the name of the output directory where iMOD stores the scenario results. Default: TMP-folder in the USER directory.

Scenario Input tab:



GHG	Enter the directory and name of the GHG-file to be used in the scenario.
GLG	Enter the directory and name of the GLG-file to be used in the scenario.
LUSE (Optional)	Enter the directory and name of the landuse type related file to be used in the scenario (**Note:** only available if at least an Agriculture-subtype is chosen, or a combination of Agriculture- and Nature-subtypes; not available if only a Nature-subtype is chosen).
NDT (Optional)	Enter the directory and name of the file with the vegetation type(s),"natuurdoeltype kaart" (**Note**: only available if a Nature-subtype is chosen, or in a combination with a Nature-subtype and Agriculture-subtype).
Result	Enter the directory and name of the result file (**Note:** not available if more than 1 subtype is selected). On default the Output Folder is selected and the name of the results file equals the chosen subtype description *.idf, e.g. subtype="natschade_landbouw" the name of the file is "natschade_landbouw_SCEN.IDF".
Legend	Enter the directory and name of the legend file (**Note:** not available if more than 1 subtype is selected).
	Click on this button to search for the needed file.

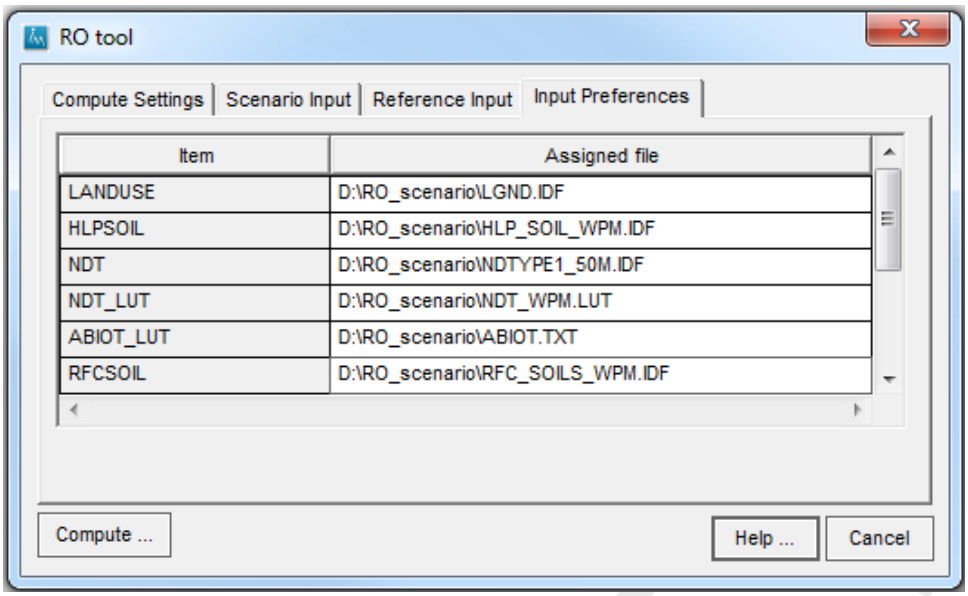
Reference Input tab:
Only available if option *Compare with reference situation* on tab *Compute Settings* is checked.



GHG	Enter the directory and name of the GHG-file to be used in the scenario.
GLG	Enter the directory and name of the GLG-file to be used in the scenario.
LUSE (Optional)	Enter the directory and name of the landuse type related map to be used in the scenario (**Note:** only available if at least an Agriculture-subtype is chosen, or a combination of Agriculture- and Nature-subtypes; not available if only a Nature-subtype is chosen).
NDT (Optional)	Enter the directory and name of the file with the vegetation type(s), "natuur-doeltype kaart" (**Note:** only available if a Nature-subtype is chosen, or in a combination with a Nature-subtype and Agriculture-subtype).
Result	Enter the directory and name of the result file (**Note:** not available if more than 1 subtype is selected). On default the Output Folder is selected and the name of the results file equals the chosen subtype description *.idf, e.g. subtype="natschade_landbouw" the name of the file is "natschade_landbouw_SCEN.IDF".
Legend	Enter the directory and name of the legend file (**Note:** not available if more than 1 subtype is selected).
	Click on this button to search for the needed file.

On the *Scenario Input* tab and *Reference Input* tab, it is not necessary to fill in the fields for *LUSE* ("landgebruik") and *NDT* ("natuurdoeltype"). When empty, iMOD uses the defined file as given in the Preference file; defined with keyword LANDUSE in case of *LUSE* and keyword NDT in case of *NDT*. As mentioned, it is not possible to fill in the fields for *Result* and *Legend* whenever more than 1 scenario objective is chosen.

Input Preferences tab:
This tab contains all information as defined in the Preference file (section 9.1), including cross reference tables, lookup tables and basic files. Files may be placed in any directory and/or different directories on user's harddisk(s).



Item	List of the preference keywords.
Assigned file	Name and directory of a file referring to *Item*.

7.12.2 Preprocessing

The needed input files to be prepared:

- Two IDF-files, one containing the GHG and one with GLG information in meters relative to surface level (positive values = below surface level). A minimum of 1 set is required for the scenario situation.
- A landuse map (IDF-format) according to the LGN5 encoding.
- A soil map (IDF-format) with HELP soil units and RFC soil units (explained later in more detail). This conversion from the 1:50.000 soil map to HELP soils and to RFC soils is done by Deltares, using a conversion table.
- A map of vegetation types (NDT, natuurdoeltypenkaart), with encoding matching the lookup-tables NDT_LUT and ABIOT_LUT.
- So called “HELP”-tables, with HELP2005-database, containing relations between crop - soiltype - GHG/GLG -respiration/moisture-stress.

As mentioned in section 7.12 “WHY?”, actual crop-yield, decrease of crop-yield and potential for development of selected vegetation types are determined by use of the two knowledge/database/ model-based tools:

- “HELP2005-tabellen”**for calculating farm crop productivity**.
- “WATERNOOD - Hydrologische randvoorwaarden natuur”**for calculating potential for development of groundwater dependent vegetation types.**

The RO-tool requires maps, tables and binary files that represent the above mentioned knowledge/database/ model-based tools.

To calculate **Agriculture-subtypes**, these files from “HELP2005-tabellen”are required:

- HLP_DRY.DAT HELP2005-database: crop|soiltype|GHG/GLG|respiration-stress.
- HLP_WET.DAT HELP2005-database: crop|soiltype|GHG/GLG|moisture-stress.
- HLP_SOIL.IDF Raster file: 1:50.000 soil map reclassified for HELP2005 (using bod2hlp.lut).
- LUSE\LGN5.IDF Raster file with land use types in LGN5-codes.
- CROP_COSTS.LUT Lookup table with crops (in LGN5-codes) and crop-yields (Euro/ha/year).

To calculate **Nature-subtypes**, these files from “WATERNOOD - Hydrologische randvoorwaarden natuur”are required:

- ◇ RFC_SOIL.IDF Raster file: 1:50.000 soil map reclassified to RFC-soils (using bod2rep.lut).
- ◇ RFC_LUT.LUT Lookup table with RFC(reprofunction)-characteristics.
- ◇ NDT.IDF Raster file with vegetation types to calculate potential development for.
- ◇ NDT_LUT.LUT Lookup table with option to aggregate vegetation types.
- ◇ ABIOT_LUT.LUT Lookup table: hydrologic requirements for vegetation types (NDT’s).

To calculate **Urban area**:

- ◇ URBAN_RANGE.LUT Lookup table with range of drainage-level (upper- and lower value in meter below surface)

Realization of the RFC soil map
A soil map containing 1:50000 soil codes is needed for this realization as well as a field with an assimilation of the fields "VOOR", "LETTER", "CIJFER", and "KALK". In case a new field needs to be added, ArcView3.x can be used by fill in the following line in the Field calculator:
[VOOR]+[LETTER]+[CIJFER].ASSTRING+[KALK]. The associations are not included in the conversion table and therefore the value of the field "EERSTE_BOD" can be used as a replacement. In the soil map, the codes related to excavated soils starting with a "|"-character are represented in the conversion table with the first two characters of the soil code (RFC-value = 99). All codes starting with a "|"-character can be selected by making use of a query function (in ArcView3.x), e.g. ([fieldname]).contains("|"). Via the calculator the specific field can be filled with the first 2 characters by making use of [CODE].LEFT(2). Similar functions are available in ArcGis. A final step in the preparation of a RFC soil map is joining the soil map with the table RFCCODES.DBF.

Realization of the HELP soil map
The same soil map (1:50000 soil codes) as was used for the realization of the RFC soil map, is needed for realization of the HELP soil map. Only now an additional field is needed, containing the groundwater tables. Accordingly, a compilation of the following fields is included in the map: "VOOR", "LETTER", "CIJFER", "KALK", and "GWT". This field is available in most of the soil maps labeled as "CODE". Next, the "BOD50_2_HLP.DBF" table needs to be joined to the "GWT"-field and converted to a raster format on field "HELPID".

7.12.3 Operational setup

To start the computation of the RO-tool, it is necessary to define at least a GHG- and/or a GLG-file (depending on chosen subtype) on the scenario tab. If iMOD cannot find all needed files, an explanatory error is raised. In case a scenario has to be compared with a reference situation, computation is only possible if all required files are defined in the Scenario Input and Reference Input tabs. At the moment everything is correctly defined, clicking on *Compute...* executes the computation.

In case of comparing a scenario with a reference situation, the output contains:

1 For both scenario and reference:

- 1.1 For **Agriculture-subtypes**: an IDF and table with actual crop-yield (relative to crop-yield under optimal abiotic conditions), and/or an IDF and table with decrease of crop-yield due to moisture-stress (“natschade landbouw”) or respiration-stress (“droogteschade landbouw”).
- 1.2 For **Nature-subtypes**: an IDF and table with the potential for development of chosen vegetation types, expressed as a percentage, relative to the potential for development under optimal abiotic conditions.
- 1.3 An IDF and table with the difference between the scenario results and the reference situation results for chosen **Agriculture and Nature subtypes** (difference = reference - scenario).

In case the computation is limited to 1 situation (either scenario, or reference), the output only contains one IDF and table with either actual crop-yield, decrease of crop-yield, or potential for development of chosen vegetation types, depending on the chosen subtype(s).

Calculation of **subtype "Urban Area"** returns an IDF and table with suitability for urban development (values 0, 50 and 100). A range for drainage-level is defined in a lookup-table. Groundwatertable above the upper range results in a 0% suitability for building, groundwatertable between upper and lower range results in a 50% suitability for building, and groundwatertable below the lower range results in a 100% suitability for building.

7.12.4 Output

The output of the computation contains:

1 IDF-files (raster files) containing:

- ◇ Agriculture: decrease of crop-yield due to moisture-stress and respiration-stress ("natschade landbouw", "droogteschade landbouw").
- ◇ Agriculture: actual crop-yield, relative to crop-yield under optimal abiotic conditions ("doelrealisatie landbouw").
- ◇ Nature: potential for development of pre-defined groundwater dependent vegetation types, expressed as a percentage, relative to the potential for development under optimal abiotic conditions ("doelrealisatie natuur").
- ◇ Urban Area: suitability for urban development ("doelrealisatie stedelijk").

2 A table with:

- ◇ Total crop-yield for all crops in scenario and reference situation (if applicable).
- ◇ Total yield per area.
- ◇ Average decrease in crop-yield, for both decrease of crop-yield due to moisture-stress and respiration-stress.
- ◇ Computed area of total extent with pro-/regression per subtype (crop-yield, vegetation type, urban class).

The output per subtype is managed over the surface area for which the computation was performed. Cells within the area but lacking any information (nodata values), are not taken into account and therefore these cells are not visible in the output totals.

7.13 Define Startpoints

WHY?

iMOD offers the possibility to trace particles throughout the model extent. For these simulations it is necessary to define the starting locations (startpoints) for these particles (pathlines).

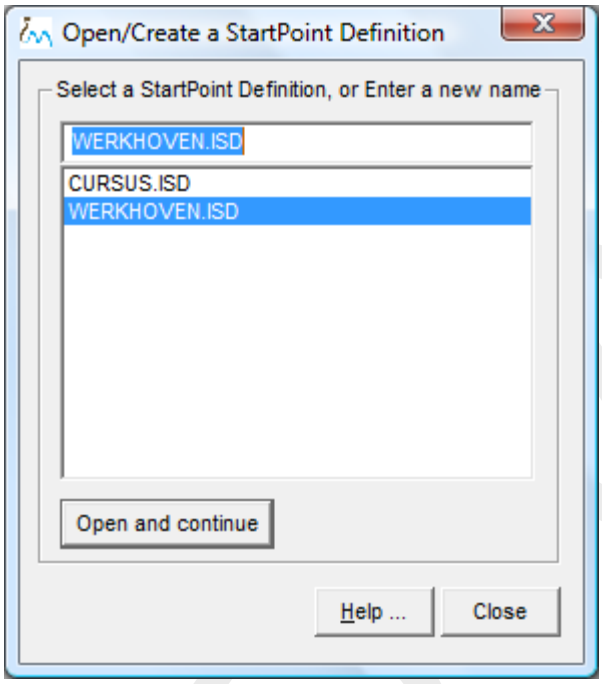
WHAT?

The starting location for particles is defined inside polygons, along lines and/or on the edge of circles around specified points. All these are specified in normal 3D-coordinates (x,y,z) and will be translated to the model at trace time. In this manner, a startpoint definition, can be easily (re) used to other model resolutions and/or model extinctions.

HOW?

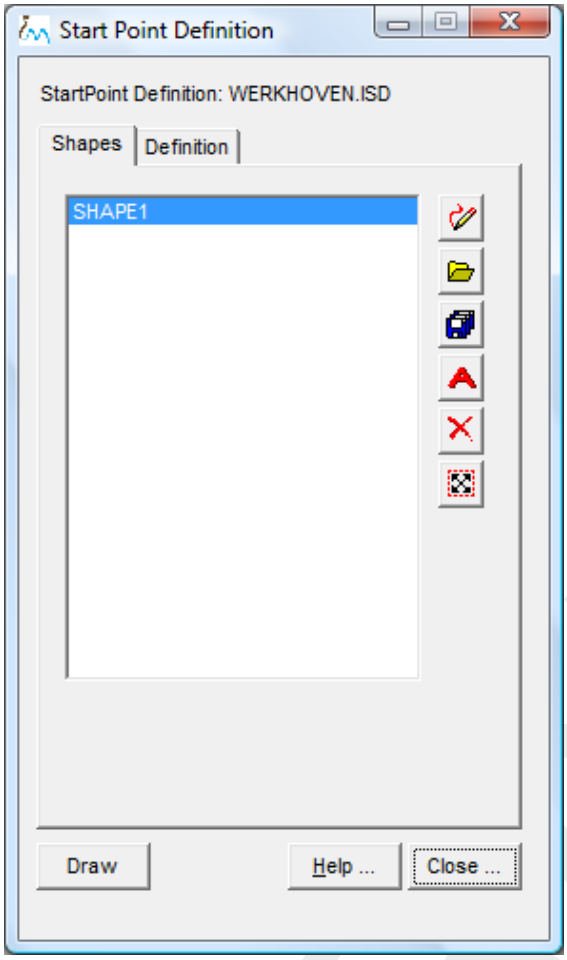
Select the *Toolbox* option from the main menu and then choose *Define Startpoints* to start the *Open/Create a Startpoint Definition* window.

Open/Create a Startpoint Definition window:



Select a StartPoint Definition, etc . . .	Enter or select an existing *.ISD file from the list. Those *.ISD files are found in the {user}\startpoints folder, see SrefiF.ISD for more detailed information on *.ISD files.
Open and continue	Click this button to open the selected *.ISD file and start the *StartPoint Definition* window.
Help . . .	Click this button to start the Help functionality
Close	Click this button to close the Open

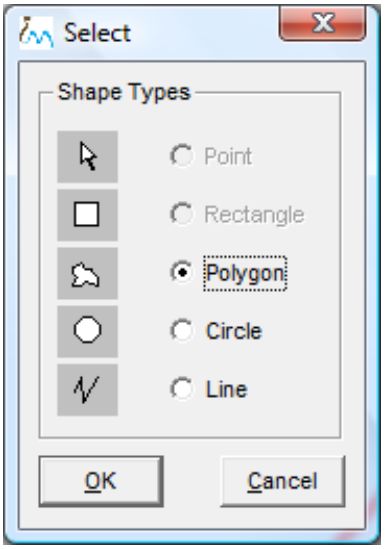
Start Point Definition window, Shapes tab:



Startpoint Definition	Displays the name of the current opened *.ISD file from the folder {user}\startpoints.
	*Draw* Click this button to start the *Select* window in which you select the shape that defines the lateral position of the startpoints (see next page).
	Click these buttons to draw, open, save, delete or rename a shape. More detailed information can be found in section 2.6.2
Draw	Click this button to draw the spatial location of the startpoints of the selected shapes.
Help ...	Click this button to start the *Help* functionality.
Close ...	Click this button to close the *StartPoint Definition* window. You will be asked to save the adjustments to the opened *.ISD file.

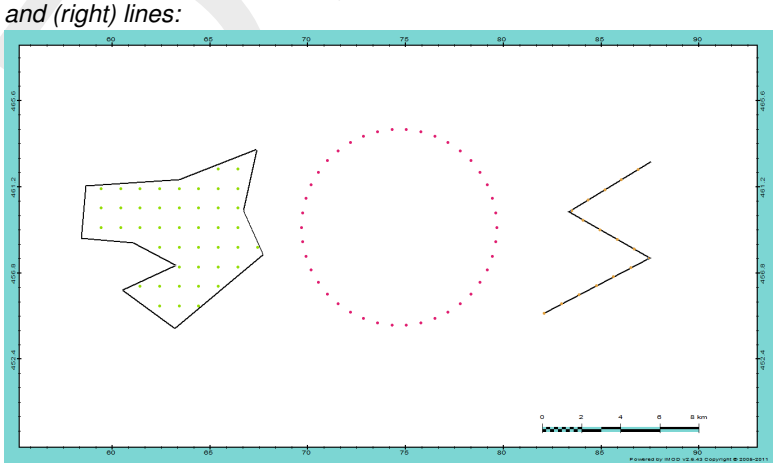
The *Select* window in which you select the shape that defines the lateral position of the startpoints.

Select window:

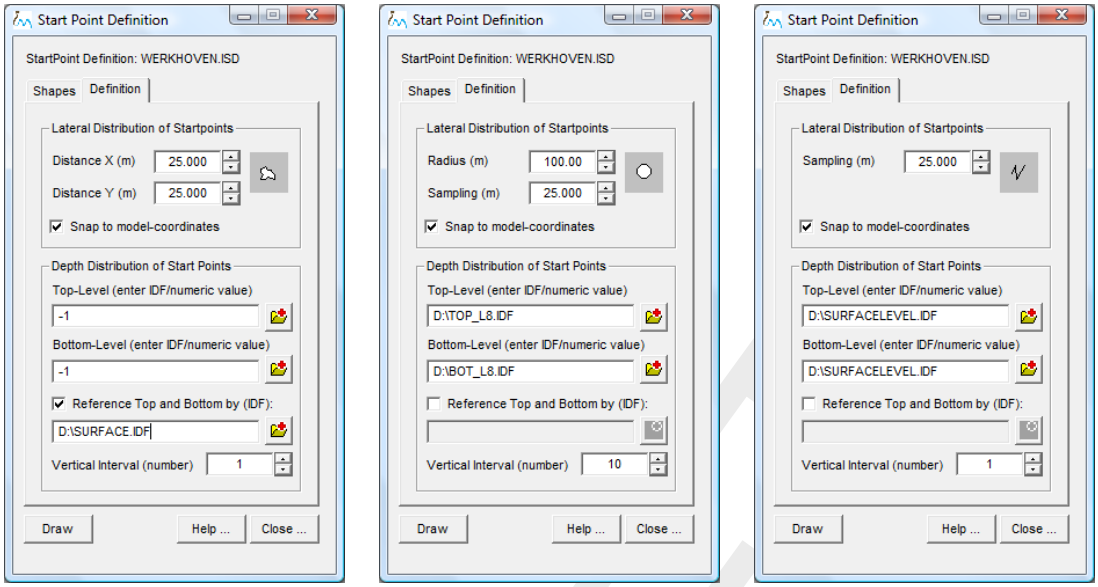


	*Point* Click this option to define startpoints for a single point
	*Rectangle* Click this option to define startpoints within a rectangle (NOT YET IMPLEMENTED)
	*Polygon* Click this option to define startpoints within a polygon
	*Circle* Click this option to define startpoints on a circle
	*Line* Click this option to define startpoints on a line

Example of startpoint created by (left) polygons, (middle) circles and (right) lines:



StartPoint Definition window, Definition tab, (1) for polygons (2) on circles and (3) along lines:



Distance X, Distance Y	Enter the lateral distances in X and Y direction for those startpoints inside the selected *Polygon* shape.
Snap to model coordinates	Click this item to snap the startpoints location to the nearest model centroid (NOT YET IMPLEMENTED)
Radius	Enter the radius of the circle.
Sampling	Enter the distance between the startpoint on the radius of the circle (*Circle* shape) or on the line (*Line* shape).
Top-Level (enter IDF/numeric value) *Bottom-Level (enter IDF/numeric value)*	Enter a IDF that represents the top and/or bottom elevation of the start-points, e.g. *D:\TOP_L8.IDF* and *D:\BOT_L8.IDF*. For each lateral position of a startpoint, the values will be read from the entered IDF-files. Moreover, you can enter a constant value (e.g., *-1*) to indicate that all values are constant.
Reference Top and Bottom by (IDF)	Check this item to enter an IDF-file as reference level. The values from this IDF will be used to add or subtract the values from the Top-level and Bottom-level. In the example in Figure 5-9d-i, the final values for the Top elevation will be those values in the *D:\SURFACE.IDF* minus 1 (the entered numeric value).
	*Open IDF* Select this button to select an IDF-file from the file selector.
Vertical Interval (number)	Enter the number of vertical intervals between the values from the top and bottom elevation. By entering the number: 1: iMOD will position one startpoint in-between the values for the top and bottom elevation; 2: will yield a startpoint equal to the top and a startpoint equal to the bottom elevation; $>$2: yields startpoints equally distributed between the top and bottom elevation.

Note: iMOD will not check, at this time, for the non-existence of any entered IDF-file. These files will be checked at runtime for the particle tracking.

7.14 Start Pathline Simulation

WHY?

Particle tracking analyses are particularly useful for delineating capture zones or areas of influence for wells.

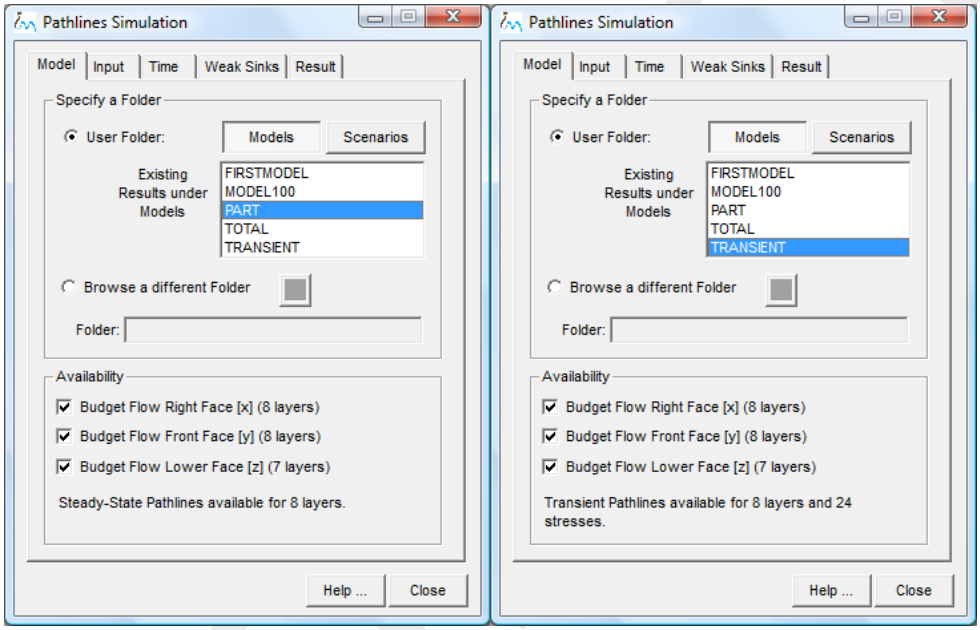
WHAT?

iMOD is equipped with iMODPATH that is a modified version of MODPATH version 3 (Pollock, 1994). iMODPATH is a particle tracking code that is used in conjunction with iMODFLOW. After running a iMODFLOW simulation, the user can designate the location of a set of particles. The particles are then tracked through time assuming they are transported by advection using the flow field computed by iMODFLOW. Particles can be tracked either forward in time or backward in time.

HOW?

Select the option *Toolbox* from the main menu and then choose the option *Start Pathline Simulation* to open the *Pathlines Simulation* window.

Pathlines Simulation window, Model tab (i) for a steady-state model (ii) for a transient model:



User Folder:	Select this radio button to select one of the following options: ***Models:*** Click this radio button to list all existing models in the folder {user}\models ***Scenarios:*** Click this radio button to list all existing models in the folder {user}\scenarios
Existing results under Models	Select one of the folders that appear in this listbox.
Browse for a different folder	Select this radio button to specify a result folder from a different location, other that the {user}**models** and/or {user}**scenarios** folder.
	*Open File* Click this button to search for a folder on disk
Folder:	Enter the name of the folder, otherwise the name of the folder will be displayed after accepting the folder from the *Open File* button. The *Availability* status will update each time an alteration is noticed in the folder name.
Availability	iMOD will check those results that are available and includes the number of modellayers.

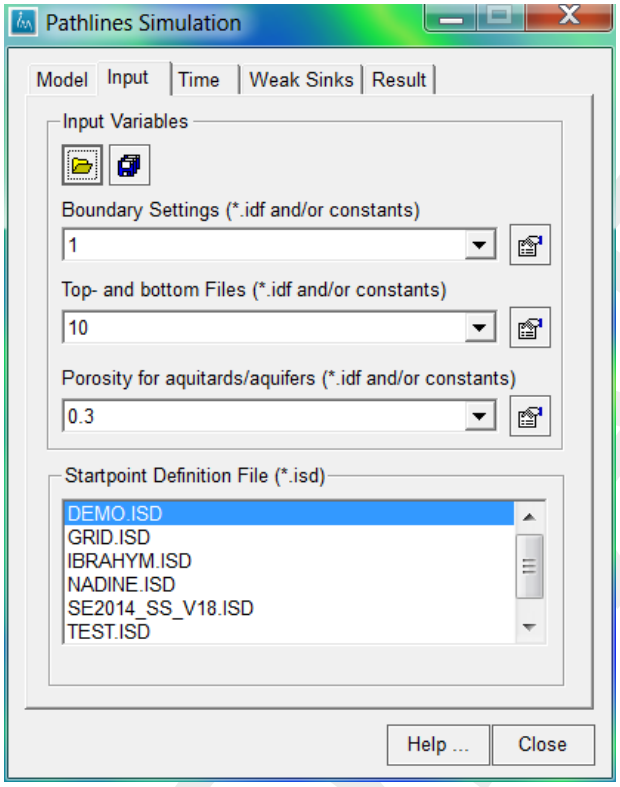
Alias	**Subfolder**

Budget Flow Lower Face [z] (.)	BDGFLF\BDGFLF_L*.IDF
Budget Flow Right Face [x] (.)	BDGFRF\BDGFRF_L*.IDF
Budget Flow Front Face [y] (.)	BDGFFF\BDGFFF_L*.IDF

Displays the status of the selected model. Whenever data is missing the other tabs are greyed out.

Help...	Click this button to start the HELP functionality.
Close	Click this button to close the Pathlines Simulation window.

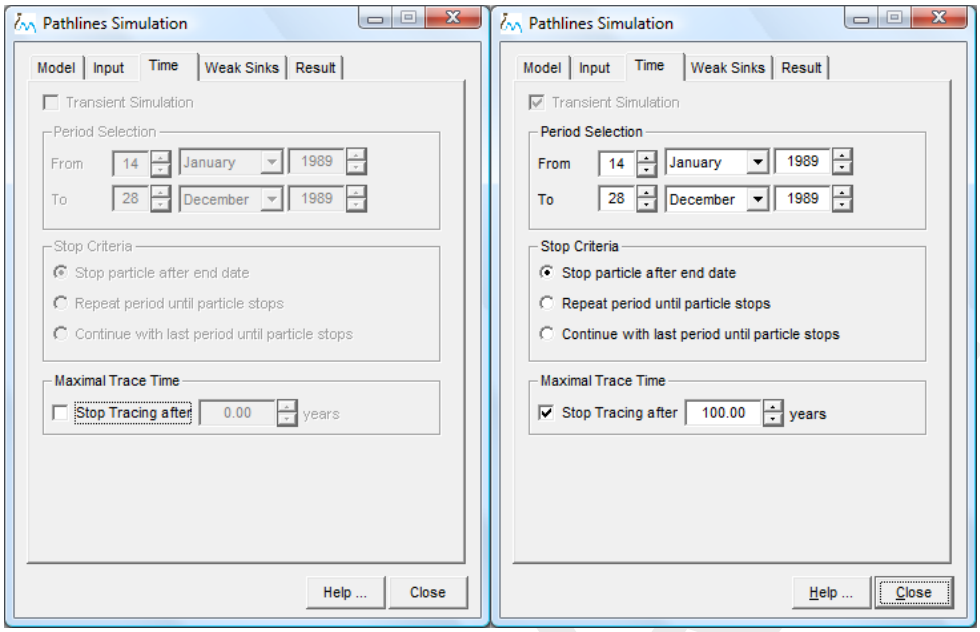
Pathlines Simulation window, Input tab:



	*Open* Click this button to open an *.IPS (iMOD Pathlines Settings) file.
	*SaveAs* Click this button to save the current input settings to an *.IPS file.
	*Properties* Click this button to open the Input Properties window (see SrefTMO.SPS.InputProp).
Boundary Settings	Click the dropdown menu to view the current files and/or values to be used as boundary settings. Any value greater than zero determines the active flow extent in which particle tracking is allowed. As a default the boundary of the flow simulation can be used (see section 7.9), however, it is not obligatory to use that particular file.
Top- and bottom	Click the dropdown menu to view the current files and/or values to be used as top- and bottom elevations of the model layers.
Porosity	Click the dropdown menu to view the current files and/or values to be used as porosity.
Startpoint Definition File	Select one of the existing *.ISD files from the list menu to use as startingpoints for the particle simulation. Those *.ISD can be created by the *Startpoint Tool* (page 297) and are located in the {user}\STARTPOINTS folder.

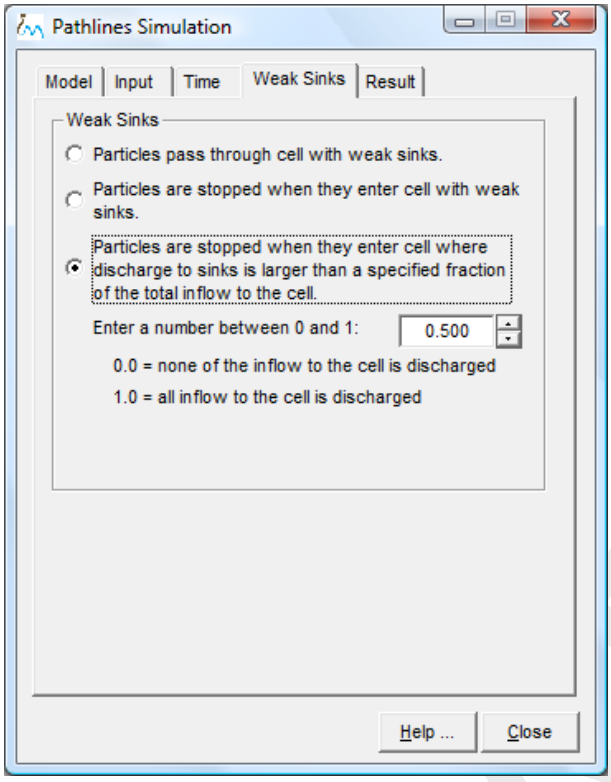
Help. . .	Click this button to start the HELP functionality.
Close	Click this button to close the Pathlines Simulation window.

Pathlines Simulation window, Time tab:



Transient Simulation	This checkbox is selected automatically for transient solutions and deselected for steady-state solutions.
From, to	Specify the start- and end period whenever a transient solution is used. The input fields are filled in automatically and defined by the content of the selected result folder on the Model tab.
Stop criteria	Select one out of three options to specify how particles are to be treated whenever they are not captured before the end of the existing solution files (only for transient simulations). ***Stop Particle after end date:*** Stop the particle simulation whenever the elapsed time of the particle exceeds the given To date. ***Repeat period until particle stops:*** Repeat the period selection (From-To) until the elapsed time of the particle exceeds the increased To date. ***Continue with last period until particle stops:*** Use the last solution within the From-To period, to simulate all particles until they are captured.
Stop tracing after	Enter the number of years for which particles need to be traced.
Help. . .	Click this button to start the HELP functionality.
Close	Click this button to close the Pathlines Simulation window.

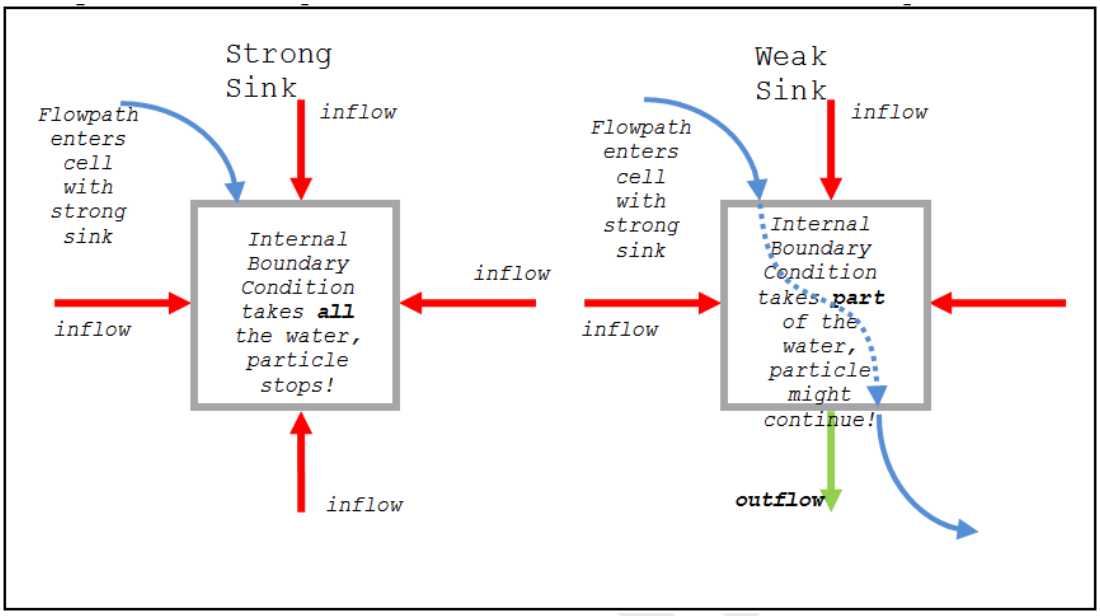
Pathlines Simulation window, Weak Sinks tab:



Particles pass . . .	Select this option to let particles pass any cell with a weak sink, no matter how "weak" they are. Be aware of the consequences of this option, since particles tend to trace over long distances until they are captured by a strong sink. This option could be wise to use whenever *Forward Tracing* option is selected on the *Result* tab.
Particles are stopped . . .	Select this option to stop particles at any cell with a weak sink, no matter how "weak" they are. Use this option whenever *Backward Tracing* is selected on the *Result* tab.
Particles are stopped when they enter . . .	Select this option to let particles stop whenever they enter a cell where the discharge is larger than a fraction of the total inflow. Whenever the fraction=1.0, particles stop at a strong sink only, as fraction=0.0, they will always stop, no matter the size of the total outflow.
Help. . .	Click this button to start the HELP functionality.
Close	Click this button to close the Pathlines Simulation window.

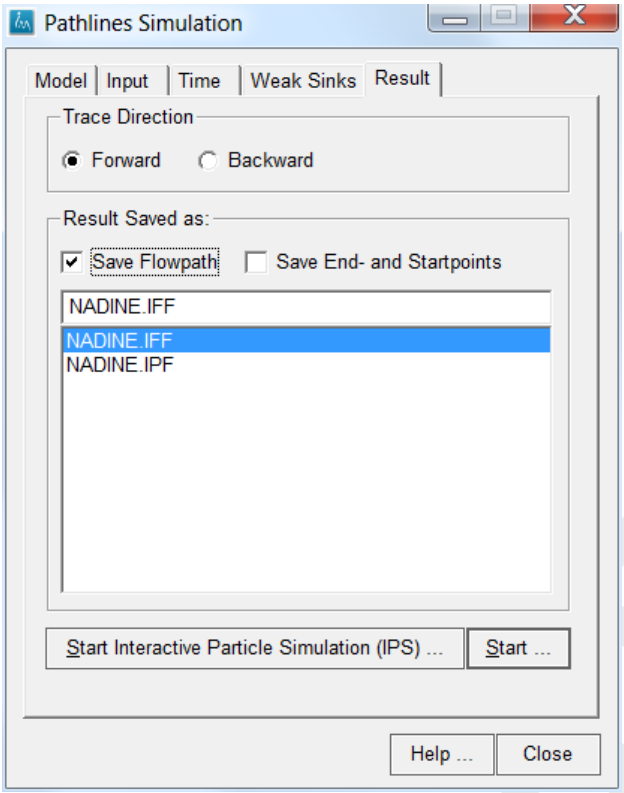
Note: The final representation of flowpaths and/or endpoints of particles is influenced significantly by the treatment of weak sinks. A strong sink is defined as a model cell in which all flowterms are directed into the model cell. Weak sinks are those that have at least one flow component that directs outside the model cell.

Example of Strong sink (left) and a Weak sink (right):



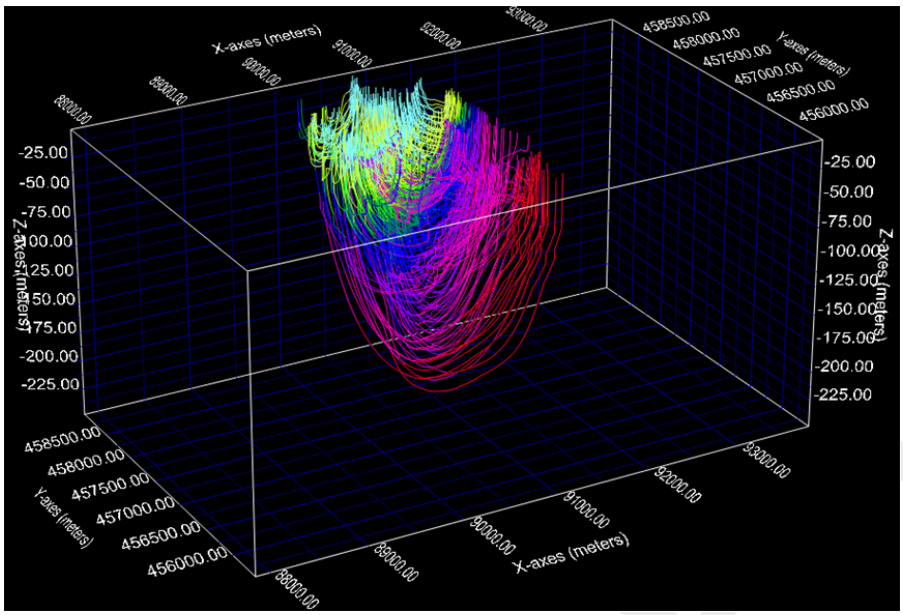
There is no way that the particle tracking algorithm itself can decide correctly whether a particle should stop or not. Moreover, it is an essential scale issue, since strong or weak sinks do not exist in reality. As the scale size (rastersize) increases, the occurrences of weak sinks in the model, will increase. This is simply caused by the phenomenon that a single coarse modelcell should represent more than one internal boundary condition and represents a larger area than the area taken by the boundary condition. So, the flowterms of these coarse cells represent an average flowfield that represent on average particles that should stop and particles that should continue. Unfortunately, that particular particles can not be simulated with the coarse model, so one should decide whether the particles should stop, continue of stop/continue depending on the ratio between the total inflow and the outflow component. These three options can be chosen in iMOD

Pathlines Simulation window, Result tab:



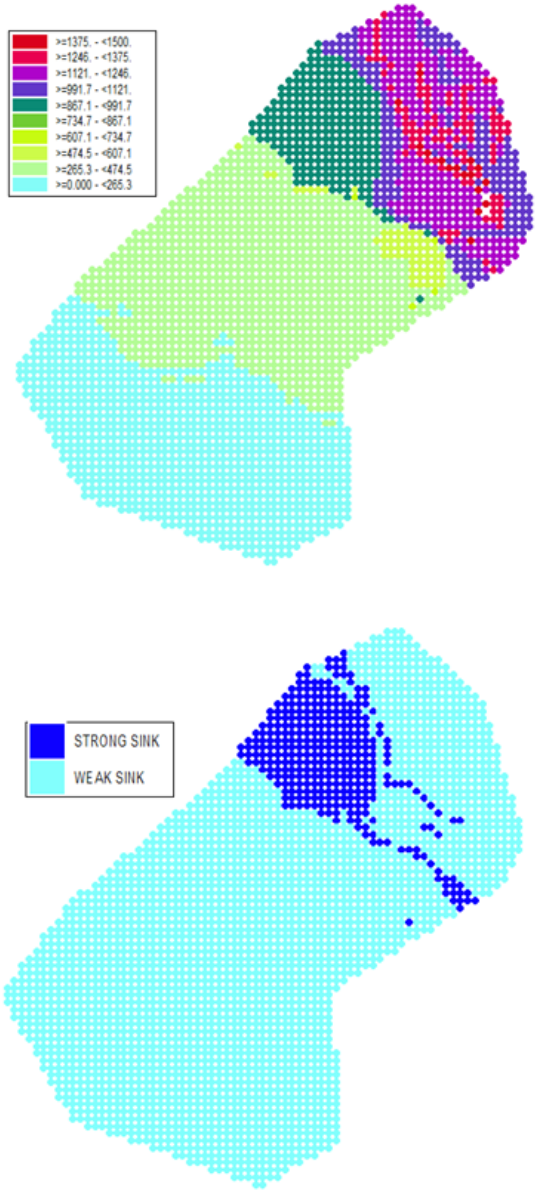
Trace Direction	Select one of the following options: ***Forward:*** This option will compute pathlines in the direction of flow ***Backward:*** This option will compute pathlines against the direction of flow
Result Save as:	Select on the following options: ***Save Entire Flowpath (*.iff):*** Select this option to save the entire flowpath in an *.IFF file (see section 9.8 for more details). The IFF has the following attributes: PARTICLE_NUMBER – number of the released particle; ILAY – modellayer of the current particle position; XCRD. – X coordinate of the current particle position; YCRD. – Y coordinate of the current particle position; ZCRD. – Z coordinate of the current particle position; TIME(YEARS) – elapsed time on the current particle since moment of release; VELOCITY(M/DAY) – current velocity of the particle; IROW – row index of current location; ICOL – column index of current location.

Example of flowpaths (IFF) plotted by the 3D Tool.



Save End- and Startpoints (*ipf):
Select this option to save the entire flowpath in an *.IPF file (see section 9.7).
The IPF has the following attributes:
SP_XCRD – X coordinate of starting location of particle;
SP_YCRD – Y coordinate of starting location of particle;
SP_ZCRD – Z coordinate of starting location of particle;
SP_ILAY – modellayer of starting location of particle;
SP_IROW – row index of starting location;
SP_ICOL – column index of starting location;
EP_XCRD – X coordinate of end location of particle;
EP_YCRD – Y coordinate of end location of particle;
EP_ZCRD – Z coordinate of end location of particle;
EP_ILAY – modellayer of end location of particle;
EP_IROW – row index of end location;
EP_ICOL – column index of end location;
TIME(YEARS) – elapsed time of particle at end location;
MAXLAYER – deepest model layer that a particles passed;
DISTANCE – traveled distance of particle from begin to end location;
IDENT.NO. – number of particle;
CAPTURED_BY – code identification of capture:
-1 – error occurred
0 – initial value
1 – inactive cell
2 – velocity is zero
3 – strong sink (no outflow)
4 – weak sink (regardless of flow)
5 – weak sink outflow greater than fraction
6 – modelboundary reached
7 – elapsed time greater that maximum time allowed.

Example of startpoints plotted by their age (upper figure) and their captured_by code (lower figure):

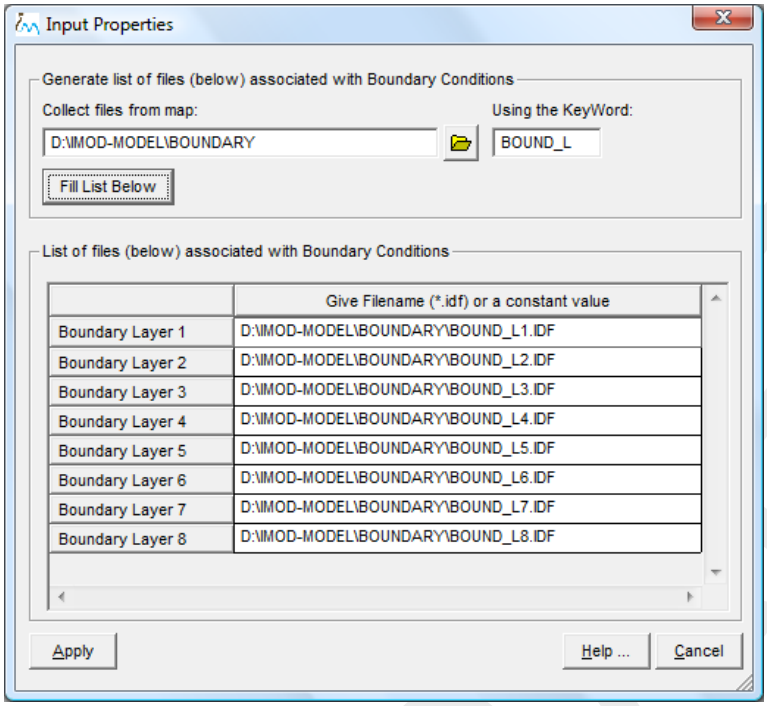


Start IPS . . .	Click this button to start the Interactive Pathline Simulation (IPS), see 7.15. iMOD will start the 3-D Tool directly and opens the IPS tab in the 3-D Tool in which pathlines might be computed interactively. The output of those simulation go straight into OpenGL and will be rendered, no results will be saved to disc, use the *Start . . .* option for that instead. Any selected SDF file will be neglected whenever the IPS is started as the starting points for the pathlines will be defined in the IPS-Tool.
Start . . .	Click this button to start the pathline simulation. iMOD will start directly, or asks for confirmation in the situation that the output file exists already. The iMODPATH runfile will be saved in the Model folder as given on tab *Model*. This runfile can be re-used easily by the iMODBatch function IMODPATH (see 8.6.6). A log-file will be saved there as well, that given information about the used files.

7.14.1 Input Properties

A pathline simulation needs particular files that can be easily defined (and stored in an *.IPS file) with the *Input Properties* function. Select the *Input Properties* button on the *Input* tab of the *Pathlines Simulation* window to start the *Input Properties* window.

Input Properties:



Collect files from Folder:	Enter the folder name to be used to construct filenames, e.g. d:\imod-model\boundary
	*Open* Click this button to select a folder from the file selector.
Using the keyword:	Enter the keyword that need to be added to the foldername, e.g. boundary_l
Fill List Below:	Click this button to construct filenames for all modellayers. The above mentioned example yield the filenames d:\imod-model\boundary\boundary_l1.idf upto modellayer 8.
List of files . . .	Display the filenames and/or constant values to be used in the pathline simulation.

7.15 Interactive Pathline Simulator

(*Extension of this functionality is in progress; we expect it will be completed and part of a Deltares-release of the iMOD-executables in February 2016.*)

WHY?

Interactive Particle tracking analyses can be essential in understanding how a geohydrological system works. For reasons of system analyses this tool might be having a great additional value. The main purpose of this tool is to gain insight in geohydrological processes by using animated groundwater flows and the ability to interactively change your point of view in a 3D-environment.

WHAT?

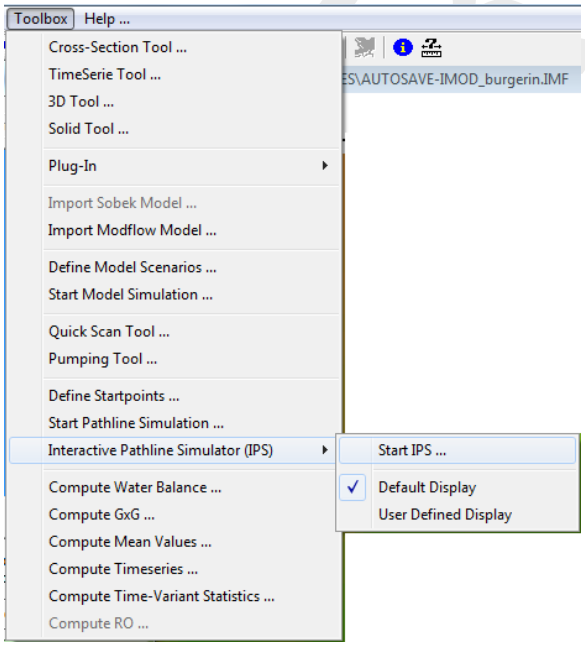
iMOD is equipped with iMODPATH that is a modified version of MODPATH version 3 (Pollock, 1994). iMODPATH is a particle tracking code that is used in conjunction with iMODFLOW. After running a iMODFLOW simulation, the user can use the runfile of iMODPATH (see section section 8.6.6) to enter the 3-D tool of iMOD (see section section 7.3). In this 3-D environment, the user is able to interactively analyze the flow path of groundwater particles, e.g. pause the flow at arbitrary moments. Particles can be tracked either forward in time or backward in time.

Note: The IPS runs only for a Steady State flow situation. In case a transient iMODPATH run file is uploaded, only the first stress period is read and used for particle tracking.

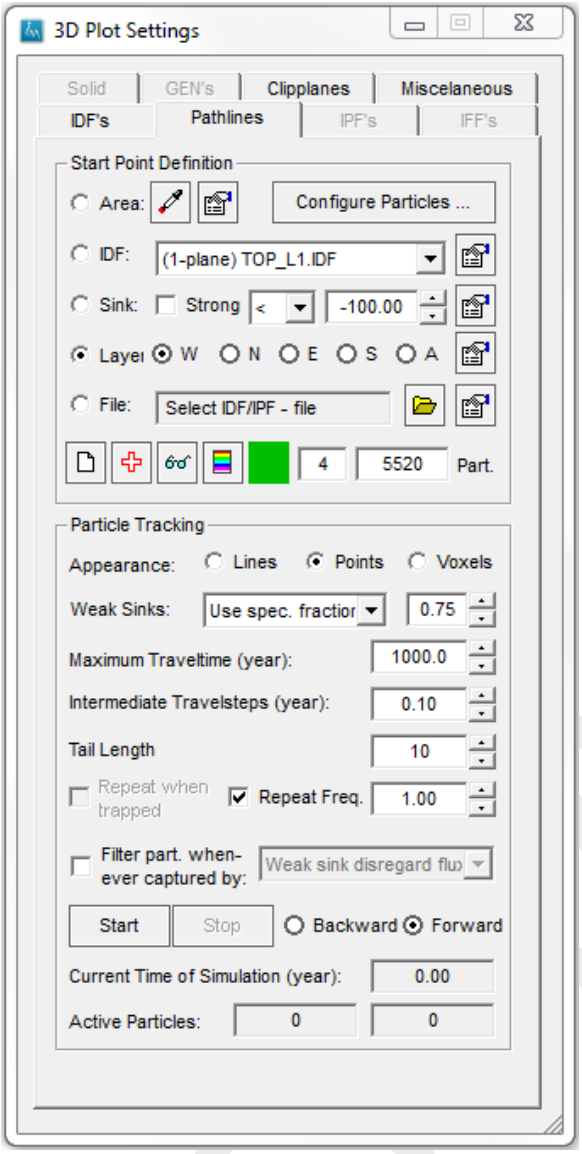
HOW?

Select the option *Toolbox* from the main menu, click on the submenu option *Interactive Pathline Simulator* and then on *Start IPS...* to select an appropriated runfile for iMODPATH. The 3-D tool will start and all the necessary files (top, bottom and fluxes) will be allocated for the current zoom extent. It is easily to start the tool for limited areas of interest, especially to be more efficient. On default iMOD starts the IPS with default display settings (checked in IPS-submenu), this means that all the files available in the runfile are loaded into the 3D-tool. Another option is to choose *User Defined Display* and iMOD only loads the files selected in the iMOD manager.

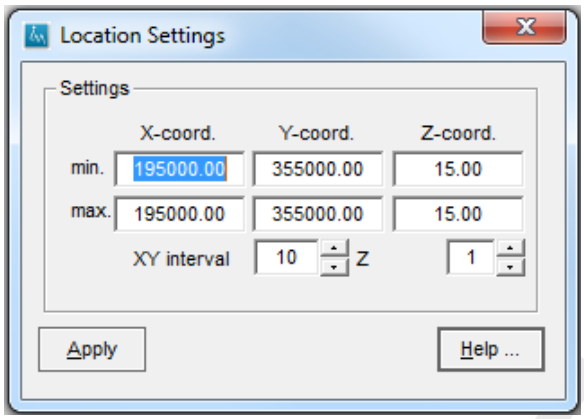
IPS-submenu:



3D Plot Settings window, Pathlines Folder tab:



Start Point Definition	With the following options the starting point of the particles can be defined.
Area:	Select this option to define a rectangular area with starting points somewhere in the modeled area.
	*Select* Click on this button to drag a rectangle (NOT AVAILABLE IN CURRENT VERSION!).
	*Attributes* Click on this button to open the *Location Settings* window:

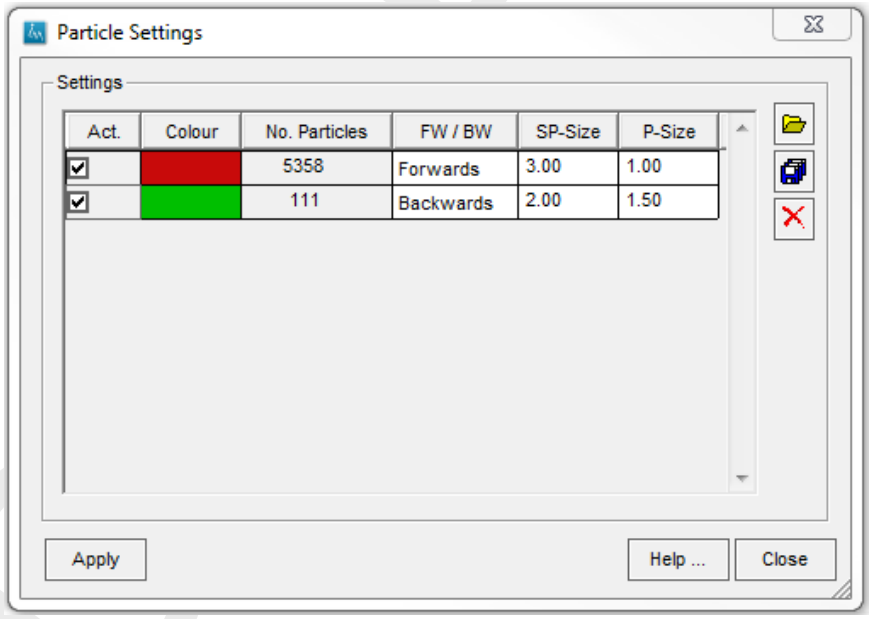


Enter the minimum and maximum x, y and z-coordinates to define the extent of the box in which the starting points needs to be placed.

XY interval	Enter a value for the amount of staring points in x- and y-direction.
Z	Enter a value for the amount of starting points in the z-direction.

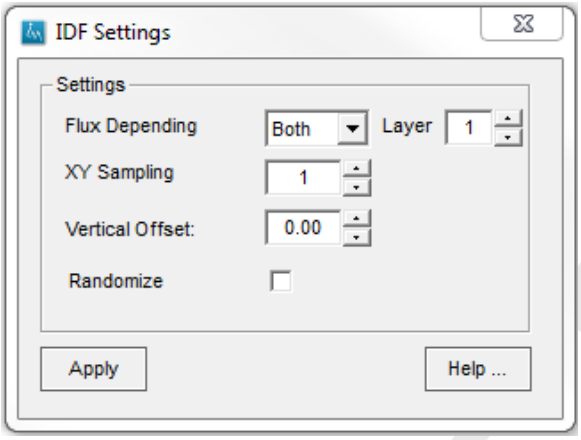
Configure Particles. . .

Click on this option to open the *Particle Settings* window:



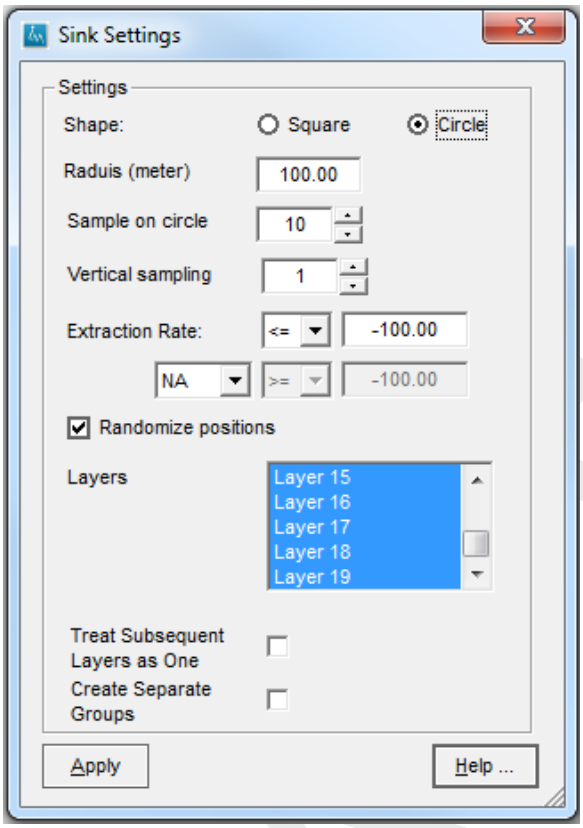
Act.	Check this option to activate a particle group.
Colour	Click on this field to change the colour of the particles and starting points.
No. Particles	Total amount of particles in a group.
BW/FW	Click on this field to change the tracking option in "Forward" or "Backward".
SP-Size	Click on this field to change the size of the starting point.
P-Size	Click on this field to change the size of the particle.
	*Open PTF* Opens an PTF(Particle Tracking File)-file containing the particle settings per group.
	*Save As* Saves the particle group settings in a IPF-file, if points needs to be analyzed in iMOD separately from the IPS-tool, or in a PTF-file for later use in IPS.

IDF:	Select this option to define the starting points based on the in a loaded IDF-file available points. Make use of the dropdown menu to select the preferred IDF-file.
	*Attributes* Click on this button to open the window *IDF Settings*:



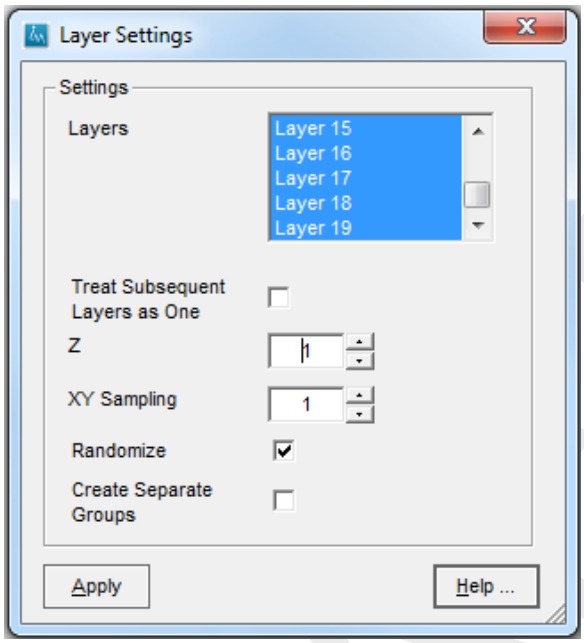
Flux Depending	Select from the dropdown menu the preferred vertical direction of the flux in a particle related cell: Up (only positive fluxes are drawn), Down (only negative fluxes are drawn) or both (all fluxes are taken into account).
Layer	iMOD uses the fluxes from the layer defined with this option, e.g. if Layer=1, iMOD uses the fluxes from model layer 1.
XY Sampling	Give the horizontal distance in meters between the starting points.
Vertical Offset	Give the vertical distance in meters between the top of the layer and the particle start position.
Randomize	Check this option to randomly place the starting points with the given settings.

Sink:	Select this option to locate the starting points on the location of the in the model area available sinks. Check *Strong* to apply the given condition, defined with the dropdown menu and the value box, only to strong sinks. The background theory about strong and weak sinks can be found in section 7.14 under *Weak Sinks tab*.
	*Attributes* Click on this button to open the window *Sink Settings*:



Shape:	Choose the shape of the points on the sink locations; *Square* or *Circle*.
Radius (meter)	Enter the size of the radius in meters of the sinks related points.
Sample on circle	Enter a value for the horizontal influence area around the sink (in meters) to be shown in the simulation.
Vertical sampling	Enter the amount of starting points per layer in the vertical direction at the sink location, e.g. if 5 layers are selected and a vertical sampling amount of 10 is chosen, there will be in total 5x10=50 starting points in the vertical direction.
Extraction rate:	Define the range of the sink extraction rate. Only sinks with discharges within the given range are labeled as sinks.
Randomize positions	Check this option to randomly place the starting points with the given settings.
Layers	Select the layers to be taken into account in the PTS at the well locations (e.g. selecting 5 layers results into 5 groups with each the total amount of defined starting points).
Treat subsequent Layers as One	Check this option to evaluate all selected layers as one layer (be aware: only successive layers are consolidated to one layer), e.g. 5 selected layers with 10 vertical samples results in 50 starting points, with this option checked on there is only 1 group with 5 layers and 10 starting points in total.
Create Separate Groups	Check this option if it is preferred to treat each selected layer as an separate subgroup within 1 group, e.g. 1 group is made with the 5 selected layers, each containing 10 starting points (50 starting points in total).

Layer:	Select this option to locate the particles at one of the displayed model boundaries. W=West boundary, N=North boundary, E=East boundary, S=South boundary and A=all boundaries.
	*Attributes* Click on this button to open the window *Layer Settings*:



In this window it is possible to i.e. select the layers to place the starting points in.

Layers	Select the layers to be taken into account in the PTS at the border location(s).
Treat Subsequent Layers as One	Check this option to evaluate all selected layers as one layer (be aware: only successive layers are consolidated to one layer).
Z	Enter the amount of starting points per layer in the vertical direction.
XY Sampling	Give the horizontal distance in meters between the starting points.
Randomize	Check this option to randomly place the starting points with the given settings.
Create Separate Groups	Check this option if it is preferred to treat each selected layer as an separate subgroup within 1 group, e.g. 1 group is made with the 5 selected layers, each containing 10 starting points (50 starting points in total).

File:	Select this option to choose an IDF- or IPF-file for the start point definition.
	*Open* Use this button to search for and open the preferred file.
	*Attributes* Click on this button to open the window *IDF Settings* which is similar to the IDF-option settings window.
	*New* Click on this button to reset the Particle settings, the *Part.* value boxes are set on 0.
	*Add* Click on this button to add a new PSP (Particle Start Point) definition group to the simulation. Every time this button is used a new group is added to the particle simulation and the value in the first *Part.* value box is increased with 1.

	*Show Selection* Click on this button to analyze the location of the defined particles in advance of or during the particle simulation.
	*Color* Click on this button the change the color of the just defined point-group.
Part.	The first value box counts the amount of PSP definition groups specified by the user. Every time the ()-button is used, this value is increased with 1 (total value = amount of defined particle groups). In this case 4 start point definitions are specified. The second value box is the sum of all the defined starting points (in this case 5520). By clicking on Configure Particles. . . the total amount of particles per start point definition can be distinguished.

Particle Tracking	With the below mentioned options it is possible to set the graphic settings of the particle tracking. A change in some of the options is directly visible on screen.
Appearance:	Choose the representation of the simulated path line points; Lines, Points or Voxels (Not Available yet).
Weak Sinks:	Make a choice between "Use spec. fraction", "Particles stop. . . " and "Particles pass. . . ". Selecting "Use spec. fraction" allows the user to enter a fraction number in between 0 and 1 (See page 377 at *Particles are stopped when they enter. . .* for a more extended description). The other two menu items use a fixed factor value in their selection (see page 377 for an extended explanation about these other two Weak Sinks options).
Maximum Travel time (year):	Give the maximum time a particle can travel through the model area during the simulation before the particle disappears from the screen or restarts from its starting point, e.g. a value of 1000 years gives the user the opportunity to follow a particle for 1000 years of simulation time.
Intermediate Travel steps (year):	Give the time step size in between two shown particles. Examples: 1. an intermediate travel step of 0.10 year and a maximum travel time of 10 years results into 100 time steps, 2. An intermediate travel step of 1 year, on the other hand, and the same maximum travel time (10 years) results in 10 time steps. The actual time of the simulation is longer in the first example compared to the second example, because the actual simulation time is based on the amount of time steps (more=longer).
Tail Length	Enter a value to define the length of the particle "shadow". The shadow shows a part (the length of the tail) of the particles pathline, in real life comparable to the contrails of aircrafts in the sky.
Repeat when trapped	Check this option to restart a particle at its starting point after the particle disappeared, e.g. it was trapped in a sink or it flews outside the model domain (only available when *Repeat Freq.* is not checked).
Repeat Freq.	Check this option to restart a particle after the entered time step value, e.g. at a frequency of 1.00 iMOD starts a new particle simulation every year of simulation time (only available when *Repeat when trapped* is not checked).
Filter part. whenever captured by:	Check this option to be able to apply a particle filter to the whole simulated particle cloud. In this refinement there are the following options: *1. Inactive cell:* restart only the particles in the simulation that are captured by an inactive cell. *2. Zero velocity:* restart only particles that have no velocity at the moment they are trapped. *3. Strong sink:* restart only the particles that are captured by strong sink(s). *4. Weak sink disregard flux:* restart only the particles that are trapped by weak sinks with a negligible flux compared to the total inflow. *5. Weak sink flux gt frac:* restart only the particles that are stopped by weak sinks with a flux larger than the fraction of the total inflow. *6. Boundary grid reached:* restart only the particles that reached the boundary of the presented model grid. *7. T greather than max T:* Restart all particles when the current simulation time is greater than the maximum Travel time.

Start	Click on this button to start the simulation.
Stop	Click on this button to stop the simulation.
Backward	Select this option to track the particles backward.
Forward	Select this option to track the particles forward.
Current Time of Simulation (year):	Counter for displaying the current time of simulation. It counts from 0 until the set *Maximum Traveltime (years)* with a time step given by textitIntermediate Travelsteps (year). The counter starts over again, when the maximum travel time is reached and a repeat-option is selected.
Active Particles:	Counters for the total amount of particles at the beginning of the simulation (left value box) and the amount of active particles during the simulation (right value box).

7.16 Waterbalance

WHY?

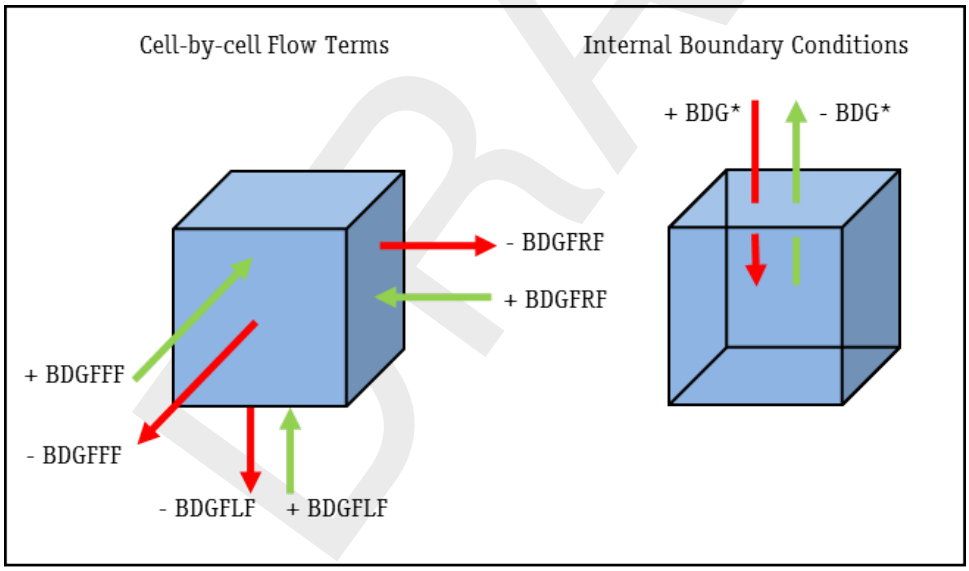
The *Waterbalance Tool* can be used to calculate the sum of budget quantities for different components for specified regions within the model extent and/or the entire model extent. Several budget items can be selected, as well as particular mode layers and/or model output periods, see section 7.16.1. The output can be analysed in the iMOD Water balance Analyser, see section 7.16.2.

WHAT?

The *Waterbalance Tool* uses specific results from iMODFLOW that start with BDG*.IDF and/or MetaSwap output file starting with MSW*.IDF. Those files are stored in folders with identical names, such as BGGFLF\BDGFLF*.IDF. All these files store the flow quantity in m^3/day (cubic meters a day). As a rule of thumb the flow quantity is negative whenever it leaves the cell and positive whenever it enters the cell. Flow that leaves the cell from the Right Face (BDGFRF), the Front Face (BDGFFF) and/or the Lower Face (BDGFLF) is negative. However, for flow terms that enter the cell from the Left Face, Back Face and/or Top Face of a model cell, the appropriate flow terms from the adjacent cells need to be multiplied with minus 1, to be consequent. For Internal Boundary Conditions (e.g. rivers, drainage systems, recharge, wells) a similar approach is valid: water that enters the cell is positive, withdrawal of water is negative. Water that flow from a specified water balance region to another region is noted netagive as it leave the corresponding water balance region.

Note: The waterbalance assumes that the dimensions for data that is included are in m^3/day. Bear in mind that fluxes that are produced by MetaSWAP are in m^3/m^2 and represent cumulative values as specified in the TIME_SIM.INP. Those may NOT overlap with the output file from iMODFLOW.

Example of budget terms inside and outside a computational cell for (left) cell-by-cell flow terms and (right) internal boundary conditions:



The Waterbalance Tool will just add all flow terms within (defined) areas and list them within a summary text files (*.TXT and *.CSV) to be opened with e.g. NotePad, TextPad and/or Excell. Alternatively, it is possible to read in a generated *.CSV file into the waterbalance analyser.

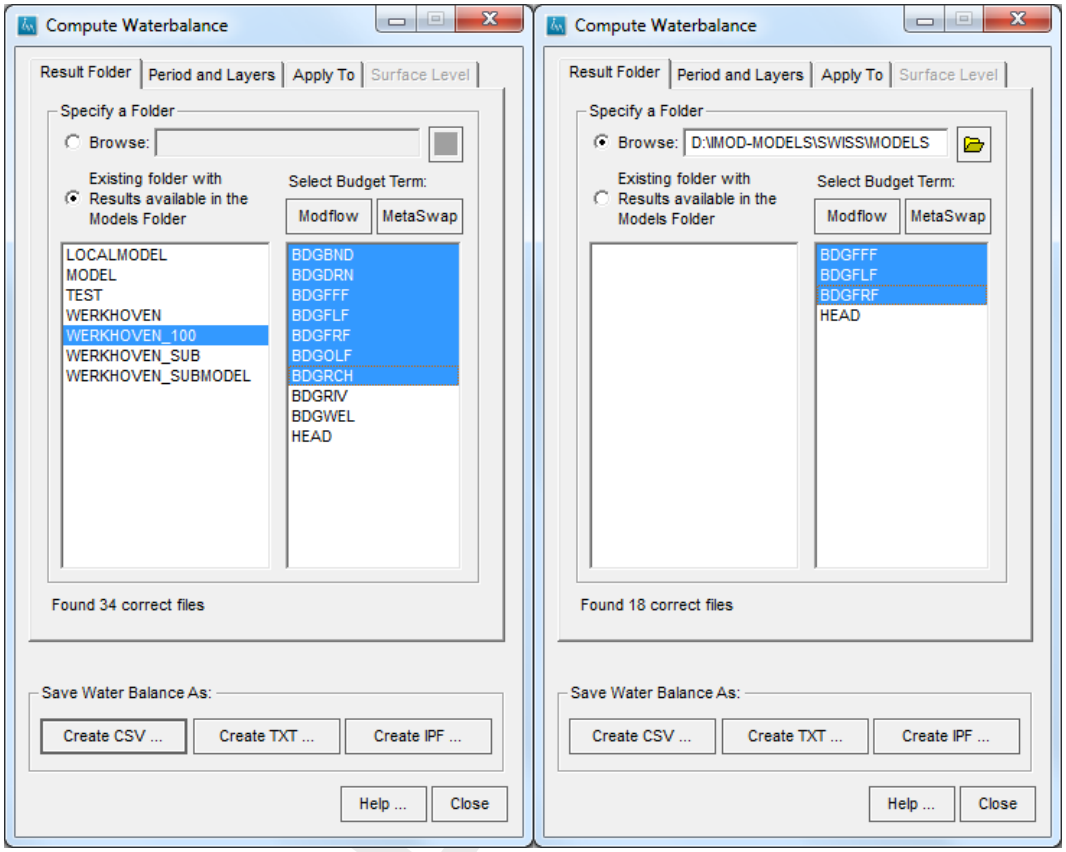
HOW?

To compute a water balance, select the option *Toolbox* from the main menu, choose *Water Balance* and then choose *Compute Waterbalance* to open the *Waterbalance* window. To analyse the results of a water balance computation, select the option *Toolbox* from the main menu, choose *Water Balance* and then choose *Analyse Waterbalance* to open the *Analyse Waterbalance* window

7.16.1 Compute Waterbalance

Start the *Compute Water Balance* window by selecting the option *Toolbox* from the main menu, choose *Water Balance* and then choose *Compute Waterbalance*.

Compute Waterbalance window, Result Folder tab; left for results from the Models folders and right from a different folder:

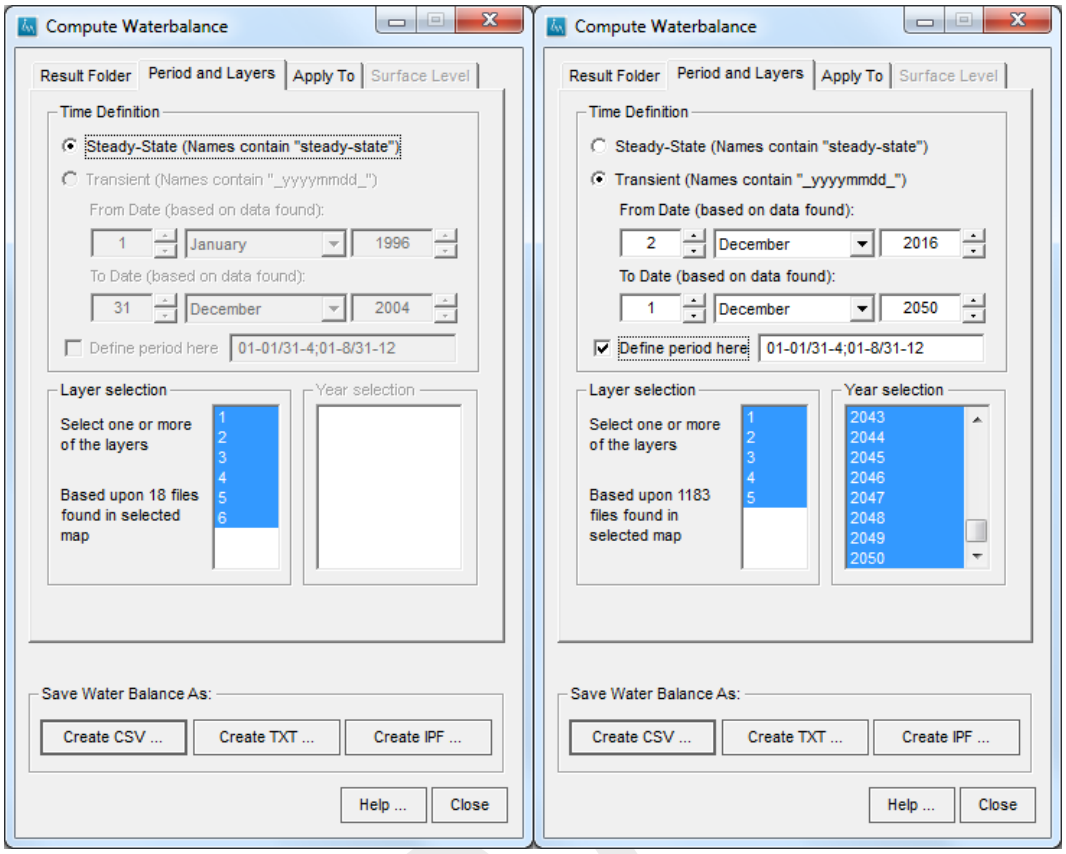


Browse:	Select this radio button to specify a result folder from a different location.
	*Select Folder* Click this button to search for a folder on disk and use that folder to list the existing budget terms. Alternatively, it is possible to enter a folder name into the string field.
Existing folder with Results ...	Select one of the folders that appear in this list box. These are the folders in the {IMOD_USER} \MODELS folder.
Select Budget Term	iMOD will list all sub folders under the current selected or entered folder. If none are selected the other tabs in this window are unavailable. Also, whenever incorrect folders (folders that doesn't contain the correct files) are selected this happens. Each time a single, or multiply budget terms are selected, iMOD starts searching for corresponding files. For large models, this can take a while, so it is often better to select the items in one click using Ctrl-Left- Mouse-Button of Shift-Left-Mouse-Button. The name convention is that the folder should start with BDG and/or MSW:

Subfolder	**Alias**
BDGBND\bdgbnd*	CONSTANT HEAD
BDGFLF\bdgflf*	FLUX LOWER FACE
BDGFRF\bdgfrf*	FLUX RIGHT FACE
BDGFFF\bdgfff*	FLUX FRONT FACE
BDGSTO\bdgsto*	STORAGE
BDGWEL\bdgwel*	WELLS

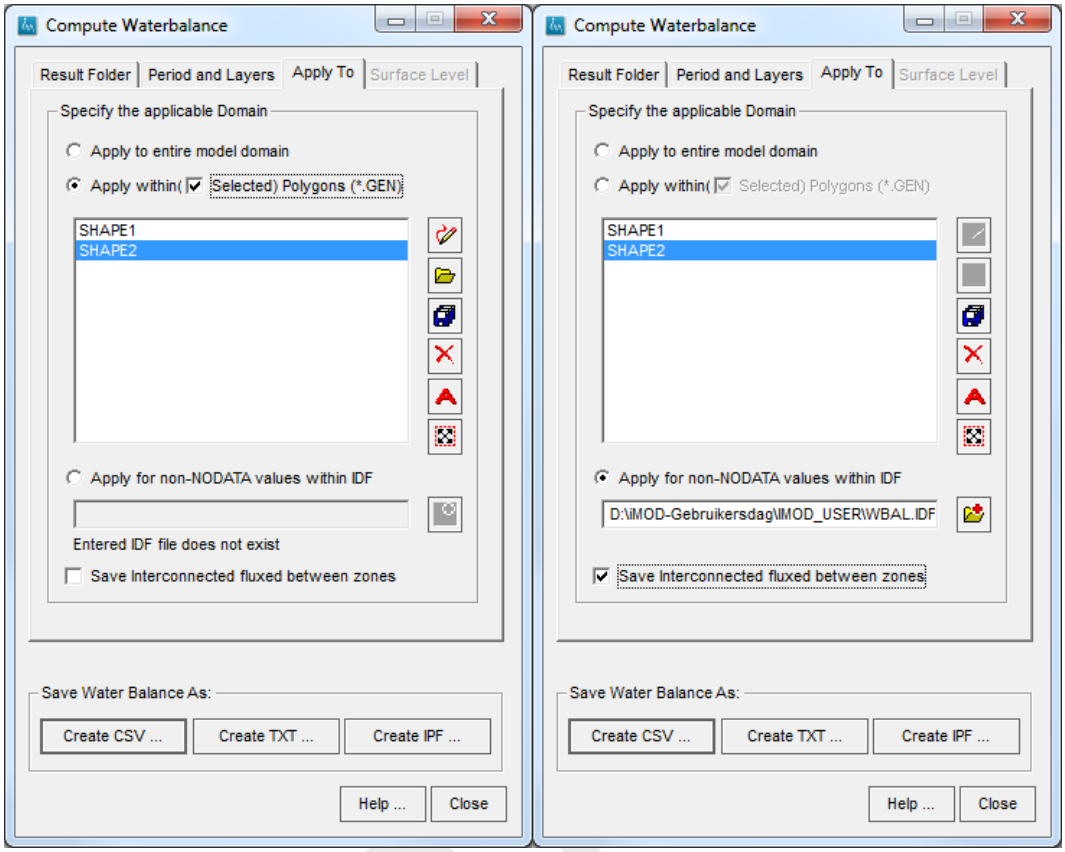
	BDGDRN\bdgdrn*	DRAINAGE
	BDGRIV\bdgriv*	RIVERS
	BDGEVT\bdgevt*	EVAPOTRANSPIRATION
	BDGGHB\bdgghb*	GENERAL HEAD BOUNDARY
	BDGOLF\bdgolf*	OVERLAND FLOW
	BDGRCH\bdgrch*	RECHARGE
	BDGISG\bdgisg*	SEGMENT RIVERS
	BDGIBS\bdgibs*	INTERBED STORAGE
	BDGCAP\bdgcap*	CAPSIM
	BDGDS\bdgds*	DECREASE WATER ST. ROOTZONE
	BDGPM\bdgpm*	MEASURED PRECIPITATION
	BDGPS\bdgps*	SPRINKLING PRECIPITATION
	BDGEVA\bdgeva*	NET EVAPORATION WATER
	BDGQRU\bdgqru*	RUNOFF
Modflow	Select this option to select budget terms that are specific for Modflow, automatically.	
MetaSwap	Select this option to select budget terms that are specific for MetaSwap, automatically.	
Found 34 correct files	Displays the number of files *(34)* that are correct (i.e. they fulfil the naming convention as described above and contain model layer information (_L*) and a date string (YYYYMMDD and/or YYYYMMDDHHMMSS) or STEADY-STATE keyword, e.g. BDGFLF_STEADY-STATE_L1.IDF or BDGFLF_20111231_L8.IDF or BDGWEL_20120414123015_L3.IDF.	
Create CSV . . . *Create TXT . . .* *Create IPF . . .*	Click this button to save the water balance computation into a CSV, TXT or IPF file, respectively. Use the extent *.TXT to define a “table” layout of the water balance in which all components are listed together. Use the extension *.CSV to define a “time series” layout of the water balance in which all component are listed as time series. This file can be read by the *Analyse Water Balance* Tool, see section 7.16.2. All default settings from the other tabs (*Period and layers* and *Apply to*) will be used. Check these before clicking this button!	
Help. . .	Click this button to start the HELP functionality	
Close	Click this button to close the *Compute Waterbalance* window	

Compute Water Balance window, Period and Layers tab (left) for a steady-state configuration and (right) for a transient configuration:



Steady-State	Click this radio button to use files only that contain the keyword steady-state, e.g. BDGFLF_STEADY-STATE_L1.IDF. This option is only available whenever these files can be found in the result folder selected on the *Result Folder* tab.
Transient	Click this radio button to use files only that contain date information (YYYYMMDD), e.g. BDGFLF_20111231_L1.IDF. This option is only available whenever these files can be found in the result folder selected on the *Result Folder* tab.
From Date:	Specify the start date (day, month and year) from which the water balance needs to be computed. It has been on default filled in with the earliest result file that could be found in the result folder selected in the *Result Folder* tab.
To Date:	Specify the end date (day, month and year) to which the water balance need to be computed. It has been on default filled in with the earliest result file that could be found in the result folder selected in the *Result Folder* tab.
Define Period here:	Enter the periods for which the water balance need to be computed solely. Each period consist of two dates delimited by a slash, e.g. dd-mm/dd-mm. In-between periods, use the ";" as a delimited. The convention is as follows: ***01-01/31-4:*** Periods starts on the first of January and ends after the 31^st of April ***01-8/31-12:*** Periods starts on the first of August and ends after the 31^st of December
Select one or more of the layers	Select one or more of the listed model layers. This content of the list box is based on the BDG*-files that are found in the selected result folder on the *Result folder* tab. Use the Ctrl-left mouse button to exclude or add individual layers, otherwise drag the mouse cursor to select layers.
Year selection	Select one or more of the listed years. The content of the list box is based on the bdg*-files that are found in the selected result folder on the *Result folder* tab.

Compute Waterbalance window, Apply To tab for (left) applied to selected shapes only or (right) for all non-NoDataValues within a selected IDF-file:

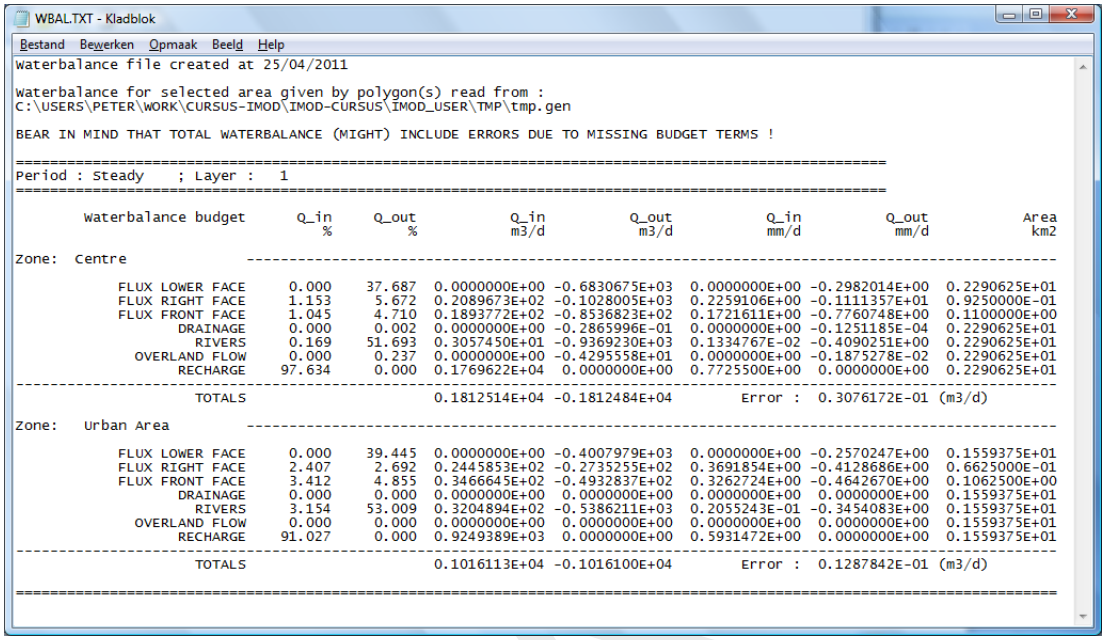


Apply to entire model domain	Select this option to compute all budget terms for the entire model extent. In fact this will be that area and dimension as described by the first BDG*.IDF to be read.
Apply within (Selected) Polygons (.GEN)*	Select this option to specify regions of interests by polygons. By default all polygons will be used for the water balance, however, if the option *Selected* is selected, only those polygons will be used that are currently selected, in the example above, this is polygon SHAPE2.
	Click these buttons to draw, open, save, delete or rename a shape. More detailed information can be found in section 4.2 *(*Create a GEN-file*).*
Apply for non-NODATA values within IDF	Select this option to use an IDF-file that describes the area of interest by its non-*NoDataValue*. For those locations of the first BDG*.IDF or MSW*.IDF read, the value within this IDF-file will be read and evaluated. A non existing IDF file name will outgrey the *Create CSV . . .*, *Create TXT . . .* and *Create IPF . . .* button.
	Open IDF-file Click this button to open an IDF-file. The selected file will be displayed, however, an IDF-file can be entered in the string field alternatively. If the IDF-file does not exists, the Apply button will grey out.
Save Interconnected fluxed between zones	Select this option to generate interconnected flux between the zones. This option is only valid whenever the flux terms BDGFRF and BDGFFF are active.

Note: After a computation for a water balance, iMOD will save an IDF-file at the location where you've saved the output file via *Create CSV*; *Create TXT*; *Create IPF*. This IDF files contains the location of the raster cells for each zone. Inspection of this file can lead to adjustment of the polygons used,

and/or reuse this particular IDF file for different model scenarios. Remember that in case of overlap in polygons, the value of the zone in the IDF-file becomes negative, so zone 1 might appear as -1. This doesn't affect the computation of the water balance volumes but has a purpose for finding overlap only.

Example of "table" layout of the water balance:

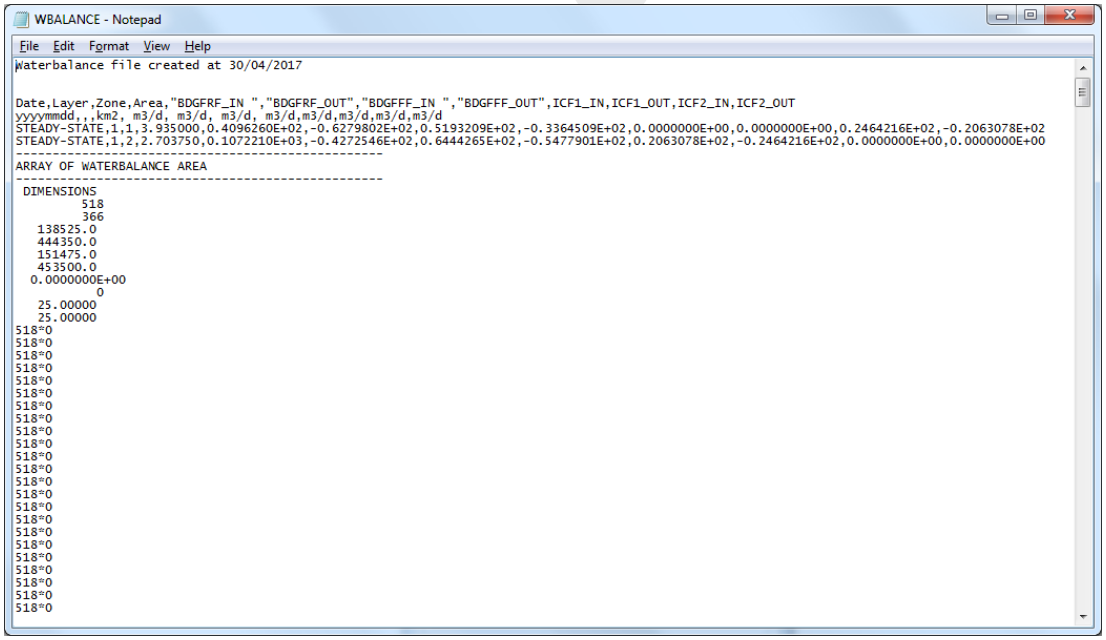


WBAL.TXT - Kladblok

Bestand Bewerken Opmaak Beeld Help

```
Waterbalance file created at 25/04/2011

Waterbalance for selected area given by polygon(s) read from :
C:\USERS\PETER\WORK\CURSUS-IMOD\IMOD-CURSUS\IMOD_USER\TMP\tmp.gen

BEAR IN MIND THAT TOTAL WATERBALANCE (MIGHT) INCLUDE ERRORS DUE TO MISSING BUDGET TERMS !

================================================================================================
Period : Steady    ; Layer :   1
================================================================================================

        Waterbalance budget      Q_in     Q_out          Q_in          Q_out          Q_in          Q_out          Area
                                    %         %          m3/d           m3/d          mm/d           mm/d           km2

Zone:  Centre              ------------------------------------------------------------------------------------------

            FLUX LOWER FACE     0.000    37.687  0.0000000E+00 -0.6830675E+03  0.0000000E+00 -0.2982014E+00  0.2290625E+01
            FLUX RIGHT FACE     1.153     5.672  0.2089673E+02 -0.1028005E+03  0.2259106E+00 -0.1111357E+01  0.9250000E-01
            FLUX FRONT FACE     1.045     4.710  0.1893772E+02 -0.8536823E+02  0.1721611E+00 -0.7760748E+00  0.1100000E+00
                   DRAINAGE     0.000     0.002  0.0000000E+00 -0.2865996E-01  0.0000000E+00 -0.1251185E-04  0.2290625E+01
                     RIVERS     0.169    51.693  0.3057450E+01 -0.9369230E+03  0.1334767E-02 -0.4090251E+00  0.2290625E+01
              OVERLAND FLOW     0.000     0.237  0.0000000E+00 -0.4295558E+01  0.0000000E+00 -0.1875278E-02  0.2290625E+01
                   RECHARGE    97.634     0.000  0.1769622E+04  0.0000000E+00  0.7725500E+00  0.0000000E+00  0.2290625E+01
------------------------------------------------------------------------------------------------------------------------
                     TOTALS                       0.1812514E+04 -0.1812484E+04        Error :  0.3076172E-01 (m3/d)

Zone:   Urban Area         ------------------------------------------------------------------------------------------

            FLUX LOWER FACE     0.000    39.445  0.0000000E+00 -0.4007979E+03  0.0000000E+00 -0.2570247E+00  0.1559375E+01
            FLUX RIGHT FACE     2.407     2.692  0.2445853E+02 -0.2735255E+02  0.3691854E+00 -0.4128686E+00  0.6625000E-01
            FLUX FRONT FACE     3.412     4.855  0.3466645E+02 -0.4932837E+02  0.3262724E+00 -0.4642670E+00  0.1062500E+00
                   DRAINAGE     0.000     0.000  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.1559375E+01
                     RIVERS     3.154    53.009  0.3204894E+02 -0.5386211E+03  0.2055243E-01 -0.3454083E+00  0.1559375E+01
              OVERLAND FLOW     0.000     0.000  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.1559375E+01
                   RECHARGE    91.027     0.000  0.9249389E+03  0.0000000E+00  0.5931472E+00  0.0000000E+00  0.1559375E+01
------------------------------------------------------------------------------------------------------------------------
                     TOTALS                       0.1016113E+04 -0.1016100E+04        Error :  0.1287842E-01 (m3/d)

================================================================================================================================
```

Example of a "time serie" layout of the water balance:



WBALANCE - Notepad

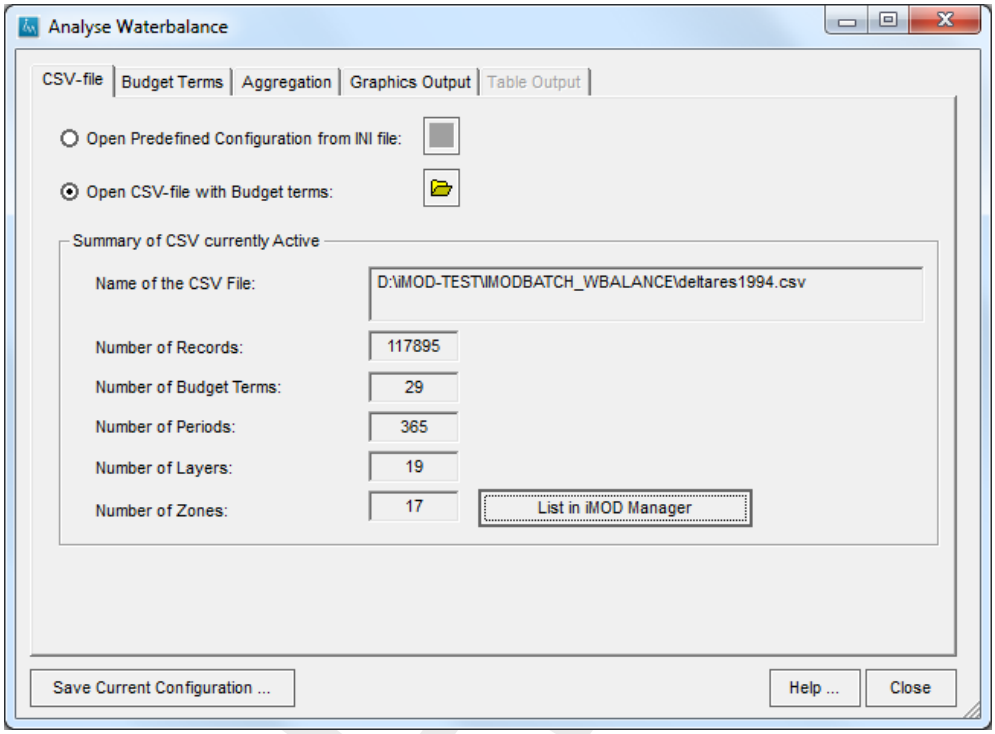
File Edit Format View Help

```
Waterbalance file created at 30/04/2017


Date,Layer,Zone,Area,"BDGFRF_IN ","BDGFRF_OUT","BDGFFF_IN ","BDGFFF_OUT",ICF1_IN,ICF1_OUT,ICF2_IN,ICF2_OUT
yyyymmdd,,,km2, m3/d, m3/d, m3/d, m3/d,m3/d,m3/d,m3/d,m3/d
STEADY-STATE,1,1,3.935000,0.4096260E+02,-0.6279802E+02,0.5193209E+02,-0.3364509E+02,0.0000000E+00,0.0000000E+00,0.2464216E+02,-0.2063078E+02
STEADY-STATE,1,2,2.703750,0.1072210E+03,-0.4272546E+02,0.6444265E+02,-0.5477901E+02,0.2063078E+02,-0.2464216E+02,0.0000000E+00,0.0000000E+00
-----------------------------------------------------
ARRAY OF WATERBALANCE AREA
-----------------------------------------------------
 DIMENSIONS
         518
         366
   138525.0
   444350.0
   151475.0
   453500.0
  0.0000000E+00
           0
   25.00000
   25.00000
518*0
518*0
518*0
518*0
518*0
518*0
518*0
518*0
518*0
518*0
518*0
518*0
518*0
518*0
518*0
518*0
518*0
518*0
518*0
518*0
518*0
518*0
518*0
518*0
518*0
518*0
```

7.16.2 Analyse Waterbalance

To analyse the results of a water balance computation, select the option *Toolbox* from the main menu, choose *Water Balance* and then choose *Analyse Waterbalance* to open the *Analyse Waterbalance* window. The key thought is that a CSV file is read in first and after that several selection can be made by the different tab on the window. Finally an output can be defined, this can be a schematic representation of the water balance, a time series, a table and/or spatial patterns of water balance term saved in IDF files.

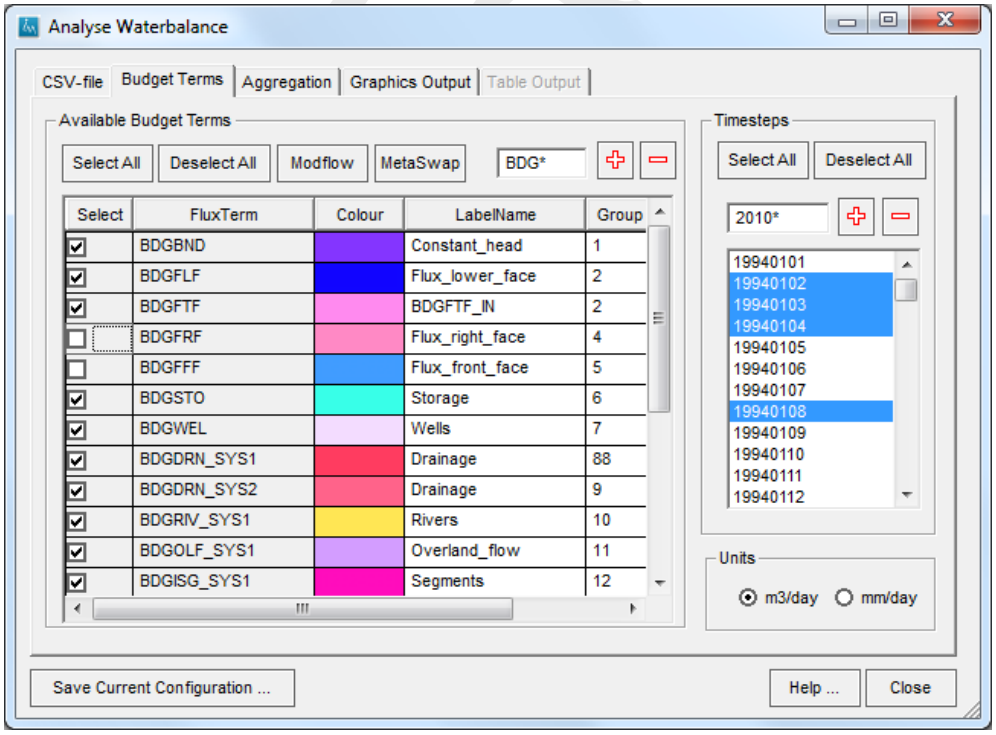
Analyse Waterbalance window, CSV-file tab:



Open Predefined Configuration from INI file:	Select this radio button to select an INI file that contains a particular set of configuration parameters.
	*Select INI file* Click this button to select an INI file from a folder. iMOD will read the configuration and read the CSV file defined in te INI file.
Open CSV-file with Budget terms:	Select this radio button to select a CSV that contains the budget terms for a particular set, default configuration parameters will be applied.

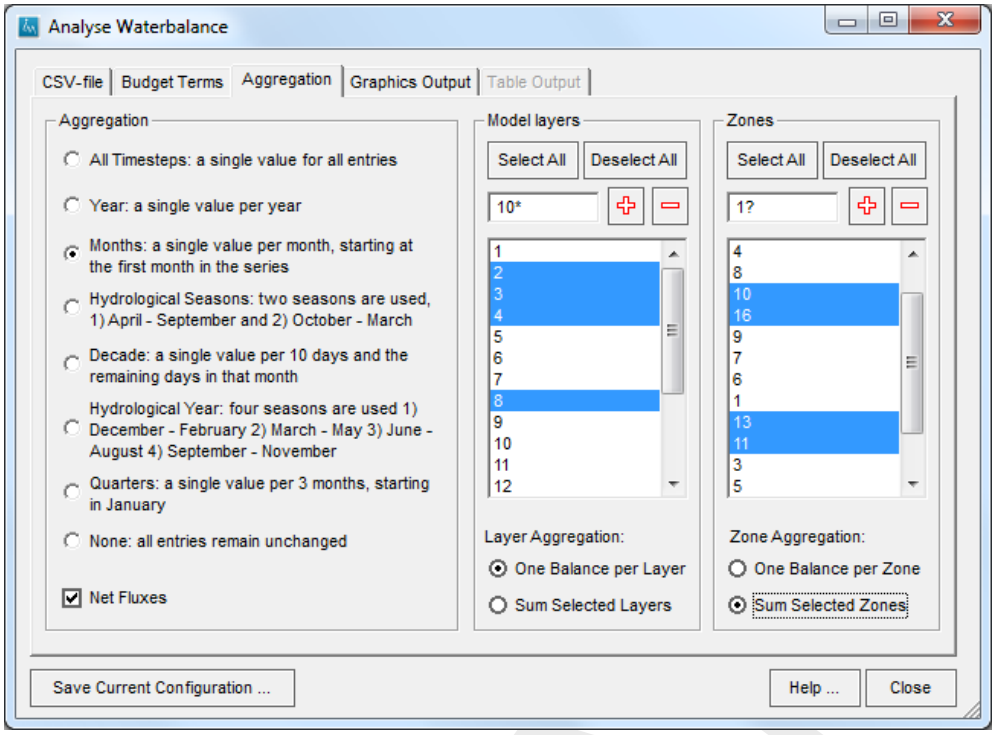
	*Select CSV file* Click this button to select a CSV-file from a folder on disk. iMOD will read the CSV completely and stores the content in memory to perform adequate and efficient selection as defined by the other tabs. The content of the CSV file will be listed as: ◇ *Name of the CSV file:* This is the name of the read CSV file; ◇ *Number of records:* This is the number of entry records, it is a multiplication of the number of periods, zones and layers; ◇ *Number of Budget Terms:* This is the number of budget terms, this can include the interconnected flow as separate entries; ◇ *Number of Periods:* This the number of periods, time series can be constructed whenever this is large than 1; ◇ *Number of Layers:* This is the number of model layers that exists in the CSV file; ◇ *Number of Zones:* This is the number of zones All the above mentioned separate quantities can be selected, aggregated and displayed by setting the appropriate selection criteria in the following tabs.
List in iMOD Manager	Click this button to export the zones as defined in the CSV file into an IDF file ({IMOD_USER} \TMP \WATERBALANCE.IDF) and listed in the iMOD Manager for viewing options, this is the only way to get the zones stored in the CSV out to re-use them and/or select the appropriate zones once the original GEN- or IDF file has been deleted.
Save Current Configuration:	Click this button to save the current configuration (the name of the CSV and the corresponding selection and settings, including changed colours to a INI file. This INI file can be loaded by the *Open Predefined Configuration from INI file:* and/or used in the iMOD Batch function *WBALANCE*, see section 8.9.2.
Help. . .	Click this button to start the HELP functionality
Close	Click this button to close the *Analyse Waterbalance* window

Analyse Waterbalance window, Budget Terms tab:



Select All	Click this button to select all budget items in the table (first column) and/or select all the available periods in the list of *Timesteps*.
Deselect All	Click this button to deselect all budget items in the table (first column) and/or deselect all the available periods in the list of *Timesteps*.
Modflow	Click this button to select all specific iMODFLOW budget terms.
MetaSwap	Click this button to select all specific MetaSwap budget terms.
*BDG** *2010**	Enter a search string to select or deselect budget terms (second column in the table FluxTerm) and/or select Timesteps. The search string is case sensitive and contain wildcards such as * and question marks to notate a fixed number of character, such as 2010??01.
	Increase Selection Click this button to increase the current selection for budget terms in the table with *Available Budget Terms* and/or periods in the list of available *Timesteps*.
	Decrease Selection Click this button to decrease the current selection for budget terms in the table with *Available Budget Terms* and/or periods in the list of available *Timesteps*.
Units	Select one of the options to define the units in the water balance. If the option *m3day* is selected no conversion is applied, whenever the option *mmday* is chosen, the budget volumes will be divided by the area of the corresponding zone and multiplied with 1000.
Available Budget Terms	The table gives all existing budget terms in the current read CSV file. The following columns are defined: ◇ *Select:* This column activates and deactivates a individual budget to more included or excluded in the presentation of a water balance; ◇ *FluxTerm:* This is a non-editable column with predefined labels for the budget terms. This is used by iMOD to be able to assign the appropriate budget terms to specific entries for the graphical representation that can be chosen on tab4; ◇ *Colour:* The colour for each row in the table is used to colour the bar in the time series graph. To change the colour, select the appropriate cell and the standard colour picking window appears in which it is possible to select a colour, see section 2.8; ◇ *LabelName:* Enter a name for each budget term that will be used in the output of the waterbalance, it will be used as a legend name in the time series and as a name in the graphical representation of the water balance. iMOD fills in a default for each distinguished budget term; ◇ *Group:* This column is filled in with a group number. By default the groups are unique numbers starting from 1 to the number of budget terms in the CSV file. The use of this group number is twofold: ▫ The group number is used to group budget terms into a single entry for the time series. So whenever two budget term receive an similar, unique group number that appear as a single budget term in the output. The name for the legend of that group will be formed by the first entry of a budget term that belongs to that group. Any number can be entered as a group number, zero is sustained as a group number as well. In the example on the figure above, the budget terms BDGFLF and BDGFTF are grouped together as group number 2; ▫ The value of the group number defines the order in which budget terms are plotted in the time series. The appearance of the time series might change significantly by changing the group numbers. In the example on the figure above the budget term BDGDRN_SYS1 is plotted at last since its group number is 88 and the highest value compared to the others.

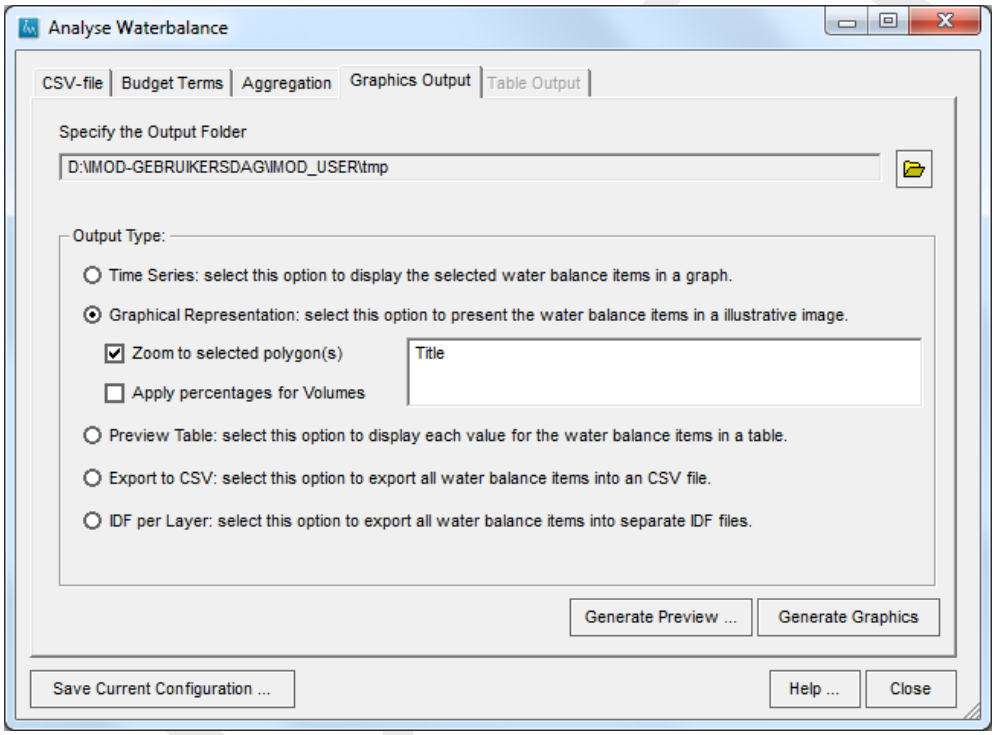
Analyse Waterbalance window, Aggregation tab:



Aggregation	Select one of the following options to aggrate the water balance for the period as selected on tab 2: ◇ *All Time steps:* a single value for all entries ◇ *Year:* a single value per year ◇ *Months:* a single value per month, starting at the first month in the series ◇ *Hydrological Seasons:* two seasons are used, 1) April - September and 2) October - March ◇ *Decade:* a single value per 10 days and the remaining days in that month ◇ *Hydrological Year:* four seasons are used 1) December - February 2) March - May 3) June - August 4) September - November ◇ *Quarters:* a single value per 3 months, starting in January ◇ *None:* all entries remain unchanged.
Net fluxes	Select this option to compute net fluxes. In that case iMOD will compute a total net flux for all selected budget terms, each budget terms receives in that particular case a single net (sum of the in- and outflow) entry in the water balance
Select All	Click this button to select all model layers in the list of available *Model layers* and/or select all the available zones in the list of *Zones*.
Deselect All	Click this button to deselect all model layers in the list of available *Model layers* and/or select all the available zones in the list of *Zones*.
10 *1?*	Enter a search string to select or deselect budget terms (second column in the table FluxTerm) and/or select Timesteps. The search string is case sensitive and contain wildcards such as * and question marks to notate a fixed number of character, such as 1?.
	Increase Selection Click this button to increase the current selection for model layers in the list of available *Model Layers* and/or zones in the list of available *Zones*.
	Decrease Selection Click this button to deecrease the current selection for model layers in the list of available *Model Layers* and/or zones in the list of available *Zones*.

Layer aggregation	Select one of the following options: ◇ *One Balance per Layer:* Select this option to treat budget terms as separate items in the selected form for output; ◇ *Sum Selected Layers:* Select this option to aggregate budget terms for selected model layers.
Zone aggregation	Select one of the following options: ◇ *One Balance per Zone:* Select this option to treat budget terms as separate items in the selected form for output; ◇ *Sum Selected Zones:* Select this option to aggregate budget terms for selected zones.

Analyse Waterbalance window, Graphics Output tab:



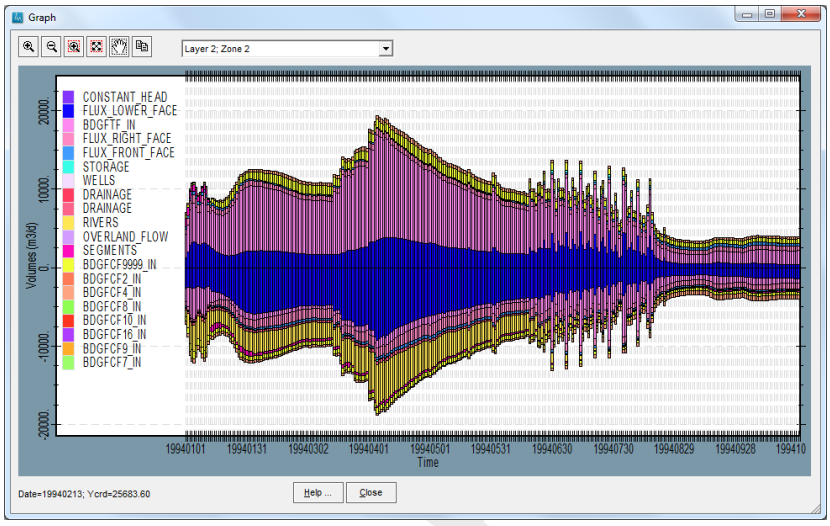
	*Select Folder* Click this button to select a folder that iMOD will uses to save images if needed. It depends on the choice of the *Output Type* whether this is applicable. By default iMOD will save the images in the TMP folder of the {IMOD_USER} folder.

Output Type

Select one of the following output options:

- *Time Series:* Select this option to display the selected water balance items in a graph. Below is an example of the time series.

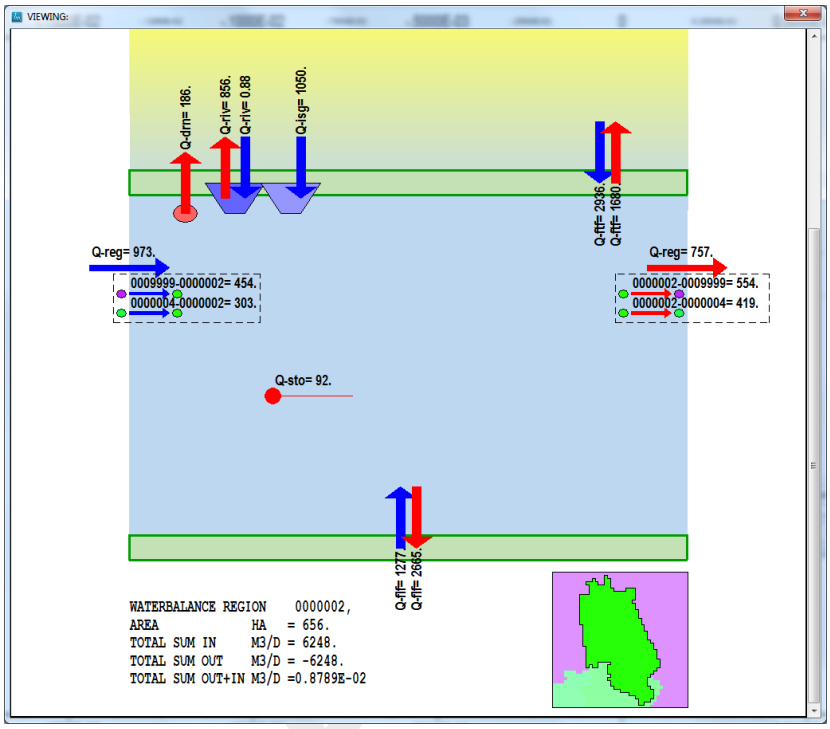
Example of the water balance time series



	*Copy* Click this icon to copy the entire content of the graphical area onto the *Clipboard* of Windows.
	*ZoomIn* Click this icon to zoom IN on the centre of the current graphical dimensions.
	*ZoomOut* Click this icon to zoom OUT on the centre of the current graphical dimensions.
	*ZoomRectangle* Click this icon to zoom in for a rectangle to be drawn. Use your left-mouse button to determine the lower-left corner of the rectangle, click again for the upper-right corner (or vice-versa).
	*ZoomFull* Click this icon to zoom in on the entire extent of the time series.
	*Move* Click this icon to move the current display. Click the left-mouse button on that location where you want to move from, repeat this after the display has been refreshed (automatically). Use the right mouse button to stop the moving process.
Layer 2; Zone 2	This dropdown menu displays all possible combination in between selected model layers and selected zone, in case no aggregation is selected for both of those entries. If a single model layer and a single zone is selected, or they are both aggregated, the dropdown menu will not appear.

◇ *Graphical Representation:* Select this option to present the water balance items in a illustrative image. Below is an example of the time series.

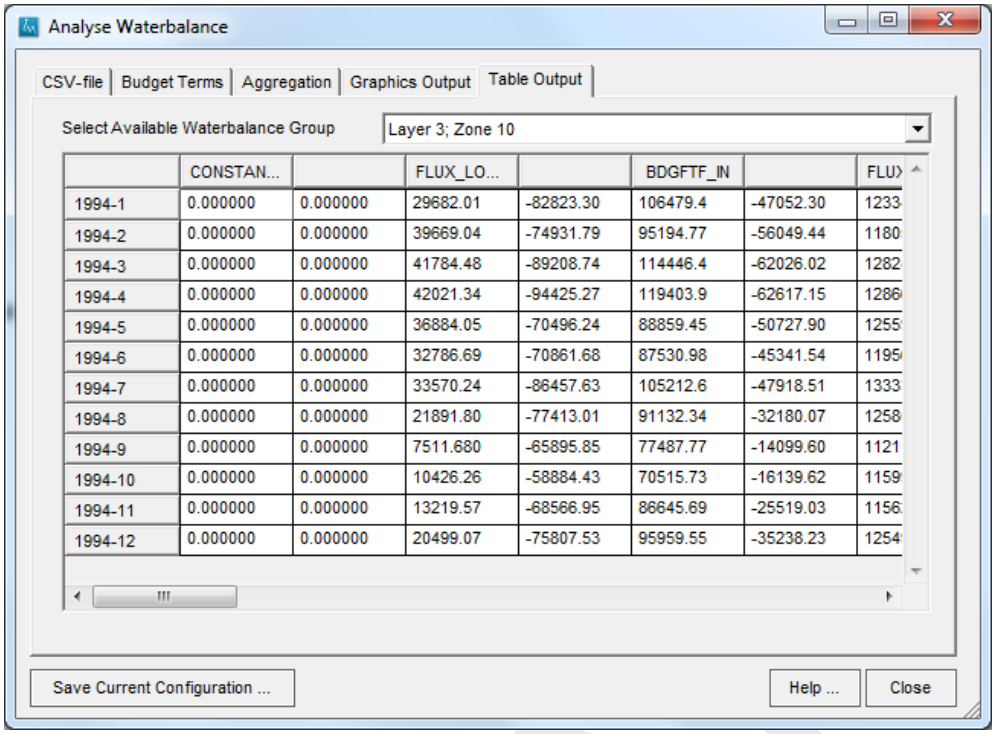
Example of the water balance as an illustrative image



◇ *Preview Table:* Select this option to display each value for the water balance items in a table, see the explanation given at the following section;
◇ *Export to CSV:* select this option to export all water balance items into an CSV file;
◇ *IDF per Layer:* Select this option to export all water balance items into separate IDF files. This yields a single IDF per budget term, per model layer and per period as selected. Any aggregation will decrease the number of output combinations. The name of each IDF file is constructed automatically and includes the budget term, the period, model layer.

Zoom to Selected Polygon(s)	Select this option to create an overview image on the graphical representation graph whereby the area is zoomed in on the selected polygon(s).
Apply percentages for Volumes	Select this option to compute percentages for each waterbalance entry in the graphical representation graph.
Title	Specify the title of the graphical representation graph. The title will automatically wrap over two line of the number of characters excceds the number for a single line. The maximum of characters is 256.
Generate Preview *Generate Table*	Click this button to generate a preview of the selected *Output Type*. Not all of the output types support a preview, this button will be out greyed for those situations. Whenever the *Generate Table* is active, the tab *Table Output* becomes accessible and display a table form of the results of the water balance.
Generate Graphics *Export CSV File* *Export IDF File*	Click this button to generate an image to be saved on disc. In this case the output folder as specified by *Output Folder* is applied. Not all of the output types support a preview, this button will be out greyed for those situations.

Analyse Waterbalance window, Table Output tab:



Select Available Waterbalance Group	Select from the dropdown the appropriate combination of selected model layers and/or zones. This dropdown menu displays all possible combination in between selected model layers and selected zone, in case no aggregation is selected for both of those entries. If a single model layer and a single zone is selected, or they are both aggregated, the drop down menu will not appear.
Table	The entries in the table are the results of any configuration settings applied in the previous tabs. Each row is a period and depend on the Aggregation type as selected on tab3. It is possible to copy the content of the entire table to the clipboard by the using the option Crtl-C.

7.17 Compute Mean Groundwaterfluctuations (GxG)

WHY?

Groundwater fluctuations are indicative for the seasonal dry or wet status of an area. The so-called GxG's are indicative of the high and low phreatic groundwater levels occurring in a period of at least 8 years. The GxG's are used frequently in the Netherlands when defining the geohydrological conditions of an area.

WHAT?

The GxG's consist of:

1. GHG ('gemiddeld hoogste grondwaterstand' / average highest groundwater level) is calculated as the average of the three highest groundwater levels (measured or simulated around every two weeks) per hydrological year (1 April – 31 March) averaged over at least eight consecutive years.
2. GLG ('gemiddeld laagste grondwaterstand' / average lowest groundwater level) is calculated as the average of the three lowest groundwater levels (measured or simulated around every two weeks) per hydrological year (1 April – 31 March) averaged over at least eight consecutive years.
3. GVG ('gemiddelde voorjaars grondwaterstand' / average spring groundwater level) is calculated as the average groundwater level of the 14th of March, the 28th of March and the 14th of April and again averaged over at least eight consecutive years.

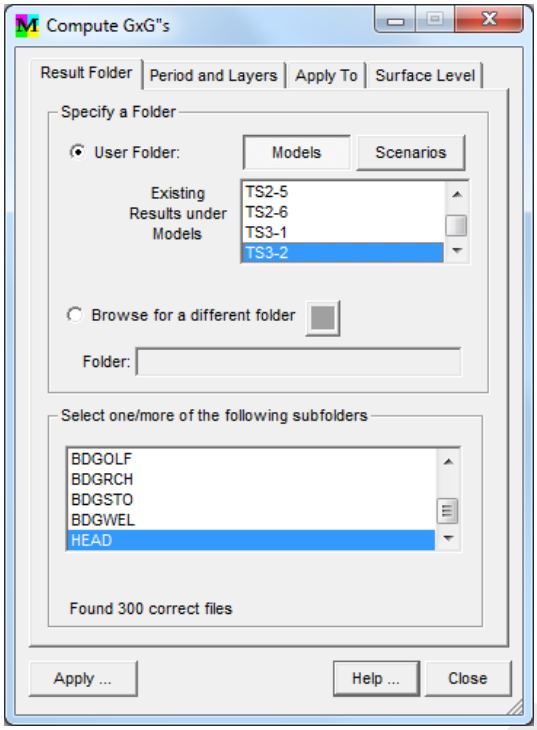
In general GXG's are expressed as groundwater depths below the land surface.

HOW?

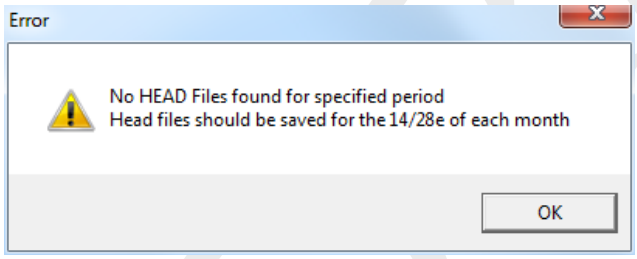
Select the option *Toolbox* from the main menu and then choose *Compute GxG* to open the *Compute GxG's* window.

This function is not described separately. The functionalities of the *Compute GXG's* window are very similar to the *Compute Waterbalance* window (see section 7.16.1). The GXG's are calculated based on the output of a transient groundwater model.

Compute GxG's window, Result Folder tab:



The computation of the GxG's will be executed using model results from the 14^{th} and 28^{th} of each month. The system will generate an error when there are no files found for these dates. The filenames will have to be *head_yyyymm14.IDF* or *head_yyyymm28.IDF*.



7.18 Compute Mean Values

WHY?
The Mean Values of budgets or heads are calculated based on the output of a transient groundwater model.

WHAT?
A new IDF is created containing the mean values.

HOW?
Select the option *Toolbox* from the main menu and then choose *Compute Mean Values* to open the *Compute Mean Values* window.

This function is not described separately. The functionalities of the *Compute Mean Values* window are very similar to the *Compute Waterbalance* window (see section 7.16.1).

7.19 Compute Timeseries

WHY?

The time-dependent results of a transient model are converted to timeseries for specified locations, e.g. at boreholes. The variation in time of the head (and of the budget-terms if desired) can be viewed in a graph and is available for processing outside iMOD.

WHAT?

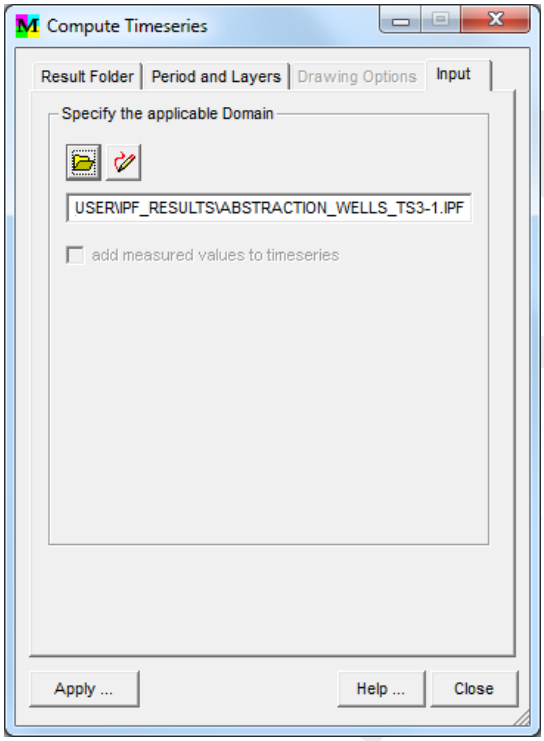
The *Compute Timeseries* option checks the available model output of the selected model and reads the IDF-files at locations read from an IPF-file or defined interactively on the map. The result is stored in an IPF-file which can be viewed in iMOD.

The functionalities of the *Compute Timeseries* window are very similar to the *Compute Waterbalance* window (see section 7.16.1). Only the specific functions in the *Input* tab of the*Compute Timeseries* window are described here.

HOW?

Select the option *Toolbox* from the main menu and then choose *Compute Timeseries* to open the *Compute Timeseries* window.

Compute Timeseries window, Input tab:



	*Open* Click this button to open an IPF-file, see section 9.7 for more details. The IPF-file contains the locations where the time-series are generated.
	*Draw* Click this button to start drawing the point locations on the map by clicking the left mouse button. Stop using the right mouse button and save the points in an IPF-file.
Apply	Click this button to start the computation of the time series. You should enter a name for the resulting IPF-file.

8 iMOD Batch functions

8.1 General introduction

This chapter contains a detailed description of a library of tools or functions that is available in iMOD-Batch mode.

8.1.1 What is an iMOD Batch Function?

An iMOD Function in general is a procedure to perform a specific task in iMOD or on iMOD files returning a new file or a series of files. Functions can be found on many places in the menu structure of the iMOD Graphical User Interface (GUI). For instance functions to:

- ◇ create, edit or analyse files (IDF, IPF ect).
- ◇ run processes like iMODFLOW and iMODPATH.
- ◇ post process modelresults (waterbalance, timeseries or plots).

An iMOD Batch Function can be used outside the iMOD GUI and its operation is controlled by parameters in an *.INI file, the initialization file.

Below you find an example of the content of a PLOT.INI file. It describes the keywords and parameters for the Function PLOT, for plotting IDF or IPF files to a PNG file.

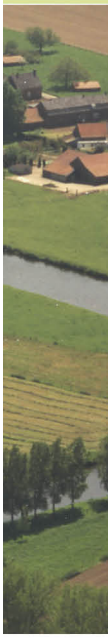
FUNCTION=PLOT
IDFFILE=D:\DATA\AHN.IDF
GENFILE=D:\DATA\PROV.GEN
OUTFILE=D:\DATA\PLOT.PNG

In general an *.INI file contains keywords some of which are compulsory (e.g. the Function name), others are optional. iMOD searches for the keywords in the entire file so there is no fixed order of keywords allowing you to insert white lines and remarks. Examples of the set up of such an *.INI file for all available iMOD Batch function is given in the following paragraphs.

The advantage of using the iMOD Batch mode instead of the iMOD Graphical User Interface (GUI) is manifold:

- ◇ it saves time when iMOD functions must be performed on many files or many times.
- ◇ the total set of *.INI files in a project can be seen as a logbook of your pre- and post-processing activities.
- ◇ minor changes to a function are easily applied by editing the *.INI file.
- ◇ *.INI files are easily shared with colleagues or clients.
- ◇ over time the *.INI files form a library, easy to re-use.
- ◇ you don't need to click around in the GUI and repeatedly type file names.

The number of functions in iMOD Batch mode is expected to grow gradually. Not all functions in the iMOD GUI are (yet) available in iMOD Batch mode (e.g. create IDF from scratch). On the other hand, some functions are available in iMOD Batch mode but not in the GUI (e.g. Modelcopy).



8.1.2 How to run an iMOD Batch Function?

To start an iMOD Batch procedure first open a Dos command window. How? Search for 'CMD' in 'All Programs' under the Windows start button). In that DOS window, simply enter the name of the iMOD executable followed by the name of the *.INI file, e.g. "iMOD_V4_3_X64R.exe IDFSCALE.INI", (see Figure 8.1) and press Enter. The program will stop after the function, described by the *.INI file, is executed.

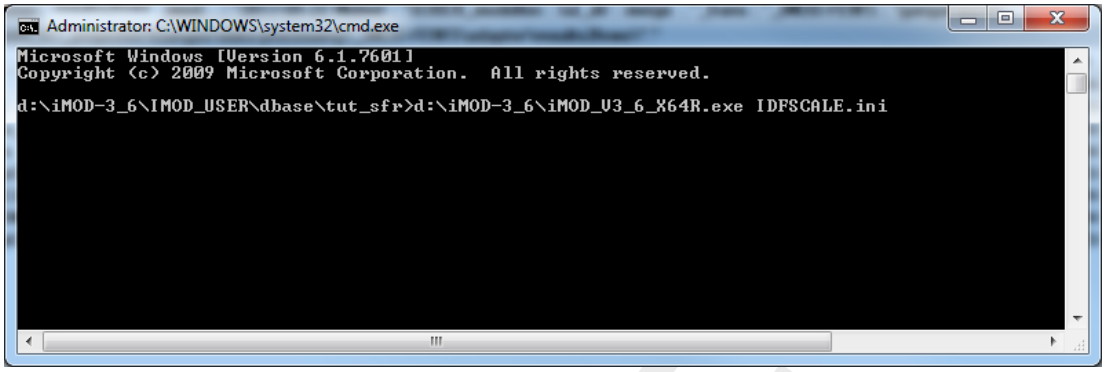


Figure 8.1: *Example of command in DOS box to run an iMOD Batch script.*

An iMOD batch function can also be started from the maim menu. See section 4.6 for an explanation.

8.1.3 Using DOS scripting (*.BAT file) to organize iMOD Batch Functions

The iMOD Batch functions can be used as described above, however, to introduce more options these functions can also be controlled in a DOS Batch file (*.BAT) using DOS scripting. Of course you can use your favourite scripting languages (e.g. Python or Matlab) to create and execute any iMOD Batch *.INI file.

This is an example of a *.BAT file creating (line 1-4) and executing (line 5) the PLOT.INI file:

ECHO FUNCTION=PLOT > PLOT.INI
ECHO IDFFILE=D:\DATA\AHN.IDF >> PLOT.INI
ECHO GENFILE=D:\DATA\PROV.GEN >> PLOT.INI
ECHO OUTFILE=D:\DATA\PLOT.PNG >> PLOT.INI
iMOD_V4_3_X64R.exe PLOT.INI

A *.BAT file is recognized in Windows as a Dos script and starts running by double clicking the *.BAT file.

8.1.4 Examples of advanced DOS scripting options

Advanced DOS scripting options enables you to make more complex use of iMOD Batch functions. This paragraph gives you some examples of advances options. On the internet there are plenty sites available giving basic or advanced DOS scripting tips and tricks in more detail (e.g. `http://www.robvanderwoude.com/batchcommands.php`).

With the installation of iMOD an example of a *.BAT file is copied to the Tutorial folder. Search for *..\TUTORIALS \TUT_Map_Analyse \SubsoilSystem \iMOD-Batch-example-IDFCALC.BAT*. Open the file, read some tips and tricks, double click and experience that this iMOD Batch function calculates the thickness of a series of 6 aquifers.

Below we describe some examples of advanced DOS scripting options.

- ◇ A (nested) loop over a list of *Text* elements
  Not more than 6 lines in the next example are necessary to create and run the iMOD Batch Function PLOT producing 12 PNG files showing the HEAD for the first 4 month of 3 successive years.

```
FOR %%A IN (1994,1995,1996) DO (
FOR %%B IN (01,02,03,04) DO (
ECHO FUNCTION=PLOT > PLOT.INI
ECHO IDFFILE=D:\MODEL\HEAD\HEAD_%%A%%B01_L1.IDF >> PLOT.INI
ECHO OUTFILE=D:\MODEL\HEAD\HEAD_%%A%%B01_L1.PNG >> PLOT.INI
iMOD_V4_3_X64R.exe PLOT.INI))
```

◇ A loop over a series of Numbers
The next example of a FOR loop (including Command Extension /L) gives you the variable %%A between 1 upto 18 with steps of 2.

```
FOR /L %%A IN (1,2,18) DO (
ECHO %%A)
)
```

◇ Using specific files from a folder
The next DOS script gives you the variable %%A for all IDF-files found in the folder D:\IMOD-MODEL\DBASE. Usage of ~n gives you the filenames only without extension.

```
SET DIR=D:\IMOD-MODEL\DBASE
FOR %%A IN (%DIR%\*.IDF) DO (
ECHO %DIR%\%%~nA.IDF
ECHO %%A
)
```

◇ Introduce an arithmetic operator within a loop
Adding a looping parameter %%b might be interesting whenever computing the thickness of aquitards since each time you need to subtract the bottom of modellayer 1 minus the top of modellayer 2.

```
SETLOCAL ENABLEDELAYEDEXPANSION
SET /A B=1
FOR /L %%C IN (1,1,18) DO (
SET /A B=B+1
ECHO FUNCTION=IDFCALC > IDFCALC.INI
ECHO FUNC=C=A-B >> IDFCALC.INI
ECHO NREPEAT=1 >> IDFCALC.INI
ECHO ABC1=BOT%%C.IDF TOP!B!.IDF TAQT%%C.IDF >> IDFCALC.INI
iMOD_V4_3_X64R.exe IDFCALC.INI
)
```

Notice that the variable B is used by bracketing it by "!" and the variable C is used by adding "%%" in front. The statement SETLOCAL ENABLEDELAYEDEXPANSION is necessary to delay the interpretation of the variable B.

◇ include IF-THEN-ELSE statements
It is often handy to include and if-then-else statement inside the batch structure, for example whenever you might want the generate a runfile for iMODPATH (see section 8.6.6). Unfortunately, the MS-DOS IF statement does not support logical operators (AND and OR) so we have to find other ways.

In the example below we only write a budget term whenever it is not equal to the BDGFLF (flow-lower-face) and the layer number is not equal to the lowest one (5).

```
SETLOCAL ENABLEDELAYEDEXPANSION
FOR /L %%A IN (1,1,5) DO (
FOR %%B IN (FFF,FRF,FLF) DO (
SET /A Flag=0
IF %%B==FLF SET /A Flag=Flag + 1
IF %%A==5 SET /A Flag=Flag + 1
IF !Flag! NEQ 2 (
ECHO ..\BDG%%B\BDG%%B_STEADY-STATE_L%%A.IDF >> IMODPATH.INI)
```

))

If an OR-statement need to be applied, do as follows:
SETLOCAL ENABLEDELAYEDEXPANSION
FOR /L %%A IN (1,1,5) DO (
FOR %%B IN (FFF,FRF,FLF) DO (
SET res=F
IF %%B==FLF SET res=T
IF %%B==FRF SET res=T
IF !res!==T (
ECHO ..\BDG%%B\BDG%%B_STEADY-STATE_L%%A.IDF >> IMODPATH.INI)
))

◇ Compare operators in DOS
Use the following compare operators:
EQU - equal
NEQ - not equal
LSS - less than
LEQ - less than or equal
GTR - greater than
GEQ - greater than or equal

The following syntax is than valid:

FOR /L %%A IN (1,1,10) DO (
IF %%A LSS 6 (ECHO value %%A is less than 6
) ELSE IF %%A GTR 6 (ECHO value %%A is greater than 6
) ELSE (ECHO value %%A is equal to 6
))

◇ Start execute process within a DOS script
Finally the iMOD executable can be started in different ways with different behavior. Any *.INI can be started with:

FOR %%A IN (1,2) DO (
iMOD_V4_3_X64R.exe {}.INI
)
however this will block the batch (*.BAT) structure that starts it until the process is ended. Below an example is given that starts iMOD and continues the batch and starts another iMOD session in another command-window.

FOR %%A IN (1,2) DO (
START iMOD_V4_3_X64R.exe {}.INI)

Batch array definition
In addition to the described batch looping commands, it is also possible to define arrays within the batch environment. This might be helpful in case e.g. there are multiple files with different filenames that can be divided into a number of subgroups and that need to be processed in a similar way. For example:

setlocal EnableDelayedExpansion
set n=0
for %%a in (KHV, KVV) do (
set var[!n!]=%%a
set /A n+=1
) for /L %%i in (0,1,1) do (
for /L %%j in (1,1,12) do (
echo FUNCTION=IDFCALC > calc_idf.ini
*echo FUNC= "C=A*B" » calc_idf.ini*
echo NREPEAT=1 » calc_idf.ini

echo ABC1= "d:\MODEL\Factor.IDF" "d:\MODEL\DATA\!var[%%i]!_L%%j.idf"
"d:\MODEL\DATA\!var[%%i]!_L%%j_factor.idf" » calc_idf.ini
d:\iMOD_versies\iMOD_v3.3\iMOD_V4_3_X64R.exe calc_idf.ini
))

In this example two type of files can be distinguished, files with 1. "KHV" and 2. "KVV" information. Each file category contains 12 files; one for each layer. All files need to be multiplied by specific factor grid, saved in Factor.idf. With the outer loop the file category is controlled and with the second loop the layers per category. Array definition can be applied to multiple cases and serve as a helpful tool to shorten your batch-scripts.

The iMOD Batch function are categorized into several topics related to IDF, IPF, ISG and GEN files and described on the following pages.

8.2 IDF-FUNCTIONS

8.2.1 IDFCALC-Function

The IDFCALC function can be used to carry out simple arithmetical operations on maximal two different IDF-files to create a new IDF-file. See for more information section 6.7.3.

FUNCTION=	**IDFCALC**
FUNC=	Enter the function, e.g. C=A-B or C=ABS(A-3.0*B), or C=A, see section 6.7.3 for more information. Whenever the symbol "/" is used, apply quotes, thus "C=A/B".
ABC{i}=	Enter the i^{th} out of NREPEAT IDF-filenames that corresponds with "A", "B" and "C" in the function FUNC.
AC{i}=	Enter the i^{th} out of NREPEAT IDF-filename that corresponds with "A" and "C" in the function FUNC.
BC{i}=	Enter the i^{th} out of NREPEAT IDF-filename that corresponds with "B" and "C" in the function FUNC.
NREPEAT=	Specify the number of times the function FUNC need to be carried out.
SOURCEDIRA=	Enter a folder that contains all the IDF-files associated to the "A" in FUNC. Apply this keyword whenever NREPEAT is absent.
SOURCEDIRB=	Enter a folder that contains all the IDF-files associated to the "B" in FUNC. Apply this keyword whenever NREPEAT is absent.
SOURCEDIRC=	Enter a folder that contains all the IDF-files associated to the "C" in FUNC. Apply this keyword whenever NREPEAT is absent.
USENODATA=	Enter USENODATA=1 to use cells that have *NoDataValues.* By default, USENODATA=0, so those cells that have *NoDataValue* will be ignored.
NODATAVALUE	Enter the value for the *NoDataValue* to be used in the computation, e.g. NODATAVALUE=0.0. This keyword is compulsory whenever USENODATA=1.
GENFILE=	Enter the name of a GEN-file, e.g. GENFILE=D:\DATA\AREA.GEN. Any computation will be carried out inside the polygons of the GENFILE. On default, GENFILE=' ', which means that no genfile will be used.
IEQUI=	Enter IEQUI=0 to construct (if needed) a non-equidistant IDF-file that counts for all raster dimensions of the entered IDF-files, this is the default. Enter IEQUI=1 to force that the resulting IDF-files are produced with equidistant cellspaces, based on the smallest cell size occurring in the IDF-files "A" and/or "B".
WINDOW=	Enter the coordinates of the window that need to be computed, solely. Enter coordinates of the lower-left corner first and then the coordinates of the upper-right corner, e.g. WINDOW=100000.0, 400000.0, 200000.0, 425000.0. When WINDOW= is absent, the entire dimensions of the first mentioned IDF-file will be used.
TRIM_VALUE=	Enter a value to be able to ignore all result values smaller than this specific absolute value after the calculation. This option gives values smaller than given/entered absolute value a *NodataValue*, e.g. C=A-B if C< 0.1 C=*NodataValue*.

Example 1

FUNCTION=IDFCALC
FUNC= "C=A/B"
NREPEAT=2
ABC1=D:\KD_L1.IDF D:\THICKNESS_L1.IDF D:\K_L1.IDF
ABC2=D:\KD_L4.IDF D:\THICKNESS_L4.IDF D:\K_L4.IDF

The above mentioned example will compute the permeability (k) by dividing the transmissivity (KD) by the thickness (THICKNESS) for modellayer 1 and modellayer 4, subsequently.

Example 2

```
FUNCTION=IDFCALC
FUNC= C=A-B
USENODATA=1
NODATAVALUE=0.0
IEQUI=1
GENFILE=D:\AREA.GEN
WINDOW=100000.0,350000.0,150000.0,450000.0
SOURCEDIRA=D:\MODEL\HEAD_*_L1.IDF
SOURCEDIRB=D:\SCENARIO\HEAD_*_L1.IDF
SOURCEDIRC=D:\EFFECT\DIFF_*_L1.IDF
```

The above mentioned example will compute the differences within the polygon(s) described by the AREA.GEN and within the given WINDOW. If any *NoDataValues* are found in the IDF-files, they will be treated as if they were NODATAVALUE=0.0. Any file that agrees with the filename HEAD_*_L1.IDF in two different folders, D:\MODEL and D:\SCENARIO will be subtracted and the results will be saved, as an equidistant IDF, in the folder D:\EFFECT. Suppose HEAD_20101231_L1.IDF is found in D:\MODEL (SOURCEDIRA), an identical filename is searched for in D:\SCENARIO (SOURCEDIRB). The yielding IDF will be DIFF_20101231_L1.IDF and will be written in D:\EFFECT.

Example 3: batch-array definition

```
setlocal EnableDelayedExpansion
set n=0
for %%a in (1.30 0.80 0.70 0.75 0.85 1.25 0.75 0.75 0.85 1.10 1.15 ) do (
set RCH_factor[!n!]=%%a
set /A n+=1
)

setlocal EnableDelayedExpansion
set n=0
for %%a in (1990 1991 1992 1993 1994 1995 1996 1997 1998 1999 2000 2001) do (
set year[!n!]=%%a
set /A n+=1
)

for /L %%i in (0,1,11) do (
echo FUNCTION=IDFCALC > calc_idf.ini
echo !RCH[%%i]!
echo FUNC= "C=!RCH_factor[%%i]!*A" » calc_idf.ini
echo SOURCEDIRA=d:\RCH_grids\RCH_averaged_1990-2001.idf » calc_idf.ini
echo SOURCEDIRC=d:\RCH_grids\RCH_!year[%%i]!.idf » calc_idf.ini
```

This example uses multiple arrays to end up with one recharge grid per year based on a given recharge factor.

8.2.2 IDFSCALE-Function

With this function it is possible to (re)scale IDF-files according to different methodologies, see section 6.7.3.

FUNCTION=	**IDFSCALE**
SCALESIZE=	Enter the cell size of the upscaled or downscaled IDF-file(s), e.g. SCALESIZE=100.0 meaning that the cellsize of the resulting IDF-file(s) will be 100 square meter uniformly.
The keyword SCLTYPE_UP and SCLTYPE_DOWN may be specified both because it is possible to execute upscaling and downscaling in one action on IDF's containing different scales. In case SCLTYPE _UP and/or SCLTYPE_DOWN are not specified than default values are used, for SCLTYPE _UP: 2 and for SCLTYPE_DOWN: 1.	
SCLTYPE_UP	Enter the scale type. Choose from the following: 1: boundary scaling (**rule:** minus values above positive values *x* above zero values); 2: arithmetic scaling (**rule:** sum *n*-values *x* within coarse cell, excluding the *NoDataValues*, and divide them by *n*); 3: geometric scaling (**rule:** take log()-function for *n*-values *x* within a coarse cell, excluding *NoDataValues* and zero values, sum them, divide them by *n* and take the exp() function); 4: sum (**rule:** sum *n*-values *x*, excluding *NoDataValues*); 5: sum conductance (**rule:** sum n-values times ratio values to calculate the average conductance over cells for upscaled cell.) 6: inverse (**rule:** take the inverse (x^{-1}) of *n*-values *x* within a coarse cell, excluding *NoDataValues* and zero values and divide them by *n*. 7: most frequent occurrence (**rule:** take that value *x* that occurs mostly within a coarse cell, excluding *NoDataValues*); 8: sum inverse (**rule:** take the inverse (x^{-1}) of *n*-values *x* within a coarse cell, excluding *NoDataValues* and zero values); 9: percentile (**rule:** take the value *x* that occurs for a given percentage within a coarse cell, excluding *NoDataValues*); 10: block value (**rule:** takes the center value of the cells that needs to be upscaled.) 11: Darcian method (**rule:** take the value *x* that occurs after a Darcian simulation of fine mesh with extent of the coarse cell, excluding *NoDataValues*); 12: homogenization (**rule:** take the value *x* that occurs after a Darcian simulation with periodic boundaries of fine mesh with extent of the coarse cell, excluding *NoDataValues*); 13: global-local method (**rule:** take the value *x* that occurs after a Darcian simulation with realistic boundary conditions of fine mesh with extent of the coarse cell, excluding *NoDataValues*); 14: 3d simulations (**rule:** Calculates with help of a numerical model the needed initial values of the upscaled model per cell; 15: zonation (**rule:** Calculates an upscaled value as the most frequent value for the integer values within the coarse grid cell, and the fraction as the averaged fraction, while ignoring those cells that do not coincide with the upscaled integer value (e.g. x_1=1.5; x_2=2.25 and x_3=1.4, means that the most frequent integer is 1, and the average fraction for 1 is (0.5+0.4+0.0)/3=0.3, so the final value is 1.3).
SCLTYPE_DOWN=	Enter the scale type. Choose from the following: 1: interpolation (**rule:** produces a good guess for al finer gridcells by a linear interpolation based on the coarse gridcells, excluding the *NoDataValues*); 2: gridvalues (**rule:** assign the value of the coarse gridcell to all finer gridcells).
SOURCEIDF	Enter the name of the IDF-file to be upscaled or downscaled, e.g. SOURCEIDF=D:\DATA\TRANSMISSIVITY.IDF.
OUTFILE	Enter the name of the upscaled or downscaled IDF-files, e.g. OUTFILE=D:\DATA\SCALED_TRANSMISSIVITY.IDF.

PERCENTILE=	Enter a percentile, e.g. PERCENTILE=0.5 (0.0<PERCENTILE<1.0). This keyword is obliged whenever SCLTYPE=9.
KMIN=	Enter a value that represents the minimal K-value. This value accounts for the upper- and lower-boundaries of the clay-layer within the voxels.
ILGROUP=	Enter a value for the amount of layers that will be taken into account to find the representative K-value at which a clay-layer begins.
WEIGHFACTOR=	Enter a weight factor, e.g. WEIGHFACTOR=0.5. This keyword is optional in case SCLTYPE in {1,3,4,5,6,9}, the default value is WEIGHFACTOR=1.0.
BLOCK=	Enter the size of the interpolation block, e.g. BLOCK=16. This keyword is optional whenever SCLTYPE=-1, meaning that a matrix of 4x4 will be used for the interpolation of each point. The default value is BLOCK=4 and other possible values are BLOCK in {4,16,36,64,100). In most situation good results are obtained with BLOCK=4.
WINDOW= (optional)	Enter the coordinates of the window that need to be computed, solely. Enter coordinates of the lower-left corner first and then the coordinates of the upper-right corner, e.g. WINDOW=100000.0, 400000.0, 200000.0, 425000.0. When WINDOW= is absent, the entire dimensions of the first mentioned IDF-file will be used.
The following are keywords that are optional in case SCLTYPE=14, solely.	
SOURCEDIR=	Enter the name of the folder that contains IDF-files that need to be used for a 3D simulation, e.g. SOURCEDIR=D:\DATA\K*.IDF.
BUFFER=	Enter the size of the buffer to be used for the Darcian simulation, BUFFER=5 (SCLTYPE in {9,10,11,12}). The default value is BUFFER=0.
ANI_X= ANI_Z=	Enter the horizontal (ANI_X) or vertical (ANI_Z) anisotropy, e.g. ANI_Z=0.3. This means that the permeability will be 3 time less permeable in vertical direction than in horizontal direction. Default value is ANI_X=ANI_Z=1.0.
DH_X= DH_Y= DH_Z=	Enter the pressure for the x, y and z direction to be imposed on the system, e.g. DH_X=1.0. This means that the hydraulic pressure difference along the x direction is equal to meter. By default DH_X=DH_Y=DH_Z=0.0.
QRATE=	Enter the strength of a extraction well positioned in the lower most model-layer, e.g. QRATE=-100 m^3/day. By default QRATE=0.0, however, in those case where QRATE<>0.0, DH_X, DH_Y and DH_Z will be ignored.
	Enter the permeability to be substituted for those cells that contain *NoDataValues*.
AQFR_KD=	

Example 1

FUNCTION=IDFSCALE
SCLTYPE_UP=1
SCALESIZE=250.0
SOURCEIDF=D:\DATA\BOUNDARY_L1.IDF
OUTFILE=D:\DATA\BOUNDARY_L1_250.IDF

This example shows how to upscale an IDF-file with boundary conditions.

Example 2

FUNCTION=IDFSCALE
SCLTYPE_DOWN=1
SCALESIZE=5.0
SOURCEIDF=D:\DATA\HEAD_STEADY-STATE_L1.IDF
OUTFILE=D:\DATA\HEAD_STEADY-STATE_L1_5X5.IDF

This example shows how to downscale an IDF-file with computed heads.

Example 3

FUNCTION=IDFSCALE
SCLTYPE_UP=3
SCALESIZE=500.0
WINDOW=100000.0,425000.0,150000.0,500000.0
SOURCEIDF=D:\DATA\HEAD_STEADY-STATE_L1.IDF
OUTFILE=D:\DATA\HEAD_STEADY-STATE_L1_500.IDF

This example shows how to upscale transmissivity for a specific window.

Example 4

FUNCTION=IDFSCALE
SCLTYPE_UP=12
SCALESIZE=100.0
SOURCEDIR=D:\GEOTOP\SEL.IDF*
OUTFILE=D:\GEOTOP\VERTICAL_C.IDF
BUFFER=5
ANI_X=3.0
DHZ=1.0
DHX=0.0
DHY=0.0

This examples show an example how to upscale permeability with a 3D Darcian simulation. The result will be vertical resistances.

8.2.3 IDFMEAN-Function

The IDFMEAN function can be used to compute a new IDF-file with the mean value (or minimum, maximum value) of different IDF-files. It is not necessary to have exactly similar IDF-files (see section 6.7.3).

FUNCTION=	**IDFMEAN**
ILAYER=	Enter the layer numbers for the IDF-files to be averaged, e.g. ILAYER=1,3.
SDATE=	Enter the starting date (yyyymmdd) for which IDF-files are used, e.g. SDATE=19980201.
EDATE=	Enter the ending date (yyyymmdd) for which IDF-files are used, e.g. EDATE=20111231.
NDIR=	Enter the number of folders to be processed repeatedly, e.g. NDIR=10.
SOURCEDIR{i}=	Enter the folder and wildcard for all files that need to be used, e.g. SOURCEDIR1=C:\DATA\HEAD\HEAD*.IDF. Repeat SOURCEDIR{i} for NDIR times. Do not include year, month or day before or after the wildcard *.
CFUNC=	Specify the name of the function to be applied. Choose out of: MEAN, to compute mean values (equal weighed); MIN to compute the minimum values; MAX to compute the maximum values. SUM to compute the sum of the values per grid cell; PERC to compute the median value (50 percentile). The default is CFUNC=MEAN.
PERCVALUE=	Specify a percentile whenever CFUNC=PERC, e.g. PERCVALUE=50.0 for median values.
IYEAR=	Specify particular year (within SDATE and EDATE) to be used exclusively, e.g. 2001,2003,2005. IYEAR is filled in for all years in-between SYEAR and EYEAR.
NPERIOD=	Enter a number of periods to be defined to use IDF-file within these periods solely, e.g. NPERIOD=2. NPERIOD=0 by default.
PERIOD{i}=	Enter a period i (ddmm-ddmm), e.g. PERIOD1=1503-3110 to express the period 15^{th} of March until the 31^{th} of October.
ISEL=	Enter a code for the area to be processed: ISEL=1 will compute the entire region ISEL=2 will compute within given polygons; ISEL=3 will compute for those cells in the given IDF-file that are not equal to the *NoDataValue* of that IDF-file.
GENFNAME=	Enter a GEN-filename for polygon(s) for which mean values need to be computed. This keyword is obliged whenever ISEL=2.
IDFNAME=	Enter an IDF-file for which mean values will be computed for those cell in the IDF-file that are not equal to the *NoDataValue* of that IDF-file. This keyword is compulsory whenever ISEL=3

Example 1

FUNCTION=IDFMEAN
ILAYER=6
SDATE=19980714
EDATE=20110728
NDIR=1
SOURCEDIR1=D:\DATA\BDGFLF.IDF*

This example shows the mimimum configuration of this function and yield the mean values for all BDGFLF*.IDF-files in the folder D:\DATA that are assigned to layer 6 (function searches for L6.IDF), and are within the periode 14^{th} of July 1998 and 28^{th} of July 2011.

The output file will be:

1 D:\DATA\BDGFLF_MEAN_1998-07-14_to_2011-07-28_L6.IDF;

2 D:\DATA\BDGFLF_COUNT_1998-07-14_to_2011-07-28_L6.IDF.

The latter shows the number of occurrences for each raster cell.

Example 2

FUNCTION=IDFMEAN
ILAYER=1,3
SDATE=19980101
EDATE=20000101
IYEAR=1999
NPERIOD=1
PERIOD1=1503-3110
ISEL=2
CFUNC=MAX
GENFILE=D:\DATA\AREA.GEN
NDIR=1
SOURCEDIR1=D:\DATA\HEAD.IDF*

This example shows a more extended configuration and will yield **maximum** values for all IDF-files inside the folder D:\DATA that meet the requirement HEAD*.IDF. Furthermore, they contain the key combination L1.IDF where “1” is defined by ILAY=1. The date expression should be within the time domain of the 1^{th} of Januari 1998 (SDATE) and 31^{th} of December 2000 (EDATE), within the year 1999 (IYEAR) and within the period between the 15^{th} of March and the 31^{th} of October (PERIOD1). Finally the mean values is computed within the polygon(s) described by the polygon AREA.GEN, solely. The output file will be:

1 D:\DATA\HEAD_MAX_19980101-20000101_L1.IDF;
2 D:\DATA\HEAD_DATEMAX_19980101-20000101_L1.IDF.

The latter shows the date (yyyymmdd) on which raster cell maximal values appeared.

8.2.4 IDFCONSISTENCY-Function

Use this function to make IDF-files consistent, meaning that the first IDF is always higher or equal to the second, which is higher or equal to the top of the IDF underneath, and so on. IDF files can represent anything, however, this tool is especially handy for consistencies applied on top- and bottom elevation of model layers.

FUNCTION=	**IDFCONSISTENCY**
NLAY=	Enter the number of model layers, e.g. NLAY=6.
TOP_L{i}=	Enter the IDF for the i^{th} modellayer that represents the top of modellayer *i*, e.g. TOP_L1=D:\INPUT\TOP_L1.IDF. Constant value may be entered as well, e.g. TOP_L1=10.0.
BOT_L{i}=	Enter the IDF for the i^{th} modellayer that represents the bottom of modellayer *i*, e.g. BOT_L1=D:\INPUT\BOT_L1.IDF. Constant value may be entered as well, e.g. BOT_L2=-43.12.
OUTPUTFOLDER=	Enter the foldername in which the adjusted IDF-files will be saved, e.g. OUTPUTFOLDER=D:\RESULT. Whenever a file is entered by a constant value, e.g. TOP_L1=23.32, a file will be created called TOP_L1.IDF that represents the (corrected) value.
ICLEAN=	Enter an option for the cleaning of the IDF files. Whenever ICLEAN=1 the procedure removes all data in all files whenever at least a single nodata value is found among them at that specific location, whenever ICLEAN=2, Removes all data whenever at least a nodata value is found for the first and second idf file at that specific location. By default ICLEAN=0 which mean that only consistency corrections are applied for cells not equal to their nodata values.
WINDOW=	Enter the coordinates of the window that needs to be computed. Enter coordinates of the lower-left corner first and then the coordinates of the upper-right corner, e.g. WINDOW=100000.0, 400000.0, 200000.0, 425000.0. When WINDOW= is absent, the entered IDF-files by TOP_L{i} and BOT_L{i} need to be equally in their dimensions. Otherwise they will be upscaled (mean) or downscaled (interpolation) to the entered CELLSIZE.
CELLSIZE=	Enter the cell size (meter) for the IDF-files that will be created, e.g. CELL_SIZE=25.0.
MINLAY_THICKNESS=	Enter the minimum layer thickness to be applied for the permeable layers in the model. Only the layers between the TOP and BOT of the same layer are taken into account, e.g. MINLAY_THICKNESS=0.10; in case the layer thickness of the second modellayer (THICKNESS_L2=TOP_L2-BOT_L2) is smaller then 0.10 m the TOP_L2 will be corrected, so the thickness of layer 2 becomes 0.10 m.
MINLTZERO_OPT	Enter this option when it is preferred to apply the given minimal layer thickness (defined with “MINLAY_THICKNESS”) also to layers that have a thickness of 0.0 m. By default this option is set to 0, which means that layers with a layer thickness equal to zero will not be corrected.

Example

FUNCTION= IDFCONSISTENCY
NLAY=2
WINDOW=120000.0,298000.0,240000.0,430000.0
CELLSIZE=100.0
TOP_L1=D:\MODEL\TOP_L1.IDF
TOP_L2=D:\MODEL\TOP_L2.IDF
BOT_L1=D:\MODEL\BOT_L1.IDF
BOT_L2=D:\MODEL\BOT_L2.IDF
OUTPUTFOLDER=D:\OUTPUT

This example corrects the top and bottom IDF-files specified by the TOP_L{i} and BOT_L{i} keywords in a top-bottom consistent manner and scales the IDF-files to the specified WINDOW and CELL_SIZE.

8.2.5 IDFSTAT-Function

The IDFSTAT function can be used to perform some elementary statistical analyses on the content of IDF-files. You can use the *IDF Info* functionality in iMOD, alternatively (see section 6.3).

FUNCTION=	**IDFSTAT**
SOURCEDIR=	Enter the name of a folder that contains a specific set of IDF-file(s), e.g. {installfolder}:\DATA\RESULTS\HEAD*.IDF. All IDF-files that agree, will be included in the analysis.
IFORMAT (optional)	Select the type of output desired, e.g. IFORMAT=1. Default value is IFORMAT=0. See for the differences in output format at the description of OUTFILE.
OUTFILE=	Specify a filename for the resulting statistical analysis, e.g. {installfolder}\DATA\RESULTS\RESULT.CSV. A result of this can look as (IFORMAT=0): 1,AHN.IDF 2,AHN_FILTERED.IDF 3,AHN_SCALED.IDF File, Population, Mean, Variance, P(0), . . . , P(100) 1, 585917, 9.2359428, 0.0248852, -6.8000002, . . . , 4.0799999 2, 40000, 1.9279687, 0.0015057, -0.1490000, . . . , 2.9757273 3, 147912, 9.2729473, 0.0498930, -6.7449999, . . . , 335.730011 All percentiles will be computed between 0 and 100 by steps of 5. The output format can also look as (IFORMAT=1): File, Population, Mean, Variance, Min, Max, Median AHN_1 , 585917, 9.2359428, 0.0248852, -6.8000002, 4.0799999, -0.324343 AHN_2, 40000, 1.9279687, 0.0015057, -0.1490000, 2.9757273, 1.35984 AHN_3, 147912, 9.2729473, 0.0498930, -6.7449999, 335.730011, 87.32234

Example 1

FUNCTION=IDFSTAT
SOURCEDIR=D:\DATA\AHN.IDF*
OUTFILE=D:\DATA\STAT.CSV

This examples illustrated how to get the statistics of all IDF-files inside the folder D:\DATA that agree with the wildcard AHN*.IDF; results will be written in the file D:\DATA\STAT.CSV.

8.2.6 IDFMERGE-Function

The MERGE function can be used to merge different IDF-files into a new IDF-file. If these IDF-files might overlap, an interpolation between the overlapping IDF-files will be carried out.

FUNCTION=	**IDFMERGE**
NMERGE=	Enter the number of IDF-files that need to be merged, e.g. NMERGE=6.
SOURCEIDF{i}=	Enter the i^{th} IDF-file, e.g. SOURCEIDF1=D:\SUBMODEL1\HEAD_L1.IDF, SOURCEIDF2= D:\SUBMODEL2\HEAD_L1.IDF. Repeat this keyword NMERGE-times. Whenever NMERGE is absent, the keyword SOURCEDIR will be used.
SOURCEDIR=	Enter the source folder and part of the filename that need to be merged, e.g. D:\DATA\HEAD*L1.IDF to merge all files that corresponds to this wild-card. This keyword SOURCEDIR is used whenever the keyword NMERGE is absent.
TARGETIDF=	Specify a filename for the resulting IDF-file, e.g. {installfolder}\TOTAL\HEAD_L1.IDF.
WINDOW=	Specify a window in which the entered IDF-files (SOURCEIDF{i}, SOURCEDIR) will be merged only. Enter coordinates of the lower-left corner first and then the coordinates of the upper-right corner, e.g. WINDOW=100000.0, 400000.0, 200000.0, 425000.0. When WINDOW= is absent, the total dimension of all selected IDF-files in the SOURCEDIR will be used.
MASKIDF=	Enter an IDF-file that needs to be mask areas (those with values equal to the *NoDataValue* in the MASKIDF) in the merged results, e.g. D:\MASK\AREA.IDF.

Example 1

FUNCTION=IDFMERGE
NMERGE=2
SOURCEIDF1=D:\MODEL1\HEAD_L1.IDF
SOURCEIDF2=D:\MODEL2\HEAD_L1.IDF
TARGETIDF=D:\RESULT\HEAD_L1.IDF

This example merges two IDF-files, HEAD_L1.IDF and HEAD_L1.IDF from two different folders, into a single one D:\RESULTS\HEAD_L1.IDF.

Example 2

FUNCTION=IDFMERGE
MASKIDF=D:\MASK\AREA.IDF
WINDOW=120000.0,425000.0,165000.0,465000.0
*SOURCEDIR=D:\DATA\HEAD*_L1.IDF*
TARGETIDF=D:\DATA\HEAD_MERGED_L1.IDF

This example merges all IDF-files in the folder D:\DATA that agree with the filename HEAD*_L1.IDF, such as HEAD_A1_L1.IDF, HEAD_A2_L1.IDF. The merged results will be “clipped” for the given extent by WINDOW and will be “masked” out by the given NoDataValues in the MASKIDF. Finally the results will be saved in HEAD_MERGED_L1.IDF.

8.2.7 IDFTRACE-Function

Use this function to make a spatial IDF file with unique zone numbers of non-connecting areas of a given IDF file.

FUNCTION=	**IDFTRACE**
IDF_IN=	You can enter the IDF file with the areas, e.g. IDF_IN=D:\DATA\LAKES.IDF. All values greates than 0.0 will be processed.
IDF_OUT=	You can enter the output IDF file with areas numbered uniquely, e.g. IDF_OUT=D:\DATA\LAKES_ID.IDF.
MINT= (optional)	Enter the minimal size of the aggregated areas to be numbered, e.g. MINT=10. By default MINT=0 and all locations with values greater than 0.0 will be used to create a numbered zone, if you enter MINT=10, only areas that are aggregated to be more than 10 locations will be numbered.

Example

FUNCTION=IDFTRACE
IDF_IN=D:\DATA\LAKE.IDF
IDF_OUT=D:\DATA\LAKE_ID.IDF
MINT=10

This example creates an IDF file that can be used by the Lake package for the identification of indivual lakes, larger than 10 gridcells from the entered IDF file at IDF_IN.

8.2.8 CREATEIDF-Function

The CREATEIDF function can be used to create IDF-files out of ESRI ASC File Formats, see section 9.13. Be aware of the fact that you can open more of these ASC files in the *iMOD Manager*, alternatively (see section 5.4).

FUNCTION=	CREATEIDF
SOURCEDIR=	Enter the name of a folder that contains a specific set of ASC file(s), e.g. {installfolder}\DATA\RESULTS\HEAD*_L*.ASC. All ASC files that agree will be converted to IDF-files.
TOPWC=	Enter the wildcard that specifies the part of the filename that represents the top elevation of the data, e.g. SEL_*.ASC. In this case, iMOD will search for the **absolute** top elevation to be defined at the location of the asterix, e.g. SEL_0.45.ASC will yield the value 0.45.
BOTEL=	Enter the **relative** bottom of the elevation to be added to the top elevation, e.g. BOTEL=-0.5 will yield an **absolute** bottom elevation of 0.45-0.5=-0.10.
ADD=	Enter a value to add to the top and the bottom elevation (TOP and BOT), e.g. ADD=3, TOP=TOP+3.
MULT=	Enter a value to multiply with the top and the bottom elevation (TOP and BOT), e.g ADD=3, TOP=TOP*3.

Example 1

FUNCTION=CREATEIDF
SOURCEDIR=D:\DATA\TOP.ASC*

The above mentioned example transforms all ESRI ASCII gridfiles that agree with the wildcard TOP*.ASC into the IDF format. The yielding files will have identical names with the extension .IDF, and will be placed in the same folder as their ASCII files, so TOP1.ASC becomes TOP1.IDF.
Files will be overwritten without questioning!

Example 2

FUNCTION=CREATEIDF
SOURCEDIR=D:\DATA\SEL.ASC*
TOPWC=SEL_*.ASC
BOTEL=-0.5

The following example translates all ESRI ASCII gridfiles that agree with the wildcard SEL*.ASC. into SEL*.IDF-files. Moreover, a top elevation (TOPWC) will be extracted from the filename at the position of the wildcard, so the function tries to read a real value at the position of the asterix, suppose the filename is SEL_0.25.ASC, the value finally read is 0.25. It will be used to enter the TOP elevation inside the IDF (see section 9.5 for the syntax of IDF-files). The bottom elevation will be equal to the top elevation (0.25 in this example) plus the given value BOTEL, in this case -0.5, thus bottom elevation is 0.25+-0.5=-0.25.

8.2.9 CREATEASC-Function

The CREATEASC function can be used to create ESRI ASC files out of IDF File, see section 9.13. This function will always replace '****" with a NODATA value.

FUNCTION=	CREATEASC
SOURCEDIR=	Enter the name of a folder that contains a specific set of IDF file(s), e.g. {installfolder}\DATA\RESULTS\HEAD*_L*.IDF. All IDF files that agree will be converted to ASC-files.

Example 1

FUNCTION=CREATEASC
SOURCEDIR=D:\DATA\TOP.IDF*

The above mentioned example transforms all IDF gridfiles that agree with the wildcard TOP*.IDF into the ESRI ASCII format. The yielding files will have identical names with the extension .ASC, and will be placed in the same folder as their IDF files, so TOP1.IDF becomes TOP1.ASC.

Files will be overwritten without questioning!

8.2.10 XYZTOIDF-Function

Use this function to create an IDF from a plain data file(s) or IPF file(s) that contain x,y,z data at least. The column ("z") can contain any type of (real) data. Also use this function to generate a 3D model of the subsoil via indicator-interpolation of various thressholds (lithology and permeability).

FUNCTION=	**XYZTOIDF**	
XYZFILE=	Enter the name of a plain text file that contains x,y,z data, e.g. XYZFILE=D:\DATA\XYZ.TXT. The format of the file should be: 1st line: header; next lines: x, y, z-data.	
IPFFILE=	Enter the name of an IPF file that contains x,y,z data, e.g. IPFFILE=D:\DATA\POINTS.IPF.	
	IXCOL= (optional)	Enter the column number of the IPF that contains the x-coordinates, e.g. IXCOL=1 (default value).
	IYCOL= (optional)	Enter the column number of the IPF that contains the y-coordinates, e.g. IYCOL=2 (default value).
	IZCOL= (optional)	Enter the column number of the IPF that contains the z-coordinates, e.g. IZCOL=3 (default value).
	ASSF_ COLUMN= (optional)	Enter the column number of the associated file of the given column IZCOL in the IPF, e.g. ASSF_COLUMN=2. In this manner, the gridding will take the values from associated files instead of those from the IPF file. This is only applied whenever the IZCOL is equal to the column in which associated files are listed in the IPF file.
	ASSF_ IDEPTH= (optional)	Enter ASSF_IDEPTH=0 to enter a the starting date from which values are picked from the associated files, e.g. ASSF_STARTDATE=20121231. Enter ASSF_IDEPTH=0 to enter an initial depth and interval to interpolate intermediate interfaces e.g. ASSF_TOP.
	Enter the following items whenever ASSF_IDEPTH=0	
	ASSF_ STARTDATE=	Enter the starting date from which values are picked from the associated files, e.g. ASSF_STARTDATE=20121231.
	ASSF_ ENDDATE=	Enter the end date to which values are picked from the associated files, e.g. ASSF_ENDDATE=20160515.
	ASSF_DDATE=	Enter the time interval for which subsequent gridding is carried out, e.g. ASSF_DDATE=14 which mean that each 14 days between the given ASSF_STARTDATE and ASSF_ENDDATE will be processed. Alternatively the keywords 'D','W','M','Y' can be applied to denote 'daily', 'weekly', 'monthly' and 'yearly'.
	Enter the following items whenever ASSF_IDEPTH=1	
	ASSF_ TOP=	Enter the uppermost elevation of the first interface, e.g. ASSF_TOP=4.0.
	ASSF_ BOT=	Enter the lowermost elevation of the last interface, e.g. ASSF_BOT=-40.0.
	ASSF_ DZ(.)=	Enter the thickness of all interfaces, e.g. ASSF_DZ=5.0. This can be a list of values as well, e.g. ASSF_DZ=5.0,2.5,10.0. In this case the first interface has a thickness of 5.0 m, the second 2.5 m and the third 10.0 m, all the other remaining interfaces will have 10.0 m as well.
	ASSF_ ZPLUS=	Enter the vertical offset to be applied to look beyond the current interface, e.g. ASSF_ZPLUS=0.5. In this case, the boreholes will be read for a length of DZ(i) $\pm$ ASSF_ZPLUS.
	Enter the following items whenever ASSF_IDEPTH=2	
	NLAY=	Enter the number of interfaces, e.g. NLAY=10. The interpolation of information will take place in between the interface i and $i+1$.

	INT_L{i}=	Enter the IDF file for each interface i up to NLAY, e.g. INT_L1=D:\DATA\INTERFACE_L1.IDF.
	KSUM= (optional)	Enter KSUM=1 to generate permeability value as a weighted sum of all individual permeability value within each vertical interval. By default KSUM=0 and the permeability is that permeability associated with the most common lithology.
	TRIMTOP_IDF= (optional)	Enter the name of an IDF file to be used to trim the interpolated values at the top, e.g. TRIMTOP_IDF=D:\DATA\DEM.IDF.
	TRIMBOT_IDF= (optional)	Enter the name of an IDF file to be used to trim the interpolated values at the bottom, e.g. TRIMBOT_IDF=D:\DATA\BEDROCK.IDF.
GENFILE= (optional)	Enter the name of a Gen-file that contains the data x,y,z data, e.g. GENFILE=D:\DATA\DATA.GEN.	
NBLNFILE= (optional)	Specify the number of GEN-files that need to be taken into account in the grid-interpolation.	
	BLNFILE_{i}=	Give the name of a specific GEN-file containing x,y-data of the faults that need to be taken into account in the grid-interpolation. Repeat this keyword for NBLNFILE times.
	FCTBLNFILE_{i}= (optional)	Enter the mutiplication factor for each BLNFILE for which the points will be moved further away if they are on both sides of a fault cq. line in the GEN file. Repeat this keyword for NBLNFILE times.
IBLNTYPE= (optional)	Specify the type of BLNFILE, e.g. IBLNTYPE=0 (default) which denotes that iMOD will use the BLNFILE or any blanked out area via a) outside the entered trim IDF files at TRIMTOP_IDF and/or TRIMBOT_IDF or b) outside the active area specified by IDFFILE_POINTER as a breakline in the interpolation (only point that can be ”seen“ without obstructed by the line will be used). Whenever IBLNTYPE=1, the BLNFILE and/or the other options a) and b), will be used as a ”polygon“ (only points within the same polygon will be used).	
SOURCEDIR= (optional)	Enter the folder and wildcard that corresponds to the XYZFILEs, e.g. D:\DATA\REGION*.XYZ. The keyword XYZFILE and IPFFILE should be absent, otherwise XYZFILE or IPFFILE will be used!	
TARGETDIR= (optional)	Enter the folder to which the IDFFILEs will be saved that correspond to the XYZFILEs or IPFFILEs found in the SOURCEDIR, e.g. TARGETDIR=D:\DATA\IDFS, the results will be called D:\DATA\IDFS\REGION*.IDF whenever SOURCEDIR=D:\DATA\REGION*.XYZ. This keyword is compulsory whenever the optional keyword SOURCEDIR is applied.	
NODATA= (optional)	Enter a *NoDataValue* for those data points that need to be excluded from the gridding, e.g. NODATA=0.0 to exclude data points equal to zero. By default, NODATA=-999.99.	
ILOG= (optional)	Enter ILOG=1 to perform a log transformation for the data points that need to be grided. By default, ILOG=0.	
WINDOW= (optional)	Enter the coordinates of the window that needs to be computed. Enter coordinates of the lower-left corner first and then the coordinates of the upper-right corner, e.g. WINDOW=100000.0, 400000.0, 200000.0, 425000.0. When WINDOW= is absent, the entire XYZFILE or IPFFILE will be gridded for its maximum extent.	
CS=	Enter the cellsize of the IDFFILE to be created, e.g. CS=100.0. This keyword is not necessary whenever IDFFILE is specified.	
IDFFILE_IN= (optional)	Enter the name of an IDF-file for which data points that are equal to its *NoDataValue* will be interpolated.	
IDFFILE_ POINTER= (optional)	Enter an IDF-file that will be needed to specify what locations will be interpolated. It temporarily blanks out the IDF-file given by IDFFILE_IN before the interpolation and resets the original value in the blanked-out area after the interpolation.	
IDFFILE	Enter the name of an IDF-file that need to be created, e.g. IDFFILE=D:\DATA\XYZ.IDF.	
GRIDFUNC=	Enter the grid function to be used:	
	MIN	minimum of all data points inside a gridcell;

MAX	maximum of all data points inside a gridcell;
MEAN	mean of all data points inside a gridcell;
PERC	percentile of all data points inside a gridcell.
PERCENTILE=	Enter the percentile (>=0 PERCENTILE <=100.0) whenever GRIDFUNC=PERC, e.g. PERCENTILE=25.0. A percentile value will be interpolated linearly, whenever the entered PERCENTILE falls in-between two values. Entering PERCENTILE=0.0 or PERCENTILE=100.0 will yield the same results as with GRIDFUNC=MIN and GRIDFUNC=MAX, respectively, however, the latter functions are faster than the function GRIDFUNC=PERC. Entered values beyond 0.0 and above 100.0, will be trimmed to 0.0 and 100.0, automatically.
BIVAR:	takes a bivariate interpolation
SKRIGING: **OKRIGING:**	takes a Kriging interpolation, SKRIGING stands for Simple Kriging (assuming a constant mean over the entire domain); OKRIGING stands for Ordinary Kriging (assuming a constant mean in the neighborhood of each estimation point). Choose one of both at one time.
PNTSEARCH= (optional)	Specify PNTSEARCH=1 to allow to search for points within the distance specified by RANGE the minimum number of points used for the interpolation. Default MINP=10 (or less whenever the dataset contains less points).
MINP= (optional)	Enter the minimal number of points needed for the interpolation, e.g. whenever MINP=50, iMOD will take the 50 nearest point in the Kriging interpolation. Whenever the keyword IQUADRANT=1, the entered MINP value is valid for each quadrant, so whenever MINP=10 and IQUADRANT=1, the total number of points will be $10 \times 4 = 40$. Default MINP=10 (or less whenever the dataset contains less points). This keyword can be entered only whenever PNTSEARCH=1.
IQUADRANT= (optional)	Select IQUADRANT=1 to force an equal distribution of point from the four quadrants around an point to be estimated. This keyword can be entered only whenever PNTSEARCH=1. By default IQUADRANT=0.
RANGE=	Enter the range that defines a neighbourhood within which all data points are related to one another, e.g. RANGE=1000 meter. It is possible to specify multiply values for RANGE whenever ASSF_IDEPTH=2 for each interface. If the number of entered values for RANGE is less than the number of interfaces to be computed, the last entered values for RANGE will be re-used for the remaining interfaces. The semivariance will become approximately equal to the variance of the whole surface itself (SILL).
SILL=	Enter the distance at which the semivariance approaches a flat region. SILL is referred as the range or span of the regionalized variable, e.g. SILL=2500. This parameter resembles a variance. The magnitude of the semivariance between points depends on the distance between the points. A smaller distance yields a smaller semivariance and a larger distance results in a larger semivariance.
NUGGET=	Enter the offset of the semivariogram, e.g. NUGGET= 10.0.

	KTYPE=	Enter the type of the Kriging model to be used to compute the value at X_i, choose from: 1 Linear Model: X_i=DIST$_i$*(SILL-NUGGET)/RANGE 2 Spherical Model: DIST$_i$<=RANGE: X_i=SILL*(1.5*(DIST$_i$/RANGE))-(0.5*(DIST^3/RANGE^3)) DIST$_i$>RANGE: X_i=SILL 3 Exponential Model: X_i=SILL*(1.0-EXP(-DIST$^3{}_i$/RANGE))
	STDEVIDF=	Enter the name for the standard deviation computed, e.g. STDEVIDF=D:\VAR.IDF.
	VARIOGRAM:	create a semivariogram, this yields no interpolation of the data, it generates a table filled in with a variogram. Whenever the WINDOW keyword is specified, a variogram will be computed for those data points that are within the bounds of the given WINDOW. The results will be written in the VARIOGRAM.TXT file, see coming pages for an example.
	LAG-INTERVAL=	Enter the number of distances over which the VARIOGRAM will be computed, e.g. LAGINTERVAL=50 will yield fifty intervals equally distributed between zero and the maximum distance between point.
	LAG-DISTANCE=	Specify the lag distance, e.g. LAGDISTANCE=50.0 to overrule the lag distance as computed by LAGINTERVAL.
	ELLIPS_LEN= (optional)	Specify the ellipsoids maximum diameter (Δx), e.g. ELLIPS_LEN=5000.0 meter. Specify this parameter only whenever IQUADRANT=0. By default ELLIPS_LEN is equal to the length specifief at RANGE.
	ELLIPS_ANI= (optional)	Specify the ellipsoids anisotropy, e.g. ELLIPS_ANI=45.0 to denote a 45 degrees shift from the north clock-wise. Specify this parameter only whenever IQUADRANT=0 and . By default ELLIPS_ANI=0.0.
	ELLIPS_RAT= (optional)	Specify the ellipsoids ratio between the semi-major axis (Δx) and the semi-minor axes (Δy), e.g. ELLIPS_RAT=0.5 denotes a flattened ellipsoid whereby the Δy is half the size of Δx. Specify this parameter only whenever IQUADRANT=0. By default ELLIPS_RAT=1.0.
	PCG:	takes a Preconditioned Conjugate Gradient interpolation
	HCLOSE= (optional)	Enter a closure criterion for the PCG solver to terminate the interpolation, e.g. HCLOSE=0.001 (this is the default).
	RCLOSE= (optional)	Enter a closure criterion for the PCG solver to terminate the interpolation, e.g. RCLOSE=1000.0 (this is the default).
	NINNER= (optional)	Enter the number of inner iteration for the PCG solver, e.g. NINNER=50 (this is the default). Use large values for NINNER to speed up the interpolation since the problem to be solved is linear.
INDICATOR (optional)	Enter INDICATOR=1 (by default INDICATOR=0) to use an indicator interpolation. Whenever this function is used, a value is assigned to a point whenever it meets a given thresshold (see NTHRESHOLD). In that case it gets score of 1.0, this is corrected for the penetration length of this in the current interval. After interpolation it yields a probability of occurrence, a fraction between 0.0 and 1.0. Use INDICATOR=-1 to skip the interpolation and recompute the probability of occurence and permeability from the existing files, in case INDICATOR=1 is applied first.	
	NTHRESHOLD	Enter the number of thresholds to be used in the indicator interpolation, e.g. NTHRESHOLD=2.

	THRESHOLD{i}	Enter for each threshold the appropriate name, e.g. THRESHOLD1=SAND. Use quotation marks for thresholds with spaces, e.g. THRESHOLD{i}="SILTY SAND". Enter NTHRESHOLD number of thresholds. iMOD will generate a probability map for each threshold for each interval. It also generates a map per intergface of the most-common threshold per grid cell.
	KH_THRESHOLD{i}	Enter for each threshold the horizontal permeability, e.g. KH_THRESHOLD1=35.0. Enter NTHRESHOLD number of horizontal permeability values. iMOD will generate a horizontal permeability map for each threshold for each interval. It also generates a total, averaged horizontal permeability map per interface as the weighted sum of all individual thresholds.
	KV_THRESHOLD{i}	Enter for each threshold the vertical permeability, e.g. KV_THRESHOLD1=10.0. Enter NTHRESHOLD number of vertical permeability values. iMOD will generate a vertical permeability map for each threshold for each interval. It also generates a total, averaged vertical permeability map per interface as the weighted sum of all individual thresholds.

Example 1

FUNCTION=XYZTOIDF
XYZFILE=D:\DATA\28BN.XYZ
IDFFILE=D:\DATA\28BN.IDF
CS=5.0
GRIDFUNC=MEAN

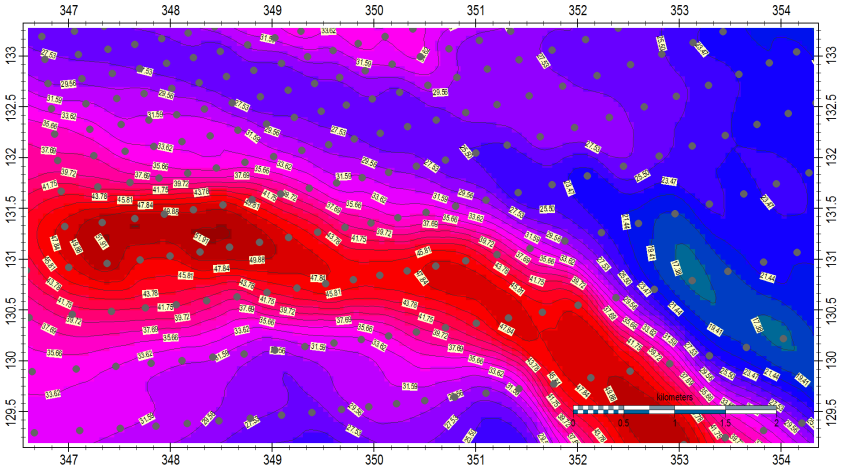
Above an example is given how to rasterize, for a 5x5 resolution (CS=5.0), the content of an XYZ file by means of its mean values (GRIDFUNC=MEAN) inside the individual rastercells. The default *NoDataValue* of -999.99 will be assigned to those rastercells that doesn't have any points inside, moreover, data points that have this particular value will be left out.

Example 2

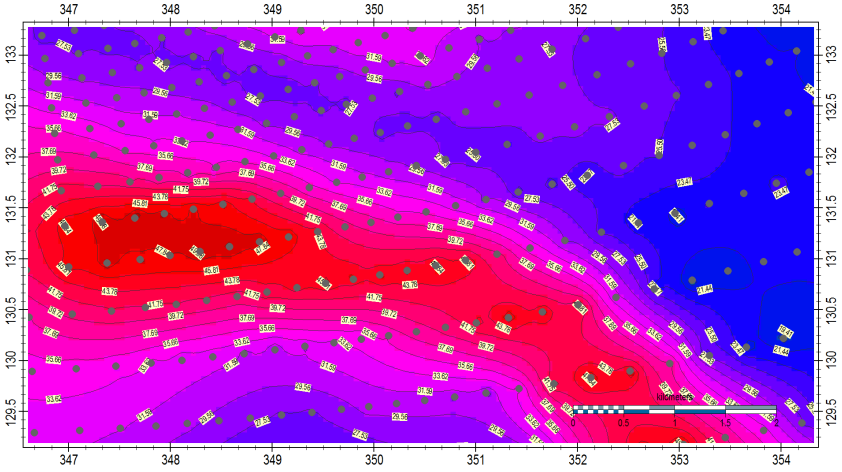
FUNCTION=XYZTOIDF
SOURCEDIR=D:\DATA.XYZ*
TARGETDIR=D:\DATA\IDF
IDFFILE=D:\DATA\28BN.IDF
CS=25.0
GRIDFUNC=PERC
PERCENTILE=5.0
NODATA=0.0

Example above shows how to rasterize, for a 25x25 resolution (CS=25.0), the content of all *.XYZ files in the folder D:\DATA, by means of its 5.0 percentile values (PERCENTILE=5.0; GRIDFUNC=PERC) inside the individual rastercells. A *NoDataValue* of 0.0 will be assigned to those rastercells that doesn't have any points inside, moreover, data points that have this particular value will be left out.

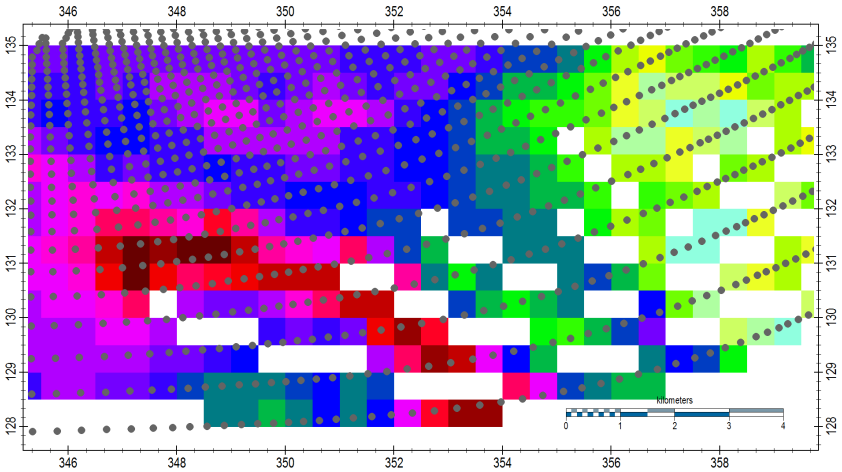
Example of Bivariant interpolation:



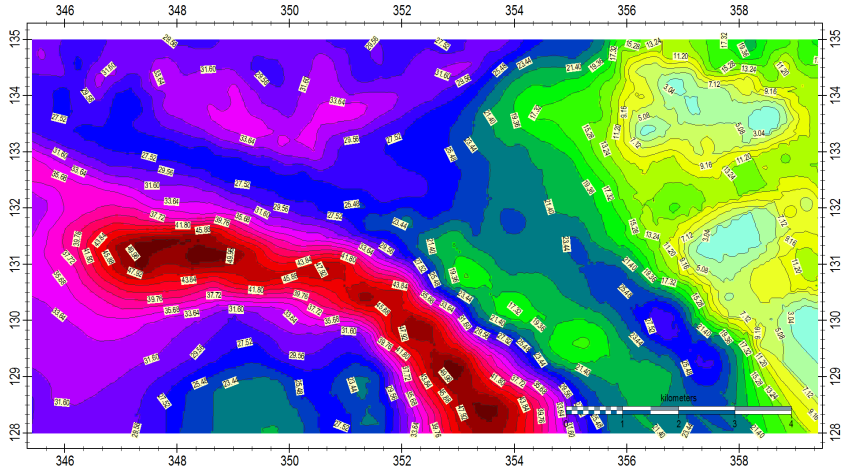
Example of PCG interpolation:



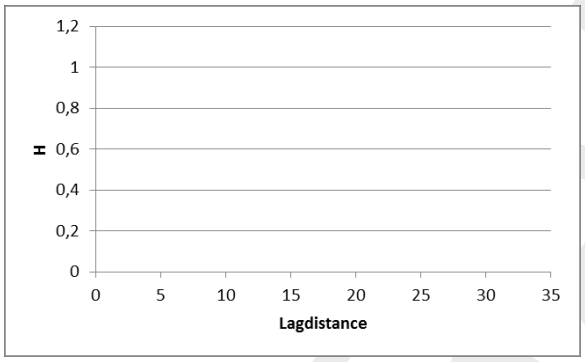
Example of MEAN sampling:



Example of Kriging interpolation (linear model):

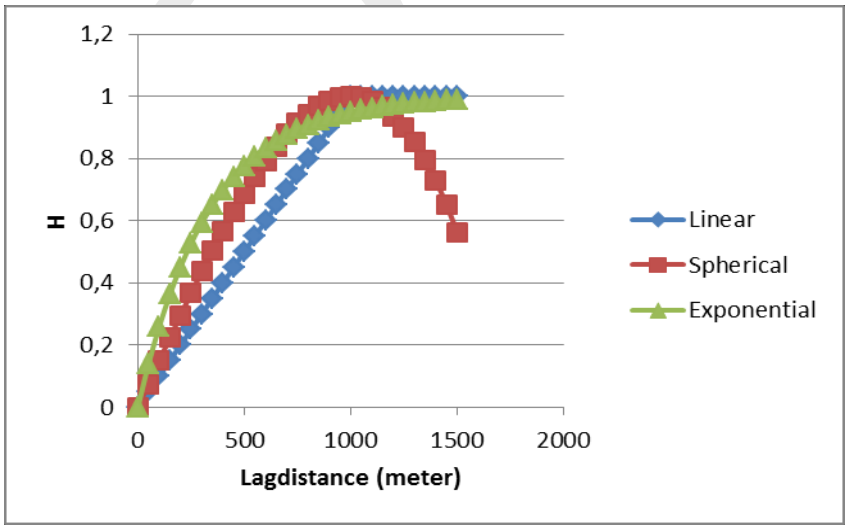


Example of a Variogram:



From the above presented variogram, the SILL would be 30 and the corresponding RANGE approximately 1000m, at that distance the SILL value flattens. The NUGGET is zero in this example.

Example of the different models to be used in the Kriging interpolation:



8.3 ISG-FUNCTIONS

8.3.1 GEN2ISG-Function

The GEN2ISG function reads a GEN file and creates a ISG file. There are two ways to use this function; IUSEDAT=1 and IUSDAT=0. These will be explained below.

Use IUSEDAT=1 to force using the *.DAT file (with GEN file information).

FUNCTION=	**GEN2ISG**
GENFNAME=	Give a GEN file containing x and y coordinates of GEN segments.
IUSEDAT=1	Enter the column number in the DAT file that represents:
STAGE1_COLUMN=	Stage on Startpoint of the Segment, e.g. STAGE1_COLUMN=1
STAGE2_COLUMN=	Stage on Endpoint of the Segment, e.g. STAGE2_COLUMN=2
BOTL1_COLUMN=	Bottom on Startpoint of the Segment, e.g. BOTL1_COLUMN=3
BOTL2_COLUMN=	Bottom on Endpoint of then Segment, e.g. BOTL2_COLUMN=4
	shape of the Crosssection of the Segment:
SLOPE_L_COLUMN=	Left slope, e.g. SLOPE_L_COLUMN=5
SLOPE_R_COLUMN=	Right slope, e.g. SLOPE_R_COLUMN=6
BWIDTH_COLUMN=	Bottom width, e.g. BWIDTH_COLUMN=7

Use IUSEDAT=0 if you do not have a *.DAT file (with GEN file information).

FUNCTION=	**GEN2ISG**
GENFAME=	Give a GEN file containing x and y coordinates of GEN segments.
IUSEDAT=0	the following keywords must be given:
IDFSTAGE=	Enter the column number in the DAT file that represents the y coordinate, e.g. STAGE1_COLUMN=1.
IDFSUMMER=	Enter the filename of the IDF-file containing summer heads.
IDFSUMMER_BACKUP=	Enter the filename of the IDF-file containing summer heads. This file is created as backup-file in case no values are found in IDFSUMMER or by using the SAMPLE_SEARCH function.
IDFWINTER=	Enter the filename of the IDF-file containing winter heads.
IDFWINTER_BACKUP=	Enter the filename of the IDF-file containing winter heads. This file is created as backup-file in case no values are found in IDFWINTER or by using the SAMPLE_SEARCH function.
SUMMERPERIOD= (optional)	Enter the day and month at which the summer period starts (e.g. default value is SUMMERPERIOD=0104 in case summer starts at the 1st of April each year).
WINTERPERIOD= (optional)	Enter the day and month at which the winter period starts (e.g. default value is WINTERPERIOD=0110 in case summer starts at the 1st of October each year)
START_YEAR=	Enter the starting year of the calculation period, e.g START_YEAR=2014.
END_YEAR=	Enter the end year of the calculation period, e.g. END_YEAR=2020.
IDFBOTTOM=	Enter the name of the IDF-file containing all bottom values of the area.
BOTTOMVALUE=	If there's no IDFBOTTOM-file available give a constant value for the bottom height, e.g. BOTTOMVALUE=12.32.
SAMPLE_DISTANCE (optional)=	Enter a value (in meters) that accounts for the distance iMOD needs to sample the IDF-file on, e.g. 100 m. On default SAMPLE_DISTANCE=250 m.
CCFFNAME=	Enter the name of a CCF-file that describes the cross-section that will be used to insert cross-section for all segments.
SEARCH_DISTANCE= (optional)	Enter a value (in meters) that accounts for the distance iMOD needs to resample the IDF-file on, e.g. 50 m. On default SEARCH_DISTANCE=250 m. This value will only be used if there were no values found by making use of the SAMPLE_DISTANCE.

IPUZZLE=	Enter a 0.0 or 1.0. If a value of 1.0 is given, iMOD tries to find all possible places where segments can be joined and join them if needed.
IDFRESISTANCE=	Enter the name of IDF-file that contains resistance values of the area.
RESISTANCE=	Enter a resistance value that will be applied to the whole area. Can only be used if IDFRESISTANCE is not available.
IDFINFILTRATIONFACTOR=	Enter the name of IDF-file that contains infiltration factors for the area.
INFILTRATIONFACTOR=	Enter an infiltration factor that will be applied to the whole area. Can only be used if IDFINFILTRATIONFACTOR is not available.
OUTFILE=	Enter the name of the ISG output-file, e.g. NEWFILE.ISG.

Example 1

FUNCTION=GEN2ISG
FUNCTION=GEN2ISG
GENFNAME=d:\Model\Data\Shape_data\river_lines.gen
IUSEDAT=0
IDFSUMMER=d:\Model\Basic_data\SUMMER_LEVEL_RIVER.IDF
IDFWINTER=d:\Model\Data\Basic_data\WINTER_LEVEL_RIVER.IDF
IDFBOTTOM=d:\Model\Data\Basic_data\BODEMHOOGTE_RIVER.IDF
SAMPLE_DISTANCE=25.0
CCFFNAME=d:\Model\Data\Basic_data\CROSS-SECTION.CCF
IDFRESISTANCE=d:\Model\Data\Basic_data\RIV_RESISTANCE.idf
IDFINFILTRATIONFACTOR=d:\Model\Data\Basic_data\INFFACTOR_RIVER.IDF
OUTFILE=d:\Model\Data\ISG_data\River.isg

Above an example is given how to use the GEN2ISG. This example will generate and ISG file based on river levels, bottom elevation, resistance, infiltration factor and the location of the river segments. The ISG will contain the following information: as many as cross-sections per river segment as there are unique ID-numbers in CCFFNAME related file, calculation nodes on each segment intersection and segment nodes as much as there are coordinate points defined in the GEN file.

8.3.2 ISGGRID-Function

Use this function to rasterize the selected ISG-files into IDF-files that can be used by iMODFLOW in a runfile.

FUNCTION=	**ISGGRID**		
ISGFILE_IN=	Enter an ISG-file that need to be simplified, e.g. ISGFILE_IN=D:\PO.ISG.		
CELL_SIZE=	Enter the cell size (meter) for the IDF-files that will be created from the ISG-file mentioned by ISGFILE_IN, e.g. CELL_SIZE=25.0.		
MINDEPTH= (optional)	Enter the minimum water depth (meter) used for the calculation of the conductance of the stream bed, e.g. MINDEPTH=1.0. The default value is MINDEPTH=0.1.		
MAXWIDTH= (optional)	Enter the maximal width of a stream (meter) used for the calculation of the conductance of the stream bed. Introducing this parameter limits any stream width larger than MAXWIDTH, e.g. MAXWIDTH=1000. The default value is MAXWIDTH=250.		
OUTPUT FOLDER=	Enter a foldername to save all rasters into, e.g. OUTPUTFOLDER=D:\OUTPUT. The following rasters will be saved:		
	1	COND	The computed stream bed conductance (m^2/day)
	2	STAGE	The interpolated riverlevel (m+MSL)
	3	BOTTOM	The interpolated riverbed height (m+MSL)
	4	INFFCT	The interpolated river infiltration factor (-)
	5	TOTAL_LENGTH	Total length if existing river segments in a single rastercell (meter)
	6	MEAN_ WPERIMETER	Mean wetted perimeter within a river segment
	7	MEAN_WIDTH	Mean stream bed width
	8	RESISTANCE	Interpolated river resistance (days). IMPORTANT to note is that a minimal resistance is applied of 0.001 days to avoid extraordinary conductance (COND) values.
	9	EROSION	Erosion matrix to be used to extent the riverbed existence over more rastercells
	The following will be created only whenever ICDIST=1		
	10	EFFECT	The computed water level that are influenced by the weirs.
	11	CUR_ID	Identification of structures for current segment.
	12	NEX_ID	Identification of following structure for current segment.
POSTFIX= (optional)	Enter a postfix to be used to add to the end of the IDF-file names mentioned above, e.g. POSTFIX=_SUMMER yields STAGE_SUMMER.IDF instead of STAGE.IDF.		
NODATA=	Enter a *NodataValue* for which water levels will be skipped in determining the waterlevels along profiles, e.g. NODATA=-999.		
WINDOW= (optional)	Enter the coordinates of the window that needs to be computed. Enter coordinates of the lower-left corner first and then the coordinates of the upper-right corner, e.g. WINDOW=100000.0, 400000.0, 200000.0, 425000.0. When WINDOW= is absent, the entire ISG will be gridded for its maximum extent.		
ISAVE=	Enter the numbers to be saved solely, e.g. ISAVE=1,1,1,0,0,0,0,0,0,0,0,0 to identify that the IDF-files COND, STAGE and BOTTOM need to be saved.		
IPERIOD= (optional)	Specify whether the waterlevels need to be calculated for a specific period, by default IPERIOD=1 which means that waterlevels will be computed as the mean over the entire existing periods within the ISG-file (which can be different among the segments). Specify IPERIOD=2 to enter a date over which the waterlevels will be averaged.		
	SDATE=	Enter a starting date to compute averaged waterlevels for, e.g. SDATE=19910101 to represent the 1^{st} of January 1991.	
	EDATE=	Enter a starting date to compute averaged waterlevels for, e.g. EDATE=19911231 to represent the 31^{st} of December 1991.	

	DDATE=	Enter a date-difference to be used to compute more rasters for different periods, e.g. DDATE=14 means that a sequence between SDATE and EDATE will be computed with length of 14 days. By default DDATE=0 which will ignore any timesteps in-between the SDATE and EDATE variables. The names of the IDF-file will be extended to include a date notification, e.g. STAGE{POSTFIX}_19910101.IDF
ICDIST= (optional)	Set this value to 1 to compute the effects of weir as stored in the ISG file. By default ICDIST=0. See keyword OUTPUTFOLDER to get the names of the extra IDF files that will be created.	
ISIMGRO= (optional)	Set this value to 1 to export the gridded values for the ISG into a MetaSWAP file svat_swnr_drng.inp.	
	SVAT2SWNR_DRNG=	Enter the name for the svatswnr_drng.inp file.
	SEGMENTCSVFNAME=	Enter the CSV that contains the list of ...
	THIESSENFNAME=	Enter an IDF file that represents the SVAT-id for MetaSWAP.
	AHNFNAME=	Enter an IDF file with the surface level.
	SYSID=	Enter a single value for the system identification.
	WDEPTH=	Enter a water depth that will be used to define the appropriate trapezia for MetaSWAP.
IAVERAGE= (optional)	Enter IAVERAGE=1 to apply an arithmetic mean for stages, bottomlevels, resistances and infiltrationfactor over time. Enter IAVERAGE=2 to apply the median value for those parameters. The default value is IAVERAGE=1.	
IEXPORT= (optional)	Set this value to 1 to export the gridded ISG into a MODFLOW river file, important to notice is that it yield a single value for each gridded cell. The export river file will be called OUTPUTFOLDER \modflow.riv. By default IEXPORT=0 and IDF files will be created.	
	NLAY=	Enter the number of model layers for which the gridded ISG file need to be assigned vertically, e.g. NLAY=3. This option is only valid whenever IEXPORT=1.
	TOP_Li=	Enter an IDF file that represents the TOP elevation of the ith layer, e.g. TOP_L1=D:\TOP_L1.IDF.
	BOT_Li=	Enter an IDF file that represents the BOT elevation of the ith layer, e.g. BOT_L2=D:\BOT_L2.IDF.
	KHV_Li=	Enter an IDF file that represents the horizontal permeability of the ith layer, e.g. KHV_L2=D:\KHV_L2.IDF.
	BND_Li=	Enter an IDF file that represents the boundary condition of the ith layer, e.g. BND_L4=D:\BND_L4.IDF.

Example:

FUNCTION=ISGGRID
ISGFILE_IN=D:\PO.ISG
CELL_SIZE=100.0
NODATA=-999.99
ISAVE=1,1,1,1,0,0,0,0,0,0,0,0
IPERIOD=1
SDATE=19980101
EDATE=19980131
OUTPUTFOLDER=D:\PO_GRIDS

The example above will rasterize the entire ISG for the period of the 1$^{\mathrm{th}}$ of January up to the 31$^{\mathrm{th}}$ of January 1998 on a 100x100 meter grid.

8.3.3 ISGADDCROSSSECTION-Function

Use this function to add cross-sections to an existing ISG-file (see section 9.9.3 for more information about the content of an ISG-file and storage of cross-sections). The methodology is twofold:

1 *One-dimensional cross-sections:*
Reading cross-sectional information from a text file for one-dimensional cross-sections.
2 *Two-dimension cross-sections:*
Reading two-dimensional bathymetry from an IDF-file for areas that are defined by a pointer IDF. All existing cross-section will be used to assign two-dimensional cross-sections. The dimension of the bathymetry will be overruling the dimensions of the pointer IDF.

FUNCTION=	**ISGADDCROSSSECTION**
ISGFILE_IN	Enter an ISG-file for which cross-sections need to be added, e.g. ISGFILE_IN=D:\DATA\MAAS.ISG.
ISGFILE_OUT=	Enter an ISG-file to save the renewed ISG for, e.g. ISGFILE_OUT=D:\DATA\MAAS_NEWCROSSSECTIONS.IDF.
2-D Cross-sections	
CROSS_PNTR=	Enter the name of an IDF-file describes the spatial distribution of two-dimensional cross-sections, e.g. CROSS_PNTR=D:\DATA\PNTR.IDF. This pointer file is used to denote areas with equal values as the pointer value at the location of the cross-section on the segmenet. The bathymetry for those areas will be read from CROSS_BATH and applied as a 2-D cross-section on the segment.
CROSS_BATH=	Enter the name of an IDF-file that describes the bathymetry for the riverbed at the locations where the values for CROSS_PNTR are $\neq$ to the pointer value at the corresponding cross-section at the segment, e.g. CROSS_BATH=D:\DATA\RIVERBED.IDF.
CROSS_ZCHK (optional)=	Enter the name of an IDF-file that describes the Reference Height to be used to distinguish between areas with positive and negative values for CROSS_PNTR, e.g. CROSS_ZCHK=D:\DATA\REFHEIGHT.IDF.
CROSS_CVAL (optional)=	Enter the name of an IDF-file that describes the Resistance values to be used to distinguish different resistance in inundation areas, e.g. CROSS_CVAL=D:\DATA\RESISTANCE.IDF. The values in this IDF act as a multiplication factor to the given resistance (attribute RESIS in the ISD2-file, see 9.9.2) at the nearest calculation point. Bear in mind that the multiplication factor will be saved in the ISG as an integer with a maximal value of 256.
1-D Cross-sections	
CROSSSECTION_IN=	Enter the filename that stores the renewed cross-sections, e.g. CROSSSECTION_IN=D:\DATA\CROSS.TXT. The syntax of the CROSSSECTION_IN file is a free-formatted, comma-separated-values file with for which each row is defined as follows (be aware that you do not include a header in the text-file):

XC,YC,LABEL,N,X_1,X_2,..,X_N,Z_1,Z_2,..,Z_N	
XC	X-coordinate (meter) for the cross-section;
YC	Y-coordinate (meter) for the cross-section
LABEL	Label for the cross-section, maximum length is 32 characters.
N	Number of cross-sections points.
X_i	Specify as many distances as needed to define the bathymetry of the riverbed. The amount of definitions *N* should be $>$3.
Z_i	Specify as many elevations as needed to define the bathymetry of the riverbed. The amount of definitions *N* would be equal the number of definitions used for X_i.

ICLEAN= (optional)	Enter ICLEAN=1 to clean ALL cross-sections before adding new ones, apply ICLEAN=2 to remove existing cross-section only for those segments where an update of the cross-section will be applied. By default ICLEAN=1.
WIDTH_IDF= (optional)	Specify an IDF that represents the width of default cross-sections to be placed on all segments and a default water depth of 5 meter where no cross-section will be placed based on the entered data by the following keywords, e.g. WIDTH_IDF=D:\DIST.IDF. This keyword is necessary only whenever ICLEAN=1.
MAXDIST= (optional)	Specify a distance (meter) over which the cross-section will be snapped to the segment, e.g. MAXDIST=5.0. By default MAXDIST=0.0 meter.

Example 1 (one-dimensional cross-sections):

FUNCTION=ISGADDCROSSSECTION
ISGFILE_IN=D:\iMOD-DATA\MAAS.ISG
CROSSSECTION_IN=D:\DATA\CROSS.TXT
WIDTH_IDF=D:\DATA\WIDTH.IDF
MAXDIST=2.5
ISGFILE_OUT=D:\iMOD-DATA\MAAS_RENEWEDCROSSSECTIONS.ISG

The example above will add cross-sections based on the entered CROSS.TXT file that specifies a cross-section for “New Cross” as follows:

12000.0,45300.0,”New Cross”,-10.0,-5.0,-2.5,2.5,7.5,12.0,5.0,3.0,2.0,1.0,2.5,5.0

the results will be saved in MAAS_RENEWEDCROSSSECTION.ISG.

Example 2 (two-dimensional cross-sections):

FUNCTION=ISGADDCROSSSECTION
CROSS_PNTR=D:\DATA\PNTR.IDF
CROSS_BATH=D:\DATA\BATHEMETRY.IDF
ISGFILE_OUT=D:\iMOD-DATA\MAAS_RENEWEDCROSSSECTIONS.ISG

The example above will transform the existing cross sections with two-dimensional definitions based on the pointerfile read in CROSS_PNTR and the corresponding bathymetry read in BATHEMETRY.IDF.

8.3.4 ISGSIMPLIFY-Function

Use this function to reduce the amount of calculation points in a ISD file (part of the ISG-files, see section 9.9.2). iMOD will eliminate calculation points that do not add significant information to the declination of waterlevels, in other words, whenever the gradient of the waterlevel can be described by less calculation points, iMOD will locate those calculation points that are able to represent the original waterlevel most optimally. iMOD will use the mean waterlevels for all calculation nodes to determine a mean descent of waterlevels along a segment. Simplification will be carried out for segments as a whole. Whenever segments will be very short, this function will have a minor effect.

FUNCTION=	**ISGSIMPLIFY**
ISGFILE_IN=	Enter an ISG-file that need to be simplified, e.g. ISGFILE_IN=D:\DATA\MAAS.ISG.
ZTOLERANCE=	Specify a distance (meter) for which the simplified waterlevel along a profile may differ from the original one, e.g. ZTOLERANCE=0.10.
NODATA=	Enter a NoDataValue for which waterlevels will be skipped in determining the waterlevels along profiles, e.g. NODATA=-999.
ISGFILE_OUT=	Enter an ISG-file to save the simplified ISG for, e.g. ISGFILE_OUT=D:\DATA\MAAS_SIMPLIFIED.IDF.

Example:

FUNCTION=ISGSIMPLIFY
ISGFILE_IN=D:\iMOD-DATA\MAAS.ISG
ZTOLERANCE=0.10
NODATA=-999.99
ISGFILE_OUT=D:\iMOD-DATA\MAAS_SIMPLIFIED.ISG

The example above will reduce the amount of calculation points such that the simplified waterlevel will be differ more than 0.10 from the original one, the results will be saved in MAAS_SIMPLIFIED.ISG.

8.3.5 ISGADJUST-Function

Use this function to perform changes to an existing ISG.

FUNCTION=	**ISGADJUST**
SESFILE=	Enter the name of the SES file, e.g. D:\iMOD-DATA\ISGEDIT\ISG-change-stage.SES.
LOGFILE=	Enter a name for the logfile showing all changes by listing both old and new parameter values , e.g. D:\iMOD-DATA\ISGEDIT\ISG-LOG.TXT default = .\log_ses.txt
OUTNAME=	Foldername for new location to save all ISG related files (*.isg, *.isp, *.isd etc), e.g. OUTNAME=D:\RIV\ISG_new

Example:

FUNCTION=ISGADJUST
SESFILE=D:\iMOD-DATA\ISGEDIT\ISG-change-stage.SES
LOGFILE=D:\iMOD-DATA\ISGEDIT\ISG-LOG.TXT
OUTNAME=D:\iMOD-DATA\RIV\ISG_new

The example above will preduce new ISG files based on new data in the SES file.

8.3.6 ISGADDSTRUCTURES-Function

Use this function will add weirs to an ISG.

FUNCTION=	**ISGADDSTRUCTURES**
ISGFILE_IN=	Enter the name of the ISG file.
IPFFILE_IN=	Enter the name of the IPF file containing weir data
	Specify the column number in the IPFFILE_IN that defines:
	IXCOL= .. the X coordinates, by default IXCOL=1.
	IYCOL= .. the Y coordinates, by default IYCOL=2.
	IDCOL= .. the ..., by default IDCOL=3.
	IOCOL= .. the ..., by default IOCOL=4.
	ISCOL= .. the ..., by default ISCOL=5.
	IWCOL= .. the ..., by default IWCOL=6.
MAXDIST=	Specify the maximum distance ...., by default MAXDIST=1000.
SYEAR=	Specify the start year, by default SYEAR=1980.
EYEAR=	Specify the end year, by default EYEAR=2012.
START_PERIOD_ SUMMER=	Specify the start date of summer, by default START_PERIOD_SUMMER='01-04'.
END_PERIOD_ SUMMER=	Specify the end date of summer, by default END_PERIOD_SUMMER='30-09'.
START_PERIOD_ WINTER=	Specify the start date of winter, by default START_PERIOD_WINTER='01-10'.
END_PERIOD_ WINTER=	Specify the end date of winter, by default END_PERIO_WINTER='31-03'.
DATE_WLEVEL=	Date of measure to be used to compute undisturbed waterlevel. Give 0-0-0 to compute the mean of all values or 28-02-1994 for a fixed date, by default DATE_WLEVEL=0-0-0.
IPFLOGFILE=	Enter a name for the log file, by default IPFLOGFILE=log.ipf.
ISGFILE_OUT=	Enter the name of the ISG output file.

Example:

FUNCTION=ISGADDSTRUCTURES
ISGFILE_IN=D:\RIV-DATA\LEGGER.ISG
IPFFILE_IN=D:\RIV-DATA\WEIR.IPF
ISGFILE_OUT=D:\RIV-DATA\LEGGER_V2.ISG

The example above will produce new ISG files based on new data in WEIR.IPF.

8.3.7 ISGADDSTAGES-Function

Use this function will add water levels to an ISG. Existing water levels wil be overwritten if the entered IPF with water levels has water levels for identical dates as already mentioned in the ISG file.

FUNCTION=	**ISGADDSTAGES**	
ISGFILE_IN=	Enter the name of the ISG file.	
IPFFILE=	Enter the name of the IPF file containing water level data via associated TXT-files. The construction of the IPFFILE_IN file must be:	
	Column 1	X coordinate.
	Column 2	Y coordinate.
	Column i	refers to the IEXT variable in the IPF-file to indicate the column with the associated TXT-files, see section 9.7 for more detailed description of the IPF-files.
ISGFILE_OUT=	Enter the name of the ISG output file.	
STAGETYPE= (optional)	Specify STAGETYPE=1 to enter river stages in the IPFFILE in depth values. By default STAGETYPE=0 and stages are entered in absolute values (m+MSL).	
ICLEAN= (optional)	Specify ICLEAN=1 to remove all existing entries for stages in the ISG file prior to adding new entries. By default ICLEAN=0 and existing entries are retained.	

Example:

FUNCTION=ISGADDSTAGES
ISGFILE_IN=D:\RIV-DATA\LEGGER.ISG
IPFFILE=D:\RIV-DATA\WATERLEVEL.IPF
ISGFILE_OUT=D:\RIV-DATA\LEGGER_V2.ISG

The example above will produce new ISG files based on new data in WATERLEVEL.IPF.

8.3.8 SFRTOISG-Function

This function can be used to convert the output of the SFR package onto an ISG file. In this way the existing functionalities of ISG Edit can be used to inspect and analyse the output of the SFR package.

FUNCTION=	**SFRTOISG**	
ISGFILE_IN=	Enter the name of the ISG file, e.g. SFRFILE_IN=D\MODEL\SFR.ISG.	
ISGFILE_OUT=	Enter the name of the ISG file that need to be created, e.g. SFRFILE_OUT=D\MODEL\SFR_RESULT.ISG.	
SFRFILE_IN=	Enter the name of the output file of the SFR package, e.g. SFRFILE_IN=D\MODEL\TUT_FSFR.TXT. The file need to contain exactly the amount of calculation points as the ISG files given at ISGFILE_IN. You should always use the ISG files that iMOD created during the export of your model to MF2005 files. This file contains per reach and per time step the following columns:	
	Column 1:	Layer number.
	Column 2:	Row number.
	Column 3:	Column number.
	Column 4:	Stream number.
	Column 5:	Segment number.
	Column 6:	Flow into Stream Reach, a positive number means flow in the stream, a negative number would not be feasible.
	Column 7:	Flow to Aquifer, negative means that groundwater migrates to surface water (stream acts as a drain), a positive value denotes that surface water infiltrates.
	Column 8:	Flow out of Stream Reach, a positive number means flow out of the stream, a negative number would not be feasible.
	Column 9:	Overland Runoff.
	Column 10:	Precipitation.
	Column 11:	Evaporation.
	Column 12:	Stream Head, the stage in the river.
	Column 13:	Stream Depth, the depth of the river.
	Column 14:	Stream Width, the width of the river.
	Column 15:	Conductance, the area of contact between the surface water and the groundwater, divided by the resistance of the riverbed.
	Column 16:	Gradient, the hydraulic gradient across the stream bed, negative gradient means that the stream acts as a drain. Large hydraulic gradients might indicate some conceptual errors. There is a difference in how the SFR package deals with unsaturated flow underneath the stream bed. As the conventional RIV package applies a hydraulic gradient as the difference between the surface water level and the river bed level, the SFR package takes the thickness of the river bed material instead.

Not all of the above mentioned columns are converted to the ISG file, the average stream discharge (half of the sum of the *Flow into Stream Reach* plus the *Flow out of Stream Reach*) is converted from m^3/d to m^3/s. Also the stream head, depth and width are transferred to the ISG file. Those four attributes are considered to be most valuable.

Example:

FUNCTION=SFRTOISG
SFRFILE_IN=D\MODEL\SFR.ISG.
SFRFILE_OUT=D\MODEL\SFR_RESULT.ISG.
SFRFILE_IN=D\MODEL\TUT_FSFR.TXT.

The example above will produce a new ISG files based on the results of the SFR package.

8.3.9 IPFTOISG-Function

This function can be used to convert an IPF file with appropriate columns into an ISG suitable for the SFR package. Two following pairs of coordinates will form a single segment in the ISG file. Also, only one calculation point and cross-section will appear on each segment. The cross-section will be rectangular (4 points) based on the given stream width (see below).

FUNCTION=	**IPFTOISG**
IPFFILE=	Enter the name of the IPF file, e.g. IPFFILE=D\DATA\STREAM.IPF.
IXCOL= (optional)	Enter the column number in the IPF file that represents the X-coordinate (m), e.g. IXCOL=2, by default IXCOL=1.
IYCOL= (optional)	Enter the column number in the IPF file that represents the Y-coordinate (m), e.g. IYCOL=3, by default IYCOL=2.
ILABELCOL= (optional)	Enter the column number in the IPF file that represents the stream label, e.g. ILABELCOL=5, by default ILABELCOL=3. This will be used to label the stream in the ISG file. If the label appeared to be empty (not available), the stream segment identification will be used as given by the keyword ISEGMCOL. In the end the segment name in the ISG file becomes "S_{*stream label*}_R_{i}" where i is the sequential order number.
ISEGMCOL= (optional)	Enter the column number in the IPF file that represents the stream segment identification, e.g. ISEGMCOL=5, by default ISEGMCOL=4. If two following pairs of coordinates in the IPF have similar stream segment identification, they become a segment in the ISG file.
IWIDTHCOL= (optional)	Enter the column number in the IPF file that represents the stream total width (m), e.g. IWIDTHCOL=6, by default IWIDTHCOL=5.
IBOTTOMCOL= (optional)	Enter the column number in the IPF file that represents the stream bottom height (m+MSL), e.g. IBOTTOMCOL=8, by default IBOTTOMCOL=6.
ISTAGECOL= (optional)	Enter the column number in the IPF file that represents the stream stage (m+MSL), e.g. ISTAGECOL=12, by default ISTAGECOL=7.
IPERMCOL= (optional)	Enter the column number in the IPF file that represents the stream bed permeability (m/d), e.g. IPERMCOL=2, by default IPERMCOL=8.
SDATE= (optional)	Enter the date for which the data will be recorde in the ISG file, e.g. SDATE=20101231123030 to represent the 31st of December 2010 at 12:30:30, by default SDATE=20000101000000.
ISGFILE=	Enter the name of the ISG file that need to be created, e.g. SFRFILE=D\MODEL\SFR_RESULT.ISG.

Example:

FUNCTION=IPFTOISG
IPFFILE=D\DATA\STREAM.IPF.
ISGFILE=D\MODEL\SFR.ISG.
ILABELCOL=4
ISEGM=3

The example above will produce a new ISG files compatible with the SFR package, for more clarification an example is given for the IPF file as well. This file will create 2 ISG segment called S_2391526_R_1 and S_2391526_R_2.

```
3
8
X
Y
SECTION
NAME
WIDTH
WATERLEVEL
BOTTOMLEVEL
```

```
PERM
0,TXT
459420.449184,5495001.214598,252-2391526,"2391526",4.000000,90.72400,89.724000,100.000
459327.814399,5495038.775708,252-2391526,"2391526",4.000000,90.27100,89.271000,100.000
459235.179516,5495076.336717,252-2391526,"2391526",2.000000,89.14100,88.141000,100.000
```

8.4 GEN-FUNCTIONS

8.4.1 GENSNAPTOGRID-Function

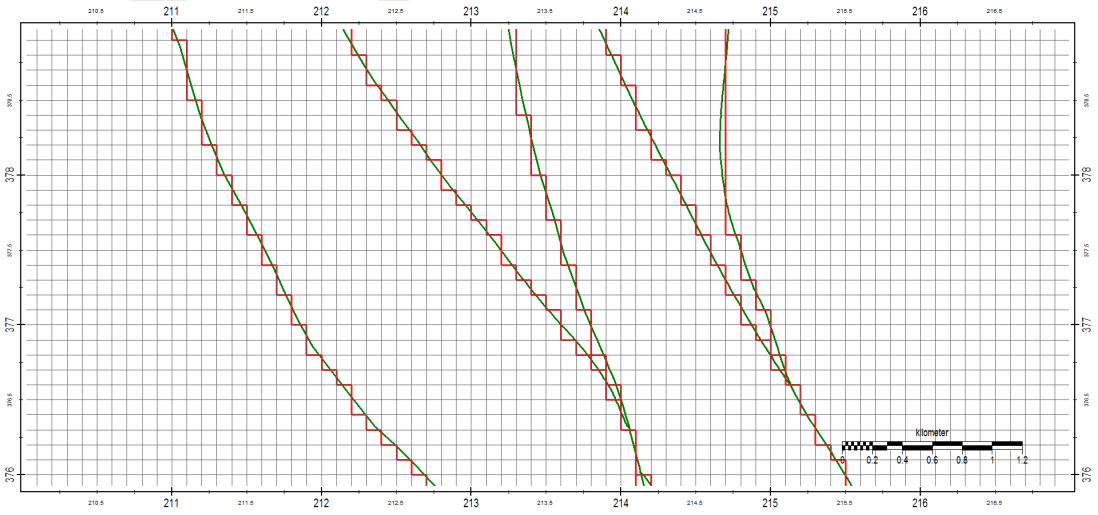
The *GENSNAPTOGRID* function can be used to rasterize a GEN file for a given raster. This gives more grip on the way these GEN files will be applied in a model for particular cell sizes. This GEN file can be transformed into a 3D GEN file as well.

FUNCTION=	**GENSNAPTOGRID**
IDFFILE= (optional)	Enter an IDF file for which the dimensions will be used to rasterize the specified GENFILE, e.g. IDFFILE=D:\BND\BND.IDF.
GENFILE=	Enter the name of a GEN-file, e.g. GENFILE=D:\DATA\AREA.GEN. These vertices in the GENFILE will be rasterized on the specified network via IDFFILE or an entered window and cell size.
WINDOW= (optional)	Enter the coordinates of the window that need to be computed, solely. Enter coordinates of the lower-left corner first and then the coordinates of the upper-right corner, e.g. WINDOW=100000.0, 400000.0, 200000.0, 425000.0. When WINDOW= is absent, the dimensions of the specified IDFFILE will be used.
CELL_SIZE= (optional)	Enter the cellsize of the grid that need to be used to rasterize the GEN file, e.g. CELL_SIZE=100.0.
GENFILE_OUT=	Enter the name of a GEN-file to be created, e.g. GENFILE_OUT=D:\DATA\AREA_RASTER.GEN.
I3D= (optional)	Specify I3D=1 whenever the GEN file need to be transformed into a 3D GEN file. By default I3D=0 and a regular, 2D GEN is created.
IDF_TOP=	Give an IDF file that represents the uppermost values of the GEN file, e.g. IDF_TOP=D:\FAULTS\BR-T-CK.IDF. Specify this keyword whenever I3D=1.
IDF_BOT=	Give an IDF file that represents the lowermost values of the GEN file, e.g. IDF_TOP=D:\FAULTS\BR-B-CK.IDF. Specify this keyword whenever I3D=1.

Example 1

FUNCTION=GENSNAPTOGRID
IDFFILE= D:\BND\BND.IDF
GENFILE=D:\DATA\AREA.GEN
GENFILE_OUT=D:\DATA\AREA_RASTER.GEN

The above mentioned example will rasterize the AREA.GEN on the dimensions of the specified BND.IDF, see the results on the next figure.



Example 2

FUNCTION=GENSNAPTOGRID
IDFFILE= D:\BND\BND.IDF
GENFILE=D:\DATA\AREA.GEN
GENFILE_OUT=D:\DATA\AREA_RASTER.GEN
I3D=1
IDF_TOP= D:\GEOLOGY\TOP_KI.IDF
IDF_BOT= D:\GEOLOGY\BOT_KI.IDF

The above mentioned example will rasterize the AREA.GEN on the dimensions of the specified BND.IDF and generates a 3D GEN using the values of the specified TOP- and BOT IDF files, see the results on the next figure.

8.4.2 GEN2GEN3D-Function

The GEN2GEN3D function reads a GEN file and creates a 3-D GEN file (see section 9.10) to be displayed in the 3-D tool or used in the runfile of iMODFLOW.

FUNCTION=	GEN2ISG
GENFILE_IN=	Give a GEN file containing x and y coordinates of GEN segments, e.g. GENFILE_IN=D:\FAULTS\FAULT.GEN.
IDF_TOP=	Give an IDF file that represents the uppermost values of the GEN file, e.g. IDF_TOP=D:\FAULTS\BR-T-CK.IDF.
IDF_BOT=	Give an IDF file that represents the lowermost values of the GEN file, e.g. IDF_TOP=D:\FAULTS\BR-B-CK.IDF.
GENFILE_OUT=	Specify the yielding GEN file, e.g. GENFILE_OUT=D:\FAULTS\FAULT_BR.GEN.
XSAMPLING= (optional)	Specify the sampling distance to add z-coordinates to the GEN file, e.g. XSAMPLING=100.0 will add a point each 100 meter. If this keyword is absent, XSAMPLING will be equal to the cell size of the IDF file entered at IDF_TOP.

Example 1

FUNCTION=GEN2GEN3D
GENFILE_IN=D:\MODEL\FAULTS\FAULT.GEN
IDF_TOP=D:\GEOLOGY\TOP_BREDA.IDF
IDF_BOT=D:\GEOLOGY\BOT_BREDA.IDF
XSAMPLING=250.0
GENFILE_OUT=D:\MODEL\FAULTS_BREDA.GEN

Above an example is given how to use the GEN2GEN3D. This example will generate and 3-D GEN file based on top- and bottome elevations of the geological formation BREDA; the result GEN is written in D:\MODEL\FAULTS_BREDA.GEN.

8.5 IPF-FUNCTIONS

8.5.1 IPFSTAT-Function

The IPFSTAT function can be used to perform statistical analyses on timeseries that are defined in IPF files as associated files.

FUNCTION=	**IPFSTAT**
IPF1=	Enter the name of an *.IPF file for which the associated timeseries need to be analysed, e.g. {installfolder}\MEASURE.IPF.
OUTFILE=	Specify a filename for the resulting IPF (or GEN whenever a GENFILE is specified), e.g. {installfolder}\RESULT.IPF.
VARIABLES=	Specify the number of variables to be computed. Only IPF1 specified: 1. Auto-Correlation - Correlation - MeanLag - NumberPoints 2. P50 over entire data period 3. (n)GxG starts at DMY1 and end at DMY2 - GHG - GLG - n(GxG) Both IPF1 and IPF2 specified: 1. Cross-Correlation - Correlation - MeanLag - NumberPoints 2. P50 IPF1 over overlapping data period IPF1 and IPF2 3. P50 IPF2 over overlapping data period IPF1 and IPF2 4. (n)GxG IPF1 starts at DMY1 and end at DMY2 - GHG - GLG - n(GxG) 5. (n)GxG IPF2 starts at DMY1 and end at DMY2 - GHG - GLG - n(GxG) Enter the variables by their subsequent numbers, e.g. VARIABLES=0, 1, 0 in case IPF1 is specified solely, it will compute the P50 only. Or VARIABLES=0, 0, 0, 1, 1 in case IPF1 and IPF2 are both specified, in this case for both the (n)GxG will be computed.
ICOLDATE1	Enter the column number that contains the date expression in the txt files associated to the first IPF file, on default ICOLDATE1=1.
ICOLVARS1	Enter the column number that contains the data, e.g. the measurement/computed head in the txt files associated to the first IPF file, on default ICOLDATE2=2.
IPF2=	Enter the name of a second *.IPF file for which the associated timeseries need to be analysed and compared to those in IPF1, e.g. {installfolder}:\COMPUTED.IPF. Be aware that the number of VARIABLES change whenever IPF2 is absent.
ICOLDATE2	Enter the column number that contains the date expression in the txt files associated to the second IPF file, on default ICOLDATE2=1.
ICOLVARS2	Enter the column number that contains the data, e.g. the measurement/computed head in the txt files associated to the second IPF file, on default ICOLDATE2=2.
GENFILE=	Enter the name of a *.GEN-file that needs to be used to aggregate value upon its individual polygons. **This option is valid in combination with IPF2 only.**

PERCENTILES=	Enter the percentiles to be computed for each individual polygon in the given GENFILE, e.g. 0.10, 0.90. If absent, PERCENTILES=0.10, 0.25, 0.50, 0.75, 0.90. **This option is valid in combination with GENFILE only.**
IINVERSE=	Enter IINVERSE=0 to use PERCENTILES as defined, however, enter IINVERSE=1 to **find** the percentiles that belong to the values entered by PERCENTILES. **This option is valid in combination with GENFILE only.**
RELATECOLIPF1= RELATECOLIPF2=	Enter the column number in IPF1 and IPF2 that need to be used to relate between the data in IPF1 and IPF2. **This option is valid in combination with IPF2 only.**
XLAG=	Specify the lagwidth to be used to compute the auto/cross correlation, e.g. XLAG=30 means that the auto/cross correlation will be computed over 30 (days). If absent XLAG=0.0 an auto/cross correlation will be computed between measurements on similar dates.
DLAG=	Specify the lag distance to be used to extent the search area, e.g. DLAG=7 means that 7 (days) before and 7 (days) after the given date+XLAG will be used to search for data. If absent DLAG=7.0 (days).
DMY1= DMY2=	Specify a starting and end date, both notated by yyyymmdd, e.g. 20110131 to express 31^{th} January 2011. If absent these values are DMY1=19000101 and DMY2=21001231, respectively. Both will be used for VARIABLES that compute (n)GxG values.
SURFACELEVEL=	Enter the name of an IDF-file that represent the surfacelevel, e.g. SURFACELEVEL=D:\DATA\AHN.IDF. This will be used to express the GxG value according to this surfacelevel.

Example 1

FUNCTION=IPFSTAT
IPF1=D:\TESTS\TEST.IPF
OUTFILE=D:\TESTS\OUT.IPF
VARIABLES=0,1,0

The example above computes the median groundwaterlevels (or equivalent) that are associated with the IPF file TEST.IPF. The result is stored in OUT.IPF.

Example 2

FUNCTION=IPFSTAT
IPF1=D:\TESTS\MEASURE.IPF
IPF2=D:\TESTS\MODEL.IPF
OUTFILE=D:\TESTS\RESIDUAL.IPF
VARIABLES=1,0,0,0,0
RELATECOLIPF1=4
RELATECOLIPF2=3
ICOLDATE2=1
ICOLVARS2=2

The example above, computes the cross-correlation between the measurements associated with the MEASURE.IPF and the computed values associates with the MODEL.IPF. The relation-columns are 4 and 3 for the MEASURE.IPF and MODEL.IPF, respectively.

8.5.2 IPFSPOTIFY-Function

The IPFSPOTIFY function can be used to *spotify* geological formations in existing model discretisations. It gives per modellayer the fraction that a geological formation exists.

FUNCTION=	**IPFSPOTIFY**	
IPFFILE_IN (optional)=	Enter the name of an *.IPF file for which the locations need to be *spotified* in the selected op geological formations, e.g IPFFILE_IN=D:\WELLS.IPF.	
	IPFFILE_OUT=	Specify an IPF file for which the fraction per geological formation will be saved, e.g. D:\RESULT.IPF.
	IXCOL=	Specify the column number in the IPFFILE_IN that defines the X coordinates, by default IXCOL=1.
	IYCOL=	Specify the column number in the IPFFILE_IN that defines the Y coordinates, by default IYCOL=2.
	IFCOL=	Specify the column number in the IPFFILE_IN that defines the attribute for which the fraction need to be computed, by default IFCOL=3.
	IZ1COL=	Specify the column number in the IPFFILE_IN that defines the top elevation to spotify underneath, by default IZ1COL=4.
	IZ2COL=	Specify the column number in the IPFFILE_IN that defines the bottom elevation to spotify above, by default IZ2COL=5.
	ILCOL=	Specify the column number in the IPFFILE_IN that defines the modellayer, by default ILCOL=6.
OUTPUT FOLDER (optional)=	Specify the output folder in which all IDF files will be saved with the individual fractions per geological formation per model layer, e.g OUTPUTFOLDER=D:\AQUIFER	
NLAY=	Specify the number of model layers to be *spotified*, e.g NLAY=10	
TOP_{i}=	Specify the top elevation for the i^{th} model layer, e.g TOP_L2=D:\MODEL\TOP_L2.IDF.	
BOT_{i}=	Specify the bottom elevation for the i^{th} model layer, e.g BOT_L2=D:\MODEL\BOT_L2.IDF.	
FORMTOP=	Specify the folder that stores the TOP elevation of geological formations, e.g. FORMTOP=D:\GEOLOGY*-T-CK.IDF. All files will be used that fit this wildcard definition.	
FORMBOT=	Specify the folder that stores the BOT elevation of geological formations, e.g. FORMBOT=D:\GEOLOGY*-B-CK.IDF. All files will be used that fit this wildcard definition.	

Example 1

FUNCTION=IPFSPOTIFY
IPFFILE_IN=D:\WELLS\TEST.IPF
IPFFILE_OUT=D:\SPOTIFIED\OUT.IPF
NLAY=2
TOP_L1=D:\MODEL\TOP_L1.IDF
BOT_L1=D:\MODEL\BOT_L1.IDF
TOP_L2=D:\MODEL\TOP_L2.IDF
BOT_L2=D:\MODEL\BOT_L2.IDF
REGISTOP=D:\REGIS-T-CK.IDF*
REGISBOT=D:\REGIS-B-CK.IDF*

The example above computes the fractions for each location in the IPFFILE_IN of all geological formations in REGISTOP and REGISBOT for each model layer.

Example 2

FUNCTION=IPFSPOTIFY
OUTPUTFOLDER=D:\FRACTIONS\AQUIFER
NLAY=2
TOP_L1=D:\MODEL\TOP_L1.IDF
BOT_L1=D:\MODEL\BOT_L1.IDF
TOP_L2=D:\MODEL\TOP_L2.IDF
BOT_L2=D:\MODEL\BOT_L2.IDF
REGISTOP=D:\REGIS-T-CK.IDF*
REGISBOT=D:\REGIS-B-CK.IDF*

The example above, computes the fractions for each cell in the model layers for each geological formation defined by the REGISTOP and REGISBOT keywords, the results are stored in the AQUIFER folder. To *spotify* aquitards in it is neccessary to switch the top and bottom elevations, e.g.

FUNCTION=IPFSPOTIFY
OUTPUTFOLDER=D:\FRACTIONS\AQUITARD
NLAY=1
TOP_L1=D:\MODEL\BOT_L1.IDF
BOT_L1=D:\MODEL\TOP_L2.IDF
REGISTOP=D:\REGIS-T-CK.IDF*
REGISBOT=D:\REGIS-B-CK.IDF*

8.5.3 IPFSAMPLE-Function

The function IPFSAMPLE samples IDF-files to add values to the points defined in an IPF file.

FUNCTION=	IPFSAMPLE
IPFFILE_IN=	Enter the name of an IPF file with minimal 2 columns that represents x- and y coordinates, e.g. D:\DATA\MEASURE.IPF.
IPFFILE_OUT=	Enter the name of an IPF file that need to be written with the results of the IDF values from the specified IDFFILE, e.g. D:\DATA\CHECK.IPF. Results read from the IDF-files in SOURCEDIR, will be stored as an extra column in IPFFILE_IN, the label will be identical to the name of the IDF-files.
SOURCEDIR=	Enter the name of an IDF-file that needs to be read by the points specified in the IPF file IPFFILE_IN, e.g. D:\DATA\RESULTS\HEAD.IDF.
IXCOL	Enter the column number in the IPF file IPFFILE_IN that represents the X-coordinate, e.g. IXCOL=4. By default IXCOL=1.
IYCOL	Enter the column number in the IPF file IPFFILE_IN that represents the Y-coordinate, e.g. IYCOL=6. By default IYCOL=2.
IACOL	Enter the column number to enter the sampled data from the IDF files, e.g. IACOL=3 which means that the entered starts at columns 3. By default IACOL=0 which means that the sampled data will be added at the end of the IPF file.

Example 1

FUNCTION=IPFSAMPLE
IPFFILE_IN=D:\WELLS.IPF
IPFFILE_OUT=D:\WELLS_KD.IPF
SOURCEDIR=D:\DATA\KD.IDF*

This example, adds values (columns) to all points in the IPF file WELLS.IPF, with the corresponding values from the KD*.IDF-files in the folder D:\DATA.

Example 2

FUNCTION=IPFSAMPLE
IPFFILE_IN=D:\WELLS.IPF
IPFFILE_OUT=D:\WELLS.IPF
SOURCEDIR=D:\DATA\KD.IDF*
IXCOL=4
IYCOL=3

This example, adds values (columns) to all points in the IPF file WELLS.IPF, with the corresponding values from the KD*.IDF-files in the folder D:\DATA. The x- and y coordinates in the IPF file WELLS.IPF, will be read from the fourth and third column, respectively.

8.6 MODEL-FUNCTIONS

8.6.1 IMPORTMODFLOW-Function

Use this function to import an existing MODFLOW configuration into iMOD files (e.g. IDFs, IPFs and GENs), see for more information section 9.7.

FUNCTION=	IMPORTMODFLOW
MVERSION=	Enter the version number of the MODFLOW configuration files, e.g. MVERSION=1988. There are four available versions supported: 1988, 1996, 2000 and 2005.
BASFILE=	Enter the location of the, so called, BAS file (use this keyword whenever MVERSION=1988), e.g. BASFILE=D:\MODEL\MODEL.BAS.
NAMFILE=	Enter the location of the, so called, NAM file (use this keyword whenever MVERSION=1988, 1996, 2000 or 2005), e.g. NAMFILE=D:\MODEL\MODEL.NAM.
OUTDIR=	Enter the folder in which all iMOD files will be saved, e.g. OUTDIR =D:\IMPORT. Subfolders will be created automatically to save the individual files, e.g. D:\IMPORT\BND\VERSION_1\BND_L1.IDF . By default OUTDIR ='.' which means that the files will be saved directly at the current location of the iMOD executable.
LLCORNER=	Enter the coordinates of the lower-left corner of your model, e.g. XMIN=200000.0,YMIN=400000.0 (all in meters). By default XMIN=YMIN=0.0.
SDATE=	Enter the starting date of your simulation, e.g. SDATE=20111027 which means 27^{th} of October 2011. By default SDATE=20110101.
PACKAGESUM=	Enter PACKAGESUM=1 to sum all existing package information into a single modelcell, this is the default. Whenever more elements occur in a single modelcell, they will be lumped together to form one value. Enter PACKAGESUM=0 to extract all elements in a single modelcell to store them, if necessary, in individual iMOD files.
RIV5TH=	Enter RIV5TH=1 to include a 5^{th} column in the river files that expresses the infiltration resistance. On default RIV5TH=0.

Example 1

FUNCTION=IMPORTMODFLOW
MVERSION=1988
BASFILE=D:\IMOD-MODEL\VELUWE\MS1L5.BAS

This is the shortest version to import the MODFLOW model MS1L5.

Example 2

FUNCTION=IMPORTMODFLOW
MVERSION=2005
NAMFILE=D:\MODEL\GWR54\MODFLOW.NAM
LLCORNER=125000.0,432000.0
SDATE=20050101
PACKAGESUM=0
RIV5TH=1

This example shows how to import a (transient) MODFLOW2005 configuration.

8.6.2 IMPORTSOBEK-Function

Use this function to import a SOBEK configuration into ISG-files for iMOD for more information.

Note: This function is only available in the X32-bits version of iMOD.

FUNCTION=	IMPORTSOBEK
ISGNAME=	Enter the name of the ISG-file to be created, e.g. ISG-NAME=D:\IMPORT\SOBEK.ISG.
SOBEKDIR=	Enter the name (location) of the SOBEK files, e.g. SOBEKDIR=D:\DATA. iMOD will search for all other files that it need in the folder D:\DATA, these files are: ◇ {SOBEKDIR}\\NETWORK.TP ◇ {SOBEKDIR}\\NETWORK.CR ◇ {SOBEKDIR}\\NETWORK.CP ◇ {SOBEKDIR}\\NETWORK.GR ◇ {SOBEKDIR}\\NETWORK.ST ◇ {SOBEKDIR}\\PROFILE.DAT ◇ {SOBEKDIR}\\PROFILE.DEF ◇ {SOBEKDIR}\\PROFILE.DEF ◇ {SOBEKDIR}\\FRICTION.DAT
CALCHIS=	Enter the name of the HIS file that contains the computed waterlevels at the calculation points, e.g. CALCHIS=D:\SOBEK\CALC.HIS.
STRUCHIS=	Enter the name of the HIS file that contains the computed waterlevels at the structures, e.g. STRUCHIS =D:\SOBEK\STRUCT.HIS.

Example 1

FUNCTION=IMPORTSOBEK
ISGNAME=D:\IMPORT\HCMC0611.ISG
SOBEKDIR=D:\SOBEK\HCMC0611
CALCHIS=D:\SOBEK\HCMC0611\CALCPNT.HIS
STRUCHIS=D:\SOBEK\HCMC0611\STRUC.HIS

The above mentioned examples imports the SOBEK model (files) in the folder D:\SOBEK\HCM0611* and combines this with the computed results from the two entered HIS files (CALCPNT.HIS and STRUC.HIS) and saves it in HCMC0611.ISG.

8.6.3 MODELCOPY-Function

The function MODELCOPY can be used to extract a separate data set for a sub model from a large model. It can also be applied to copy the entire dataset as specified by the entered runfile into a separate folder. In this process, all IDF and IPF files that can be identified in a given runfile, will be clipped to the given window. Other files that are mentioned in the runfile will be copied. As a result a complete copy of a part of the original model will be saved and can be simulated separately.

Note: Other files that might be referred to from files other than the specified runfile, will not be copied.

FUNCTION=	MODELCOPY
RUNFILE=	Enter the name of a runfile that contains a specific set of IDF-file(s), e.g. RUNFILE=D:\RUNFILES\MODEL.RUN.
TARGETDIR=	Enter the name of a folder in which the resulting files will be copied, e.g. TARGETDIR=D:\SUBMODEL.
WINDOW= (optional)	Specify a window (X1,Y1,X2,Y2) for which the entered RUNFILE will be clipped, WINDOW=125100.0,345000.0,135000.0,355000.0. By absence of this keyword, the extent of the model will be read from the runfile, including the corresponding cell size.
	CELL_SIZE= Specify a cellsize whenever the keyword WINDOW is specified, e.g. CELL_SIZE=250.0.
CLIPDIR= (optional)	Enter a foldername for which all filenames will be trimmed, e.g. CLIPDIR=D:\MODEL. If the original filenames are D:\MODEL\DRN\SYS1\DRN_EL_L1.IDF and D:\MODEL\DRN\SYS2\DRN_EL_L1.IDF, they will be saved in {TARGETDIR}\DRN\SYS1\DRN_EL_L1.IDF and {TARGETDIR}\DRN\SYS2\DRN_EL_L1.IDF, respectively. By omitting CLIPDIR, both files will be stored in {TARGETDIR}\DRN\DRN_EL_L1.IDF instead.

Example 1

FUNCTION=MODELCOPY
RUNFILE=D:\RUNFILES\MODEL.RUN
TARGETDIR=D:\MODEL\SUBMODEL

The above mentioned example copies all IDF and IPF files from the runfile D:\RUNFILES\MODEL.RUN and the result is saved in D:\MODEL\SUBMODEL. A new runfile is created that will be saved in D:\MODEL\SUBMODEL\MODEL.RUN. Use this configuration to create a cleaned up folder structure of the model.

Example 2

FUNCTION=MODELCOPY
RUNFILE=D:\RUNFILES\MODEL.RUN
TARGETDIR=D:\MODEL\SUBMODEL
WINDOW=147000.0 448000.0 155000.0 452000.0
CLIPDIR=D:\MODEL

The above mentioned example is equal to example 1 except that it clips all IDF and IPF files from the runfile D:\RUNFILES\MODEL.RUN to the window 147000.0 448000.0 155000.0 452000.0 and the files remain their original filename under D:\MODEL. In this way complex structured in filename will be preserved.

8.6.4 CREATESUBMODEL-Function

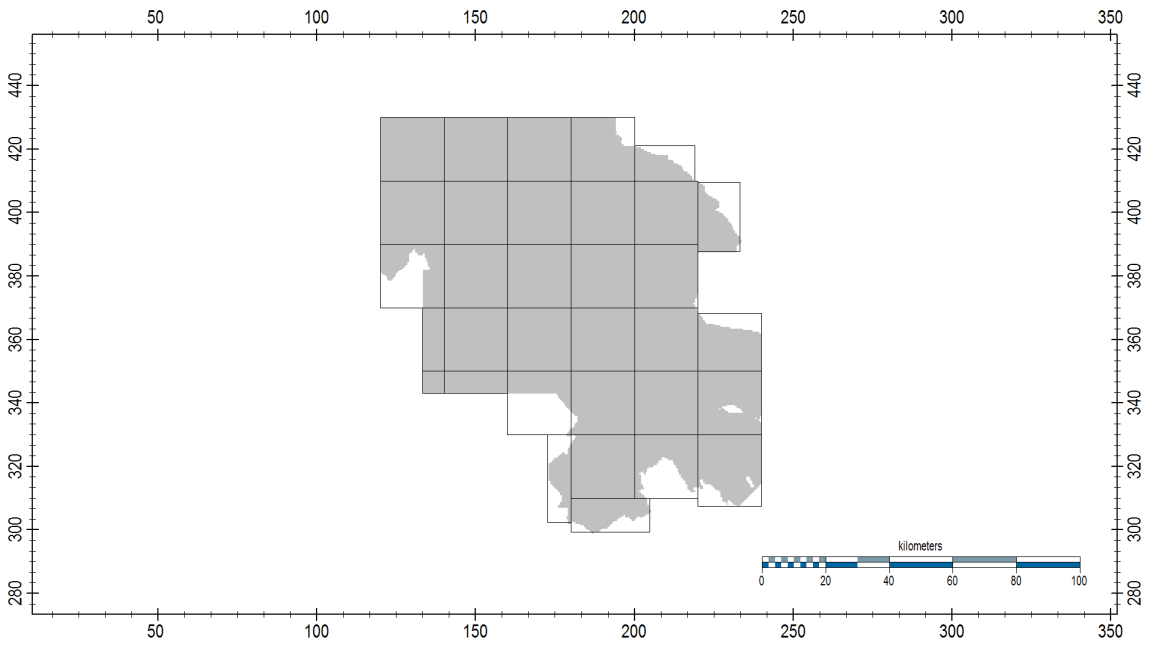
Use this function to create submodels for iMODFLOW based on a pointer IDF that determines the active area to be simulated.

FUNCTION=	CREATESUBMODEL
DSIZE=	Enter the maximum size of a submodel in meters, e.g. DSIZE=10000.0.
CSIZE=	Enter the cellsize to be used in the submodels, this will be used to fill in the appropriate column in the runfile, e.g. CSIZE=25.
IBOUND=	Enter the IDF-file that describes the location of active area to be simulated, e.g. IBOUND=D:\IBOUND_L1.IDF.
SUBMODELFILE=	Enter the name of the text file that will be created that stores the header of a runfile that describes the submodels, e.g. SUBMODELFILE=D:\SUBMODELS. iMOD will create a SUBMODELFILE.RUN to be used in a runfile and a SUBMODELFILE.GEN of the submodels to be displayed in iMOD.

Example 1

FUNCTION=CREATESUBMODEL
DSIZE=10000.0
CSIZE=25.0
IBOUND=D:\DBASE\IBOUND_L1.IDF
SUBMODELFILE=D:\RUNFILE\SUBMODEL

The example above will create submodels with a maximum extent of 10000m (10km) and will write a runfile header with cellsizes of 25m. The IDF-file IBOUND.IDF will be used to determine the active areas of the model. iMOD creates boxes with 10x10km first and then decreases submodels whenever this is possible, moreover, whenever submodels become too small (25% of 10km), they will be joined together.



8.6.5 RUNFILE-Function

The function RUNFILE can be used to create a runfile (*.RUN) from a projectfile (*.PRJ), or create a projectfile from a runfile.

FUNCTION=	**RUNFILE**	
RUNFILE_IN= (optional)	Enter the name of a runfile that contains a specific set of IDF-file(s), e.g. RUNFILE=D:\RUNFILES\MODEL.RUN.	
PRJFILE_OUT= (optional)	Enter the name of a projectfile that need to be created based on the content of the runfile specified by RUNFILE_IN, e.g. PRJFILE_OUT=D:\PRJFILES\MODEL.PRJ."	
PRJFILE_IN= (optional)	Enter the name of a projectfile that need to be used to create a runfile specified by RUNFILE_OUT, e.g. PRJFILE_IN=D:\PRJFILES\MODEL.PRJ.	
RUNFILE_OUT= (optional)	Enter the name of a runfile that will be created, e.g. RUNFILE_OUT=D:\RUNFILES\MODEL.RUN.	
NAMFILE_OUT= (optional)	Enter the name of a namfile that will be created, e.g. NAMFILE_OUT=D:\NAMFILES\MODEL.NAM.	
OUTPUT_FOLDER= (optional)	Enter the name of an output folder that will be created to save the results of the model simulation, e.g. OUTPUT_FOLDER=D:\MODEL\OUTPUT. In the situation that RUNFILE_OUT is entered, this keyword will add the output folder at the first line of the runfile, in the situation that NAMFILE_OUT is entered, the OUTPUT_FOLDER will be written in the *.MET file.	
	IDEBUG= (optional)	Enter IDEBUG=0 to generate a MF2005 compatible model, all arrays are listed in *.ARR files if needed. Use IDEBUG=1 or IDEBUG=2 to export all model input to ASC or IDF, respectively.
ISS= (optional)	Specify the type of time configuration to be added to the RUNFILE or NAMFILE; for transient enter ISS=1 and for steady state enter ISS=0. By default ISS=0.	
	Specify following keywords for transient configurations (ISS=1)	
	TIMFNAME= (optional)	Specify the name of the *.TIM file that contains the time discretisation of the model to be created, TIMFNAME=D:\RUNFILES\MODEL.TIM. For more information on the content of a *.TIM file see section 9.4.
	SDATE= (optional)	Specify a starting date of the simulation in yyyymmddhhmmss format, e.g. SDATE=20120101080000 to denote the 1^{st} of January 2012 at 08:00:00 am. This keyword is only compulsory whenever TIMFNAME is absent.
	EDATE= (optional)	Specify a starting date of the simulation in yyyymmddhhmmss format, e.g. EDATE=20121231153030 to denote the 31^{st} of December 2012 at 15:30:30. This keyword is only compulsory whenever TIMFNAME is absent.

ITT= (optional)	Specify a time interval category, e.g. ITT=2 to denote days. The other are: ◇ **1 Hourly** Select this option to generate hourly stress-periods; ◇ **2 Daily** Select this option to generate daily stress-periods; ◇ **3 Weekly** Select this option to generate weekly stress-periods; ◇ **4 Decade** Select this option to generate stress-periods per decade; ◇ **5 14/28** Select this option to generate stress-periods on the 14th and 28th day of each month; ◇ **6 Monthly** Select this option to generate monthly stress-periods; ◇ **7 Yearly** Select this option to generate yearly stress-periods; ◇ **8 Packages** Select this option to generate stress-periods that are determined by the input data This keyword is only compulsory whenever TIMFNAME is absent.
IDT= (optional)	Specify a time interval of the time steps corresponding to the chosen time interval category ITT, e.g. IDT=7 to denote the 7 days whenever ITT=2. This keyword is only compulsory whenever TIMFNAME is absent.
ISTEADY= (optional)	Specify ISTEADY=1 to include an initial steady-state time step to the model. This will add packages with the time stamp STEADY-STATE to the first stress-period of your model. By default ISTEADY=0.
NSTEP= (optional)	Specify the number time step within each stress period, e.g. NSTEP=10. Whenever a model suffers some convergence issues, increase the number of time steps might help. Also, steady-state convergence problems can be overcome by creating a transient model with enough time step. By default NSTEP=1.
NMULT= (optional)	Specify the multiplication factor in which the step size of each subsequent time step will increase, e.g. NMULT=1.2. The factor need to $\geq$ 1.0. The higher the number to more explosive the size will increase in subsequent time steps according to: $$\Delta t_i = \Delta T_j \left(\frac{\mathrm{NMULT} - 1}{\mathrm{NMULT}^{\mathrm{NSTP}} - 1} \right) \quad (8.1)$$ wherein Δ_i is the current length of the time step i, ΔT_j is the total length of the current stress period j. By default NMULT=1.0.
SSYSTEM= (optional)	Specify SSYSTEM=1 to save each system for a package by a separate file. By default SSYSTEM=0.
ISAVEEND DATE= (optional)	Specify ISAVEENDDATE=1 to save each file with a time stamp equal to the end of the corresponding stress period (and/or time step). By default ISAVEENDDATE=0 and the time stamp will be equal to the start date of each stress period (and/or time step). This option is applicable whenever NAMFILE_OUT is specified, whenever RUNFILE_OUT is specified, this keyword doesn't have any effect.

IPKS= (optional)	Specify IPKS=1 to apply the PKS package instead of the PCG package. The PKS seems to be more robust than the PCG solver, so you might want to use this PKLS solver in case of non-convergency due to huge contrasts in conductivities and/or using the multi-core applications. This option is applicable whenever NAMFILE_OUT is specified, whenever RUNFILE_OUT is specified, this keyword doesn't have any effect.
ICHKCHD= (optional)	Specify ICHKCHD=1 to convert constant head cells (from the read starting heads) that are not belonging to the layer to which they are assigned. If a value exceeds the top of a model layer to which it is assigned, the boundary value is turned into 99 and is converted to an active node. This option is applicable whenever NAMFILE_OUT is specified, whenever RUNFILE_OUT is specified, this keyword doesn't have any effect.
ICONCHK= (optional)	Use this keyword to correct the drainage levels automatically during a simulation. Whenever ICONCHK=1 the drainage level will be higher or equal to existing level from the RIV and/or ISG package. This prevents undesired circulation of groundwater between drainage system underlying an infiltrating river system. By default ICONCHK=0 and no corrections are performed.
IDOUBLE= (optional)	Use this keyword to save all results from the simulation in double precision. Whenever IDOUBLE=0 (default) a single precision value is saves, whenever IDOUBLE=1 a double precision is saved. Be ware that all files will be affected by this and therefore the total space that will be occupied by the results will be doubled.
IFVDL= (optional)	Use this keyword to use the Formulae of "Van de Lange" for the correction of river conductances. This keyword can be used whenever the PRJ file contains TOP and BOT definitions as well as KHV. The SFT package might be used to include permeabilities and thickness used by the formulae, otherwise, if this SFT package is absent, de values from the model will be used. By default IFVDL=0 and no corrections are performed.
MINKD= (optional)	Use this keyword to assign a minimal horizontal conductance KD (m^2/d) to maximize the computed conductances, internally, e.g. MINKD=0.01, by default MINKD=0.0.
MINC= (optional)	Use this keyword to assign a minimal vertical resistance C (d) to maximize the computed vertical resistances, internally, e.g. MINC=1.0, by default MINC=0.0.
UNCONFINED= (optional)	Use this keyword to include unconfined conditions for model layers, e.g. UNCONFINED=1,1,1,0,0,0 by default UNCONFINED=0 and model layers are confined. So, if NLAY=10 and UNCONFINED=1,1,1 this means that the first three model layers are unconfined, the remaining layers are confined. The values for UNCONFINED are: ◇ **0** The model layer is confined, the saturated thickness is defined by the top- and bottom elevation per model layer; ◇ **1** The model layer is unconfined, the saturated thickness is defined by the compute hydraulic head and the bottom elevation per model layer; ◇ **2** The model layer is confined by the saturated thickness is defined by the starting heads and the bottom elevation per model layer. It is obligatory to include the modules STO and SPY whenever unconfined conditioned are simulated for a transient model. iMOD will add and configure the WETDRY option automatically.
IPEST= (optional)	Use this keyword to include the parameter optimisation package PST, e.g. IPEST=1, by default IPEST=0 and the model is simulated conventionally.

SAVESHD= SAVEWEL= SAVEDRN= SAVERIV= SAVEGHB= SAVERCH= SAVEEVT= SAVEMNW= SAVELAK= SAVESFR= SAVEUZF= SAVEFHB= (all optional)	Use these keywords to save the hydraulic head per layer or/and results for the WEL, DRN, RIV, GHB, RCH, EVT, LAK, MNW, SFR, FHB and UZF package, e.g. SAVESHD=3,4,10 to note that model layers 3, 4 and 10 will be saved only, by default all keyword are 0, meaning no layers will be saved. Specify SAVESHD=-1 to denote that ALL layers will be saved, this is similar for the other packages, except for the SFR and UZF package, specifying more layers does not have any effect.	
SAVEFLX= (optional)	Use this keyword to include layers to be saved for the spatial fluxes in x,y and z direction, e.g. SAVEFLX=3,4,10 to note that model layers 3, 4 and 10 will be saved only, by default SAVEFLX=0, meaning no layers will be saved. Specify SAVEFLX=-1 to denote that ALL layers will be saved. Part of this, the BDGFFF, BDGFRF, BDGFLF and BDGBND will be saved.	
NETWORKIDF= (optional)	Specify an IDF file that represents the network for the simulation, e.g. NETWORKIDF=D:\MODEL \NETWORK.IDF. This keyword is optional, whenever this keyword is absent the network is defined by the first IDF file in the entered PRJ file or specified by the keyword WINDOW.	
WINDOW= (optional)	Specify a window (X1,Y1,X2,Y2) for which the constructed RUNFILE will be clipped, e.g. WINDOW=125100.0,345000.0,135000.0,355000.0.	
	CELLSIZE=	Specify a cell size to be used, e.g. CELLSIZE=25.0. This keyword is necessary and read whenever the WINDOW keyword is entered.
	BUFFER= (optional)	Specify a buffer to be added to the specified window (WINDOW), e.g. BUFFER=1500.0. This keyword is optional and the default value is BUFFER=0.0.
	BUFFERCS= (optional)	Specify a maximal cell size in the buffer, e.g. BUFFERCS=100.0. This keyword is optional and read whenever BUFFER is specified and greater than 0.0 meter.
ISOLVE= (optional)	Enter ISOLVE=1 to start a simulation after generating a RUNFILE or NAMFILE, by default ISOLVE=0.	
	MODFLOW=	Whenever ISOLVE=1, enter the simulator, e.g. MODFLOW=D:\PROGRAMS\IMODFLOW.EXE.

Example 1

FUNCTION=RUNFILE
RUNFILE_IN=D:\RUNFILES\MODEL.RUN
PRJFILE_OUT=D:\PRJFILES\MODEL.PRJ

The above mentioned example creates a projectfile D:\PRJFILES\MODEL.PRJ file out of the runfile D:\RUNFILES\MODEL.RUN.

Example 2

FUNCTION=RUNFILE
PRJFILE_IN=D:\PRJFILES\MODEL.PRJ
RUNFILE_OUT=D:\RUNFILES\MODEL.RUN
WINDOW=147000.0 448000.0 155000.0 452000.0
CELLSIZE=25.0
BUFFER=1500.0
SDATE=19940101120000
EDATE=20121231235959
ITT=3

IDT=2

The above mentioned example creates runfile D:\RUNFILES\MODEL.RUN, based on the content of the projectfile D:\PRJFILES\MODEL.PRJ for a specified window. The model starts at the 1^{st} of January 1994 at 12:00:00 am and ends at the 31^{st} of December 2012 at 23:59:59 am at uses two-weekly time steps.

Example 3

FUNCTION=RUNFILE
PRJFILE_IN=D:\PRJFILES\MODEL.PRJ
NAMFILE_OUT=D:\NAMFILES\MODEL.NAM
TIMFILE_OUT=D:\TIMFILES\MODEL.TIM
ISOLVE=1
MODFLOW=D:\PROGRAM\IMODFLOW.EXE

The above mentioned example creates a Modflow2005 configuration NAMFILE D:\NAMFILES\MODEL.NAM, based on the content of the projectfile D:\PRJFILES\MODEL.PRJ for a times discretisation specified in the *.TIM file D:\TIMFILES\MODEL.TIM. After that, it starts the simulation using the simulator D:\PROGRAM\IMODFLOW.EXE.

8.6.6 IMODPATH-Function

The function IMODPATH computes flowlines based on the budget terms that result from the iMODFLOW computation. The IMODPATH function uses a very simple runfile. For more information see section 7.14.

FUNCTION=	**IMODPATH**	
IRUN= (optional)	Specify IRUN=1 to start a particle simulation, apply IRUN=0 to skip the particle simulation and perform a post processing solely, if IPOSTP=1. By default IRUN=1.	
RUNFILE= (optional)	Enter the name of the runfile that describes the files needed for the iMODPATH simulation, e.g. RUNFILE=D:\MODEL\SIM.RUN. This keyword is compulsory whenever IRUN=1. The content of such a runfile is as follows:	
	NLAY	Enter the number of model layers, e.g. NLAY=8
	NPER,ISTO	Enter the number of stress periods, .e.g. NPER=1 and whether fluxes from storage need to be read in, in case a transient simulation is carried out. The storage flux will correct the total water balance and influence whether an unbalance will denote a weak/strong sink. Whenever the absolute unbalance is larger than 0.01 m^3/d, iMODPATH will treat the location as a potential weak/strong sink. The parameter ISTO is optional and whenever it is absent the flow from storage is as assumed to be zero.
	NSDF (optional)	Enter the number of SDF files to be computed, sequentially. Enter a value for this only whenever NSDF$\geq$2. Leave this keyword out, whenever a single ISDFILE and OUTFILE is entered.
	ISDFILE / IPFFILE	Enter the name of the startpoint file, see section 7.13 for more information about this type of file (see section 9.19 for the actual syntax). Repeat this, together with OUTFILE for NSDF-times whenever NSDF is specified and NSDF$\geq$2. Alternatively an IPF file can be specified. It is compulsory to specify at least three columns whereby the first three columns are reseverd for the x-, y- and z-coordinate.
	OUTFILE	Enter the name of the result file. The extent will be added or replaced to the appropriate output format (IFF and/or IPF), e.g. D:\d:\RESULT. Repeat this, together with ISDFILE for NSDF-times whenever NSDF is specified and NSDF$\geq$2.
	IMODE	Enter the mode of the results to be achieved, use IMODE=1,0 for flowlines and IMODE=0,1 for endpoints only. For example IMODE=1,1 will save both particles in the IFF and IPF format.
	IFWBW	Enter the direction of the tracing, use IFWBW=0 for a forward tracing and IFWBW=1 for a backward tracing.
	ISNK	Specification on how to handle “weak”-sinks. Particles will continue at weak sinks for ISNK=1 as they stop at weak sinks for ISNK=2. The latter can be specified as a fraction for ISNK=3, see keyword FRACTION.
	FRACTION	Specify the fraction of the total outflow to be a measure to determine whether particles should stop or continue in a model with a “weak”-sink. FRACTION=1.0 to let particles stop at a strong sink only, as FRACTION=0.0 act as particles will always stop, no matter the size of the total outflow (ISNK=1).
	STOPCRIT	Enter the stop criteria. Specify STOPCRIT=1 to stop the particle as its age becomes equal to MAXT; specify STOPCRIT=2 to repeat the transient period in the time window as specified by the keywords SWINDOW and EWINDOW until the particles meets the MAXT criterion or stops in a weak/strong sink. Or, alternatively set STOPCRIT=3 to continue with the last results at the end of the time window until the particle terminates. This keyword STOPCRIT is only applicable whenever NPER$>$ 1.
	MAXT	Enter the maximum tracing time (days).

STARTDATE	Enter the startdate for the particle tracing, e.g. 19960414 to express the 14^{th} of April 1994. This keyword is used only whenever NPER $>$ 1, but you need to specify an artificial STARTDATE for NPER=1.
SWINDOW	Enter the start date for the time window in which the particle tracing will operate, e.g. 19960414 to express the 1^{st} of April 1994. Only used whenever NPER $>$ 1, but you need to specify an artificial SWINDOW for NPER=1.
EWINDOW	Enter the end date for the time window in which the particle tracing will operate, e.g. 20040328 to express the 28^{th} of March 2004. Only necessary whenever NPER $>$ 1, but you need to specify an artificial EWINDOW for NPER=1.

Repeat the following NLAY-times

IBOUND	Enter the boundary condition (IDF). Particle tracing will pass through boundary values $>$ 0 only.
TOP	Enter the top elevation (IDF or constant value) of a modellayer (m+MSL).
BOT	Enter the bottom elevation (IDF or constant value) of a modellayer (m+MSL).
PORAQF	Enter the porosity (IDF or constant value) of the aquifer (-).
PORAQT	Enter the porosity (IDF or constant value) of the aquitard (-). Specify this keyword for model layers < NLAY.

Repeat the following NLAY times and NPER times

BDGFRF	Enter the IDF file that represents the water budget (m^3/day) along the x axes (columns) at the eastern border of each cell; positive flow is westwards, negative is eastwards.
BDGFFF	Enter the IDF file that represents the water budget (m^3/day) along the y axes (rows) at the southern border of each cell; positive flow is northwards, negative flow is southwards.
BDGFLF	Enter the IDF file that represents the water budget (m^3/day) along the z axes (layers) at the lower border of each cell; positive flow is upwards, negative flow is downwards. Specify this keyword for model layers 1 up to NLAY-1.
BDGSTO (optional)	Enter the IDF file that represents the water budget (m^3/day) that goes into storage; positive flow is going out of the storage, negative flow is going into the storage. This parameter is only needed whenever ISTO=1.

WINDOW= (optional)	Specify a window (X1,Y1,X2,Y2) for which the constructed model will be simulated, e.g. WINDOW=125100.0,345000.0,135000.0,355000.0. This might increase the efficiency as only a part of the model need to be allocated and read in.
ICONVERTGEN= (optional)	Specify ICONVERTGEN=1 to convert the results of the pathline simulation as well into a GEN (see 9.10) and DAT file (see 9.11). Bij default ICONVERTGEN=0 and no conversion will occur.
IPOSTP= (optional)	Use IPOSTP=1 to include a post processing direct after the pathline simulation. This post processing will separate the IFF and/or IPF files to a given selection criterion. By default IPOSTP=0.

The following keyword are applicable only whenever IPOSTP=1

IFFFLOW= (optional)	Enter the name of the IFF file (generated by the pathline simulation) to be processed, e.g. IPFFLOW=D:\RESULT\PATHLINES.IFF. By default no IFF file will be processed.
IPFFLOW= (optional)	Enter the name of the IPF file (generated by the pathline simulation) to be processes, e.g. IPFFLOW=D:\RESULT\PATHLINES.IPF. By default no IPF file will be processed.

IPFFNAME= (optional)	Enter the IPF file to be used to separate the content of the IFFFLOW and or IPFFLOW file names to the labels (ILABELCOL) as specified in the IPFFNAME, e.g. D:\INPUT\WELLS.IPF	
	IDFFLOW=	Specify an IDF file that will be used to map the points from the IPF file given at IPFFNAME on a network in order to match them with the results in the IPF and/or IFF file given at IPFFLOW and IFFFLOW, respectively, most common is to use a random IDF from the result map of the simulation results, used for the particle simulation, e.g. IDFFLOW=D:\RESULT\HEAD_STEADY-STATE_L1.IDF.
	IXCOL=	Enter the column number in the given IPF file IPFFNAME that represents the X-coordinate, e.g. IXCOL=1.
	IYCOL=	Enter the column number in the given IPF file IPFFNAME that represents the Y-coordinate, e.g. IYCOL=2.
	ILABELCOL=	Enter the column number in the given IPF file IPFFNAME that represents the label, e.g. ILABELCOL=3. For each unique label, a separate IPF and/or IFF will be constructed.
	ILAYCOL=	Enter the column number in the given IPF file IPFFNAME that represents the model layer, e.g. ILAYCOL=4.
TOPFNAME= (optional)	Enter the IDF file to be used to identify the top of an interface that need to be used to separate IPF and/or IFF file that are underneath the interface, e.g. D:\INPUT\TOP_L1.IDF, by default no top is given and in that case all is valid. Any *NodataValue* in the IDF file will be used to exclude any particle.	
BOTFNAME= (optional)	Enter the IDF file to be used to identify the bot of an interface that need to be used to separate IPF and/or IFF file that are above the interface, e.g. D:\INPUT\BOT_L7.IDF, by default no bottom is given and in that case all is valid. Any *NodataValue* in the IDF file will be used to exclude any particle.	
IEXTRACT= (optional)	Enter the extract option for the particles, choose from the following: ◇ IEXTRACT=1 Choose this option to extract the entire particle that satisfies the top- and/or bottom criterion, if only a single segment of the particle meets the criterion the entire particle is exported; ◇ IEXTRACT=2 Choose this option to extract the particle until it satisfies the top- and/or bottom criterion, if only a single segment meets the criterion, the particle up to that location is extracted, the particle is not examined after that anymore, so a multiply agreement to the selection criterion does not hold; ◇ IEXTRACT=3 Choose this option to extract the entire particle onwards after it satisfies the top- and/or bottom criterion lastly at first agreement, so whenever a particle hits the selection criterion the extraction will start after the last segment that met this selection criterion; ◇ IEXTRACT=4 Choose this option to extract the entire particle that ends within any of the locations that satisfies the top- and/or bottom criterion. By default IEXTRACT=4. This option is also only valid for IPF files that are entered at IPFFLOW.	

Example 1

FUNCTION=IMODPATH
RUNFILE=D:\IMOD\IMODPATH.RUN

and the content of the IMODPATH.RUN file: *2, !## NLAY*
1, !## NPER
2
D:\STARTPOINTS\LAYER2.ISD
D:\MODEL\CAPTURE_LAYER2.IFF
D:\STARTPOINTS\LAYER3.ISD
D:\MODEL\CAPTURE_LAYER3.IFF
1,1 !## IMODE
0, !## IFWBW
2, !## ISNK
0.50, !## FRACTION
1, !## ISTOP
0.1000E+31, !## MAXT
19960414, !## SDATE
19960401, !## SWINDOW
20040328, !## EWINDOW
D:\IMOD-MODEL\IBOUND1.IDF, !## IBOUND
D:\IMOD-MODEL\TOP1.IDF, !## TOP
D:\IMOD-MODEL\BOT1.IDF, !## BOT
0.3, !## PORAQF
0.1, !## PORAQT
D:\IMOD-MODEL\IBOUND2.IDF, !##
D:\IMOD-MODEL\TOP2.IDF, !##
D:\IMOD-MODEL\BOT2.IDF, !##
0.3, !## PORAQF
D:\MODEL\BDGFRF\BDGFRF_STEADY-STATE_L1.IDF, !## BDGFRF
D:\MODEL\BDGFFF\BDGFFF_STEADY-STATE_L1.IDF, !## BDGFFF
D:\MODEL\BDGFLF\BDGFLF_STEADY-STATE_L1.IDF, !## BDGFLF
D:\MODEL\BDGFRF\BDGFRF_STEADY-STATE_L2.IDF, !## BDGFRF
D:\MODEL\BDGFFF\BDGFFF_STEADY-STATE_L2.IDF, !## BDGFFF

The above mentioned example will do a particle simulation.

Example 2

FUNCTION=IMODPATH
IRUN=0
IPOSTP=1
IFFFLOW=D:\RESULT\PATHLINES.IFF
IPFFLOW=D:\RESULT\PATHLINES.IPF
IDFFLOW=D:\RESULT\HEAD_STEADY-STATE_L1.IDF
IPFFNAME=D:\INPUT\WELLS.IPF
IXCOL=1
IYCOL=2
ILAYCOL=3
ILABELCOL=4

The above mentioned example will NOT do a particle simulation (IRUN=0), but performs a post processing solely.

8.7 GEO-FUNCTIONS

8.7.1 DINO2IPF-Function

This function will extract from a CSV file exported from DINO (TNO) appropriate data to generate an IPF file with borehole information attached to it. The content of the CSV file is prescribed on the next page.

FUNCTION=	DINO2IPF
CSVFILE=	Enter a CSV file that contains the necessary information from the DINO database, e.g. CSVFILE=D:\DATA\DINO.CSV. The output file (IPF file) will be named after the CSVFILE. Moreover, you can specify a wildcard to transform more CSV files into a single IPF file, e.g. CSVFILE=D:\DATA*.CSV. In this case you need to specify an IPF filename with the keyword IPFFILE=.
WINDOW=	Specify a window (X1,Y1,X2,Y2) for which the entered RUNFILE will be clipped, WINDOW=125100.0,345000.0,135000.0,355000.0.
GENFILE=	Enter a name for a GEN-file that contains a polygon that determines the area for which the CSV files need to be converted into the IPF files.
IPFFILE=	Enter the name of the IPF file to be created, e.g. IPFFILE=D:\DINO.IPF. This keyword is obliged only whenever the CSVFILE contains a wildcard "*".

Example 1:

FUNCTION=DINO2IPF
CSVFILE=D:\iMOD-DATA\DINO.csv*
IPFFILE=D:\iMOD-DATA\DINO\AREA.IPF
WINDOW=130000.0,450000.0,141000.0,461000.0

This example imports all CSVFILES (*.csv) into the IPF file AREA.IPF for a particular window.

Example 2:

FUNCTION=DINO2IPF
CSVFILE=D:\iMOD-DATA\DINO\BOX12.CSV
GENFILE=D:\IMOD-DATA\AREA.GEN
WINDOW=130000.0,450000.0,141000.0,461000.0

The example above imports the boreholes from the BOX12.CSV for the area within the specified polygon(s) in AREA.GEN.

8.7.2 GEOTOP-Function

This function will replace the top of a groundwatermodel with a GEOTOP schematization.

FUNCTION=	**GEOTOP**
RESULTFOLDER=	Enter the name of the folder that will store the merged model
NLAYG=	Enter the number of GEOTOP model layers.
KVG_L{i}=	Enter the IDF for the i^{th} modellayer of GEOTOP that represents the KV *i*, e.g. TOP_L1=D:\GEOTOP\KVG_L1.IDF.
KHG_L{i}=	Enter the IDF for the i^{th} modellayer of GEOTOP that represents the KH *i*, e.g. TOP_L1=D:\GEOTOP\KHG_L1.IDF.
NLAYM=	Enter the number of model layers in the groundwatermodel
Enter the IDF for the j^{th} model layer of the model that represents: ...	
IBM_L{j}=	... the iBOUND for layer *j*, e.g. IBM_L1=D:\UTRECHT\iBOUND_L1.IDF
SHM_L{j}=	... the starting head for layer *j*, e.g. SHM_L1=D:\UTRECHT\SHEAD_L1.ID
TPM_L{j}=	... the top of layer *j*, e.g. TPM_L1=D:\UTRECHT\TOP_L1.IDF
BTM_L{j}=	... the bot of layer *j*, e.g. BTM_L1=D:\UTRECHT\BOT_L1.IDF
KHM_L{j}=	... the KH for layer *j*, e.g. KHM_L1=D:\UTRECHT\KH_L1.IDF
KAM_L{j}=	... the KA for layer *j*, e.g. KAM_L1=D:\UTRECHT\KA_L1.IDF
KVM_L{j}=	... the KV for layer *j*, e.g. KVM_L1=D:\UTRECHT\KV_L1.IDF
WINDOW=	Specify a window (X1,Y1,X2,Y2) for which the entered RUNFILE will be clipped, e.g. WINDOW=125100,345000,135000,355000
	CELLSIZE= Enter the cell size (meter) for the IDF-files that will be created, e.g. CELLSIZE=25

Example 1:

FUNCTION=GEOTOP
RESULTFOLDER=D:\MODEL
NLAYG=95
WINDOW=130000,450000,141000,461000
kvg_l1 =D:\MODEL\kv1.idf
khg_l1 =D:\MODEL\kh1.idf
.....
kvg_l95=D:\MODEL\kv95.idf
khg_l95=D:\MODEL\kh95.idf
NLAYM=9
tpm_l1=d:\modelutrecht\GEOHYDROLOGY\VERSION1\TOP_l1.IDF
btm_l1=d:\modelutrecht\GEOHYDROLOGY\VERSION1\BOTTOM_l1.IDF
kdm_l1=d:\modelutrecht\TRANSMISSIVITY\VERSION1\TRANSMISSIVITY_l1.IDF
vcm_l1=d:\modelutrecht\VERTICALRESISTANCE\VERSION1\VERTICALRESISTANCE_l1.IDF
ibm_l1=d:\modelutrecht\IBOUND\VERSION1\IBOUND_l1.IDF
shm_l1=d:\modelutrecht\STARTINGHEADS\VERSION1\SHEAD_l1.IDF
....
tpm_l9=d:\modelutrecht\GEOHYDROLOGY\VERSION1\TOP_l9.IDF
btm_l9=d:\modelutrecht\GEOHYDROLOGY\VERSION1\BOTTOM_l9.IDF
kdm_l9=d:\modelutrecht\TRANSMISSIVITY\VERSION1\TRANSMISSIVITY_l9.IDF
vcm_l9=d:\modelutrecht\VERTICALRESISTANCE\VERSION1\VERTICALRESISTANCE_l9.IDF
ibm_l9=d:\modelutrecht\IBOUND\VERSION1\IBOUND_l9.IDF
shm_l9=d:\modelutrecht\STARTINGHEADS\VERSION1\SHEAD_l9.IDF

8.7.3 GEF2IPF-Function

This function will combine information from a series of GEF files to generate an IPF file with borehole or cone penetration test (CPT) information attached to it. The content of the GEF file is prescribed in section 9.23.1.

FUNCTION=	GEF2IPF
GEFDIR=	Enter a DIR name that contains the GEF files, e.g. GEFDIR=D:\DATA\.
IPFFILE=	Enter the name of the IPF file to be created, e.g. IPFFILE=D:\DINO.IPF.
GEFTYPE=	Enter a number for the type of file you prefer to read in. 1=CPT, 2=Borehole.
WINDOW=	Specify a window (X1,Y1,X2,Y2) for which the GEF files will be selected, WINDOW=125100.0,345000.0,135000.0,355000.0.
GENFILE=	Enter a name for a GEN-file that contains a polygon that determines the area for which GEF files will be selected for conversion.

Example 1:

FUNCTION=GEF2IPF
GEFDIR=D:\iMOD-DATA\DINO
IPFFILE=D:\iMOD-DATA\DINO\AREA.IPF
GEFTYPE=1

This example imports all GEFFILES (*.GEF) into the IPF file AREA.IPF.

Example 2:

FUNCTION=GEF2IPF
*GEFDIR=D:\iMOD-DATA\DINO*DELFT*.GEF*
GENFILE=D:\IMOD-DATA\AREA.GEN
GEFTYPE=1

The example above imports the GEFFILES for the area within the specified polygon(s) in AREA.GEN.

Note: Before the calculation is started, iMOD asks you what type of GEF-file you want to convert to IPF; one that contains CPT (Cone Penetration Test) information or one that contain Borehole information. Below you can find a brief description of both GEF-file types.

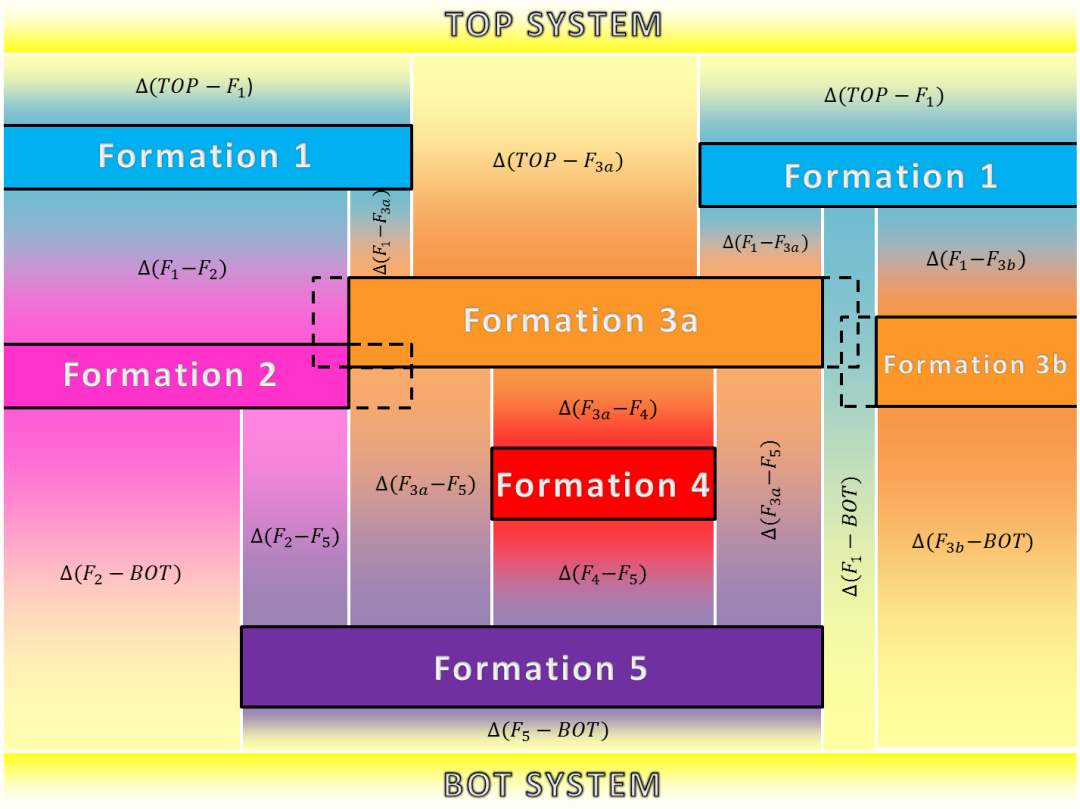
8.7.4 CUS-Function

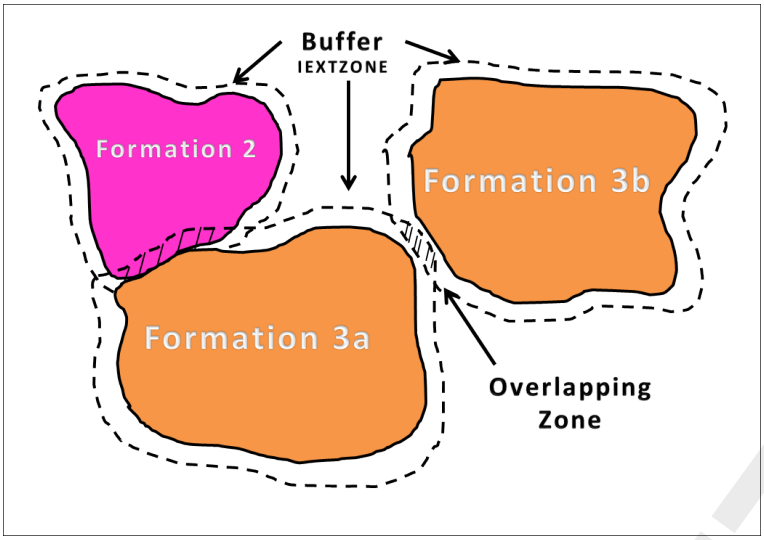
Use this function to determine a minimal number of model layers (aquifers) based on layers that describe geological formations (mainly aquitards). The concept idea is that the vertical distribution of aquitards (or other distinguishing layers), determine the minimal number of model layers to represent the aquifers.

The methodology computes the interrelation ship of all the individual parts within a geological formation and those from other geological formations. This interrelation ship is used to compute the minimal number of model layers to capture all of them without any loss of information. In fact is depends mainly on the lateral distribution of the geological formation whether the number of model layers becomes less than the number of geological formations. The more spread in the distribution, probably less model layers are necessary. In figure an overview is given of the methodology as it shows all the interrelation ships that are computed, e.g. $\Delta F_1 - F_5$ represents the distance between a individual element on the the first and fifth geological formation, and moreover, the vertical position of both elements as the first element should always be in a higher model layer than the fifth element. All these interrelation ships are fed into a linear-programming algorithm than find a model layer for each of them such that the total number of model layers is minimized.

The CUS function is fully automatic, which means that a) the vertical order of given geological formations is irrelevant, b) overlapping geological formations may be clipped and c) interrelation ships in vertical and horizontal direction are computed automatically that yield a minimal set of model layers. With some variables it is possible to set the threshold at what distance of interrelation ship geological formation may join together, in this way aquitards can be combined whenever they are separated less than a given distance.

Schematic overview working of CUS





FUNCTION=	CUS	
ICPOINTERS= ICPOINTERS=1	Enter a value whether or not use a given CUS pointer IDF (0 or 1).	
	FDISTANCES=	file containing predefined vertical distances that is created in a former CUS-action.
	or	
	CRIT_THICKNESS=	maximum vertical step size (e.g. CRIT_THICKNESS=25.0 m) to combine elements laterally.
	MIN_THICKNESS=	minimal thickness of the element to be included in the final model, e.g. MIN_THICKNESS=0.5 will include elements thicker than 0.5 meter only.
	ZCRIT=	Critical vertical distance. Layers will be connected vertically whenever a percentage of their vertical distance is less than ZCRIT, e.g. ZCRIT=0.5 m.
	PERCENTAGE=	Give the percentile for which ZCRIT needs to be taken into account, e.g. if a percentage of 90 is given, layers will be connected if 90% of the distance is less than ZCRIT.
	ICLIP=	Enter the name of an IDF file (at least at the dimension of the given IDF files are FORMTOP_Li or FORMTOP keyword) that denote the zone for which an entry is not need to be blanked out. E.g. ICLIP=D:\CUS\ZONES.IDF. Whenever a value of 1 is found in the IDF file, all geological formations that refer to this zone 1, will be blanked out for areas not equal to 1.
	IEXPZONE=	Enter a number of additional cells around each individual element in each formation (IEXPZONE>1 to have any effect), to be used to include any element laterally to determine the most optimal model layer, e.g. IEXPZONE=2. Adding a value of IEXPZONE will have the effect that elements that are horizontally nearby (less than 2 cells in this case), will be tried to vertically positioned in the same model layer.
ICPOINTERS=0		
	NLAY=	Enter the fixed number of model layers to be constructed based on the IDf file with pointer values given at PNT_L{i}.
NFORM=	Enter number of geological formations, e.g. NFORM=19.	
FORMTOP_L{i}=	Enter an optional zone number after the file name whenever the keyword ICLIP is used, e.g. FORMTOP_L1=D:\INPUT\BEK1_CK.IDF,1.	

FORMBOT_L{i}=	Enter an optional zone number after the file name whenever the keyword ICLIP is used, e.g. FORMBOT_L1=D:\INPUT\BEK1_CK.IDF,1.	
ICPOINTERS=0		
	PNT_L{i}=	Enter an IDF for the i^{th} formation that gives a pointer values that refers to a modellayer *i*, e.g. PNT_L1=D:\INPUT\PNT_L1.IDF. This file contains for example the values 1-5. These values serve as a label of a specific aquitard layer (1-5).
or		
FORMTOP=	Enter a path and wildcard to specify for a collection of IDF files containing information about the TOP of the geological formations to be included, e.g. FORMTOP_L1=D:\FORMATIONS*_TOP.IDF.	
FORMBOT=	Enter a path and wildcard to specify for a collection of IDF files containing information about the BOT of the geological formations to be included, e.g. FORMBOT_L1=D:\FORMATIONS*_BOT.IDF.	
OUTPUTFOLDER=	Enter the foldername in which the results will be saved, e.g. OUTPUTFOLDER=D:\RESULT.	
WINDOW=	Enter the coordinates of the window that needs to be computed. Enter coordinates of the lower-left corner first and then the coordinates of the upper-right corner, e.g. WINDOW=100000, 400000, 200000, 425000. When WINDOW= is absent iMOD will take the WINDOW-extent of the input IDF's.	
CELLSIZE=	Enter the cell size (meter) for the IDF-files that will be created, e.g. CELL_SIZE=25.0.	
TOPSYSTEM=	Enter the name of the IDF-file containing the AHN for the specific model area.	
BOTSYSTEM=	Enter the name of the IDF-file containing the bottom-depth of the lowest bottom layer of the model.	

Example

FUNCTION= CUS
NLAY=2
WINDOW=120000.0,298000.0,240000.0,430000.0
CELLSIZE=100.0
FORMTOP_L1=D:\MODEL\BEK1_T.IDF
FORMTOP_L2=D:\MODEL\BEK2_T.IDF
FORMBOT_L1=D:\MODEL\BEK1_B.IDF
FORMBOT_L2=D:\MODEL\BEK2_B.IDF
OUTPUTFOLDER=D:\OUTPUT
TOPSYSTEM=D:\MODEL\AHN250.IDF
BOTSYSTEM=D:\MODEL\BEDROCK_TOP.IDF

This example corrects the top and bottom IDF-files specified by the FORMTOP_L{i} and FORMBOT_L{i} keywords in a top-bottom consistent manner and scales the IDF-files to the specified WINDOW and CELLSIZE.

8.7.5 SOLID-Function

Use this function to generate hypothetical interfaces (i.e. line in between model layers that represent an artificial interface since any resistance layer (e.g. clay). This function computes the transmissivities and vertical resistance between model layers as well. It uses the PCG solver algorithm for the interpolation of the hypothetical interfaces and uses the existence of permeability field and the top- and bottom elevation to compute the nett transmissivity for each a modellayer and vertical resistance between those model layers. By means of *masks* it is possible to define those areas for which hypothethical interfaces need to be computed.

FUNCTION=	**SOLID**	
NLAY=	Enter the number of modellayers, e.g. NLAY=6.	
OUTPUT-FOLDER=	Enter the foldername in which the results IDF-files will be saved, e.g. OUTPUT-FOLDER=D:\RESULT. The following results will be saved:	
	\MASK	If IMASK=1, for each interface a mask IDF will be created and saved in this folder. Those may be adjusted afterwards, but set IMASK=0 to avoid that those modified mask files will be overwritten
if ICKDC=1		
	\FFRAC	For each model layer, the fraction of each geological formation will be saved. It represents the fraction $(0.0-1.0)$ of the geological that is present in the model layer.
	\CFRAC	For each in between model layer (aquitard), the fraction of each geological formation will be saved. It represents the fraction $(0.0-1.0)$ of the geological that is present in the aquitard.
	MDL_TOP_{i}	The TOP elevation for each model layer;
	MDL_BOT_{i}	The BOT elevation for each model layer;
	MDL_KD_{i}	The total transmissivity for each model layer, it becomes zero when the thickness of the aquifer (model layer) is zero;
	MDL_VC_{i}	The vertical resistance over aquitards in between each model layer, **excluding** the resistance due to the vertical resistance in the above- and beneath lying aquifers. Its value becomes zero if the aquitard is absent;
	MDL_KHV_{i}	The horizontal permeability for each model layer, it can be zero for layer thicknesses of zero;
	MDL_KVA_{i}	The vertical anisotropy for each model layer, it will always have a value larger than 0 and smaller equal to 1. For non existing model layers (aquifers) this parameter will be one;
	MDL_KVV_{i}	The vertical permeability for each aquitard in between each model layer, it becomes zero when the aquitard does not exists;
	MDL_KDFRAC_{i}	The total fraction of the model layers that has been parameterised by the permeabilities found by the REGISKHV and/or REGISKVV files, whenever the fraction is 1.0 is means that the entire model layer has been filled in correctly, lower values indicate that areas in the model layers have not been filled in properly.
	MDL_CFRAC_{i}	The total fraction of the aquitards in between the model layers that has been parameterised by the permeabilities found by the REGISKHV and/or REGISKVV files. So more comment above;
TOP_L{i}=	Enter the IDF for the i^{th} modellayer that represents the top of modellayer i, e.g. TOP_L1=D:\INPUT\TOP_L1.IDF.	
ICLC_TL{i}=	Enter the option 0 or 1 to define whether this TOP modellayer needs to be interpolated, e.g. ICLC_TL1=1. This is optional, the default is 1.	
BOT_L{i}=	Enter the IDF for the i^{th} modellayer that represents the bottom of modellayer i, e.g. BOT_L1=D:\INPUT\BOT_L1.IDF.	

ICLC_BL{i}=	Enter the option 0 or 1 to define whether this BOTTOM modellayer needs to be interpolated, e.g. ICLC_BL1=1. This is optional, the default is 1.	
IMASK=	Specify IMASK=1 to (re)compute masks. Those are IDF files that contain a pointer value that indicates how the interfaces need to be computed. A mask value can have the following values: 0 means that this particular location will be excluded, those locations are initially formed by non-existence of the upper- and lowermost interface; -1 means that for that particular area no interface will be computed, the original value will be used; +1 means that the interface will be computed for this locations. Each mask IDF file will be saved in the MASK folder under the given OUTPUTFOLDER. Whenever IMASK=0, iMOD will look in this particular folder to read the mask IDF files, make sure that those files are in that folder.	
if IMASK=1		
	ZOFFSET=	Specify a vertical offset (meters) for which mask values need to be set af -1. In other words, whenever the difference between the TOP_L{i} and BOT_L{i+1} is larger than ZOFFSET the mask will be put on -1. Small aquitards can be removed in this manner. By default ZOFFSET=0.0 meter.
IHYPO=	Specify IHYPO=1 to compute the hypothetical interfaces, for mask values of 1. As the values for TOP_L|[i} and BOT_L{i+1} need to be identical for mask values of +1, only the interface for TOP_L|[i} will be computed and BOT_L{i+1} will be set equal to that value.	
if IHYPO=1		
	DZ(.)= (optional)	Specify for each model layer the minimal thickness (meter), e.g. DZ(1)=1.0 means that the minimal thickness will be 1.0 meter. In this way it is possible to have continuous thicknesses for model layers. This minimal thickness requirement can not be met whenever the distance between two aquitards is less than this DZ. In that case a smaller thickness is forced. By default DZ=0.0 for each model layer.
	IMIDELEV= (optional)	Specify IMIDELEV > 0.0 to force te PCG solver to position the hypothetical interface more-or-less such that model layers have uniform thicknesses. By default IMIDELEV=1.0, however, IMIDELEV=0.0 will deactivate this feature. The higher the value for IMIDELEV the more the hypothetical interface is a true average for all appropriate interfaces, the lower the value, the more smooth is the hypothetical interface will be probably, but the constraint of even distributed interfaces is more violated.
	IINT_IDF= (optional)	This keyword can be defined as IINT_IDF=1 (Default) to use upper and lower situated clay layers by the investigation of the hypothetical interfaces.
	IBNDCHK= (optional)	Specify IBNDCHK=1 to check internally for isolated cells that are NOT connected to constant value cells. By default IBNDCHK=0.
	HCLOSE= (optional)	Specify the closure criterion of the PCG solver, e.g. HCLOSE=0.1 m. By default HCLOSE=0.001 meter.
	MICNVG= (optional)	Specify the number of subsequent inner convergences of the PCG solver, e.g. MICNVG=25. Use this whenever the PCG solver does not find a unique solution. By default MICVNG=5.
ICKDC=	Specify ICKDC=1 to compute transmissivities for model layers and vertical resistances for in between model layers.	
if ICKDC=1		

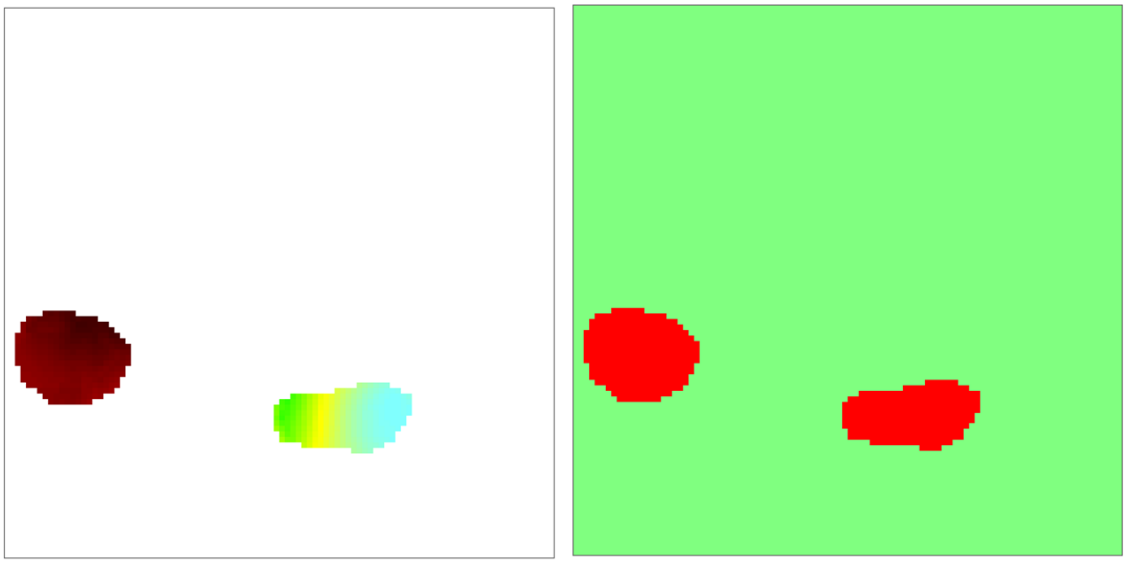
FNLAY= (optional)	Number of formation to be specified separately, e.g. FNLAY=10.	
if FNLAY specified		
	FTOP_L{i}=	Specify the IDF file for the i^{th} TOP elevation of a geological formations, e.g. FTOP_L1=D:\FORMATION\BEK1_TOP.IDF.
	FBOT_L{i}=	Specify the IDF file for the i^{th} BOT elevation of a geological formations, e.g. FBOT_L1=D:\FORMATION\BEK1_BOT.IDF.
	FKHV_L{i}=	Specify the IDF file for the i^{th} horizontal permeability of a geological formations, e.g. FKHV_L1=D:\FORMATION\BEK1_KHV.IDF. If no file is defined, this permeability will be assumed to be 3.0 the given vertical permeability at FKVV_L{i}.
	FKVV_L{i}=	Specify the IDF file for the i^{th} vertical permeability of a geological formations, e.g. FKVV_L1=D:\FORMATION\BEK1_KVV.IDF. If no file is defined, this permeability will be assumed to be 0.3 the horizontal permeability at FKHV_L{i}.
if FNLAY is not specified		
	FOLDERTOP=	Specify the folder that stores the TOP elevation of geological formations, e.g. FOLDERTOP=D:\REGIS*-TCK.IDF. All files will be used that fit this wildcard definition.
	FOLDERBOT=	Specify the folder that stores the BOT elevation of geological formations, e.g. FOLDERBOT=D:\REGIS*-BCK.IDF. All files will be used that fit this wildcard definition.
	FOLDERKHV=	Specify the folder that stores the horizontal permeability of geological formations, e.g. FOLDERKHV=D:\REGIS*-KH-SK.IDF. All files will be used that fit this wildcard definition. If no file is found for the horizontal permeability for a particular geological formation, this permeability will be assumed to be 3.0 the vertical permeability.
	FOLDERKVV=	Specify the folder that stores the vertical permeability of geological formations, e.g. FOLDERKVV=D:\REGIS*-KV-SK.IDF. All files will be used that fit this wildcard definition. If no file is found for the vertical permeability for a particular geological formation, this permeability will be assumed to be 0.3 the horizontal permeability.
WINDOW= (optional)	Enter the coordinates of the window that needs to be computed. Enter coordinates of the lower-left corner first and then the coordinates of the upper-right corner, e.g. WINDOW=100000.0, 400000.0, 200000.0, 425000.0. When WINDOW= is absent, the entered IDF-files by TOP_L{i} and BOT_L{i} need to be equally in their dimensions. Otherwise they will be upscaled (mean) or downscaled (interpolation) to the entered CELLSIZE.	
	CELLSIZE=	Enter the cell size (meter) for the IDF-files that will be created, e.g. CELLSIZE=25.0.
NGEN= (optional)	Enter a value of the amount of GEN-files containing fault information to be taken into account in the interpolation.	
if NGEN $\geq$ 1		
	GEN_i=	Related to NGEN with this keyword all the GEN-files are included in the interpolation. Enter first a value for the interface number, followed by the name of the GEN-file. Example: GEN_1=3,faults_l3.GEN.

Example

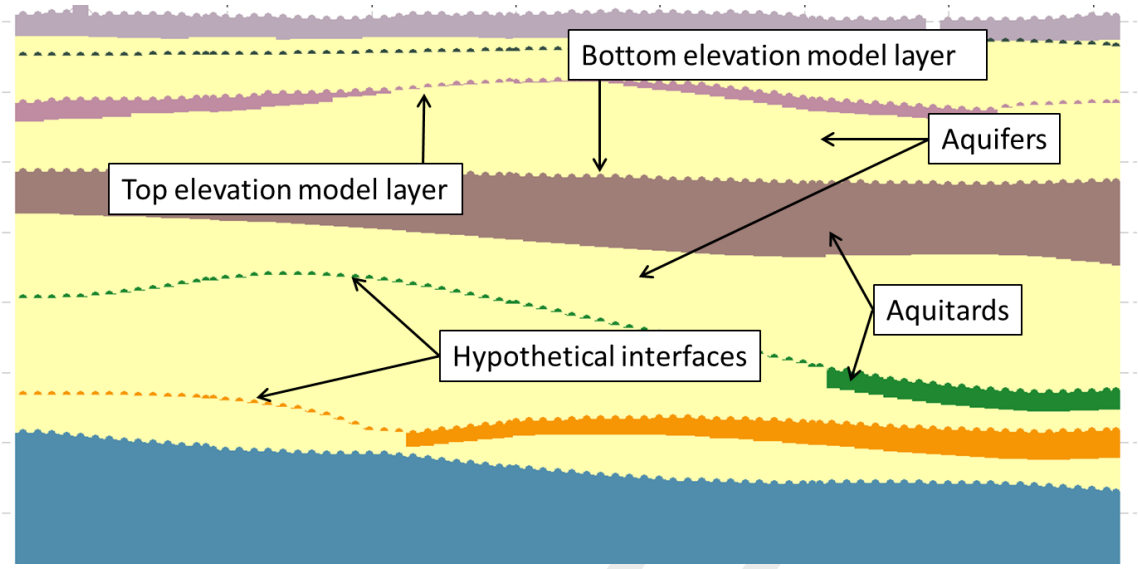
FUNCTION=SOLID
NLAY=4
TOP_L1=D:\MODEL\TOP_L1.IDF
TOP_L2=D:\MODEL\TOP_L2.IDF
TOP_L3=D:\MODEL\TOP_L3.IDF
TOP_L4=D:\MODEL\TOP_L4.IDF
BOT_L1=D:\MODEL\BOT_L1.IDF
BOT_L2=D:\MODEL\BOT_L2.IDF
BOT_L3=D:\MODEL\BOT_L3.IDF
BOT_L4=D:\MODEL\BOT_L4.IDF
IMASK=1
IHYPO=1
ICKDC=1
FOLDERTOP=D:\REGIS-T-CK.IDF.*
FOLDERBOT=D:\REGIS-B-CK.IDF.*
FOLDERKHV=D:\REGIS-KH-SK.IDF.*
FOLDERKVV=D:\REGIS-KV-SK.IDF.*
OUTPUTFOLDER=D:\OUTPUT

This example creates a mask files based on the specified top and bottom IDF-files specified by the TOP_L{i} and BOT_L{i} keywords. For those areas that does not contain an aquitard, the hypothetical interfaces will be computed. After that the transmissivities and vertical resistances will be computed and the other output as specified at the ketword OUTPUTFOLDER.

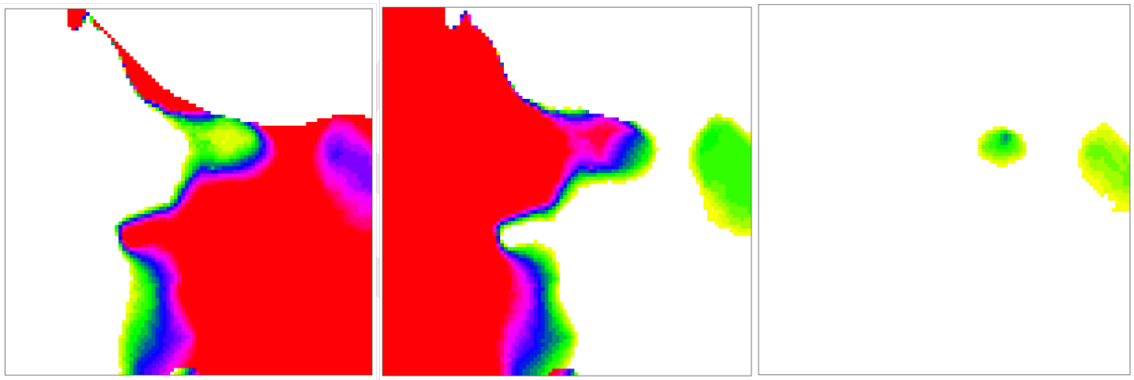
Example of (left) the computed thickness of an aquitard; (right) the corresponding values for the mask IDF (green is +1 and red = -1). The green area will be filled in by hypothetical interfaces.



Example of computed hypothetical interfaces as TOP and BOTTOM elevation for the model layers.



Example of computed fractions of a geological formation in three different model layers. The formation has been part of three model layers and the total transmissivity of each model layers depends on the given fraction of the geological formation. Red represents a higher fraction than yellow.



8.7.6 FLUMY-Function

Use this function to create Flumy textfiles with borehole information out of IPF-file related textfiles. Flumy textfiles contain information about the fluvial deposits in (former) riverbeds (Flumy is a Geovariances software program).

FUNCTION=	**FLUMY**
IPFFILE=	Enter the name of the IPF-file that contains the borehole information to be converted to a Flumy readable format.
OFFSET=	Optional variable. Enter a value to elevate the depth of the borehole, e.g. current depth of borehole is -50 m, on OFFSET=50 results in a borehole reference depth of 0 m.
NPARAM=	Enter the number of parameters {i} that needs to be distinguished in the Flumy-textfile(s).
GRAIN{i}=	Enter the name of each grain type as given in IPF related textfile, e.g. GRAIN1=SILT or GRAIN2='Sandy Clay'
FACIESL{i}=	Enter the verb for the location in the fluvial area where the i^{th} Grain type will be located, e.g. FACIESL1=OB, in case "SILT" needs to be located in the Overbanks.
FACIESN{i}=	Enter the number related to the facies layer as defined with FACIESL, e.g. FACIESL1=OB, FACIESN1=8.

The created Flumy textfile(s) are(is) stored in the working directory of the FLUMY-batchfile within a new generated folder “FLUMY”.

Example 1

FUNCTION=FLUMY
IPFFILE='D:\flumy\Boreholes\Borelogs.ipf'
NPARAM=2
GRAIN1=SILT
FACIESL1=OB
FACIESN1=8
GRAIN2='SANDY Clay'
FACIESL2=PB
FACIESN2=2

Note: Be aware that quotes are obligatory around a “GRAIN”-description containing more than one word, e.g. 'SANDY CLAY'. Otherwise iMOD is not able to read this parameter properly.

8.7.7 GEOCONNECT-function

Use this function to:

1 Compute (new) KHV, KVV and KVA grids, and/or KDW and VCW grids based on a given factor (iPEST) per formation, or
2 Aggregate model output based on given REGIS formations.This functionality is also available as a GUI-function (see section 7.6).
3 Identify fraction grids based on REGIS formations at a chosen model location and for the selected model layers (not available yet, only in the iMOD GUI).

General Settings
These are the general settings needed in each *.ini file using the GEOCONNECT-Batch function:

FUNCTION=	**GEOCONNECT**
NLAY=	Enter the amount of model layers, e.g. NLAY=10.
ACTLAYERS= (optional)	Enter a string of values to include or exclude a specific model layer from the computation; 0=inactive, 1=active, on default all layers are used in de computation (similar to e.g.: ACTLAYERS=1111111111). E.g. in case of the amount of model layers is 10 and it is preferred to only take the first 6 layers into account: ACTLAYERS=1111110000.
REGISFOLDER=	Give the directory and name of the folder where all REGIS-files are stored. **Note:** subdirectories are not allowed and the filenames need to be of the following format: abbreviation formation name-t/b/ks/kv-ck/sk.idf ('t' and 'b' need to be combined with 'ck', and 'ks' and 'kv' with 'sk'), e.g. d:\Model\REGIS\bez1-b-ck.idf.
TOPFOLDER=	Give the directory and name of the folder of the model TOP-files, e.g. d:\Model\TOP\TOP.
BOTFOLDER=	Give the directory and name of the folder of the model BOT-files, e.g. d:\Model\BOT\BOT.
WINDOW=	Enter the coordinates of the window that needs to be computed. Enter coordinates of the lower-left corner first and then the coordinates of the upper-right corner, e.g. WINDOW=100000, 400000, 200000, 425000. When WINDOW= is absent iMOD will take the WINDOW-extent of the input IDF's.
CELLSIZE=	Enter the cell size (meter) for the IDF-files that will be created, e.g. CELL_SIZE=25.0.
NFORM=	Enter a value for the total amount of formation factors to be read from this *.ini file, e.g. NFORM=127.
FORM{i}=	Give the name of the i'th formation and the corresponding factor, e.g. FORM1=HLC,1.000. The largest i-number needs to correspond with the total amount of factors as defined with NFORM.
OUTPUTFOLDER=	Give the directory and name of the folder to store the results of the preprocessing computation.
IFLAG=	Enter a value for the specific GeoConnect function to be used: 1=Preprocessing, 2=Postprocessing. E.g. IFLAG=1.

Options
In case of IFLAG=1 (Preprocessing option), the *.ini file with specific options needs to contain:

ISAVEK= (optional)	Use this keyword to save KHV, KVV and KVA values in IDF-format; 1=saved and 0=not saved, on default ISAVEK=1.
ISAVEC= (optional)	Use this keyword to save KDW and VCW values in IDF-format; 1=saved and 0=not saved, on default ISAVEC=0.

In case of IFLAG=2 (Postprocessing option), the *.ini file with specific options needs to contain:

DBASEFOLDER=	Give the directory and name of the folder containing DBASE model information, e.g. p:\1221301-ibrahym-dld\DBASE_V1
IAGGR=	Give the value related to the aggregation option to be used, IAGGR=1: apply to model output, IAGGR=2: apply to model input, IAGGR=3: apply to ipf-file
if IAGGR=1...	
MODELFOLDER=	Give the directory and name of the folder containing the preferred model output information.
MODELTYPE=	Choose the favored variable to apply the aggregation to, e.g. MODELTYPE=2. Options are: 1="HEAD", 2="BDGWEL", 3="BDGRIV", 4="BDGDRN".
if IAGGR=2...	
INPUTTYPE=	Choose the preferred variable to apply the aggregation to, e.g. INPUTTYPE=3. Options are: 1="KDW", 2="VCW", 3="KHV", 4="KVV".
if IAGGR=3...	
IPFFILE=	Give the directory and name of the IPF-file to apply the aggregation to.
IDUPLICATES=	Give the type of aggregation. When IDUPLICATES=1 the maximum value per grid cell is taken from the files to be aggregated, IDUPLICATES=2: maximum value, IDUPLICATES=3: average value, and IDUPLICATES=4 the sum of the value per grid cell for all files is taken as new value in the aggregated grid cell in the file.
ISAVETB= (*Optional*)	Use this option to include TOP- and BOT elevation IDF-files in the aggregation and save these files in the given Output folder. In case ISAVETB=1, TOP and BOT elevation are saved, otherwhise when ISAVETB=0 these files are not saved in the Output folder.
OUTPUTFOLDER=	Give the directory and name of the folder to store the results of the preprocessing computation.

Be sure that the *geostratigraphy.txt* file is placed in the defined DBASEFOLDER!

Example 1
FUNCTION=GEOCONNECT
NLAY=19
ACTLAYERS=1111111100011100000
REGISFOLDER=d:\Model_Ibrahym2\REGIS21
TOPFOLDER=d:\Model_Ibrahym2\DBASE_V2\TOP\VERSION_1
BOTFOLDER=d:\Model_Ibrahym2\DBASE_V2\BOT\VERSION_1
IFLAG=1
NFORM=8
FORM1=HLC,1.000
FORM2=BXSCK1,1.300
FORM3=BXZ1,1.000
FORM4=BXK1,1.000
FORM5=BXLMK1,2.500
FORM6=BXZ2,1.000
FORM7=BXK2,1.000
FORM8=BXZ3,0.910
OUTPUTFOLDER=d:\Model_Ibrahym2\Results\
ISAVEK=1
ISAVEC=0

With this example the KHV, KVV and KVA grids are (re)calculated (preprocessing) based on a given factor per formation (FORM{i}) for the first 8 formation seen from the top of the Ibrahym model.

Example 2
FUNCTION=GEOCONNECT
NLAY=19
ACTLAYERS=1111111100011100000
REGISFOLDER=d:\Model_Ibrahym2\REGIS21
TOPFOLDER=d:\Model_Ibrahym2\DBASE_V2\TOP\VERSION_1
BOTFOLDER=d:\Model_Ibrahym2\DBASE_V2\BOT\VERSION_1
IFLAG=2
DBASEFOLDER=d:\Model_Ibrahym2\DBASE_V2
NFORM=8
FORM1=HLC,1
FORM2=BXSCK1,4
FORM3=BXZ1,4
FORM4=BXK1,4
FORM5=BXLMK1,4
FORM6=BXZ2,4
FORM7=BXK2,4
FORM8=BXZ3,4
IAGGR=2
INPUTTYPE=3
IDUPLICATES=3
ISAVETB=0
OUTPUTFOLDER=d:\Model_Ibrahym2\DBASE_V4\

In this example the (postprocessing) aggregation is applied to the input model variable "KHV" for which all BX-formations are taken together as one formation. An average value is calculated per grid cell for all BX-formations. Top and Bot elevations are not saved.

8.7.8 CREATEIZONE-Function

Use this function to calculate zones and corresponding fractions per model layer based on geologic formations, these IDF files can be used during an iPEST optimization.

FUNCTION=	**CREATEIZONE**
OFOLDER=	Give an output folder name to store all the fractions per model layer.
PFOLDER=	Give a folder name that contains the fraction per model layers per geological formations. This is the results
NLAY=	Enter the number of modellayers, e.g. NLAY=6
MINF=	Minimum fraction to assign .... to a zone....
IZONEOFFSET=	Enter the number of zones .....
IGROUPOFFSET=	Enter the number of zones .....
NFORMATIONS=	Enter the number of formations.
FORMATION{i}=	Enter the ... for the i^{th} formation.
TPARAMETER=	Enter the name of the ...

Example

FUNCTION=CREATEIZONE
OFOLDER=D:\MODEL
PFOLDER=D:\RESULTS
NLAY=2
MINF=...
IZONEOFFSET=...
IGROUPOFFSET=...
NFORMATIONS=2
FORMATION1=...
FORMATION2=...
TPARAMETER=5

The example above will .....

8.8 PREPROCESSING-FUNCTIONS

8.8.1 CREATEIBOUND-Function

Use this function to create an IBOUND IDF-file that takes into account non existing cells on the bottom of a layer system. The IBOUND value in a cell is set to 0 whenever this cell and all underlying cells do not exist (thickness = 0.). Otherwise the IBOUND value is set to 1.

FUNCTION=	**CREATEIBOUND**
NLAY=	Enter the number of model layers, e.g. NLAY=6.
TOP_L{i}=	Enter the IDF for the i^{th} modellayer that represents the top of modellayer *i*, e.g. TOP_L1=D:\MODEL\TOP_L1.IDF.
BOT_L{i}=	Enter the IDF for the i^{th} modellayer that represents the bottom of modellayer *i*, e.g. BOT_L1=D:\MODEL\BOT_L1.IDF.
RESULTDIR=	Enter the foldername in which the adjusted IDF-files will be saved, e.g. RESULTDIR=D:\RESULT.

Example

FUNCTION= CREATEIBOUND
NLAY=3
TOP_L1=D:\MODEL\TOP_L1.IDF
TOP_L2=D:\MODEL\TOP_L2.IDF
TOP_L3=D:\MODEL\TOP_L3.IDF
BOT_L1=D:\MODEL\BOT_L1.IDF
BOT_L2=D:\MODEL\BOT_L2.IDF
BOT_L3=D:\MODEL\BOT_L3.IDF
RESULTDIR=D:\RESULT

This example checks for a 3 layer system whether there are non existing cell on the bottom of the system. These cells get an IBOUND value of 0.
The result of this function is an IBOUND file for each layer D:\RESULT\IBOUND_L{i}.IDF.

8.8.2 AHNFILTER-Function

This function filters artificial elements out of a digital terrain model (or equivalent). The method searches for areas that are connected by small vertical thresholds and denote them as true surface whenever the extent is significant. The remaining areas with less extent are marked as potential surface and will become true surface whenever they differ minorly with the interpolated surface at those locations. Moreover, the algorithm searches for local upconing (trees, buildings) and depressions (streams) and removes them from the surface. The function yields two output files: the filtered original file and a pointer file. The pointer file indicates the type of change by the filtering to the original file. The values in the pointer file indicate: 2: no change; <2: filtered cell.

FUNCTION=	**AHNFILTER**
NAHN=	Enter the number of the IDF-files to be used during the AHN filtering, e.g. NAHN=2.
IDFFILE{i}=	Enter the name of the i^{th} IDF-file. The IDF-files should contain data points that represent some kind of elevation data, e.g. IDFFILE2=D:\DATA\AHN_REGION2.IDF.
XCRIT=	Enter the vertical offset between adjacent cells that are allowed to group together, e.g. XCRIT=0.5 (default value). In this case all adjacent cells that have an offset of less or equal 0.25 (unit of IDFFILE{i}). Increasing XCRIT will group more cells together, decreasing XCRIT will group them more difficult. The size of the group will determine whether the group is assigned a surfacelevel directly, or not. The default value is XCRIT=0.5.
NSCRIT=	Enter the number of cells that behave like a threshold whether the current group of cells can be denoted as surface level, e.g. NSCRIT=1500 (default value).
DPW=	Enter the size of the window that will be used to determine local upconing and depressions in the area, e.g. DPW=5 (default value). In this case a squared 5x5 window will be applied .
LOCCRIT=	Enter the max difference of those values (not equal to the *NoDataValue*) in the DPW window, e.g. LOCCRIT=2.0 (default value). Whenever this difference exceeds LOCCRIT, no local upconing and depression will be computed.
DP1=	Enter the percentile for which all data points in the DPW window that are less or equal to DP1 will be assigned to local depression, e.g. DP1=30.0 (default value).
DP2=	Enter the percentile for which all data points in the DPW window that are greater or equal to DP2 will be assigned to local upconing, e.g. DP2=30.0 (default value).
INTXCRIT=	Enter the maximum residual change in the interpolation of the intermediate surface level, INTXCRIT=0.05 (default value).
CORXCRIT=	Enter the difference between the original surface level, as read by the IDFFILE{i}-files, and the intermediate surface level, e.g. CORXCRIT=0.10 (default value).
NCORXCRIT=	Enter the minimum number of changes by CORXCRIT, e.g. NCORXCRIT=50 (default value).
IGNORENODATA=	Enter IGNORENODATA=1 (default value) to ignore all data points equal to the *NoDataValue* of the IDFFILES{i}. Enter IGNORENODATA =0 to interpolate all data points equal to the *NoDataValue*.
NWINDOW=	Enter the number of windows to filter, e.g. NWINDOW=5. By default the entire dimensions of the IDFFILES{i} will be processed. The default value is NWINDOW=0.
WINDOW{i}=	Enter the coordinates of the lower-left corner first and then the coordinates of the upper-right corner, e.g. WINDOW5=100000.0, 400000.0, 200000.0, 425000.0. This keyword is compulsory whenever NWINDOW>0.
OUTFILE= OUTFILE{i}=	Enter the name of the IDF-file that contains the filtered surface level, e.g. OUTFILE=D:\DATA\AHN.IDF. In case NWINDOW>0, it is obliged to specify the OUTFILE for all windows, e.g. OUTFILE5=D:\DATA\AHN5.IDF. In the latter case, you can use the FUNCTION=IDFMERGE (see section 8.2.4) to merge all outcomes.

BUFFER=	Enter a buffersize to overlap the different WINDOW{i}'s, e.g. BUFFER=1500.0 (default value).
IAGGREGATEY=	Enter IAGGREGATEY=1 to join adjacent intermediate surface levels to form larger ones, with a change that they become greater than NSCRIT and become surfacelevel. Default value is IAGGREGATEY=0.

Example 1

FUNCTION=AHNFILTER
NAHN =1
IDFFILE1=D:\DATA\AHN_ORG.IDF
OUTFILE=D:\DATA\AHN_FILTERED.IDF

The above mentioned example is the most simple one, it filters the AHN_ORG.IDF with all default setting values and saves the result in AHN_FILTERED.IDF.

Example 2

FUNCTION=AHNFILTER
NAHN =2
IDFFILE1=D:\DATA\AHN_ORG1.IDF
IDFFILE2=D:\DATA\AHN_ORG2.IDF
NWINDOW=2
WINDOW1=125000.0 426000.0 130000.0 430000.0
WINDOW2=130000.0 426000.0 135000.0 430000.0
OUTFILE1=D:\DATA\AHN_FILTERED1.IDF
OUTFILE2=D:\DATA\AHN_FILTERED2.IDF
BUFFER=2500
IGNORENODATA=0
NSCRIT=1250
LOCCRIT=200.0
XCRIT=100.0
DPW=5
DP1=30.0
DP2=90.0
CORXCRIT=10
CORCRIT =75
INTXCRIT=5

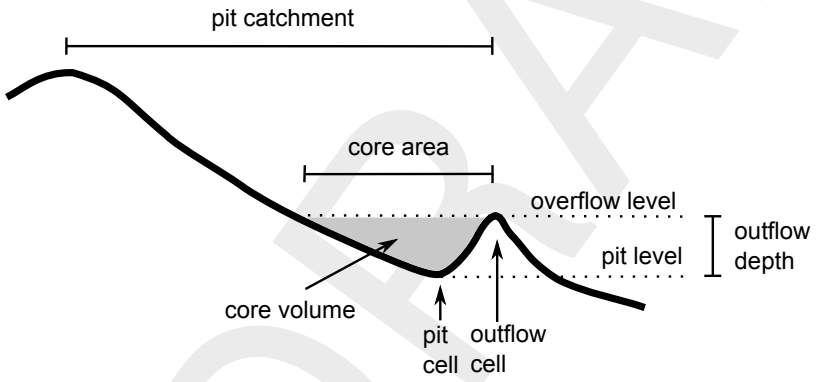
The above mentioned example filters two windows and uses two different IDF-files (AHN_ORG1.IDF and AHN_ORG2.IDF). The main reason for using most of the setting variables is that the dimension of the original IDF-files (IDFFILE1 and IDFFILE2) is centimeter instead of meter.

8.8.3 CREATESOF-Function

The iMOD-Batch function SOF (Surface Overland Flow) is able to compute “spill” levels (surface overlandflow levels) for large regions with or without supplied outflow or outlet locations. First of all, the “pit”-locations are identified, these are defined as those locations that are surrounded by higher values all around in a 3×3 matrix of grid cells. There is no other escape possible other than the lowest neighbouring level, the so called “spill”-level. All these identified “pitts” are sorted from the lowest to the highest “pitt” and processed in that order. Given a “pitt” location, the surrounding grid cells will be stored in a boundary list. From the boundary list the lowest “spill”-level will be found and stored in a boundary list. Whenever this “spill”-level is lower than the current “spill”-level, the process stops, because than probably the water can flow outside the current “core-volume”. If not, the lowest level on the boundary becomes the new “spill”level and the node will be remove from the boundary list and added to the “pitt”-list. The process repeats itself. All grid cells that are mentioned in the pitt-list will belong to the same “core-volume” and receive the final “spill”-level. At the end of this section, an example is given. As the “core-area” will be flat, the discharge occurs on that area as follows: it first follows the steepest gradient along the digital elevation model into the pit location and from there it follows the shortest way up towards the outflow cell. As the “core-area” will be flat, it cannot be used for a SFR package (see section 12.28). To avoid flat-surfaces, the SOF function can be generate a slight adapted surface level for flat areas as a gradient from where the flat area is entered to the elevation next to the exitpoint. Those corrected flat areas (that can be a stretch along a stream) can be used for the SFR discretisation, at the same time the original slope of the flat area (0.0) is adjusted as saved. This slope is used as well for the SFR to define the stream slopes.

In the following figure the mentioned variables are explained.

Example of the concepts for the “spill”-level procedure.



The steepest gradient of the digital terrain model is computed for a 3×3 neighbourhood, so the slope s and aspect a are at one point estimated from elevations of it and surrounding 8 points i, thus:

$$s_i = \tan^{-1} \frac{z_i - z_0}{\sqrt{(x_i - x_0)^2 + (y_i - y_0)^2}}$$
$$a_i = \arctan^{-1} (x_i - x_0, y_i - y_0)$$

where the slope s_i is in radians as well as the aspect a_i, and z_0, x_0 and y_0 are the elevation, x-coordinate and y-coordinate at the current node respectively, for which the gradient and aspect need to be computed. For reasons of conveniences, the aspect a is taken by the arctangent function with two arguments. The purpose of using two arguments instead of one is to gather information on the signs of the inputs in order to return the appropriate quadrant of the computed angle, which is not possible for the single-argument arctangent function. The final value for the aspect a is $-\pi < a < \pi$, meaning that π is pointing to the east, $0.5 \times \pi$ points to the south, $0.0 \times \pi$ points to the west and $-0.5 \times \pi$ points to the north. For perfect flat areas, a south direction is chosen arbitrarily. Also *NodataValues* are threated as “pitts”.

The CREATESOF function continues to follow a drop of water to its down slope neighbour, we call this a trace and all cells that are passed are stored in a thread. This method uses the steepest descent

direction from a particular location and moves to the next ceel along that direction, so the new x-location x_i is found from the previous x-location x_{i-1} by:

$$x_i = x_{i-1} + \Delta x \cdot \cos(a_{i-1}) \; ; \; y_i = y_{i-1} \Delta y \cdot \sin(a_{i-1})$$

where Δx and Δy are the cellsizes of the digital elevation model.

FUNCTION=	**CREATESOF**	
IFLOW=	Enter a code to specify the way the CREATESOF function need to work. Below the options are described	
IFLOW=0	Apply IFLOW=0 to compute the "spill"-levels, slopes and aspect angles. The following files will be saved:	
	*_COPY.IDF	used digital elevation data after clipping and/or up- or down-scaling, this file is only save whenever the keyword WINDOW is given;
	*_PITT.IDF	describes the location of the individual "pitts"-cells (value 1 for a "pitts"-cell; 0 for all non-"pitts"-cells);
	*_SLOPE.IDF	gradient ($\frac{\Delta x}{\Delta z}$) of the steepest slope of each grid cell (only whenever IGRAD=1);
	*_ASPECT.IDF	steepest flow angle (azimuth) in radians, north=$-\frac{1}{2}\pi$; west=0; south=$\frac{1}{2}\pi$; east=π; (only whenever IGRAD=1).
	LEVELIDF=	Enter the surface level (Digital Terrain Model DTM) IDF-file that need to be processed, e.g. LEVELIDF=D:\DATA\DTM.IDF.
	PITTSIZE= (optional)	Enter the minimal size of a pit to become a natural outlet. If the size of the pit exceeds the number given for e.g. PITTSIZE=100, this area becomes a natural outlet and all streams will terminate at that location.
	OUTLETIDF= (optional)	Enter an IDF file that describes the outlet locations, e.g. OUTLETIDF=D:\MODEL\RIVERS.IDF. Each location not equal to the *NodataValue* in the given OUTLETIDF will be used to terminate the further search for a "spill"-level.
	WINDOW= (optional)	Specify a window for which the entered LEVELIDF will be clipped and resized whenever the entered CELLSIZE $/ne$ the cellsize of the given IDF file at LEVELIDF. Enter the coordinates of the lower-left corner first and then the coordinates of the upper-right corner, e.g. WINDOW=100000,400000,200000,425000. When WINDOW is absent, the internal dimensions of LEVELIDF will be used.
	CELLSIZE=	Specify a cell size for the WINDOW to be clipped out or resized, e.g. CELLSIZE=100.0.
	IGRAD= (optional)	Enter the IGRAD=0 to ignore the computation of the aspects of the DTM as these computations can take significant amount of time, especially the flat areas and the elevation on the core volumes. Choose IGRAD=1 to include the aspect computations. The default value is IGRAD=0 whenever this keyword is absent.
	SOFIDF=	Enter the SOFIDF to save the computed Surface Overland Flow level; this elevation describes the elevation at which surface water will discharge water. For "pitts" this is the overflow level, for all others it is the original DTM-level. By subtracting the original DTM- and SOF-level the size and depth ("pitts"-volume) of the can be computed.
IFLOW=1	Specify IFLOW=1 to compute the entire "flow path" of the particle flowing over the surface. The following files will be saved:	
	*.GEN	describes the entire flow path only if IWRITE=1;
	*_ZONE.IDF	describes the number of discharge zone as specified by the IPF file given by the optional keyword DISZONEIPF.
	LEVELIDF=	Enter the surface level (Digital Terrain Model DTM) IDF-file that need to be corrected for flat surfaces, it is logical to use the SOFIDF from a IFLOW=0 run; e.g. LEVELIDF=D:\DATA\SOF.IDF.

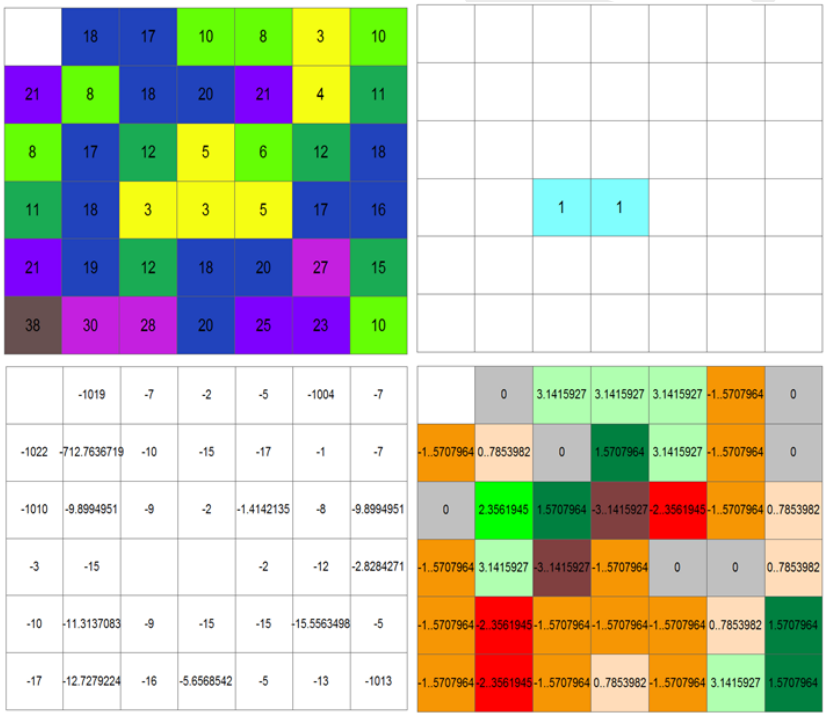
	SLOPEIDF=	Enter the slope IDF file that need to be processed, it is logical to use the slope IDF from a IFLOW=0 run; e.g. SLOPEIDF=D:\DATA\SLOPE.IDF.
	LEVEL_ OUTIDF=	Enter the IDF file for the corrected surface level (Digital Terrain Model DTM) for flat areas, e.g. LEVEL_OUTIDF=D:\DATA\SOF_FLAT.IDF.
	SLOPE_ OUTIDF=	Enter the slope IDF file corrected for flat areas, e.g. SLOPE_OUTIDF=D:\DATA\SLOPE_FLAT.IDF.
	COUNTIDF	Enter the IDF file for which the total number of flow paths (friction) that passes each grid cell, e.g. COUNTIDF=D:\DATA\FRICTION.IDF;
	ASPECTIDF=	Enter the name of the IDF file that contains the aspects for each grid cell, e.g. ASPECTIDF=D:\RESULTS\ASPECT.IDF. Normally, this is the result of the simulation with IFLOW=0 (*_ASPECT.IDF).
	DISZONEIPF= (optional)	Enter the name of the IPF file that describes the location of discharge measurement station, e.g. DISZONEIPF=D:\INPUT\DIS.IPF. The minimal requirement of the data in the IPF file is that the first three columns need to describe the x, y and station number (integer). The resulting {SOFIDF}_ZONE.IDF will present the areas that discharge to the given station numbers. This DISZONEIPF is optional.
	IWRITE= (optional)	Enter IWRITE=1 to write a GEN file of all "flow paths" of particles flowing over the DTM. This will yield the file {ASPECTIDF}.GEN. Writing this file will reduce the performance and it often will yield an enormous file. The default option is IWRITE=0.
IFLOW=2	Specify IFLOW=2 to compute cumulative volumes for catchments for each location and for given percentiles.	
	COUNTIDF	Enter the IDF file with the total number of flow paths that passes each grid cell this is equal to the output file *_COUNT.IDF whenever IFLOW=1;
	ITQP	Enter ITQP=0 to compute the percentiles and ITQP=1 to use the percentiles. Suppose, ITQP=0, the IDF files given at TQPIDF{i} will be created, and whenever ITQP=1, those files at TQPIDF{i} will be used.
	MINQ	Enter a minimal discharge volume to be taken into account in the comutation of percentiles, e.g. MINQ=1000 m^3/day will exclude those with total discharges less than this amount.
	TTQP	Enter TTQP=0 to compute a total percentile, or enter ITQP=1 to compute monthly percentiles.
	NTQP	Enter the number of percentiles of discharges to be computed, e.g. NTQP=5 means that you need to enter 5 percentiles at PTQP{i}..
	PTQP{i}	Enter the percentile value (0.0 $\geq$ PTQP{i} $\leq$ 100.0) for the i^{th} percentile out of NTQP, e.g. PTQP2=50.0.
	RESULTIDF	Enter a folder of the results IDF files to be used for the total volume computation, e.g., RESULTIDF=D:\RESULTS\BDGRIV\BDGRIV_*_L1.IDF. This means that all files will be used to compute a percentile, it depends on the amount of files that meet this wildcard. Whenever TTQP=0 a single percentile will be computed for all files, whenever TTQP=1, a monthly percentile will be computed, this means that a monthly output will be created at OUTPUTFOLDER.

	OUTPUT-FOLDER	Enter a directory to store all results, e.g. OUTPUT-FOLDER=D:\RESULTS\VOLUMES. Here, for each inputfile (RESULTIDF) a percentile will be computed of the current discharge compared to the total (TTQP=0) or monthly (TTQP=1) percentile. The files are called TQ_PERCENT_{yyyymmdd}.IDF. This files gives a unique value of the class at which the current discharge belong. Whenever the value is 3, this means a percentile value that belongs in between the 3^{rd} and 4^{th} class. A value of 0 means that the current percentile undercounts the given percentiles in the TQPIDF{i} files, a value above NTQP, means that it exceeds the given percentiles.
	TQPIDF{i}	Enter the name of the IDF to save the percentiles, e.g. TQPIDF2=D:\RESULTS\VOLUMES\TQP_50.IDF. Whenever TTQP=1, these names will be enhanced by a month identification, e.g. the final name becomes D:\RESULTS\VOLUMES\TQP_50_AUG.IDF to denote the 50^{th} percentile for the month August.
IFLOW=3	Specify IFLOW=2 to compute cumulative volumes for catchments for each location and for given percentiles.	
	LEVELIDF=	Enter the surface level (Digital Terrain Model DTM) IDF-file, it is logical to use the SOFIDF from a IFLOW=1 run; e.g. LEVELIDF=D:\DATA\SOF_FLAT.IDF.
	SLOPEIDF=	Enter the slope IDF file that need to be processed, it is logical to use the slope IDF from a IFLOW=1 run; e.g. SLOPEIDF=D:\DATA\SLOPE_FLAT.IDF.
	COUNTIDF	Enter the IDF file for which the total number of flow paths (friction) that passes each grid cell, e.g. COUNTIDF=D:\DATA\FRICTION.IDF;
	ASPECTIDF=	Enter the name of the IDF file that contains the aspects for each grid cell, e.g. ASPECTIDF=D:\RESULTS\ASPECT.IDF. Normally, this is the result of the simulation with IFLOW=0 (*_ASPECT.IDF).
	IFORMAT=	Enter the format of the output file structure. Apply IFORMAT=1 whenever a conventional RIVER package need to be constructed with river conductances, -stage, -bottomlevels and infiltration factors. Apply IFORMAT=2 to construct a ISG file for the SFR package.
	RCND_IDF=	Enter the IDF file that represented to river conductance; e.g. RCND_IDF=D:\DATA\RCOND.IDF.
	RSTG_IDF=	Enter the IDF file that represented to river stage; e.g. RSTG_IDF=D:\DATA\RSTAGE.IDF.
	RBOT_IDF=	Enter the IDF file that represented to river bottom; e.g. RBOT_IDF=D:\DATA\RBOTTOM.IDF.
	RINF_IDF=	Enter the IDF file that represented to river stage; e.g. RINF_IDF=D:\DATA\RINF.IDF.
	RAIN= (optional)	Enter the size of the rainfall used to compute the dimension of the stream, e.g. RAIN=1.0 means an constant rain event of 1 mm/day. By default RAIN=1.0 mm/day.
	MINFRICTION= (optional)	Enter the size of the minimal value for the friction (number of passes in the COUNTIDF file) to be used to generate a stream, e.g. MINFRICTION=5.0 means all grid sizes with a friction value of less than 5.0 will not be processed to a IDF-files (see above) or an ISG file. By default MINFRICTION=0.0.

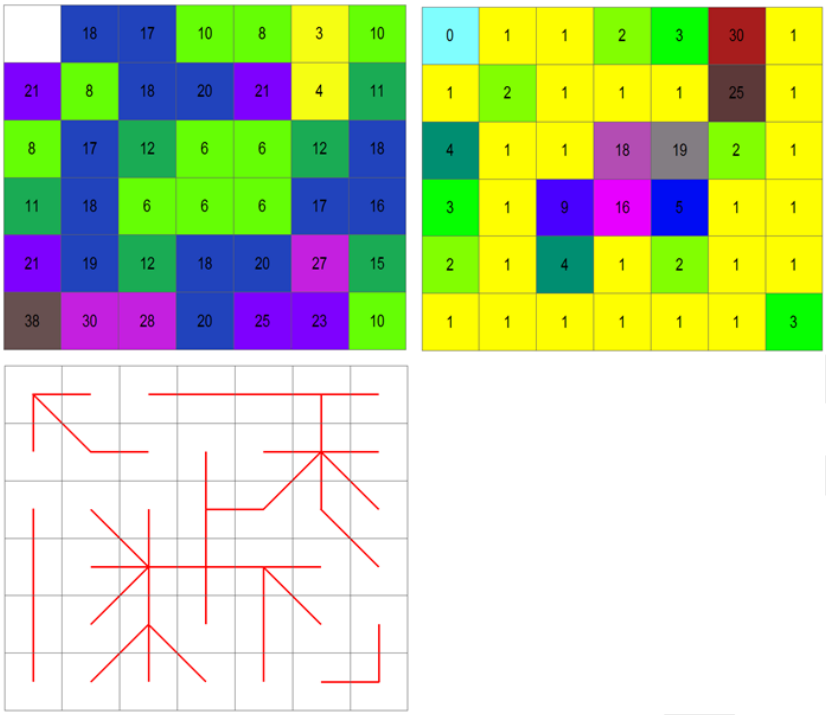
NQDW= (optional)	Enter the number of discharge-depth-width relations, e.g. NQDW=3. iMOD will interpolate between three given relations between discharge, depth and width to determine the dimensions at each gridcell based on the friction in the COUNTIDF. By default NQDW=0 and iMOD uses the Manning equation to determine the dimensions at each gridcell. $$d = \left(\frac{Q \times n}{w \times \sqrt{S}} \right)^{\frac{3}{5}} \quad (8.2)$$ where d is the depth, Q the discharge, n Mannings roughness coefficient (n=0.03), w is the width (w=1.0) and S is the slope between two gridcells. It is advisable to use the QDP-relations as that will results in a more reliable conductance than the Manning Equation since the latter is very sensitive to local gradients in the DTM (slopes).
QDW_{i}(.)=	Enter for each relation the discharge (m^3/day), depth (m) and width (m), e.g. QDW_3=100.0,1.5,20.0. Start with QDW_1=0.0,0.0,0.0.

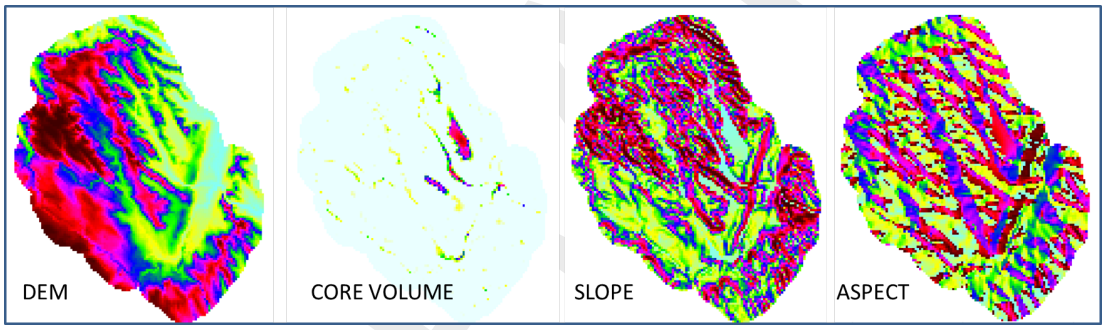
Below an example is given for the output variables for IFLOW=0 and IFLOW=1 for an artificial DTM.

Example of (upperleft) a DTM (upperright) the PITTs (lowerleft) the SLOPE and (lowerright) the ASPECT.



Example of (upperleft) a SOF (upperright) the number of passes and (lowerleft) the flowpaths GEN).





Example 1

FUNCTION=CREATESOF
IFLOW=0
LEVELIDF=D:\DATA\DTM.IDF
SOFIDF=D:\OUTPUT\SOF.IDF

Example 2

FUNCTION=CREATESOF
IFLOW=1
ASPECTIDF=D:\OUTPUT\DTM_ASPECT.IDF
IFLOW=1

Example 3

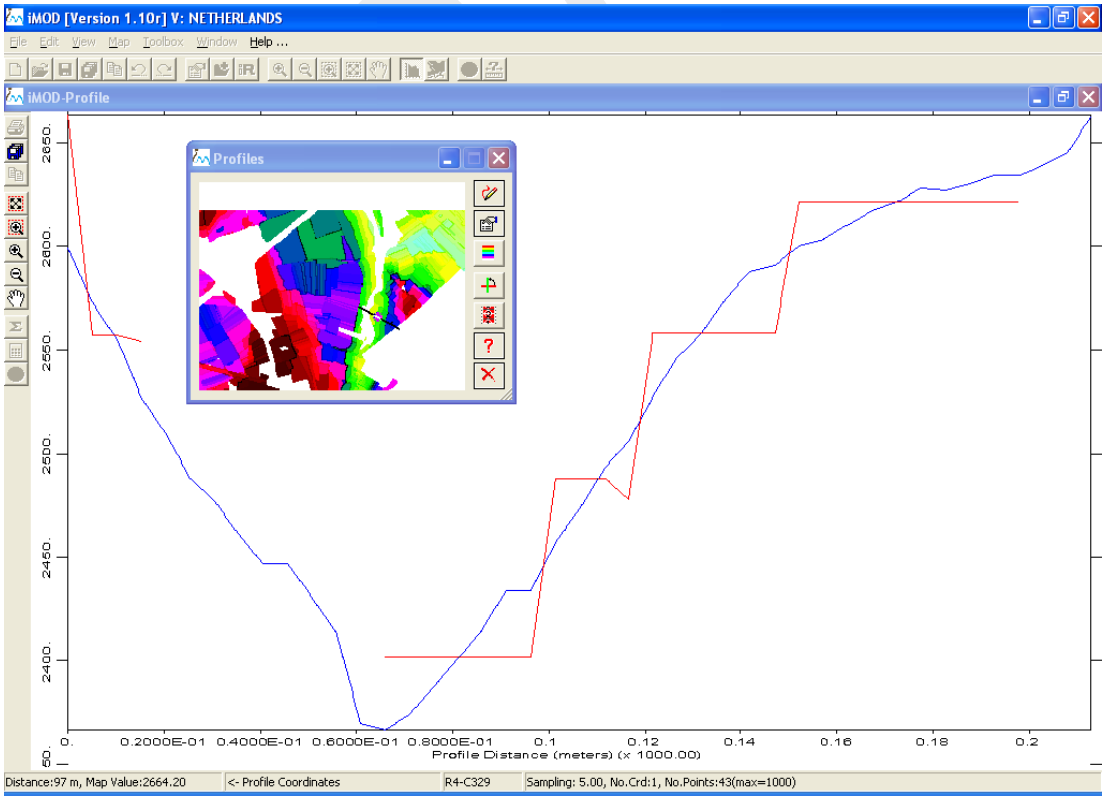
FUNCTION=CREATESOF
IFLOW=2

8.8.4 DRNSURF-Function

The DRNSURF function is used to calculate the drainage level for surface runoff based on info about landuse, DEM and a pointer IDF with info on buildings.

FUNCTION=	DRNSURF
SURFIDF=	Enter the surface level IDF-files that need to be processed, e.g. SURFIDF=D:\DATA\DEM.IDF.
PNTRIDF=	Enter the pointer IDF with built area = 1, e.g. PNTRIDF=D:\DATA\KDSTR.IDF.
LUSEIDF=	Enter the Landuse IDF file, e.g. LUSEIDF=D:\DATA\LGN6.IDF.
NLUSE=	Number of landuse zones to be distinguished.
ILUSE{i}=	Enter the i^{th} landuse code, e.g. ILUSE1 = 3
TDRAINAGE=	Give the percentage of maximum change in drainage level in the area. Used as a treshold.
TSURFLEVEL=	Give the percentage that accounts for the maximum change in surface level. This percentage will be used to define zones.
PERCENTILE=	Give a percentile value (0-100, 50=median) that accounts for the agricultural area that needs to contain drainage within the zones.
OUTIDF=	Enter the name of the output IDF file.
WINDOW=	Enter the coordinates of the window that need to be computed, solely. Enter coordinates of the lower-left corner first and then the coordinates of the upper-right corner, e.g. WINDOW=100000, 400000, 200000, 425000. When WINDOW= is absent, the entire dimensions of the IDF-window will be used.
CELL_SIZE=	Enter the cell size (meter) for the IDF-files that will be created, e.g. CELL_SIZE=25.0.

This figure shows the differences in surface level relative to the defined drainage levels.



Example 1

FUNCTION=DRNSURF
SURFIDF=D:\DATABASE\AHN2.IDF
PNTRIDF=d:\DATABASE\KDSTR_2012.IDF
LUSEIDF=d:\DATABASE\LGN6.IDF
NLUSE=3
ILUSE1=1
ILUSE2=2
ILUSE3=3
TDRAINAGE=50.0
TSURFLEVEL=50.0
PERCENTILE=50.0
OUTIDF=d:\MODEL\DRN\DRN_SL.idf

This example creates an IDF with the drainage elevation for the surface level.

8.9 POSTPROCESSING-FUNCTIONS

8.9.1 GXG-Function

The GXG function calculates the maximum and minimum groundwaterhead during the hydrological year (from 1^{st} April) based on the mean of the three highest, c.q. lowest observed groundwaterheads. The GXG is an indicator used in the Netherlands to indicate the seasonal variation of the groundwaterhead.

FUNCTION=	**GXG**
ILAYER=	Enter the layer numbers to be used in the GxG computation, subsequently; e.g. ILAYER=1,3,6.
NDIR=	Enter the number of folders to be processed repeatedly, e.g. NDIR=10.
SOURCEDIR{i}=	Enter the folder and first part of the filename for all files that need to be used, e.g. SOURCEDIR1=C:\DATA\HEAD. This mean that the GXG function will search for IDF-files that meet the name syntax requirement of C:\DATA\HEAD_{yyyymmdd}_L{ILAY}.IDF.
SURFACEIDF=	Enter a name for the surface, e.g. SURFACEIDF=C:\DATA\AHN.IDF. By default a surface elevation of 0.0m+MSL will be considered.
SYEAR=	Enter the start year (yyyy) for which IDF-files are used, e.g. SYEAR=1998.
EYEAR=	Enter the end year (yyyy) for which IDF-files are used, e.g. EYEAR=2011. This keyword will be read whenever SYEAR is included.
IYEAR=	Specify particular years to be used, e.g. IYEAR=2001,2003,2004. This keyword will be read whenever the keyword SYEAR is absent.
STARTMONTH=	Enter the start month from the which the hydrological year starts. Default STARTMONTH=4.
IPERIOD=	Enter two integers (0 or 1) for each month to express the inclusion of the 14^{th} and 28^{th} of that particular month, e.g. IPERIOD=010101010101010101010101, which mean to use the 14^{th} of each month solely. On default IPERIOD=111111111111111111111111.
ISEL=	Enter a code for the area to be processed: ISEL=1 will compute the entire region ISEL=2 will compute within given polygons; ISEL=3 will compute for those cells in the given IDF-file that are not equal to the *NoDataValue* of that IDF-file.
GENFNAME=	Enter a GEN-filename for polygon(s) for which mean values need to be computed. This keyword is obliged whenever ISEL=2.
IDFNAME=	Enter an IDF-file for which mean values will be computed for those cell in the IDF-file that are not equal to the *NoDataValue* of that IDF-file. This keyword is compulsory whenever ISEL=3
HGLG3=	Indicates whether or not the HG3 and LG3 needs to be computed output per year. HG3(LG3)=the groundwater stage maps of the 3 days with the highest(lowest) stages per year. **Note:** Only the first layer indicated by keyword ILAYER, will be taken into account. Its not possible to get the HG3 and LG3 for more layers within one GXG-calculation. **Note 2:** in case of SYEAR=2011, EYEAR=2013 and startmonth=4, the HG3 and LG3 are computed for year 2011 (=04-2011 till 03-2012) and 2012 (=04-2012 till 03-2013).

Example 1

FUNCTION=GXG
ILAYER=1
NDIR=1
SOURCEDIR1=D:\MODEL\HEAD
SYEAR=1991
EYEAR=2000

This illustrates a simple example of a GxG computation over the years 1991 (actually starts at 14^{th} of

April 1991) until 2000 (actually 28^{th} of March 2000), for all the HEAD* files that are within the folder D:\MODEL. Since the keyword SURFACEIDF is absent, the GxG will be expressed according to 0.0 instead of a true surface level, moreover, ILAY is absent too, but ILAY=1 will be used as default.

Example 2

FUNCTION=GXG
ILAYER=1,2
SURFACEIDF=D:\DATA\AHN.IDF
IYEAR=1994,1995,2000,2001
IPERIOD=000000001111111100000000
ISEL=3
IDFNAME=D:\DATA\ZONE.IDF
NDIR=1
SOURCEDIR1=D:\MODEL\HEAD

This example computes the GxG for the years 1994, 1995, 2000 and 2001 only. This means two hydrological years, namely 14-4-1994-upto 28-3-1995 and 14-4-2000 upto 28-3-2001. In this period the summer months May, June, July, August are included as expressed by the IPERIOD keyword.

8.9.2 WBALANCE-Function

The WBALANCE function calculates the water balance based on the model output for the steady state condition or for a specific period and area. Alternatively, this function can create images, IDF files and/or CVS files from aggregation on existing CSV files.

FUNCTION=	**WBALANCE**
CSVFNAME= (optional)	Enter the name of the CSV file that contains a water balance created previously by this function, e.g. CSVFNAME=D:\MODELRESULTS\WBAL.CSV.
	Use the following keywords whenever CSVFNAME is entered
DIR=	Enter the name of the output folder that will be used to save the resulting pictures, e.g. DIR=D:\MODELRESULTS\FIGURES.
BDGIACT= (optional)	Enter BDGIACT=1 to denote that the budget is present in the output file (IDF, CSV and/or time series/graphical representation image). By default BDGIACT=1 for all budget terms.
BDGIGRP= (optional)	Enter the group numbers for the individual budget terms, e.g. BDGIGRP=1,2,5,5. By default BDGIGRP is a sequence of the numbers 1,2,3 etc.
BDGICLR= (optional)	Enter the colour number (combination of the individual colour red,green en blue - each ranging between 0-255) of the particular budget term, BDGICLR=2443254.
IAVG= (optional)	Enter the method for averaging: ◇ 1 = All Time steps: a single value for all entries; ◇ 2 = Year: a single value per year; ◇ 3 = Months: a single value per month, starting at the first month in the series; ◇ 4 = Hydrological Seasons: two seasons are used, 1) April - September and 2) October - March; ◇ 5 = Decade: a single value per 10 days and the remaining days in that month; ◇ 6 = Hydrological Year: four seasons are used 1) December - February 2) March - May 3) June - August 4) September - November; ◇ 7 = Quarters: a single value per 3 months, starting in January; ◇ 8 = None: all entries remain unchanged, this is the default.
NETFLUX= (optional)	Enter NETFLUX=1 to apply net fluxes in the output, by default NETFLUX=0.
IUNIT= (optional)	Enter INUIT=1 to apply net fluxes in the output, by default IUNIT=0.
LSUM= (optional)	Enter LSUM=1 to aggregate all selected model layers into a single water balance, by default LSUM=0.
ZSUM= (optional)	Enter ZSUM=1 to aggregate all selected zones into a single water balance, by default ZSUM=0.
LAYERS= (optional)	Enter the number of layer(s) to be used for the water balance, e.g. LAYERS=1,2,5. By default ALL layers are selected.
ZONES= (optional)	Enter the number of zones(s) to be used for the water balance, e.g. ZONES=10,23. By default ALL zones are selected.
DATES= (optional)	Enter the number of dates(s) to be used for the water balance, e.g. DATES=20100114,20100128. By default ALL dates are selected.

IOPT= (optional)	Enter the output option: ◇ 1 = Time Series: select this option to display the selected water balance items in a graph, this is the default; ◇ 2 = Graphical Representation: select this option to present the water balance items in a illustrative image; ◇ 3 = Preview Table: select this option to display each value for the water balance items in a table; ◇ 4 = Export to CSV: select this option to export all water balance items into an CSV file; ◇ 5 = IDF per Layer: select this option to export all water balance items into separate IDF files.
OUTPUTFNAME= (optional)	Enter the name of the CSV file to be created whenever IOPT=4, e.g. OUTPUTFNAME=D:\OUTPUT\SUMMARY.CSV.
	Use the following keywords whenever CSVFNAME is NOT entered
NDIR=	Enter the number of folders to be processed repeatedly, e.g. NDIR=10.
SOURCEDIR{i}=	Enter the main source folder for which underlying files need to be used; e.g. SOURCEDIR1=C:\DATA\MODEL. It depends on the following keywords: BAL{i}, BAL{i}ISYS, ILAYERS and SDATE/EDATE, what specific files the WBALANCE function will obtain.
ILAYER=	Enter the layer numbers to be included in the waterbalance, e.g. ILAYER=1,3,6. It is also possible to specify the layers as LAYERS=4:45 to indicate that these layers 4 up to 45 need to used.
SDATE= (optional)	Enter the starting date (yyyymmdd or yyyymmddhhmmss) for which IDF-files are used, e.g. SDATE=19980201 or SDATE=20141231123015 (the latter expresses the 31$^{\text{th}}$ of December 2014 at 12 hours, 30 minutes and 15 seconds.
EDATE= (optional)	Enter the ending date (yyyymmdd or yyyymmddhhmmss) for which IDF-files are used, e.g. EDATE=20111231 or EDATE=20180715084500 (the latter expresses the 15$^{\text{th}}$ of July 2018 at 08 hours, 45 minutes and 00 seconds. In case SDATE is specified, EDATE is compulsory as well.
IYEAR= (optional)	Specify a particular year (within SDATE and EDATE) to be used exclusively, e.g. IYEAR=2001,2003,2005. IYEAR is filled in for all years in-between SYEAR and EYEAR.
NPERIOD= (optional)	Enter a number of periods to be defined to use IDF-file within these periods solely, e.g. NPERIOD=2. NPERIOD=0 by default.
PERIOD{i}= (optional)	Enter a period i (ddmm-ddmm), e.g. PERIOD1=1503-3110 to express the period 15th of March until the 31th of October.
NBAL=	Enter the number of water balance topics, e.g. NBAL=2.
BAL{i}=	Enter for each of NBAL topics one of the folder name, e.g. BDGBND, BDGSFR. iMOD will look for files that are in the folder SOURCEDIR{i}\BAL{i}\BAL{i}_{time}_{layer}.IDF. whenever SDATE/EDATE is absent, {time} is the keyword STEADY_STATE. Repeat BAL{i} for NBAL times. E.g., whenever BAL1=BDGWEL and the simulation is steady state, the following file is appropriate: SOURCEDIR{i}\BDGWEL\BDGWEL_STEADY-STATE_L1.IDF. For transient simulations, the iMOD Batch function will search for SOURCEDIR{i}\BDGWEL\BDGWEL_????????_L1.IDF and SOURCEDIR{i}\BDGWEL\BDGWEL_????????????_L1.IDF files. After that, the selected set of files will be matched against the given SDATE and EDATE keywords, and if necessary against NPERIOD.

BAL{i}ISYS= (optional)	Enter the number of systems to be included in the water balance, e.g. BAL1ISYS=1,2,3. This mean to add the systems 1,2 and 3 for the first entered water balance item, specified by BAL1. E.g. iMOD will look for files as BDGDRN_STEADY-STATE_SYS1_L1.IDF. By default, no systems will be distinguished and iMOD will look for files as BDGDRN_STEADY-STATE_L1.IDF.
OUTPUTNAME{i}=	Enter the output filename (*.TXT, *.CSV or *.IPF), e.g. OUTPUT-NAME1=C:\DATA\HEAD\WBAL_MIPWA.CSV
ISEL= (optional)	Enter a code for the area to be processed: ISEL=1 will compute the entire region ISEL=2 will compute within given polygons; ISEL=3 will compute for those cells in the given IDF-file that are not equal to the *NoDataValue* of that IDF-file. By default ISEL=1.
GENFILE= (optional)	Enter a GEN-filename for polygon(s) for which mean values need to be computed. This keyword is obliged whenever ISEL=2.
IDFNAME= (optional)	Enter an IDF-file for which mean values will be computed for those cell in the IDF-file that are not equal to the *NoDataValue* of that IDF-file. This keyword is compulsory whenever ISEL=3
WBEX= optional)	Enter WBEX=1 to generate interconnected flux between the zones. This option is only valid whenever the flux terms BDGFRF and BDGFFF are active. By default WBEX=0 and none of the interconnected fluxes is computed.

Example 1

FUNCTION=WBALANCE
NBAL=3
BAL1=BDGFRF
BAL2=BDGFFF
BAL3=BDGFLF
ILAYER=3
NDIR=1
ISEL=2
GENFILE=D:\MODEL\zone.gen
SOURCEDIR1=D:\MODEL
OUTPUTNAME1=D:\MODEL\WBAL.TXT

The above mentioned simple example will give a waterbalance for the BDGFRF, BDGFFF and BDGFLF, respectively, for modellayer 3.
The IDF-files will be D:\MODEL\BDGFRF\BDGFRF_STEADY-STATE_L3.IDF;
D:\MODEL\BDGFFF\BDGFFF_STEADY-STATE_L3.IDF; and
D:\MODEL\BDGFLF\BDGFLF_STEADY-STATE_L3.IDF.
The result is written in D:\MODEL\WBAL.TXT.

Example 2

FUNCTION=WBALANCE
NBAL=2
BAL1=BDGRIV
BAL1SYS=1,2
BAL2=BDGDRN
ILAYER=1,2
SDATE=19900101
EDATE=20001231
IYEAR=1990,1995,1997,2000
NPERIOD=1
PERIOD1=0104-3107
ISEL=2

GENFNAME=D:\DATA\ZONES.GEN
NDIR=2
SOURCEDIR1=D:\MODEL
SOURCEDIR2=D:\SCENARIO
OUTPUTNAME1=D:\OUTPUT\WBAL_MODEL.CSV
OUTPUTNAME2=D:\OUTPUT\WBAL_SCENARIO.CSV

The example above will compute a waterbalance for two modellayers (1,2) for the budgetfiles BDGRIV*SYS1 and BDGRIV*SYS2 and BDGDRN in the period from 1^{th} of April until the 31^{th} of July for the years 1990,1995,1997,2000. The waterbalance will be summed for the zones that are described by the polygon(s) inside the file ZONES.GEN. Finally, the computation will be executed twice, for those results in D:\MODEL and those in D:\SCENARIO. Results are stored in the folder D:\OUTPUT.

8.9.3 PWTCOUNT-Function

Use this iMODFLOW post-processing function to count the number of moments where a PWT situation occurs.

FUNCTION=	PWTCOUNT
SDLIDF=	Enter the name of the IDF file containing the layer number of the first Aquitard.
ILAYIDF=	Enter the name of the IDF file containing the deepest modellayers to be processed.
SDATE=	Enter the starting date (yyyymmdd) for which IDF-files are used.
EDATE=	Enter the ending date (yyyymmdd) for which IDF-files are used.
SOURCEDIR{i}=	Enter the folder and wildcard for all files that need to be used.
OUTPUTIDF=	Enter the name of the IDF file that contains calculated sum of moments where PWT situations occur.

Example 1

FUNCTION=PWTCOUNT
SDLIDF=C:\RESULTS2\PWT\SDL_LAYER.IDF
ILAYIDF=C:\RESULTS2\PWT\PWT_LAYER.IDF
SDATE=19980201
EDATE=20111231
SOURCEDIR=C:\RESULTS\PWT\PWT.IDF*
OUTPUTIDF=D:\RESULTS\PWT_COUNT.IDF

The above mentioned examples creates the IDF file PWT_COUNT.IDF based on a timeseries of PWT result files in directory C:\DATA\

8.9.4 IDFTIMESERIE-Function

Use this function to generate timeseries out of IDF-files that have the notation {item}_yyyymmdd_l{ilay}.idf. These are IDF-files that yield from a normal iMODFLOW simulation.

FUNCTION=	IDFTIMESERIE
IPF1=	Enter the name of an IPF file that contains the locations of the measurements, e.g. IPF1=D:\DATA\MEASURE.IPF.
IPF2=	Enter the name of an IPF file that will be used to store the computed time series, e.g. IPF2=D:\IMOD\MODEL.IPF.
ILAY=	Enter the modellayer, e.g. ILAY=2.
SOURCEDIR=	Enter the directory name of the folder that contains the specific files + the first (similar) part of the name of the files, e.g. SOURCEDIR=D:\MODEL\HEAD\HEAD. This will yield IDF-files that belong to D:\MODEL\HEAD\HEAD_{yyyymmdd}_L{ilay}.IDF .
SDATE= (optional)	Enter the start date of the time series to be computed, e.g. SDATE=19700803000000 to express the 3$^{\mathrm{rd}}$ of August 1970. By default SDATE=0 and will not be used, the series starts at the earliest file that confirms the SOURCEDIR.
EDATE= (optional)	Enter the end date of the time series to be computed, e.g. SDATE=20120601133015 to express the 1$^{\mathrm{st}}$ of June 2012 at 13 hours, 30 minutes and 15 seconds. By default EDATE=0 and will not be used, the series ends at the latest file that confirms the SOURCEDIR.
LABELCOL= (optional)	Enter the column to be used for labelling the associated text files. Default LABELCOL=0 and will not be used.

Example 1

FUNCTION=IDFTIMESERIE
IPF1=D:\MODEL\HEAD_TSERIES.IPF
IPF2=D:\MODEL\HEAD_TSERIES_IMODBATCH.IPF
ILAY=1
SDATE=19500101000000
EDATE=20120101123005
SOURCEDIR=D:\RESULT\HEAD\HEAD

The example above will yield time series from the results in D:\RESULT\HEAD_*.IDF for the period between the 1$^{\mathrm{st}}$ of January 1950 and the 1$^{\mathrm{st}}$ of January 2012 at 12 hours, 30 minutes and 5 seconds.

Example 2

FUNCTION=IDFTIMESERIE
IPF1=D:\MODEL\HEAD_TSERIES.IPF
IPF2=D:\MODEL\HEAD_TSERIES_IMODBATCH.IPF
ILAY=1
SDATE=19500101000000
EDATE=20120101000000
SOURCEDIR=D:\RESULT\HEAD\HEAD
LABELCOL=3

The example above differs for LABELCOL only. The 3$^{\mathrm{rd}}$ column will be used to generate the name of the text file that stores the time series.

8.9.5 IPFRESIDUAL-Function

Use this function to calculate residuals based on IPF files containing statistics.

FUNCTION=	**IPFRESIDUAL**
NIPF=	Enter the number of IPF files to be handled.
IPFFILE{i}=	Enter the name of the i^{th} IPF file containing measurement and computed values, or refer to associated TXT files with time series of measurements and computed values.
OUTNAME=	Enter the name of the output filename, the statistics will be written in this file. . Also, per IZONE and per LAYER, a separate IPF file will be created called residual_ILAY{i}_IZONE{j}
ILCOL{i}= (optional)	Enter the column number in the IPF{i} that represents the model layer, by default ILCOL{i}=3.
IMCOL{i}= (optional)	Enter the column number in the IPF{i} that represents the measurement. In the case that the measurement is given by associated TXT files, it is not necessary to enter this keyword, in the other case the default value is IMCOL{i}=3.
IHCOL{i}= (optional)	Enter the column number in the IPF{i} that represents the computed head. In the case that the computed head is given by associated TXT files, it is not necessary to enter this keyword, in the other case the default value is IHCOL{i}=3.
W_TYPE{i}= (optional)	Give whether the load is entered as variance (W_TYPE1=1) or weights (W_TYPE1=2). Whenever variances are entered, a weight is computed internally as: $w = \frac{1}{\sqrt{v}}$ in which w is the weight and v is the variance. By absent of the keyword, so NO weight are expected in that case (in fact the weight will be 1.0 for all locations).
IWCOL{i}= (optional)	Enter the column number in the IPF{i} that represents the variance or weight, by default IWCOL{i}=3 and is only read whenever W_TYPE{I} $\neq$ 0.0.
SDATE= (optional)	Enter the starting date for which statistics need to be gathered, e.g. SDATE=20020726 to express the 26$^{\text{th}}$ of July 2002. By default SDATE=-10^{10}.
EDATE= (optional)	Enter the starting date for which statistics need to be gathered, e.g. EDATE=20041226 to express the 26$^{\text{th}}$ of December 2004. By default EDATE=10^{10}.
POINTERIDF= (optional)	Enter the name of an IDF file containing zones, those zones will be used to distinguish in different statistics per zone.
	NZONE= Enter the number of zones in the IDF file to be used, e.g. NZONE=3 will use three zone form the IDF files mentioned at POINTERIDF.
	IZONE{i}= Enter the value in the IDF file for the i^{th} zone, e.g. IZONE1=10 denotes that the first zone will be number 10 from the POINTERIDF.
ICOLLECT= (optional)	Specify ICOLLECT=1 to add the associated TXT files to the created IPF files, by default ICOLLECT=0.
HNODATA= (optional)	NodataValue of the computed head, by default HNODATA=-999.99. This value will . be only used for IPF files with W_TYPE{i}=0

Example

FUNCTION=IPFRESIDUAL
NIPF=2
POINTERIDF=D:\MODEL\POINTER.IDF
IPF1=D:\MODEL\HEAD_TSERIES_1.IPF
IPF2=D:\MODEL\HEAD_TSERIES_2.IPF
W_TYPE2=1
NZONE=2
IZONE1=5
IZONE2=6
OUTNAME=D:\RESULTS\STATISTICS.TXT

The example above will give a file containing all the residual values per given IPF-file based on the predefined conditions.

8.9.6 PLOTRESIDUAL-Function

Use this function to make a scatter plot or histogram plot of the observations and calculated heads and residuals (calculated-observed) from the iPEST-output data file(s).

FUNCTION=	**PLOTRESIDUAL**
INPUTFILE=	You can either enter the iPEST output text file name or an IPF file with appropriate information, such as x, y, measurement, computed head and weight values. For IPF files with associated text files, you need to specify ITRANSIENT=1 .
IXCOL= (optional)	Enter the column number in the IPF file for the X-coordinates, by default this IXCOL=1.
IYCOL= (optional)	Enter the column number in the IPF file for the Y-coordinates, by default this IYCOL=2.
IMCOL= (optional)	Enter the column number in the IPF file for the measurement, for TRANSIENT=1, it means the column in the associated text file, by default this IMCOL=2.
IHCOL= (optional)	Enter the column number in the IPF file for the computed head, for TRANSIENT=1, it means the column in the associated text file, by default this IHCOL=3.
IWCOL= (optional)	Enter the column number in the IPF file for the weight values, by default this IWCOL=0 and the weight values are assumed to be all equal to 1.0.
ILCOL= (optional)	Enter the column number in the IPF file for the layer values, by default this ILCOL=0 and the layer is assumed to be all equal to 1.
IPLOT=	Choose the preferred plot type (see for examples further this function description): ◇ 1=Scatter plot; ◇ 2=Histogram plot; ◇ 3=IPF file.
BMPNAME= (IPLOT=1,2)	Give the name of the output plot file (*.BMP. *.PNG, *.JPG or *.PCX), e.g. d:\SCATTERPLOT_LAYER2.BMP.
IPFNAME= (IPLOT=3)	Give the name of the output IPF file (*.IPF), e.g. d:\RESIDUAL.IPF.
ITRANSIENT= (optional)	Choose the type of input file you use: ◇ 0=Steady-state input (this is the default value ITRANSIENT=0); ◇ 1=Transient input. A transient input file contains date values in the first column whenever the INPUTFILE is an iPEST output file, in the case the INPUTFILE is an IPF-file, the measurement and computed heads are expected to be in associated text files.
ILAYER= (optional)	Choose the layers you prefer to plot, e.g. ILAYER=1,4,8 plots data points related to layer 1, 4 and 8. On default all layers are plotted.
IIPF= (optional)	Choose the IPF-file(s) for which the data needs to be plotted, e.g. IIPF=1,3 plots the data points for the first and 3$^{\mathrm{rd}}$ given IPF-file from the input file. This keyword is not used in case the INPUTFILE is an IPF file.
WC1= (optional)	Specify the lower limit of a weight value to be included in the selection for the statistics, e.g. WC1=1000.0 means that only points with weight value larger than 1000 will be included in the statistics. By default WC1 is absent, so all will be included.
WC2= (optional)	Specify the upper limit of a weight value to be included in the selection for the statistics, e.g. WC2=5000.0 means that only points with weight value smaller than 5000 will be included in the statistics. By default WC2 is absent, so all will be included.

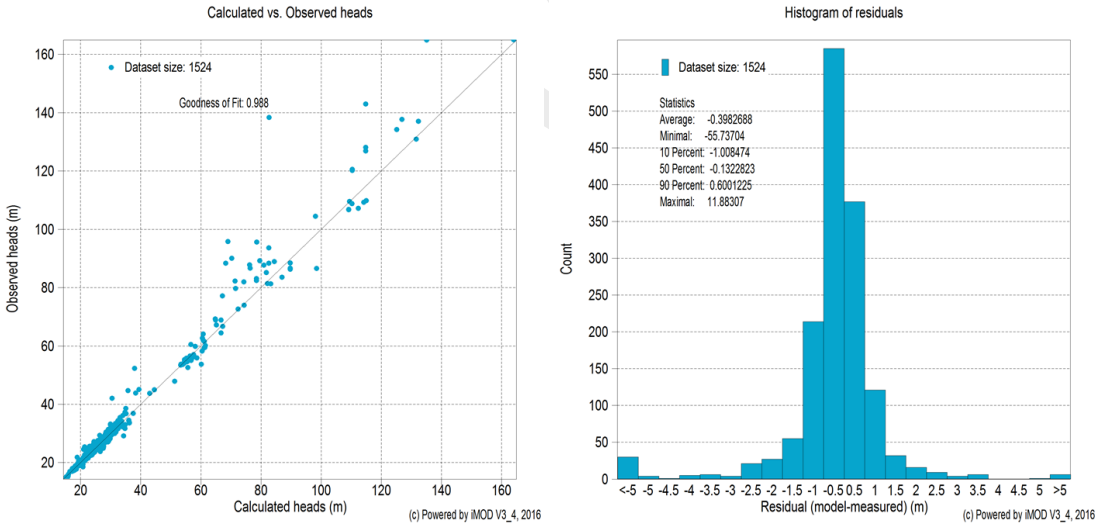
HCLASSES= (optional)	Specify the classes for the histogram, by default this keyword is absent and the following classes are used: -10.0E10,-5.0,-4.5,-4.0,. . . ,4.0,4.5,5.0,10.0E10. However, it is possible to specify a user defined class via e.g. HCLASSES=-10.0,-1.0,-0.5,0.5,1.0,10. In this case the first class is for > -10.0 and ≤ -1.0, the second for > -1.0 - and ≤ -0.5 and so on.
IWEIGHT= (optional)	Choose whether you will include the weight factors in the calculation or not. IWEIGHT=0 the weight factor is not included, IWEIGHT=1 the weight factor is included. In case the weight factor equals 0 the related data point will not be plotted.

Example

FUNCTION=PLOTRESIDUAL
INPUTFILE=D:\LOG_PEST_RESIDUAL_10V2.6.66.TXT
IPLOT=1
BMPNAME=d:\SCATTERPLOT.BMP
ITRANSIENT=1
ILAYER=16
IIPF=1
IWEIGHT=1

This example makes a scatter plot with the name SCATTERPLOT.BMP of the transient data available in the LOG_PEST_RESIDUAL_10V2.6.66.TXT file for a selection of layer 16 and the first IPF-file. The plotted values are multiplied with the given weight factor.

Result of PLOTRESIDUAL, left a scatter plot (IPLOT=1) , right a histogram (IPLOT=2)



8.10 WELL-FUNCTIONS

8.10.1 DEVWELLTOIPF-Function

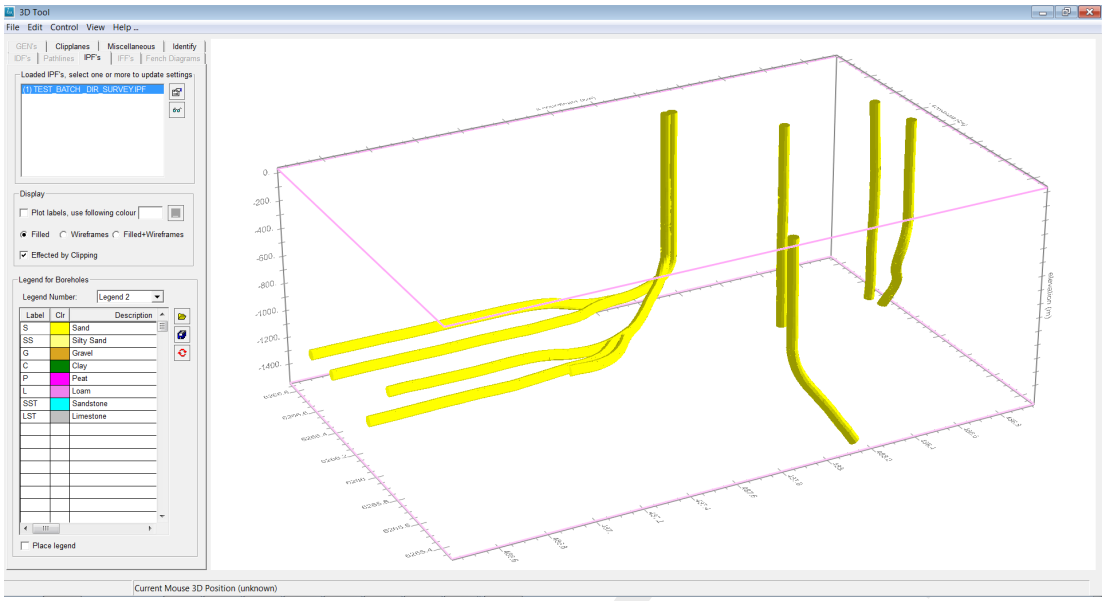
The function DEVWELLTOIPF converts Deviated Wells, described in CSV format, to an IPF file.

FUNCTION=	**DEVWELLTOIPF**
CSVFNAME=	Enter the name of an CSV file with minimal 6 columns that represents x- and y coordinates, azimuth and inclination, e.g. CSVFNAME=D:\DATA\AGS.CSV.
IPFFNAME=	Enter the name of an IPF file that need to be created for the data in the CSVFNAME„ e.g. IPFFNAME=D:\DATA\AGS.IPF. For each unique borehole in the CSV file, another point will be created in the IPF. The associated TXT files for the boreholes will be stored in the same folder as the IPFFNAME.
NCOL (optional)	Enter the column number in the CSV file that represents the unique NAME of the borehole, e.g. NCOL=4. By default NCOL=1. The NAME of the borehole will be used to create the file name for the associated TXT files. Therefore, the character set ‘: /’ will be replaced by a ‘_’ (a space is part of the replacement as well, denoted as the second character in the set).
XCOL (optional)	Enter the column number in the CSV file that represents the x-coordinate, e.g. XCOL=4. By default XCOL=2.
YCOL (optional)	Enter the column number in the CSV file that represents the y-coordinate, e.g. YCOL=6. By default YCOL=3.
ZCOL (optional)	Enter the column number in the CSV file that represents the z-coordinate, e.g. ZCOL=6. By default ZCOL=0 that means that the z-coordinate will be 0.0 m+MSL.
DCOL (optional)	Enter the column number in the CSV file that represents the depth, e.g. DCOL=6. By default DCOL=5. Remember that the DEPTH is measured as the net distance (meter) through the borehole.
ACOL (optional)	Enter the column number in the CSV file that represents the azimuth, e.g. ACOL=8. By default ACOL=6. The azimuth is defined as the angle with the z-axes measured clockwise with a zero angle pointing to the north and 90 to the east.
ICOL (optional)	Enter the column number in the CSV file that represents the inclination, e.g. ICOL=12. By default ICOL=7. The inclination is defined as the angle from the surface (xy-plane) downwards by a positive angle whereby 90.0 degrees is perpendicular downwards.
NLCOL (optional)	Enter the number of labels to be additional added to the TXT files of the boreholes, e.g. NLCOL=2. By default NLCOL=1 and a default label column is added (see LCOL{i}).
LCOL{i} (optional)	Enter the column number in the CSV file that represents the label, e.g. LCOL1=4. By default LCOL1=0 and a default label is used. The label can be used to colour particular trajectories along a borehole differently. If this keyword is absent, iMOD will add a default label ‘S’.

Example 1

FUNCTION=DEVWELLTOIPF
CSVFNAME=D:\AGS.CSV
IPFFNAME=D:\IPF.IPF

This example, converts the columns in the CSV file via the default columns setting to an IPF file, the outcome in 3D could look like the following image.


3D Tool
File Edit Control View Help ...
GENs Clipplanes Miscellaneous Identify
IDFs Pathlines IPFs IFFs Fench Diagrams
Loaded IPF's, select one or more to update settings
(1) TEST_BATCH_DIR_SURVEY.IPF
Display
Plot labels, use following colour
Filled Wireframes Filled+Wireframes
Effected by Clipping
Legend for Boreholes
Legend Number: Legend 2
Label Clr Description
S Sand
SS Silty Sand
G Gravel
C Clay
P Peat
L Loam
SST Sandstone
LST Limestone
Place legend
Current Mouse 3D Position (unknown)

8.10.2 ASSIGNWELL-Function

The ASSIGNWELL function reads a geological model (TOP/BOT) in order to assign well screens to the right formation layer.

FUNCTION=	**ASSIGNWELL**
IPFFILE_IN=	Give IPF file containing WELL or MEASUREMENT information.
IPFFILE_out=	Give IPF file outputfile.
IXCOL=	Enter the column number in the IPF file IPF{i} that represents the x coordinate, e.g. IXCOL=4. By default IXCOL=1.
IYCOL=	Enter the column number in the IPF file IPF{i} that represents the y coordinate, e.g. IYCOL=6. By default IYCOL=2.
IDCOL=	Enter the column number in the IPF file IPF{i} that represents the extraction rate of the well, e.g. IQCOL=12. By default IQCOL=3.
IZ1COL=	Enter the column number in the IPF file IPF{i} that represents the top of the well screen, e.g. ITCOL=4. By default ITCOL=4.
IZ2COL=	Enter the column number in the IPF file IPF{i} that represents the bottom of the well screen, e.g. IBCOL=6. By default IBCOL=5.
NFORMATIONS=	Enter the number of formations
FORMATION{i}=	Enter the number of the column in the IPF for the ..... of the i^{th} formation
TOP_L{i}=	Enter the IDF for the i^{th} modellayer that represents the top of modellayer *i*, e.g. TOP_L1=D:\INPUT\TOP_L1.IDF.
BOT_L{i}=	Enter the IDF for the i^{th} modellayer that represents the bottom of modellayer *i*, e.g. BOT_L1=D:\INPUT\BOT_L1.IDF.

Example 1

FUNCTION=ASSIGNWELL
IPFFILE_IN=D:\DATA\WELL.IPF
IPFFILE_OUT=D:\DATA\WELL_ASSIGNED.IPF
NFORMATIONS=2
FORMATION1=1
FORMATION2=2
TOP_L1=D:\GEOHYDROLOGY\TOP1.IDF
BOT_L1=D:\GEOHYDROLOGY\BOT1.IDF
TOP_L2=D:\GEOHYDROLOGY\TOP2.IDF
BOT_L2=D:\GEOHYDROLOGY\BOT2.IDF

Above an example is given how to divide well filters in the file WELL.IPF over 2 model layers resulting in the file WELL_ASSIGNED.IPF.

8.10.3 MKWELLIPF-Function

The MKWELLIPF function computes the extraction strength for each well based on a weighed value according to their length and permeability of the penetrated model layer. At the end of the procedure iMOD echoes a summary of the total and average extraction per model layer.

FUNCTION=	**MKWELLIPF**
NLAY= (optional)	Enter the number of layers from which well may be organized, e.g. NLAY=7, by default NLAY=0 which means that only extraction rates are computed from the associated TXT files. Following keywords are needed/optional whenever NLAY greater than zero
TOPIDF{i}=	Enter the name of an IDF-file that represents the top elevation of the i^{th} modellayer, e.g. TOPIDF1=D:\MODEL\TOP1.IDF.
BOTIDF{i}=	Enter the name of an IDF-file that represents the bottom elevation of the i^{th} modellayer, e.g. BOTIDF3=D:\MODEL\BOT_LAYER3.IDF.
KDIDF{i}= (optional)	Enter the name of an IDF-file that represents the transmissivity of a particular i^{th} modellayer, e.g. KDIDF2=D:\MODEL\TRAN_L2.IDF.
CIDF{i}= (optional)	Enter the name of an IDF-file that represents the vertical resistance between two adjacent modellayers i and i+1, e.g. CIDF1=D:\MODEL\C_L1.IDF.
ITCOL= (optional)	Enter the column number in the IPF file IPF{i} that represents the top of the well screen, e.g. ITCOL=4. By default ITCOL=4.
IBCOL= (optional)	Enter the column number in the IPF file IPF{i} that represents the bottom of the well screen, e.g. IBCOL=6. By default IBCOL=5.
MINKHT= (optional)	Specify the minimum horizontal transmissivity (m^2/d) that will receive a well. By default MINKHT=0.0 m^2/day. This parameters is used only whenever values are entered for KDIDF{i}.
MINKD= (optional)	Specify the horizontal transmissivity that is used te define the model layer of the well. The first model layer with a transmissivity of more than the specified MINKD, will receive the complete well. By default MINKD=0.0 m^2/day. This parameters is used only whenever values are entered for KDIDF{i} and will be active for those wells that cannot be assigned due to missing values for ITCOL and IBCOL.
IMIDF= (optional)	Whenever IMIDF=0, the mid of a well screen is computed by the top and bottom screen heights if both available (not equal to the parameter FNODATA). Whenever IMODF=1, the mid of the screen is equal to the top of the screen whenever the bottom height might be absent, and equal to the bottom whenever the top is absent. It both are available, the computation of the mid of the well screen is equal to the method described by IMIDF=0. By default IMIDF=0.
ICLAY= (optional)	Whenever wells might fall completely in an aquitard (in between two model layers), specify ICLAY=1 to shift the well vertically to that model layer that is most nearby (above- or beneath), this is the default value. Specify ICLAY=0 and the well will be removed whenever completely in an aquitard.
FNODATA= (optional)	Specify the *NoDataValue* for the top and bottom of the well screen, denoted by ITCOL and IBCOL. By default FNODATA=-99999.0, values equal to this will be discarded.
	Rest of keywords are applicable for all values of NLAY.
NIPF	Enter the number of IPF files to be organized, e.g. NIPF=3.
IPF{i}=	Enter the name for the i^{th} IPF file, e.g. D:\DATA\WELL.IPF. The resulting IPF files will be save in the folder D:\DATA\WELL\IMOD_MKIPF_WELLS_L*.IPF for each model layer that has extraction rate $<>$ 0.0.
IXCOL= (optional)	Enter the column number in the IPF file IPF{i} that represents the x coordinate, e.g. IXCOL=4. By default IXCOL=1.
IYCOL= (optional)	Enter the column number in the IPF file IPF{i} that represents the y coordinate, e.g. IYCOL=6. By default IYCOL=2.

IQCOL= (optional whenever ISS=0)	Enter the column number in the IPF file IPF{i} that represents the extraction rate of the well, e.g. IQCOL=12. By default IQCOL=3.
ISS= (optional)	This flags determines whether an time average extraction volume need to be computed for a specified period of time, for that case ISS need to be 1. By default ISS=0 and an average value is computed for the time series as a whole.
SDATE= (if ISS=1)	Specify a starting date (YYYYMMDD) from which the determination of a well strength/head will be computed. This keyword is compulsory whenever ISS=1.
EDATE= (if ISS=1)	Specify an end date (YYYYMMDD) from which the determination of a well strength/head will be computed. This keyword is compulsory whenever ISS=1.
HNODATA= (optional)	Specify the *NoDataValue* for the extraction rate, values equal to this will be discarded. By default HNODATA=0.0.

The IPF file IMOD_MKIPF_WELLS_ALL.IPF contains all rows from the original IPF. This file is easier to analyse whether the well screen assigned have been computer properly. There is an attribute ERROR_CODE in the IPF that denotes the way the well has been assigned. The following codes are applicable; "@" means that the well is assigned to a nearby model layer; "#" means that the well could not be assigned; and "-" means that the well has been assigned appropriately. The IPF file IMOD_MKIPF_WELLS_UNASSIGNED.IPF contains all rows from the original IPF that could not be assigned to model layer.

Methodology

The following steps are carried out for each individual record in the IPF file (IPF{i}):

1 Compute the individual length of the well screen between the ITCOL and IBCOL into well screen segments, that penetrate any model layer. Well screen segments that are above the surface elevation (TOPIDF1) or below the lowest bottom elevation (BOTIDF{NLAY}) will be clipped off;
2 Compute the horizontal permeability (k-value) for all model layers that are penetrated by the remaining well screen segments. Assign a ratio to all well screen segments based on their individual length multiplied by the k-values of the corresponding model layer divided by their total summed value;
3 Correct any ratio for a mismatch between the centre of the penetrating model layer z_c and the vertical midpoint of the well screen segment f_c, by:

$$f = 1.0 - \frac{|z_c - f_c|}{0.5\Delta z},$$

where Δz is the thickness of the corresponding aquifer.
4 Remove ratio that are smaller than 5%.
5 If in aquitard, move it to the nearest aquifer above or below the aquitard, only whenever ICLAY=1;
6 If nothing in model, whenever system on top of model, put them in first model layer with thickness and permeability larger than MINKH.

Example 1

FUNCTION=MKWELLIPF
NLAY=3
TOPIDF1=D:\DATA\TOP1.IDF
TOPIDF2= D:\DATA\TOP2.IDF
TOPIDF3= D:\DATA\TOP3.IDF
BOTIDF1= D:\DATA\BOT1.IDF
BOTIDF2= D:\DATA\BOT2.IDF
BOTIDF3= D:\DATA\BOT3.IDF
KDIDF1= D:\DATA\KD1.IDF
KDIDF2= D:\DATA\KD2.IDF
KDIDF3= D:\DATA\KD3.IDF
CIDF1=D:\DATA\C1.IDF
CIDF2=D:\DATA\C2.IDF
NIPF=1
IPF1=D:\DATA\WELL.IPF

The example above, will classify each location in the IPF file D:\DATA\WELL.IPF according their length and associated transmissivity, within any penetrating modellayer.

Example 2

FUNCTION=MKWELLIPF
NLAY=3
TOPIDF1=D:\DATA\TOP1.IDF
TOPIDF2= D:\DATA\TOP2.IDF
TOPIDF3= D:\DATA\TOP3.IDF
BOTIDF1= D:\DATA\BOT1.IDF
BOTIDF2= D:\DATA\BOT2.IDF
BOTIDF3= D:\DATA\BOT3.IDF
KDIDF1= D:\DATA\KD1.IDF
KDIDF2= D:\DATA\KD2.IDF
KDIDF3= D:\DATA\KD3.IDF
CIDF1=D:\DATA\C1.IDF
CIDF2=D:\DATA\C2.IDF
IXCOL=1
IYCOL=2
ITCOL=3
IBCOL=4
IQCOL=8
NIPF=3
IPF1=D:\DATA\INDUSTRY.IPF
IPF2= D:\DATA\DRINKINGCOOPERATION.IPF
IPF3= D:\DATA\AGRICULTURE.IPF

The example above, will classify each location in three IPF files according their length and associated transmissivity, within any penetrating model layer. The content of the IPF files is different than the default IXCOL, IYCOL, ITCOL, IBCOL and IQCOL column identifications, and therefore added here.

8.11 BMPTILING-Function

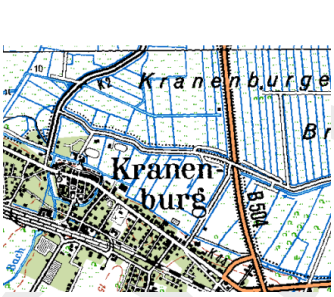
Use this function to create a set of topographical bitmaps out of a single one, to be used as background topology. The function assumes that one single bitmap needs to be split in small “tiles” for different resolutions. Those different resolutions can be used at different zoom levels to maintain a high performance while plotting these bitmaps on the graphical canvas.

FUNCTION=	BMPTILING
BMPFILE=	Enter the name of a BMP file, e.g. D:\DATA\AIRPHOTO.BMP.The function assumes that a worldfile (*.BMPW) is accompanied by the bitmap for the syntax of a worldfile.
OUTPUTFOLDER=	Enter the name of the folder that will store all the generated bitmaps (tiles) at different resolutions (5), e.g. D:\DATA\BMP. The function will generate a *.CRD file too and associated TXT files. Referring to the *.CRD file from the iMOD preference file will ensure that the bitmaps can be used directly in iMOD.

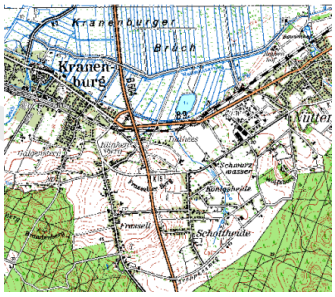
Example

FUNCTION=BMPTILING
BMPFILE=D:\DATA\AIRPHOTO.BMP
OUTPUTFOLDER=D:\DATA\BMP

The example above generates different tiles from the bitmap AIRPHOTO.BMP in the folder D:\DATA\BMP. In this folder the file BMP.CRD will be stored that can be used to direct with a keyword TOP25 in the iMOD preference file.



Tile 1



Tile2

Tile 3

Tile 4

Tile 5

8.12 PLOT-Function

The *Plot* function can be used to construct figures that are normally displayed on the graphical display of iMOD.

FUNCTION=	PLOT		
OUTFILE=	Enter the name of the output filename. The format of the image depends on the extension of the filename: ◇ *.BMP : Windows Bitmap; ◇ *.PCX : ZSoft PC Paintbrush; ◇ *.PNG : Portable Network Graphic image; ◇ *.JPG : JPEG/JFIF image.		
IDFFILE= (optional)	Enter the name of an *.IDF-file that needs to be plotted, for example IDFFILE=D:\AHN.IDF.		
	IDFLEGFILE= (optional)	Enter the name of a *.LEG file that needs to be used for colouring the *.IDF-file. If no IDFLEGFILE is given, iMOD will apply a default iMOD Legend based on a linear distribution of IDF values.	
	IDFLEGTXT (optional)	Enter the text to be added to the legend, e.g. IDFLEGTXT="Transmissivity (m2/day)". By default no legend text will be displayed.	
	IDFSTYLE= (optional)	Specify whether the IDF need to be displayed as gridvalues, contourlines, vectors or all three combined. For example IDFSTYLE=111 will plot the IDF-file by all styles, default IDFSTYLE=100. Use the following syntax to specify the style to be used, any combination is possible:	
		100	IDF will be displayed as filled rectangles.
		010	IDF will be contoured.
		001	IDF will be displayed as arrows based on the gradient in the IDF file.
IPFFILE= (optional)	Enter the name of an IPF file to be plotted, for example IPFFILE=D:\DATA.IPF. iMOD will use the first and second column in the IPF for the x- and y coordinate and displays the point in red circles. Use the other keywords to change these settings.		
	IPFXCOL= (optional)	Specify the column to be used for the x-coordinate, by default IPFXCOL=1	
	IPFYCOL= (optional)	Specify the column to be used for the y-coordinate, by default IPFYCOL=2	
	IPFHCOL= (optional)	Specify the column to *scale* the dots that have larger values than others, those will be displayed as an increased marker symbol. iMOD will scale the values based on the entries at the keyword IPHCOL_M. By default IPFHCOL=0.	
	IPFHCOL_M= (optional)	Specify the methodology to scale the symbol, by default IPFHCOL=0 and no scaling applies. iMOD displays them, such that small symbol sizes will be plotted upon large symbols, choose a method from the following:	
		1	iMOD scales the values for IPFHCOL linearly from large up to small using 4 sizes (sizes 1,2,3 and 4).
		2	iMOD scales the values for IPFHCOL linearly from large up to small using 4 sizes (sizes 1,2,3 and 4) within the range of the entered legend.
		3	iMOD scales the absolute values for IPFHCOL linearly from large up to small using 4 sizes (sizes 1,2,3 and 4).
		4	iMOD scales the symbols using the direct value for IPFHCOL, sizes need to be positive, negative entries will be as treated as positive sizes.

	IPFASSFILES= (optional)	Specify whether to plot the graph (e.g. timeseries, boreholes) described in the associated TXT files linked by the IPFFILE. Specify IPFASSFILES=1 to plot a simple graph (without axes) and IPFASSFILES=2 to plot a advanced graph with axes. By default IPFASSFILES=0 and no associated text files are displayed.	
	IPFASSFILES_ ALL= (optional)	Specify whether to plot the content of all associated TXT files in a single image (IPFASSFILES_ALL=1) or alternatively, plot each TXT file in a separate image (IPFASSFILES_ALL=2). In that case the OUTFILE will be created automatically by the name of the individual points given by the column denoted by the IEXT, see section 9.7 for more information on IPF files.	
	IPFSTYLE= (optional)	Specify the style to be used to plot the points. Use the following options:	
		0	Use this style to display the points as solid circles.
		1	Use this style to colour the points by the labels in the column IPFLCOL in combination with the specified legend in LEGFILE.
		Specify the following whenever IPFSTYLE=1	
		IPFLEGFILE= (optional)	Enter the appropriate legend file to be used, e.g. IPFLEGFILE=D:\RESIDUAL.LEG
		IPFLCOL=	Specify the column in the IPF file for the colouring of points, needed whenever IPFLEGFILE is specified.
		IPFLEGTXT= (optional)	Enter the text to be added to the legend, e.g. IPFLEGTXT="Residual (m)". By default, whenever the keyword is absent, no legend text will be displayed.
		NLABELS= (optional)	Specify the total number of labels to be plotted, e.g. NLABELS=3. By default NLABELS=0 and no labels (i.e. columns in the IPF file) will be plotted.
		ILABELS=	Specify the number of individual labels (i.e. the columns in the IPF file) to be plotted, e.g. ILABELS=3,5,7 and NLABELS=3.
IFFFILE= (optional)	Enter the name of an IFF file to be plotted, for example IFFFILE=D:\MODEL\FLOW.IFF. iMOD will plot the lines in coloured by the sixth column in the IFF file (normally that is the attribute CUMTT).		
	IFFLEGFILE= (optional)	Enter the appropriate legend file to be used, e.g. IFFLEGFILE=D:\CUMTT.LEG. By default no legend is used.	
	IFFLEGTEXT= (optional)	Enter the text to be added to the legend, e.g. IDFLEGTXT="Cumulative Time (years)". By default no legend text will be displayed.	
LEGTSIZE= (optional)	Enter the size for the legend text, enter the size in fraction of the plotting box, e.g. LEGTSIZE=0.05.		
NGEN= (optional)	Enter the number of GEN files to be included, e.g. NGEN=3.		
	GENFILE{i}=	Enter the name of a *.GEN-file that needs to be plotted on the background. On default the line, points and/or polygons in the GENFILE, will be drawn as black features, e.g. GENFILE1=D:\GENS\TOPOGRAPHY.GEN. This keyword need to be repeated for NGEN-times.	

	GENCOLOUR{i}= (optional)	Enter a colour number to be used colouring the ith GEN file specified by GENFILE{i}. By default a black line will be plotted. Specify a colour by **red**, **green** and **blue** components, e.g. GENCOLOUR1=255,0,0 to express full red. Use GENCOLOUR1=0,0,0 to specify black and GENCOLOUR1=255,255,255 to set white.
	GENTHICKNESS{i}= (optional)	Enter a line thickness to be used for the ith GEN file specified by GENFILE{i}. By default GENTHICKNESS1=1 and a line thickness of 1 will be applied, e.g. GENTHICKNESS1=5 to express a line thickness of 5.
TOP25= (optional)	Enter the location of the CRD-file (see section 9.18 for the syntax of such a file), used by iMOD to display ... background images, e.g. TOP25=d:\TOP25\BMPCRD.CRD.	
WINDOW= (optional)	Enter the coordinates of the window that needs to be displayed. Enter coordinates of the lower-left corner first and then the coordinates of the upper-right corner, e.g. WINDOW=100000.0, 400000.0, 200000.0, 425000.0. When the keyword is absent, the figure will be displayed at the maximum extent of the entered filenames at the keywords IDFFILE, IPFFILE and/or IFFFILE.	

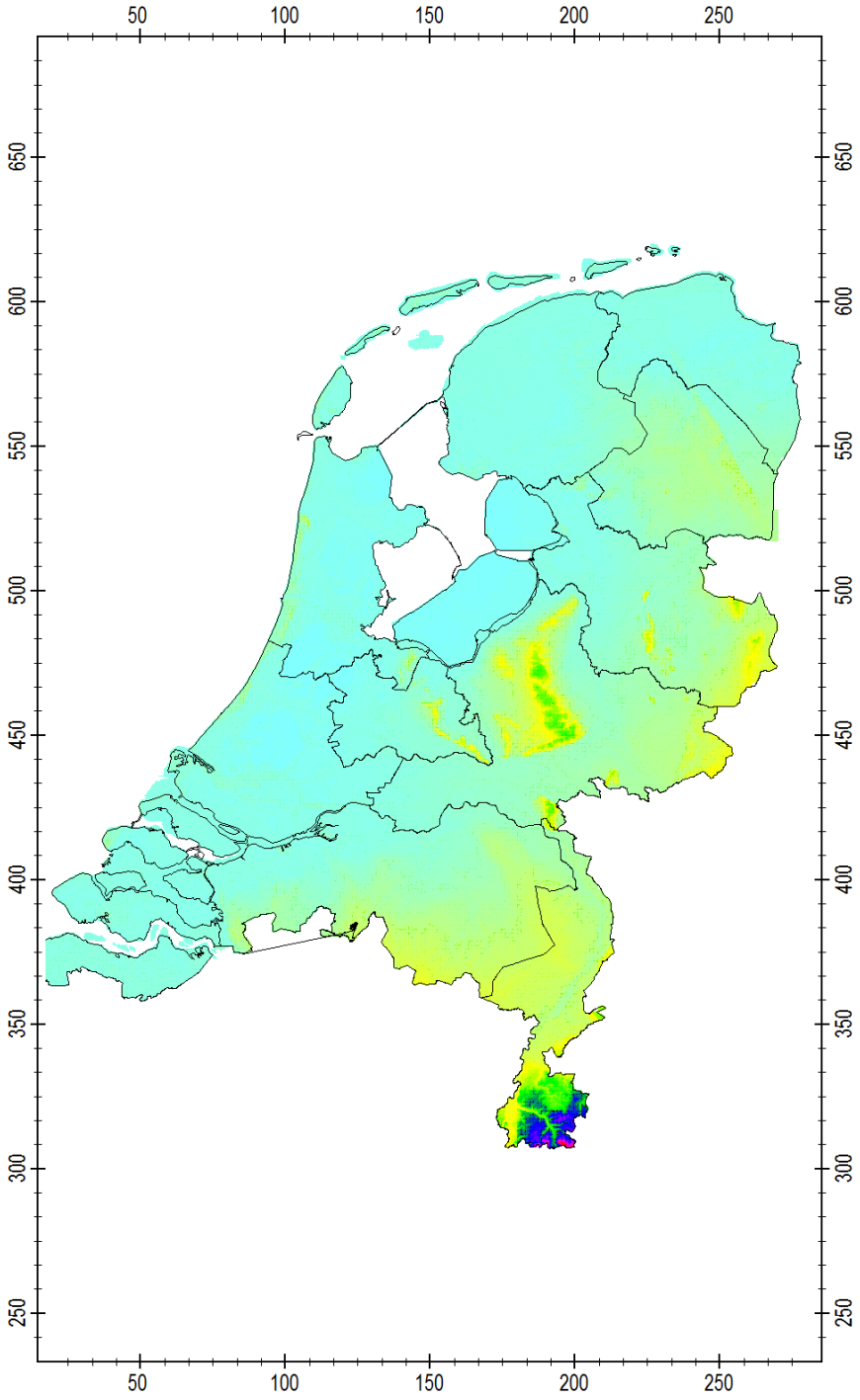
The following keywords need to be **all** defined in order to generate more sophisticated display, including legend boxes and figure labels.

TITLE=	Enter a name for the main title of the figure, e.g. TITLE="Effect of an Increased Well Capacity on Groundwater Levels". Use quotes for titles that contain spaces.
SUBTITLE=	Enter a string for a subtitle, e.g. SUBTITLE="Financed by European Union". Use quotes for titles that contain spaces.
FIGTXT=	Enter a figure identification, e.g. FIGTXT="Figure 1-a". Use quotes for titles that contain spaces.
PRJTXT=	Enter a description of the project, e.g. PRJTXT="iMOD Groundwater Flow Model". Use quotes for titles that contain spaces.
YFRACLEGEND= (optional)	Enter the percentage (0-100%) of the legend occupation in the legend area, e.g. YFRACLEGEND=50.0 means that the legend will be placed in 50% of the total legend area. On default YFRACLEGEND=100.0, so the entire legend area will be used.
RESOLUTION= (optional)	Enter the resolution of the bitmap, i.e. the number of pixels. On default RESOLUTION=3200. Higher resolutions will yield more accurate images.

Example 1

FUNCTION=PLOT
IDFFILE=D:\TUTORIAL\iMODBATCH\AHN.IDF
IDFSTYLE=100
NGEN=1
GENFILE1=D:\TUTORIAL\iMODBATCH\PROVINCES.GEN
OUTFILE=D:\TUTORIAL\iMODBATCH\AHN.png

As a result of the above described content the following figure will be created:

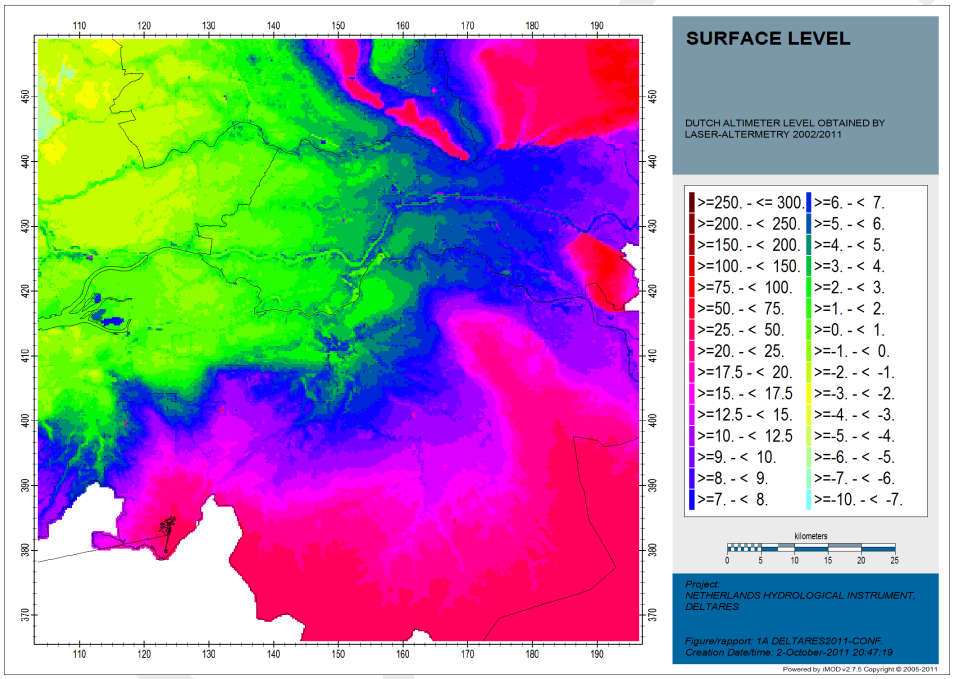


Example 2

FUNCTION=PLOT
IDFFILE=D:\TUTORIAL\iMODBATCH\AHN.IDF
IDFSTYLE=100
IDFLEGFILE=TUTORIAL\AHN.LEG
GENFILE1=D:\TUTORIAL\iMODBATCH\PROVINCES.GEN
OUTFILE=D:\TUTORIAL\iMODBATCH\AHN.png
RESOLUTION=3200
WINDOW=100000.0,400000.0,200000.0,425000.0
TITLE="Surface Level"
SUBTITLE="Dutch Altimeter Level obtained by Laser-Altimetry 2002/2011"
FIGTXT="1A DELTARES2011-Conf. "
PRJTXT="Netherlands Hydrological Instrument, Deltares"

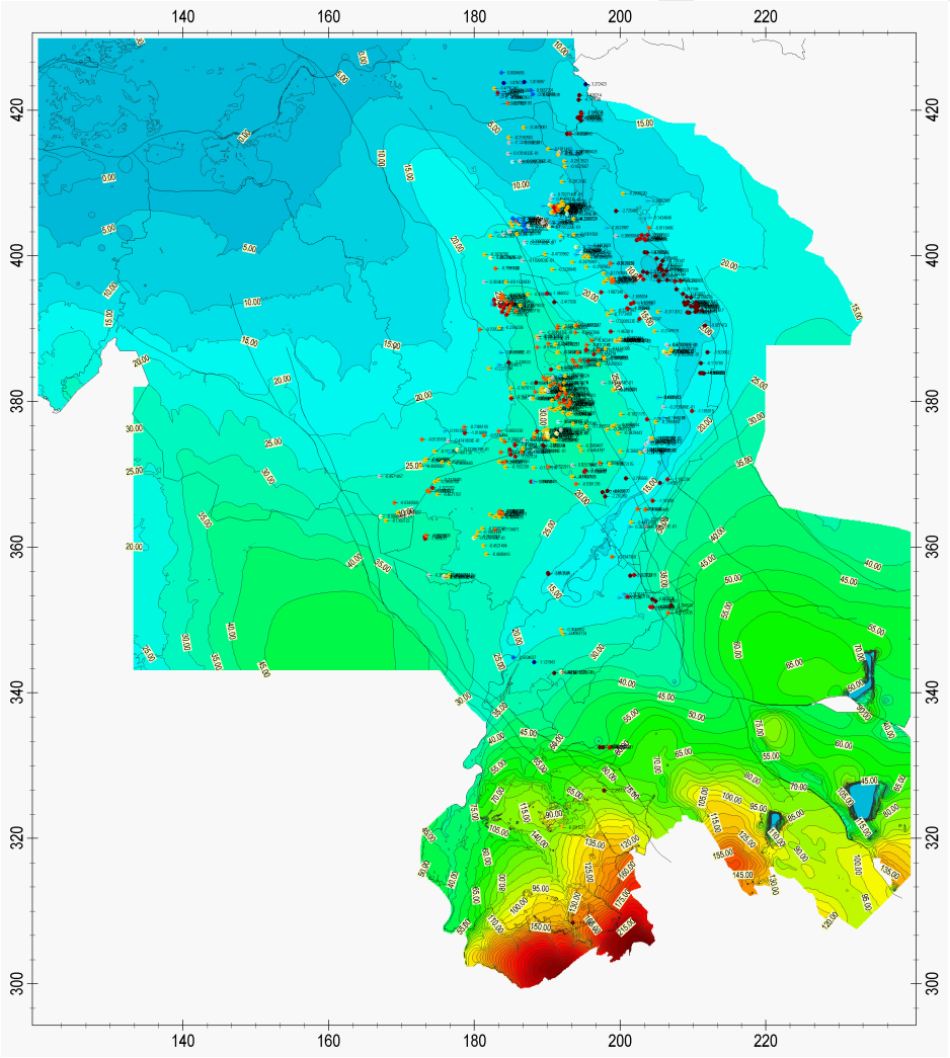
As a result of the above described content the following figure will be created.

Advanced example of a resulting bitmap:



Example 3

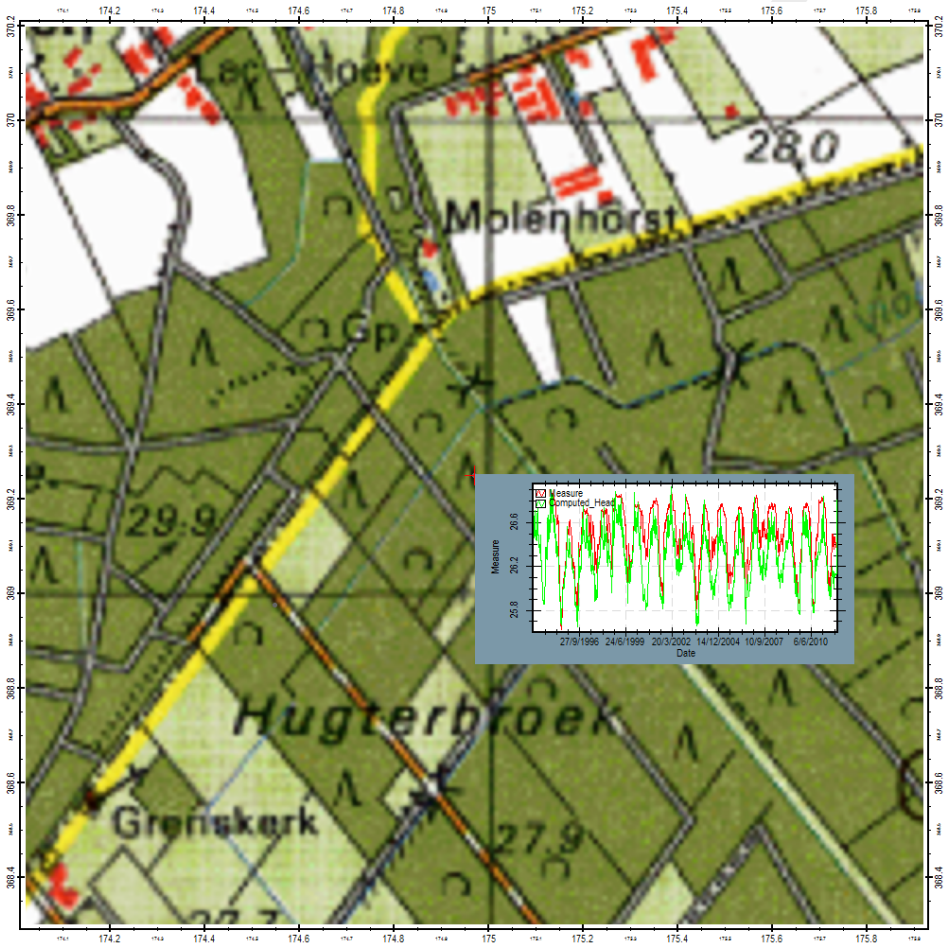
FUNCTION=PLOT
IDFFILE=D:\IMOD-TEST\PLOT\BATHEMETRY.IDF
IDFLEGFILE=D:\iMOD-TEST\PLOT\LEGEND.LEG
IDFSTYLE=010
IPFFILE=D:\iMOD-TEST\PLOT\BATHYMETRY.IPF
IPFXCOL=2
IPFYCOL=1
IPFLCOL=3
IPFSTYLE=1
IPFLEGFILE=D:\iMOD-TEST\PLOT\LEGEND.LEG
NLABELS=1
ILABELS=3
NGEN=2
GENFILE1=D:\PROVINCES.GEN
GENFILE2=D:\FAULTS.GEN
OUTFILE=D:\IMOD-TEST\PLOT\BATHEMETRY.png



Example 4

FUNCTION=PLOT
IPFFILE=D:\TUTORIAL\iMODBATCH\MEASURE.IPF
IPFASSFILES=2
IPFASSFILES_ALL=1
TOP25=D:\TUTORIAL\iMODBATCH\BMP.CRD
OUTFILE=D:\TUTORIAL\iMODBATCH\TIMESERIES.png

As a result of the above described content the following figure will be created.

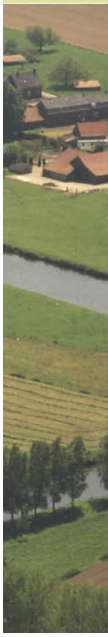


9 iMOD Files

This chapter describes the types of files used by iMOD.

Below a summarizing table of all file types used by iMOD is given; per file type it includes a short description of it's purpose. iMOD uses both generally known file types (e.g. bitmap BMP-files, ESRI-shape files, NetCDF-files) and iMOD-specific file types; the latter group of files is described in detail in section 9.1 - section 9.4. The table includes links to the related sections in this chapter or other sections in this user manual, or links to external sources.

Extent	Associated Extension(s)	Reference in this manual	Description
*.prf		section 9.1	iMOD Preference File (ASCII): File containing the initial settings for iMOD.
*.imf		section 9.2	iMOD-MetaFile (ASCII): File containing information to display selected maps including legend and topographical overlays.
*.idf		section 9.5	iMOD Data-File (BINARY): Raster file, containing information on a raster with evenly or non-evenly distributed rows and columns. Format is specific for iMOD and developed to handle large sized data sets in a time-efficient way. Besides geographical information, IDF-files can handle meta-data as well (such as descriptive information).
*.mdf		section 9.6	iMOD Multi-Data-File (ASCII): This file contains several references to IDF-files. Whenever IDF will be grouped they will be collected into a MDF-file.
*.ipf	*.txt *.dat	section 9.7	iMOD Point-File (ASCII): File containing the information on points. An IPF-file can direct to several *.TXT files that contain timevariant information (such as timeseries) or vertical descriptions (such as drilling logs or cone penetration test logs).
*.iff		section 9.8	iMOD Flow-File (ASCII): File containing the information that result after computing flow-lines within iMOD. It describes mainly lines in 3D-coordinates with their age and velocity.
*.isg	*.isp (BINARY) *.isd1 (BINARY) *.isd2 (BINARY) *.isc2 (BINARY) *.ist2 (BINARY)	section 9.9	iMOD Segment-File (ASCII): File containing the information to describe a line/segment for river modeling. An ISG-file directs to an ISP-file (containing the actual coordinates of the segment), an ISD#-file (containing timevariant information on e.g. waterlevels), an ISC#-file (containing information on the cross-section of the segments) and *.IST# (containing timevariant information on e.g. waterlevels at weirs and/or water structures)
*.gen	*.dat	section 9.10	(both ASCII): ESRI Generate File that described line elements, e.g. lines, polygons (closed lines). For polygons, *.DAT can be included that contain information for polygons.



Extent	Associated Extension(s)	Reference in this manual	Description
*.csv		section 9.12	Comma Seperated Values file. It stores tabular data (numbers and text) in plain text. Each line of the file is a data record. Each record consists of one or more fields, separated by commas. The use of the comma as a field separator is the source of the name for this file format.
*.asc		section 9.13	An ASCII format based ESRI grid, also known as an ARC/INFO ASCII GRID. (A binary format is widely used within Esri programs, such as ArcGIS.) The ASCII format is used as an exchange, or export format, due to the simple and portable ASCII file structure.
*.leg		section 9.15	iMOD Legend File (ASCII): File containing information on classes and colours used by iMOD to display an IDF, IPF, IFF and/or GEN.
*.clr		section 9.16	iMOD Colour File (ASCII): File containing the initial colour used by iMOD
*.dlf		section 9.17	File containing color information to display boreholes
*.crd	*.*	section 9.18	iMOD Coordinate File (ASCII): File that directs to other files depending on the zoom levels.
*.isd		section 9.19	An ISD-file contains information about the location of startpoints used in the calculation of pathlines from model output.
*.sol		section 9.20	iMOD SOLid project file (ASCII): File containing the layer definitions of a solid
*.spf		section 9.21	A SPF-file describes the variation in the top and bottom elevation along a cross-sectional line.
*.ses		section 9.22	File describing the operations that need to be carried out by ISG Edit.
*.tim		section 9.4	A time step configuration file.
*.prj		section 5.5	iMOD Project File (ASCII): File containing the characteristics of files used in a model simulation.
*.run		chapter 10	iMOD Runfile (ASCII): File used to run a model simulation with iMODFLOW. An iMOD Runfile may be generated from an iMOD Project File
*.ini		section 7.10.1, section 7.11.1	iMOD Initialization File (ASCII): File containing specific information for particular parts of iMOD, such as the *ScenarioTool* and the *QuickScanTool*. The syntax is comparable to the *.INI-files of Windows.
*.msk		section 5.1	iMOD Mask-File (BINARY): File contains coordinates of a rectangular area that can be loaded into iMOD to zoom to that particular area.
*.isd		section 7.13	iMOD Startpoint Definition File (ASCII):
*.bmp	*.png *.pcx *.ps		Window Bitmap (BMP), Portable Network Graphics Image (*.png), ZSoft PC Paintbrush, PostScript (*.ps) (all BINARY): Export file containing the image of the current window.
*.nc			NetCDF File (BINARY). More info: www.unidata.ucar.edu/software/netcdf.

Extent	Associated Extension(s)	Reference in this manual	Description
*.map			Map file (BINARY): A Map file is a binary data text file, which is mainly used as in-/output file of PCRaster. For more information about PCRaster go to: http://pcraster.geo.uu.nl/.
*.arr		section 9.14	Array file (ASCII): An array file is a text file, which is used as input file for iMODFLOW. It is generated by iMOD once the input is created for iMODFLOW. You can read in such a file directly in iMOD.
*.shp			ESRI Shape File (BINARY). File containing topological information on lines, points, polygons.

9.1 PRF-files

During the start-up of iMOD the iMOD preference file is read to instruct iMOD to operate with the correct settings. These settings are placed in the IMOD_INIT.PRF file, or in a *.PRF file saved by the user.

Example of a PRF-file:

```
USER "D:\IMOD\IMOD_USER"
DBASE "D:\IMOD\DBASE"
MODFLOW "D:\OSSDELTARES\IMOD_X64_R.EXE"
TOP25 "D:\TOP25 \BMPCRD.CRD"
SUBSURFEXDBASE "N:\Units\\DBASE"
7ZIP "C:\PROGRAM FILES\7-ZIP\7Z.EXE"
PLUGIN1 "D:\IMOD\PLUGINS
FFMPEG "D:\THIRD_PARTY_SOFTWARE\FFMPEG.EXE"
FFMPLAY "D:\THIRD_PARTY_SOFTWARE\FFMPLAY.EXE"
VLCPLAYER "C:\PROGRAM FILES (X86)\VIDEOLAN \VLC.EXE"
```

The following keywords can be included in the *.PRF file:

KeyWord	Folder/ File	Description
USER (compulsory)	Folder	User Map.
TOP25	File	Name of the *.CRD file to be able to position topographical bitmaps at the right coordinates.
VECTOR	Folder	Map to direct to the map that stores topographical vector information (GEN, IPF, SHP files).
HELPFILE	File	iMOD Help file (*.PDF) that can be used by the *Help ...* button throughout the application.
DBASE	Folder	Map that directs to the location of the model data. This map is used to replace the string $DBASE$ in a runfile.
IRDBASE	File	Map that directs to the location of the Quick Scan Tool command (see section 7.10). Used for obtaining an approximate result from a database filled with model results.
TAGS	File	Map to store all the Tags (Comments), make sure this location is accessible by all relevant iMOD users.
MODFLOW	File	Name of the iMODFLOW executable.
SCENTOOL	File	Initialization file (*.INI) for usage of the *ScenarioTool*.
PLUGIN1	Folder	First folder that directs to the location of plugin subfolders. Such a subfolder at least contains the executable-file to be used in iMOD.
PLUGIN2	Folder	Second folder that directs to another location of plugin subfolders. Such a subfolder at least contains the executable-file to be used in iMOD.
NORTH-ARROW	File	Bitmap (BMP, PNG) that represents a North arrow that can be placed on the graphical window.
ACROBAT-READER	Exe	Executable for the Acrobat Reader to be used to read the iMOD Help file as specified by the Keyword HELPFILE.
IR_COSTS	File	Give the name of the map file containing action related cost to be used in the IR-database.
7ZIP	Exe	Executable that will be used to unzip the files used by the Subsurface Explorer, see section 5.6.
SUBSURF-EXDBASE	Folder	Folder in which the corresponding files are organized for the Subsurface Explorer, see section 5.6.
SOLIDTOOL	File	Initialization file (*.INI) for usage of the *SolidTool*.
HELPFILE	File	iMOD Help file (*.PDF) that can be used by the *Help ...* button throughout the application.

KeyWord	Folder/ File	Description
FFMPEG	Exe	Give the name of the FFMPEG executable, this program is necessary to compute playable movie files from bitmaps. This functionality is supported via the *Movie Tool*, see section 7.5. You can download the program for free at: https://ffmpeg.org.
FFMPLAY	Exe	Give the name of the FFMPLAY executable, this program is necessary to play movie files, such as *.AVI and/or *.MPG. This functionality is supported via the *Movie Tool*, see section 7.5. You can download the program for free at: https://ffmpeg.org.
VLCPLAYER	Exe	Give the name of the VLC-player, this program is necessary to play movie files, see section 7.5.2. You can download the program for free at: http://www.videolan.org/vlc/index.nl.html.
To calculate **Agriculture-subtypes**, these files from "HELP2005-tabellen"are required:		
HLP_DRY	File	Give the name of the map file (HLP_DRY.dat) containing help dry values. HELP2005-database: crop|soiltype|GHG/GLG|respiration-stress.
HLP_WET	File	Give the name of the map file (HLP_WET.dat) containing help wet values.
HLPSOIL	File	Give the name of the Raster file (HLP_SOIL.IDF); 1:50.000 soil map reclassified for HELP2005 (using bod2hlp.lut).
LANDUSE	File	Give the name of the Landuse Raster file (*.INP) containing landuse types in LGN5-codes, to be able to read in the landuse distribution over the area.
CROP_COSTS	File	Give the name of the map file containing lookup table with crops (in LGN5-codes) and crop-yields (Euro/ha/year).
To calculate **Nature-subtypes**, these files from "WATERNOOD - Hydrologische randvoorwaarden natuur"are required:		
RFC_SOIL	File	Give the name of the Raster file:1:50.000 soil map reclassified to RFC-soils (using bod2rep.lut).
RFC_LUT	File	Give the name of the file containing lookup table with RFC(reprofunction)-characteristics.
NDT	File	Give the name of the Raster file with vegetation types to calculate potential development for.
NDT_LUT	File	Give the name of the file (*.LUT) containing lookup table with option to aggregate vegetation types.
ABIOT_LUT	File	Give the name of the file containing a lookup table with hydrologic requirements for vegetation types (NDT's).
To calculate **Urban-subtypes**, these files from "WATERNOOD - Hydrologische randvoorwaarden stedelijk"are required:		
URBAN_RANGE	File	In case of urban area, give name of map file to be able to read ranges of groundwater level. Values are in meters below surface.

Note: iMOD will search for the IMOD_INIT.PRF file in the same folder from where iMOD is started. If such a file can not be found, iMOD will ask to create its own, with the minimal required keyword: USER. In the USER folder the following subdirectories are created when you start working in iMOD:

{USER}\imffiles – storage of iMOD MetaFiles (*.IMF)
{USER}\tmp – storage of temporary files;
{USER}\legend – storage of legend files (*.LEG);
{USER}\masks – storage of maskfiles (*.MSK);
{USER}\runfiles – storage of runfiles (*.RUN);
{USER}\scenarios – storage of scenario folders with scenario files (*.SCN;*.SDF);
{USER}\solids – storage of SOLID-folders (*.SOL and *.SPF);
{USER}\qsresults – storage of results of the QuickScanTool;
{USER}\scentool – storage of results of the ScenTool;
{USER}\settings – storage of setting-files (*.*);

{USER}\shapes – storage of shape-file (*.GEN; *.SHP);
{USER}\startpoints – storage of startingpoint (*.ISP).

9.2 IMF-files

All information of an iMOD project as shown in the iMOD Manager is saved in an iMOD Meta File (IMF). The file enables to save the project contents for later use. The IMF-file contains information to display the selected maps including legend and topographical overlays.

The IMF-file is saved in ASCII-format and it has a logical structure. However it is not advised to change this file outside iMOD due to the long list of map properties.

On default, iMOD saves the content of the iMOD Manager each minute whenever the option Autosave On (1 minute) from the File menu is checked. This file is called AUTOSAVE-IMOD.IMF and is located in the folder {USER}\imffiles.

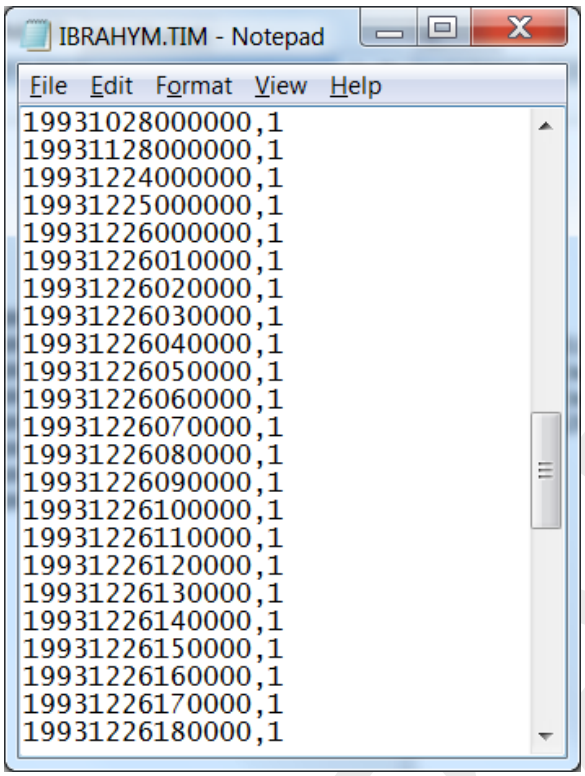
9.3 PRJ-files

A PRJ file describes the configuration of a model, it is a list of files that are associated to model layers and/or time steps. From a PRJ a MF2005 or Runfile can be configured for a specified number of model layers and/or transient periods. The PRJ file is the base file from which numerical simulations can be carried out. The syntax of the PRJ is simple and straightforward as all individual modules and packages are listed uniformly.

9.4 TIM-files

A TIM-file describes the time sequence to be used constructing a runfile and/or Modflow2005 configuration, see section 5.5.4.

Example of a TIM-file:



9.5 IDF-files

The file syntax of IDF-files (iMOD Data Files) is based on a file structure without any line feeds. This means that all data is written direct after each other. It is not written on a different *line*, but in a different *record*. An advantage of such a file structure is the possibility to access data randomly throughout the file (known as *direct-access*). The fileformat is unformatted and can therefore not be read in a normal TextEditor. It is based on the *little-endian* data format, as we read the file with records of 4 bytes (or 1 word). Since iMOD 5.0, the IDF file can be written in double-precision as well. In that case, the format of the real values are double precision (float 8) instead of single precision (float 4).

Cond.	Record	Format (bytes)	Variable	Description
	1	Integer 4	1271 2296	Lahey Record Length Identification; 1271 is a single precision IDF, 2296 a double precision.
	2	Integer 4	NCOL	Number of columns
	3	Integer 4	NROW	Number of rows
	4	Float 4/8	XMIN	X lower-left-coordinate
	5	Float 4/8	XMAX	X upper-right-coordinate
	6	Float 4/8	YMIN	Y lower-left-coordinate
	7	Float 4/8	YMAX	Y upper-right-coordinate
	8	Float 4/8	DMIN	Minimal data value
	9	Float 4/8	DMAX	Maximal data value
	10	Float 4/8	NODATA	NoData value
	11	Integer 1	IEQ	0: equidistant IDF 1: nonequidistant IDF
		Integer 1	ITB	0: no usage of TOP and BOT Values 1: usage of TOP and BOT Values
		Integer 1	—	Not used
		Integer 1	—	Not used
IEQ=0	12	Float 4/8	DX	Column width.
	13	Float 4/8	DY	Row height.
ITB=1	12+ABS(IEQ-1)*2	Float 4/8	TOP	Top value if ITB=1
	13+ABS(IEQ-1)*2	Float 4/8	BOT	Bot value if ITB=1
IEQ=1	12+ITB*2	Float 4/8	DX(NCOL)	Column width for each column, ranging from west to east
	13+ITB*2+NCOL	Float 4/8	DY(NROW)	Row height for each row, ranging from north to south
	IREC=10 + ABS(IEQ-1)*2 + IEQ*(NROW+NCOL) + ITB*2 + 1			
	IREC	Float 4/8	X(:,:)	Value for each cell
	IREC +(NROW*NCOL)	Integer 1	IADIT	Binary number to store optional arguments: IP1=1: Comments added IP?=?:
IP1=1	IREC +(NROW*NCOL)+1	Integer 1	NLINE	Number of lines that contain comments
		Char. 4	COMM(NLINE)	Comment for NLINE

9.6 MDF-files

iMOD map files can be grouped into a Multi Data File (MDF) which contains the references to all grouped IDF-files. The MDF-file helps to control the number of files in the iMOD Manager.

The MDF-file is saved in ASCII-format and it has a logical structure. However it is not advised to change this file outside iMOD.

The content of the MDF-file can be displayed by the option Info on the Map Info window

9.7 IPF-files

The file syntax for IPF-files (iMOD Point File) is very straightforward and stored in ASCII-format. In this way this type of file can be easily edited and/or created outside iMOD with any other type of (commercial) software package. The syntax of the file is twofold and depends on whether the IPF file need to be associated with additional files (see next subsections).

1 IPF file that is configured such that is cannot be used with additional, associated, files;
2 IPF file that is configured such that it can be used with additional, associated, files.

The first type of IPF file is very easy and simple. The syntax is equal to any regular comma-separated file.

Variable	Description
NLABELS	Specify for each column the label names.
DATABLOCK	For NLABELS, specify individual entries per column. Entries with empty spaces need to be bracketed by single quotes.

An example of such an IPF file as a comma-separated file is given below:

```
X,Y,Z,"City of Holland"
100.0,435.0,-32.3,Amsterdam
553.0,143.0,-7.3,"Den Bosch"
```

For such an IPF file it is not necessary to define the number of columns or row to be read in, iMOD will distinguish these itself. Another IPF format, for which it more desired to explicitly define the number of attributes and entries is given below, as an advantage it allows to attach additional data to the IPF file:

Variable	Description
NRECORDS	Number of records
NFIELDS	Number of fields
FIELDNAME$_i$	Name of the field number *i*, data is stored in column number *i* in the DATABLOCK. Repeat this item for NFIELDS on a separate line.
INDEXCOLUMN,EXTENT	Number of the index column, to be used for assess an extra file. Use INDEXCOLUMN=0, to indicate that there are no extra files associated. If INDEXCOLUMN $>$0, the EXTENT (e.g. TXT), will be added to the names in the INDEXCOLUMN to form the actual filename to be read. The maximum length for the *EXTENT* is 10 characters. It is not necessary to choose the extension *.TXT for these type of files, moreover, any extension can be chosen as long as the right EXTENT is given in the IPF-file that should call these additional files.
DATABLOCK	For NRECORDS each record (line) will contain data for each field. It is not sustained to leave data out, whenever no data is known for that particular field. For each data entry the maximum is 50 characters!

Example of an IPF-file without a reference to an additional file associated to it (INDEXCOLUMN=0,EXTENT=TXT):

```
2
4
X
Y
Z
"City of Holland"
0,TXT
100.0,435.0,-32.3,Amsterdam
553.0,143.0,-7.3,"Den Bosch"
```

The different data for each field should be delimited by a single (or more) space(s), or a comma. Do

not use tabs as delimiters! Entries that contain spaces should be encapsulated by quotes, e.g. City of Holland should be entered as 'City of Holland'.

Associated Files

Associated files can contain different types of data that are processed differently by iMOD. The syntax of each of those type of files is similar each time and described as follows:

Variable	Description
NRECORDS	Number of records
NFIELDS,ITYPE	Number of fields and the type of this file. 1 Timevariant information such as timeseries of measurements, extraction rates and so on (ITYPE=1); 2 1D Borehole information that are oriented downwards perpendicularly by a z value (ITYPE=2); 3 Cone Penetration Test Information that are oriented downwards perpendicularly (ITYPE=3); 4 3D Borehole information that is truly 3D as each interval is expressed by x,y and z coordinates (ITYPE=4).
FIELDNAME$_i$, NODATA$_i$	Name of the field number *i*, data is stored in column number *i* in the DATABLOCK. Missing data per field is determined by their corresponding *NoDataValue*. Repeat this item for NFIELDS.
DATABLOCK	For *NRECORDS* each record (line) will contain data for each field. It is not sustained to leave data out, whenever no data is known for that particular field use the corresponding Field *NoDataValue*.

In the next subsections each of those type of files will explained in more detail.

9.7.1 Associated Files with Timevariant Information

Timevariant information of timeseries can be stored in *.TXT files and their location and other spatial attributes (e.g. depth of the screen, surfacelevel) or non-dimensional information (e.g. id, name) can be stored in the IPF-file. The syntax of the *.TXT file is as follows:

```
4
3,1
DATE,-9999.0
MEASUREMENT,-9999.0
PREDICTION,-9999.0
19940114 5.70 5.70
19940128 5.73 5.71
19940214 4.95 5.10
19940228 5.01 5.15
```

Note: It is compulsory to use the first column to enter the date, expressed by a [yyyymmdd] notation or alternatively [yyyymmddhhmmss].

9.7.2 Associated File with 1D Borehole Information

Boreholes can be stored in *.TXT files and their location and other spatial attributes (e.g. surfacelevel) or non-dimensional information (e.g. id, name) can be stored in the IPF-file. The syntax of these *.TXT files is equal to the syntax of the TimeVariant information, however, ITYPE=2.

It is compulsory to use the first column to enter the vertical coordinate, expressed by meter+MSL (mean-sea-level). Use comma's and/or space(s) as delimiters. Moreover, any other column can be used for colouring the interval i and i+1. In the example below, the *Lithology=S* will be used to colour

the interval between 3.90 and -3.10 m+MSL. The colours that will be used are defined in iMOD or can be read by iMOD from a *.DLF-file.

```
6
4,2
"Z-COORDINATE,M+MSL",-9999.99
"LITHOLOGY",-9999.99
"SANDCLASS NEN5104",-9999.99
"DISTORTION",-9999.99
3.90,S,NONE,NONE
-3.10,S,NONE,SPARSE
-11.10,S,ZFC,NONE
-23.10,C,NONE,NONE
-26.10,S,ZFC,NONE
-38.10,-,-,-
```

9.7.3 Associated File with Cone Penetration Test Information

Cone Penetration Test(or borelog) information can be stored in *.TXT files and their location and other spatial attributes (e.g. surfacelevel) or non-dimensional information (e.g. id, name) can be stored in the IPF-file. The syntax of these *.TXT files is equal to the syntax of the TimeVariant information, however, ITYPE=3. It is compulsory to use the first column to enter the vertical coordinate, expressed by meter+MSL (mean-sea-level). Use comma's and/or space(s) as delimiters.

```
6
8,3
"M,LENGTH",-9999.99
"MPA,CONUSRESISTANCE",-9999.99
"MPA,RESISTANCE",-9999.99
0.32,0.721,-9999.99
0.30,0.760,-9999.99
0.28,0.783,0.048
0.26,0.828,0.061
0.24,0.865,0.066
0.22,0.893,0.073
0.20,0.838,0.073
0.16,0.930,0.070
```

In the example above, the fields that contain values equal to the *Field NoDataValue* will not be drawn and are excluded in the graph.

9.7.4 Associated File with 3D Borehole Information

3D Boreholes information (dx,dy,z) can be stored in *.TXT files and their origin location and other spatial attributes (e.g. surfacelevel) or non-dimensional information (e.g. id, name) can be stored in the IPF-file. It is compulsory to use the first three columns to enter the offset in x and y direction and the vertical coordinate, expressed by meter+MSL. Use comma's and/or space(s) as delimiters. Moreover, any other column can be used for colouring the interval i and i+1. In the example below, the *Lithology=S* will be used to colour the interval between (x,y,z) 5.0,5.0,-1.0 and (x,y,z) 7.5,5.0,-2.5. The colours that will be used are defined in iMOD or can be read by iMOD from a *.DLF-file.

```
9
4,4
"OFFSET X",-9999.99
"OFFSET Y",-9999.99
"Z-COORDINATE,M+MSL",-9999.99
"PERF_TREATMENT_TYPE",-9999.99
0.0,0.0,0.0,CEMENT
```

```
5.0,5.0,-1.0,CEMENT_PLUG
7.5,5.0,-2.5,PERFORATION
12.1,6.0,-4.5,PERFORATION
3.1,2.0,-12.3,"ACID TREATMENT"
-2.1,-3.1,-32.3,"MULTI-STAGE FRACTURE"
-21.1,-43.1,-12.3,"PACKING DEVICE"
-2.1,-5.6,-4.3,"PACKING DEVICE"
-5.1,3.6,-7.3,-
```

Note: It is possible with this type of boreholes to change azimuth and angle throughout the borehole, in this manner the borehole can move in any direction through the subsoil and even move up- and downwards.

9.8 IFF-files

The File syntax for IFF-files (iMOD Flowpath File) is very simple and stored in ASCII-format. In this way these type of files can be easily edited and/or created outside iMOD with other (commercial) software. The formal syntax is as follows and prescribed:

NFIELDS	Number of fields; NFIELDS=9.
PARTICLENUMBER	Number of the particle in the particle tracking.
ILAY	Modellayer number of the current position of the particle.
XCRD.	X-coordinate of the current position of the particle.
YCRD.	Y-coordinate of the current position of the particle.
ZCRD.	Z-coordinate of the current position of the particle.
TIME(YEARS)	Age at the current position of the particle.
VELOCITY(M/D)	Velocity at the current position of the particle.
IROW	Row number at the current position of the particle.
ICOL	Column number at the current position of the particle.
DATABLOCK	Each record (line) will contain data for each field.

iMOD will draw the flowpath using the XCRD, YCRD and ZCRD (the latter is used within the *Cross-Section Tool* and the *3DTool*). Whenever the PARTICLENUMBER changes, iMOD will start drawing another line until the end of the IFF-file is reached.

```
9
PARTICLE_NUMBER
ILAY
XCRD.
YCRD.
ZCRD.
TIME(YEARS)
VELOCITY(M/DAY)
IROW
ICOL
1 1 0.5000000E-01 0.5000000E-01 -0.9999871E-05 0.0000000E+00 0.0000000E+00
1 1
1 1 0.5000000E-01 0.5000000E-01 -0.1000000E-01 0.5673637E-01 0.3333346E-05
1 1
1 2 0.5000000E-01 0.5000000E-01 -0.2000000E-01 0.6494979E-01 0.3333346E-05
1 1
1 2 0.1024045E-01 0.5000000E-01 -0.5000000E-01 0.7477629E-01 0.2234009E-01
1 1
1 3 0.1024045E-01 0.5000000E-01 -0.6000000E-01 0.7639593E-01 0.2234009E-01
1 1
1 3 0.3189280E-02 0.5000000E-01 -0.1000000E+00 0.8600117E-01 0.1724361E-01
1 1
1 4 0.3189280E-02 0.5000000E-01 -0.1100000E+00 0.8977942E-01 0.1724361E-01
1 1
1 4 0.5000000E+00 0.5000000E+00 -0.1300000E+00 0.8977942E-01 0.0000000E+00
1 1
2 1 0.1050000E+01 0.5000000E-01 -0.9999871E-05 0.0000000E+00 0.0000000E+00
1 2
2 1 0.1050000E+01 0.5000000E-01 -0.1000000E-01 0.5673701E-01 0.3333308E-05
1 2
2 2 0.1050000E+01 0.5000000E-01 -0.2000000E-01 0.6495053E-01 0.3333308E-05
1 2
2 2 0.1000000E+01 0.5000000E-01 -0.2037707E-01 0.6526318E-01 0.4339061E+00
1 1
2 2 0.2150384E+00 0.5000000E-01 -0.5000000E-01 0.7478765E-01 0.4418139E+00
1 1
2 3 0.2150384E+00 0.5000000E-01 -0.6000000E-01 0.7640729E-01 0.4418139E+00
1 1
```

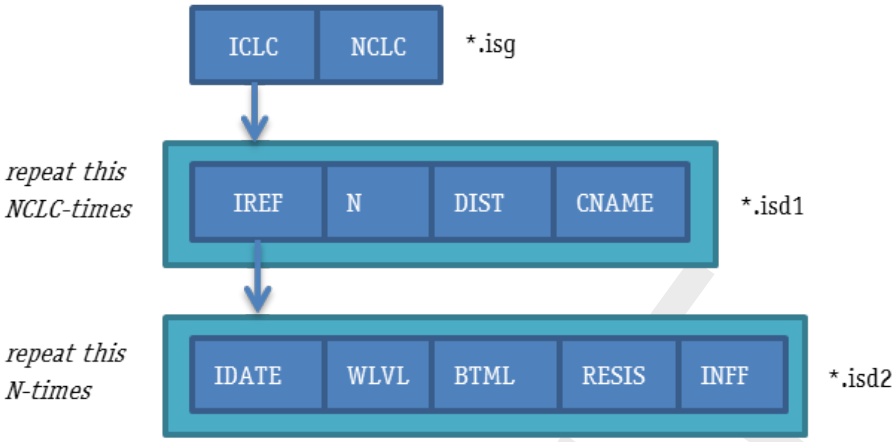
```
2 3 0.6697147E-01 0.5000000E-01 -0.1000000E+00 0.8601253E-01 0.7347348E-01
1 1
2 4 0.6697147E-01 0.5000000E-01 -0.1100000E+00 0.8979079E-01 0.7347348E-01
1 1
2 4 0.5000000E+00 0.5000000E+00 -0.1300000E+00 0.8979079E-01 0.0000000E+00
1 1
3 1 0.2050000E+01 0.5000000E-01 -0.9999871E-05 0.0000000E+00 0.0000000E+00
1 3
3 1 0.2050000E+01 0.5000000E-01 -0.1000000E-01 0.5673680E-01 0.3333321E-05
1 3
3 2 0.2050000E+01 0.5000000E-01 -0.2000000E-01 0.6495028E-01 0.3333321E-05
1 3
3 2 0.2000000E+01 0.5000000E-01 -0.2058719E-01 0.6543636E-01 0.2796153E+00
1 2
3 2 0.1000000E+01 0.5000000E-01 -0.2783243E-01 0.7310726E-01 0.2836847E+00
1 1
3 2 0.4154203E+00 0.5000000E-01 -0.5000000E-01 0.7855114E-01 0.4418535E+00
1 1
3 3 0.4154203E+00 0.5000000E-01 -0.6000000E-01 0.8017078E-01 0.4418535E+00
1 1
3 3 0.1293783E+00 0.5000000E-01 -0.1000000E+00 0.8977602E-01 0.1391620E+00
1 1
3 4 0.1293783E+00 0.5000000E-01 -0.1100000E+00 0.9355428E-01 0.1391620E+00
1 1
3 4 0.5000000E+00 0.5000000E+00 -0.1300000E+00 0.9355428E-01 0.0000000E+00
1 1
```

9.9 ISG-files

The ISG-file format is developed to capture all relevant information used by surface water elements in direct relation with groundwater. It stores stages, bottom heights, infiltration factors, resistances, and moreover, the actual outline of the surface water element. To store all these different types of information the ISG-file format consists of associated files that are connected by the ISG-file. This ISG-file is the one that will be actually read by iMOD, the other files will be opened by iMOD automatically. The syntax of the ISG-file format, and its associated files is as follows:

Attribute	Description
First line add the following columns	
NSEG	Number of segments.
ASFR	Usage of surface-flow routing (SFR) ability from the associated file *.ISD*. If ASFR=0, a default ISG file is used, if ASFR=1, a SFR compliant version of the ISG is used.
LABEL	Enter the name of the labels that are present in the ISD file, whenever ASFR=0 the following labels are present: ◇ **Date** The date of the entry; ◇ **Water level** The Waterlevel at the calculation node ◇ **Bottom level** The Bottom level at the calculation node; ◇ **Resistance** The Resistance at the calculation node; ◇ **Inf.factor** The Infiltration factor at the calculation node. whenever ASFR=1 the following labels are present: ◇ **Date** The date of the entry;
For each segment add the following columns	
LABEL	Name of the segment, use quotes to distinguish names with empty spaces. Maximum size of each label is 52 characters.
ISEG	Record number that defines the first coordinate (node) in the associated ISP-file.
NSEG	Number of records in the ISP-file that describes the segment by coordinates.
ICLC	Record number that defines the first calculation points on the segment ISEG within the associated ISD1-file.
NCLC	Number of calculation points on segment ISEG.
ICRS	Record number that defines the first cross-section on the segment ISEG within the associated ISC1-file.
NCRS	Number of cross-sections on segment ISEG.
ISTW	Record number that defines the first weir/structure on the segment ISEG within the associated IST1-file.
NSTW	Number of weirs/structures on segment ISEG.
IQHR	Number of discharge-water level relationships.
NQHR	Record number that defines the first discharge-water level relationships on the segment ISEG within the associated ISQ1-file.

The structure of the ISG-file can be illustrated by the following figure:



Example of an ISG-file:

```
16,0,"Date","Water level","Bottom level","Resistance","Inf.factor"
"NOM_1",1,120,1,15,1,8,1,2,1,16
"NOM_2",121,18,16,7,9,3,3,2,17,8
"NOM_3",139,7,23,13,12,3,5,9,25,21
"NOM_4",146,6,36,4,15,1,14,2,46,5
"NOM_5",152,44,40,15,16,4,16,2,51,16
"NOM_6",196,16,55,8,20,3,18,2,67,9
"NOM_7",212,27,63,9,23,3,20,0,76,9
"NOM_8",239,31,72,12,26,4,20,4,85,16
"NOM_9",270,88,84,13,30,4,24,1,101,13
"NOM_10",358,10,97,6,34,1,25,1,114,6
"NOM_11",368,71,103,15,35,5,26,4,120,18
"NOM_12",439,11,118,32,40,10,30,0,138,31
"NOM_13",450,12,150,4,50,2,30,2,169,5
"NOM_14",462,30,154,6,52,2,32,1,174,6
"NOM_15",492,4,160,2,54,1,33,0,180,1
"NOM_16",496,15,162,5,55,2,33,1,181,5
```

Note: Warning: To maintain consistency do not edit a ISG-file outside iMOD.

For nodes, calculations points, cross-section, structures a similar setup is used. A first reference is made from the *.ISG file to the record in the *.ISD1 file. From there another reference is made to the *ISD2 file that contains the specific configuration parameters.

The files associated from the ISG-file (ISP-, ISD-, ISC-, IST- and ISQ-files) are all binary and indexed files and cannot be edited in regular text editors. In the following sections these file-types will be described in detail.

9.9.1 ISP fileformat

The ISP-file is built with a record length of 8 bytes/2 words. The first record of the ISP is reserved to store the record length (2295). The ISEG variable in the ISG points to the record number that determines the first coordinate of the segment. Since the first record is reserved already, iMOD actually reads the ISEG+1 record instead. From each record two reals will be read that represent the x and y coordinate of the current node on the segment, see table below:

Attributes for each record in an ISP-file:

Attribute	*Width (bytes)*	*Description*
X	4 (real)	X-coordinate of node (meter)
Y	4 (real)	Y-coordinate of node (meter)

9.9.2 ISD1 and ISD2 fileformat

The ISD1-file is built with a record length of 44 bytes/11 words. The first record is solely reserved to store the record length (11511). The ICLC variable in the ISG-file points to the record number that determines the first calculation point on the segment. Since the first record is reserved already, iMOD actually reads the ICLC+1 record instead. Each record contains the following attributes:

Attributes for each record in an ISD1-file:

Attribute	*Width (bytes)*	*Description*
N	4 (int)	Number of data records in the ISD2-file that describes the timeserie of the calculation point.
IREF	4 (int)	Record number within the ISD2-file for the first data record that describes the timeserie for the calculation point.
DIST	4 (real)	Distance (meters) measured from the beginning of the segment (node 1) that located the calculation point.
CNAME	32 (char)	Name of the calculation point.

Note: For the SFR-option, only two calculation points are allowed, one at the beginning of a segment and one at the end.

It depends on the ASFR whether this file contains information for the RIV package (ASFR=0) or alternatively for the SFR package (ASFR=1). The ISD2-file is built with a record length of 20 bytes/5 words for ASFR=0. For ASFR=1, the record length is 44 bytes/13 words of 4 bytes each. The first record is solely reserved to store the record length (ASFR=0: 5367 and ASFR=1: 12535). The IREF variable in the ISD1-file points to the record number that determines the data for the calculation point on the segment. Since the first record is reserved already, iMOD actually reads the IREF+1 record instead. Each record contains the following attributes:

Attributes for each record in an ISD2-file (ASFR=0):

Attribute	*Width (bytes)*	*Description*
IDATE	4 (int)	Date representation as yyyymmdd.
WLVL	4 (int)	Waterlevel of the river (m+MSL)
BTML	4 (real)	Bottom level of the riverbed (m+MSL).
RESIS	4 (real)	Resistance of the riverbed (days).
INFF	4 (real)	Infiltration factor (-)

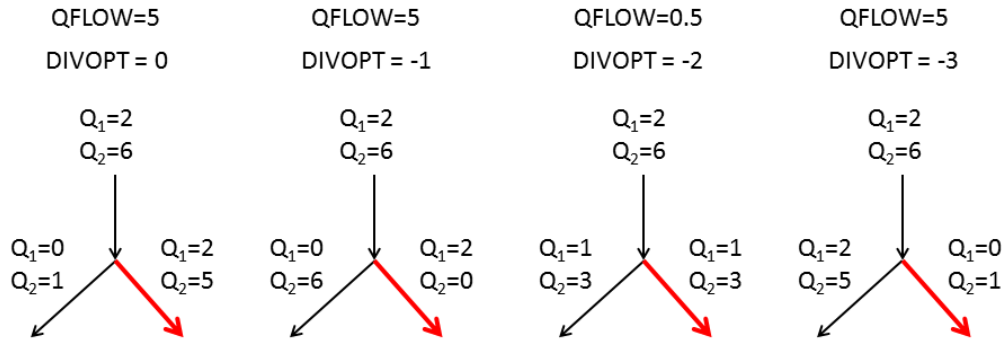
Attributes for each record in an ISD2-file (ASFR=1):

Attribute	*Width (bytes)*	*Description*
IDATE	4 (int)	Date representation as yyyymmdd.
CTIME	8 (char)	Time representation as hh:mm:ss.
BOTTOM-LEVEL	4 (real)	Bottom level of the riverbed (m+MSL).
THICK-NESS	4 (real)	Thickness of the river bed (m).
HCFACT	4 (real)	Conductivity of the river bed (md^{-1}).

IDOWN-SEG	4 (int)	An integer value of the downstream stream segment (-) that receives tributary inflow from the last downstream reach of this segment. ◇ IDOWNSEG > 0 If this segment feeds (or discharges into) another downstream (tributary) segment, then enter the number of that stream; ◇ IDOWNSEG $= 0$ If this segment does not feed (or discharge into) another downstream (tributary) segment, then enter a value of 0 for this variable. If the segment ends within the modeled grid and IDOWNSEG $= 0$, outflow from the segment is not routed anywhere and is no longer part of the stream network. One may wish to use this if all flow in the stream gets diverted into a lined canal or into a pipe; ◇ IDOWNSEG < 0 If the flow out of this segment discharges into a lake, IDOWNSEG will be equal to the negative value of the lake identification number; where the minus sign is used as a flag to tell the model that flow enters a lake rather than a tributary stream segment.
IUPSEG	4 (int)	An integer value of the upstream segment (-) from which water is diverted (or withdrawn) to supply inflow to this stream segment if this segment originates as a diversion from an upstream segment. ◇ IUPSEG > 0 If this stream segment receives inflow as a diversion from an upstream segment, enter the stream number; ◇ IUPSEG < 0 If the source of a stream segment is discharge from a lake, set IUPSEG equal to the negative value of the lake identification number; where the minus sign is used as a flag to tell the model that streamflow into this segment is derived from a lake rather than a stream segment; ◇ IUPSEG $= 0$ If this stream segment does not receive inflow as a diversion from an upstream segment, then set IUPSEG $= 0$.
CALCOPT	4 (int)	Method determines how stream depth and width are calculated for each reach in a segment (-). ◇ CALCOPT=1 Stream depth in each reach is specified at the beginning of a stress period and remains unchanged unless flow at the midpoint of a reach is zero, then depth is set to zero in that reach; ◇ CALCOPT=2 Stream depth is calculated and updated each iteration of the solver within a time step and is calculated from Manning's equation assuming a wide rectangular channel; ◇ CALCOPT=3 Stream depth and width is calculated and updated each iteration of the solver within a time step and are calculated from Manning's equation using an eight-point cross section; ◇ CALCOPT=4 Stream depth and width is calculated and updated each iteration of the solver within a time step and are calculated from a power function; ◇ CALCOPT=5 Stream depth and width is calculated and updated each iteration of the solver within a time step and are calculated from a table of values as entered in the Q-DEPTH/WIDTH relation ships.

DIVOPT	4 (int)	An integer value that only is specified if IUPSEG > 0 (do not specify a value in this field if IUPSEG $= 0$ or IUPSEG < 0). DIVOPT defines the prioritization system for diversion, such as when insufficient water is available to meet all diversion stipulations, and is used in conjunction with the value of QFLOW (specified below) (-). ◇ DIVOPT=1 then if the specified diversion flow (QFLOW) is greater than the flow available in the stream segment from which the diversion is made, the diversion is reduced to the amount available, which will leave no flow available for tributary flow into a downstream tributary of segment IUPSEG; ◇ DIVOPT=2 then if the specified diversion flow (QFLOW) is greater than the flow available in the stream segment from which the diversion is made, no water is diverted from the stream. This approach assumes that once flow in the stream is sufficiently low, diversions from the stream cease, and is the ”priority“ algorithm that originally was programmed into the STR1 Package (Prudic, 1989); ◇ DIVOPT=3 then the amount of the diversion is computed as a fraction of the available flow in segment IUPSEG; in this case, $0.0 <$ QFLOW < 1.0; ◇ DIVOPT=4 then a diversion is made only if the streamflow leaving segment IUPSEG exceeds the value of QFLOW. If this occurs, then the quantity of water diverted is the excess flow and the quantity that flows from the last reach of segment IUPSEG into its downstream tributary (IDOWNSEG) is equal to QFLOW. This represents a flood-control type of diversion, as described by Danskin and Hanson (2002).

Illustration of the diversion rates per DIVOPT category that is assigned to the red segment in the figure.



QFLOW	4 (real)	Streamflow entering or leaving the upstream end of a stream segment (i.e. the first reach) (m^3s^{-1}). iMOD will check whether you have entered a value larger than 100000.0, as this might indicate that the entered volume is m^3d^{-1} instead of m^3s^{-1}. Be aware of the fact that the meaning of QFLOW depends on the fact whether the stream is a ◇ If the stream is a headwater stream (IUPSEG = 0) which is the first stream of a segment without any segment upstreams, QFLOW defines the total inflow to the first reach of the segment. The value can be any number ≥ 0.0; ◇ If the stream is a tributary stream (IUPSEG = 0) with an upstream dewatering stream, QFLOW defines additional specified inflow to or withdrawal from the first reach of the segment (that is, in addition to the discharge from the upstream segment of which this is a tributary). This additional flow does not interact with the groundwater system. For example, a positive number might be used to represent direct outflow into a stream from a sewage treatment plant, whereas a negative number might be used to represent pumpage directly from a stream into an intake pipe for a municipal water treatment plant; ◇ If the stream is a diversionary stream (IUPSEG $\neq$ 0), and the diversion is from another stream segment, QFLOW defines the streamflow diverted from the last reach of stream segment IUPSEG into the first reach of this segment. The diversion is computed or adjusted according to the value of DIVOPT; ◇ If the stream is a diversionary stream (IUPSEG $\neq$ 0), and the diversion is from a lake, QFLOW defines a fixed rate of discharge diverted from the lake into the first reach of this stream segment (unless the lake goes dry) and flow from the lake is not dependent on the value of ICALC. However, if QFLOW $= 0$, then the lake outflow into the first reach of this segment will be calculated on the basis of lake stage relative to the top of the streambed for the first reach using one of the methods defined by ICALC.
QRUNOFF	4 (real)	volumetric rate (m^3s^{-1}) of the diffuse overland flow runoff that enters the stream segment, the rate is apportioned to each reach of the segment.
PPTSW	4 (real)	precipitation (mmd^{-1}) that is the volumetric rate per unit area of water added by precipitation directly on the stream channel (in units of length (millimeter) per time (day)).
ETSW	4 (real)	evaporation (mmd^{-1}) that is the volumetric rate per unit area of water removed by evapotranspiration directly from the stream channel (in units of length (millimeter) per time (day)). ETSW is always defined as a positive value.

9.9.3 ISC1 and ISC2 fileformat

The ISC1-file is built with a record length of 44 bytes/11 words. The first record is solely reserved to store the record length (11511). The ICRS variable in the ISG-file points to the record number that determines the first calculation point on the segment. Since the first record is reserved already, iMOD actually reads the ICRS+1 record instead. Each record contains the following attributes:

Attributes for each record in an ISC1-file:

Attribute	***Width (bytes)***	***Description***	
N	4 (int)	The meaning of this attribute is twofold:	
		>0	Number of data records in the ISC2-file that describes the actual cross-section.

		<0	The absolute number of data records in the ISC2-file that describes the riverbed as a collection of x,y,z points including an extra record to describe the dimensions (*DX,DY*) of the network that captured the x,y,z points.
IREF	4 (int)	Record number within the ISC2-file for the first data record that describes the cross-section.	
DIST	4 (real)	Distance (meters) measured from the beginning of the segment (node 1) that locates the cross-section.	
CNAME	32 (char)	Name of the cross-section.	

The ISC2-file is built with a record length of 12 bytes/3 words. The first record is solely reserved to store the record length (3319). The IREF variable in the ISC1-file points to the record number that determines the data for the calculation point on the segment. Since the first record is reserved already, iMOD actually reads the IREF+1 record instead. Each record contains the following attributes whenever N$>$0:

Attributes for each record in an ISC2-file:

Attribute	***Width (bytes)***	***Description***
DISTANCE	4 (real)	Distance of the cross-section measured from the centre of the riverbed (minus to the left en positive to the right).
BOTTOM	4 (real)	Bottom level of the riverbed (meter), whereby zero will be assigned to the lowest riverbed level.
MRC	4 (real)	Manning's roughness coefficient (-).

Alternatively, the record can have a different meaning whenever N$<$0:

Attribute	***Width (bytes)***	***Description***
First record at IREF+1		
DX	4 (real)	Absolute width $\|\|DX\|\|$ in meters of the rectangular raster that follows.
DY	4 (real)	Absolute height $\|\|DY\|\|$ in meters of the rectangular raster that follows.
HREF	4 bytes	Reference Height in meters. Whenever specified (*DX* < 0.0 and *DY* < 0.0), the attribute Z (specified below) is organized differently.
Following records starting at IREF+2		
X	4 (real)	X coordinate (meter) for a riverbed "pixel", these coordinates need to be on a rectangular network with spatial distance of $\|\|DX\|\|$ measured at the centre of the "pixel".
Y	4 (real)	Y coordinate (meter) for a riverbed "pixel" , these coordinates need to be on a rectangular network with spatial distance of $\|\|DY\|\|$ measured at the centre of the "pixel".
Following record is valid whenever *DX* > 0.0 and *DY* > 0.0		
Z	4 (real)	Bottom level of the riverbed (meter).
Following record is valid whenever *DX* < 0.0 and *DY* < 0.0		
Z_m	2 (integer)	Integer value of bottom level of the riverbed (integer of meters), e.g. bottom level is -23.43, Z_m=-23.
Z_c	1 (integer)	Integer value of remaining digits of bottom level of the riverbed (remaining centimeter), e.g. bottom level is -23.43, Z_c=43 centimeters.

Z_p	1 (integer)	Integer value of area affected by HREF, e.g. areas with $Z_p < 0$ will be inundated only whenever the current river stage is higher than the Reference Height (HREF) and the river stage is higher than the corresponding riverbed. Areas with $Z_p > 0$, will be inundated whenever the river stage is higher than the current riverbed. The absolute value of Z_p is used as a multiplication factor for the river bed resistances for the attribute RESIS in the ISD2-file (see 9.9.2).

9.9.4 IST1 and IST2 fileformat

The IST1-file is built with a record length of 44 bytes/11 words. The first record is solely reserved to store the record length (11511). The ISTW variable in the ISG-file points to the record number that determines the first calculation point on the segment. Since the first record is reserved already, iMOD actually reads the ISTW+1 record instead. Each record contains the following attributes:

Attributes for each record in an IST1-file:

Attribute	***Width (bytes)***	***Description***
N	4 (int)	Number of data records in the IST2-file that describes the actual timeserie for the weir/structure.
IREF	4 (int)	Record number within the IST2-file for the first data record that describes the weirs/structure.
DIST	4 (real)	Distance (meters) measured from the beginning of the segment (node 1) that locates the weir/structure.
CNAME	32 (char)	Name of the weir/structure.

The IST2-file is built with a record length of 12 bytes/3 words. The first record is solely reserved to store the record length (3319). The IREF variable in the IST1-file points to the record number that determines the data for the calculation point on the segment. Since the first record is reserved already, iMOD actually reads the IREF+1 record instead. Each record contains the following attributes:

Attributes for each record in an IST2-file:

Attribute	***Width (bytes)***	***Description***
IDATE	4 (int)	Date representation as yyyymmdd.
WLVL_UP	4 (real)	Water level for the upstream side of the weir/structure (m+MSL).
WLVL_DWN	4 (real)	Water level for the downstream side of the weir/structure (m+MSL).

9.9.5 ISQ1 and ISQ2 fileformat

The ISQ1-file is built with a record length of 44 bytes/11 words. The first record is solely reserved to store the record length (11511). The IQHR variable in the ISG-file points to the record number that determines the first calculation point on the segment. Since the first record is reserved already, iMOD actually reads the IQHR+1 record instead. Each record contains the following attributes:

Attributes for each record in an ISQ1-file:

Attribute	***Width (bytes)***	***Description***
N	4 (int)	Number of data records in the ISQ2-file that describes the actual timeserie for the q-width/depth relation ship.
IREF	4 (int)	Record number within the ISQ2-file for the first data record that describes the q-width/depth relation ship.
DIST	4 (real)	Distance (meters) measured from the beginning of the segment (node 1) that locates the q-width/depth relation ship.

CNAME	32 (char)	Name of the q-width/depth relation ship.

The ISQ2-file is built with a record length of 12 bytes/3 words. The first record is solely reserved to store the record length (3319). The IREF variable in the ISQ1-file points to the record number that determines the data for the calculation point on the segment. Since the first record is reserved already, iMOD actually reads the IREF+1 record instead. Each record contains the following attributes:

Attributes for each record in an ISQ2-file:

Attribute	***Width (bytes)***	***Description***
Q	4 (int)	Discharge in m^3d^{-1}.
WIDTH	4 (real)	Width of the stream at the given discharge Q (m).
DEPTH	4 (real)	Depth of the stream at the given discharge Q (m).
FACTOR	4 (real)	obsolete factor at the given discharge Q (-).

9.10 GEN-files

In iMOD two types of GEN files are distinguished a) the standard GEN file format as developed by ESRI, maker of ArcINFO, ARCGIS, ArcView and b) the GEN file developed for iMOD to generate images from large GEN-files most efficiently.

9.10.1 Standard GEN-files

Creating a GEN-file can be done in ArcView3.x by means of the sample script shp2gen.ave (installdirectory \ESRI\AV_GIS30\ARCVIEW\Samples\scripts\shp2gen.ave). Within ArcGIS this can be performed only by a conversion of the ArcGIS shapefile to a ArcINFO coverage and finally using the command UNGENERATE to create a GEN-file. The syntax of a GEN-file should be as follows:

Points

ID_1, X_1, Y_1
ID_2, X_2, Y_2
..
ID_n, X_n, Y_n
END

Lines

ID_1
X_1,Y_1
X_2,Y_2
X_3,Y_3
...
X_n,Y_n
END
ID_2
X_1,Y_1
X_2,Y_2
X_3,Y_3
...
X_n,Y_n
END
END

Polygons

ID_1
X_1,Y_1
X_2,Y_2
X_3,Y_3
...
X_1,Y_1
END
ID_2
X_1,Y_1
X_2,Y_2
X_3,Y_3
...
X_1,Y_1
END
END

or 3-D representatives as

3-D Polygons

```
ID1
X1,Y1,Z1
X2,Y2,Z2
X3,Y3,Z3
...
X1,Y1,Z1
END
ID2
X1,Y1,Z1
X2,Y2,Z2
X3,Y3,Z3
...
X1,Y1,Z1
END
END
```

Note: iMOD will display 3-D Polygons as filled surfaces in the 3-D tool.

3-D Lines

```
ID1
X1,Y1,Z1
X2,Y2,Z2
X3,Y3,Z3
...
Xn,Yn,Zn
END
ID2
X1,Y1,Z1
X2,Y2,Z2
X3,Y3,Z3
...
Xn,Yn,Zn
END
END
```

Note: The ID field is read by iMOD as a character-type, thus ID can be an integer, real or character, e.g. ID=1, or ID=3.22 or ID=Area1. Make sure that quotes are use for ID field with spaces or commas; such as ID="Area 1".

9.10.2 iMOD GEN-files

The GEN file format for iMOD is optimized for usage in iMOD and to represent large GEN-files efficiently. The file format is a binary format and consists out of coordinates as well as attributes for existing labels.

```
XMIN(real*8),YMIN(real*8),XMAX(real*8),YMAX(real*8)
MAXPOL(integer*4),MAXCOL(integer*4)
```

add following in case MAXCOL > 0

```
COLWIDTH(integer*4)[dimension MAXCOL]
LABELS(character*11)[dimension MAXCOL]
```

repeat following for MAXPOL

```
NPOINTS(integer*4)
```

add following in case MAXCOL > 0

```
LABELS(character*COLWIDTH[dimension MAXCOL])[dimension MAXCOL]
```

add these representing the point,polygon,lines

```
XMIN(real*8),YMIN(real*8),XMAX(real*8),YMAX(real*8)
X(real*8),Y(real*8))[dimension NPOINTS]
```

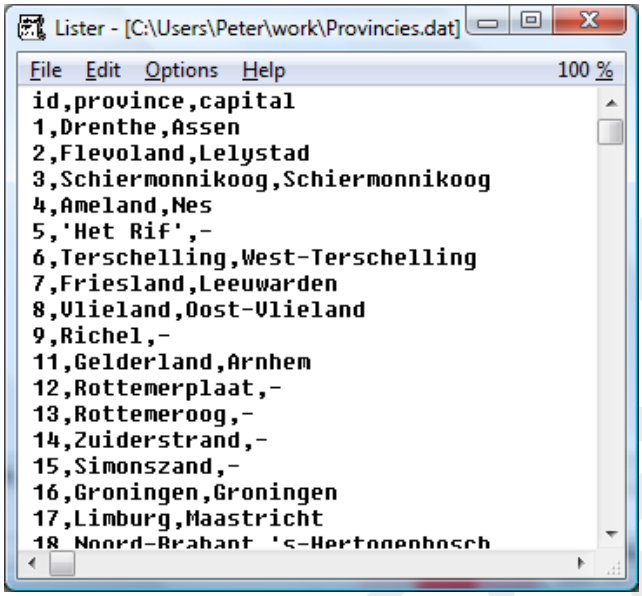
The above mentioned format is slightly different whenever labels are absent, some of the entries can be skipped for those cases. The size of the labels for the attributes is limited to 11 characters, this is similar are present in the DBF-files from ESRI-ArcGIS. Whenever NPOINTS=1, iMOD will display a points, whenever the first and last coordinates are similar a polygon will be plotted, in others cases a line.

9.11 DAT-files

DAT-files can be used to associate information to GEN-files (see section section 9.10.1). The DAT-file should have the same name as the GEN-file. The ID_i number(s) for the polygons is used to relate to the proper *ID* in the DAT-file. The syntax of a DAT-file is simple.

Header	Header label for each column. The first column is reserved for the *ID* number to relate to the *ID* number of the associated GEN-file.
Values	Enter a value for each column for each unique *ID* value in the associated GEN-file.

Example of a DAT-file:



Note: It is possible, however, to relate more polygons with identical *ID* numbers, to the same *ID* in the DAT-file.

Note: The different data for each field should be delimited by a single (or more) space(s), or a comma. Do not use tabs as delimiters! Entries that contain spaces should be encapsulated by quotes, e.g. Het Rif should be entered as 'Het Rif'.

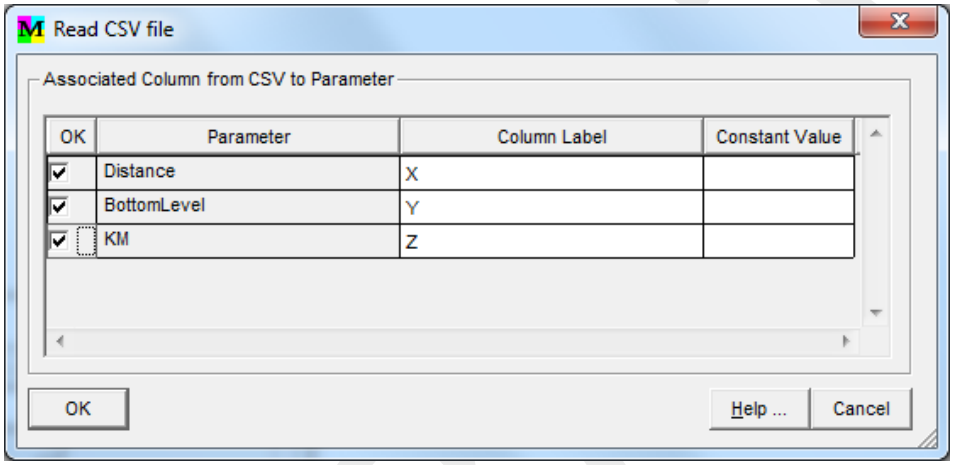
9.12 CSV-files

Throughout the iMOD application it is possible to import or export data for and into a CSV-file format (comma-separated-values file). The syntax for those files is simple and straightforward and equal to a DAT-file format (see section 9.11).

Header	Header label for each column.
Values	Enter a value for each column, use quote for entry fields that contain spaces, comma's, e.g. *Klompen Plein* should be noted as *"Klompen Plein"*.

Whenever a CSV is imported in iMOD the *Read CSV-file* window is shown. For each parameter that needs to be assigned (depending on the calling interface, see section 6.10.3.9), this window links the parameter read from the CSV-file to the column in the CSV-file.

Read CSV-file:



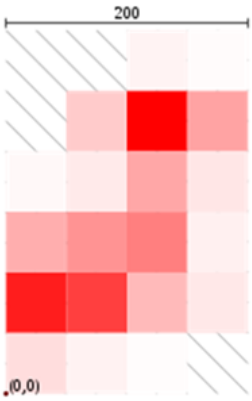
Field in the Table:	
OK	Click this checkbox to use the associated column for the corresponding column, e.g. the *Distance* parameter will be read from the column with the label *X*.
Parameter	The rows for this column will be filled in automatically based on the parameters needed from the calling interface.
Column Label	Select the appropriate column in the CSV-file to be assigned to the parameter listed in the *Parameter* column.
Constant Value	Enter a value to be used as a constant value for all rows in the CSV-file.
OK	Click this button to import the data from the CSV-file and use the read value for the appropriate *Parameters*.
Help . . .	
Cancel	Click this button the cancel the import from the selected CSV-file.

9.13 ASC-files

The format is relatively straight-forward: the first six lines indicate the reference of the grid, followed by the values listed in the order they would naturally appear (left-right, top-down). For example, consider a grid, shown to the left. This could be encoded into an ASCII grid file that would look like:

ESRI ASCII Format:

Grid image



Values

	25	75	125	175
275	NA	NA	5	2
225	NA	20	100	36
175	3	8	35	10
125	32	42	50	6
75	88	75	27	9
25	13	5	1	NA

ASCII grid format

```
ncols         4
nrows         6
xllcorner     0.0
yllcorner     0.0
cellsize      50.0
NODATA_value  -9999
-9999 -9999 5 2
-9999 20 100 36
3 8 35 10
32 42 50 6
88 75 27 9
13 5 1 -9999
```

Variable	Format	Description
#	-	Comments
Ncols	Integer	Numbers of columns
Nrows	Integer	Numbers of rows
Xllcorner	Real	The western (left) x-coordinate
Yllcorner	Real	The southern (bottom) y-coordinate
Cellsize	Real	The length of one side of a square cell
NODATA_ value	Real	The value that is regarded as “missing” or “not applicable”; this line is optional, but highly recommended as iMOD will expect this line to be declared
DataBlock	Integer Real	Listing of the raster values for each cell, starting at the upper-left corner (north-west). These number are delimited using a single (or more) space(s) character(s).

Note: These ESRI ASCII rasters will be converted into IDF-files whenever they are read into iMOD. However, iMOD can export IDF-files into ESRI ASCII files again.

9.14 ARR-files

The format is relatively straight-forward: it is a free-formatted file format generate by iMOD to create the input per package for iMODFLOW. For example, consider a grid with boundary conditions:

```
-1,1,1,1,1,1,-1
0,0,0,1,1,1,-1
0,0,0,1,1,1,-1
0,0,0,1,1,1,-1
0,0,0,0,0,1,-1
```

This could be encoded into an ARR grid file that would look like:

```
-1
5*1
-1
3*0
3*1
-1
3*0
3*1
-1
3*0
3*1
-1
5*0,
1
-1
```

At the end of the file, there is a list of the dimensions of the file with:

```
 DIMENSIONS
120
132
120000.0
298000.0
240000.0
430000.0
3.4028235E+38
0
1000.000
1000.000
```

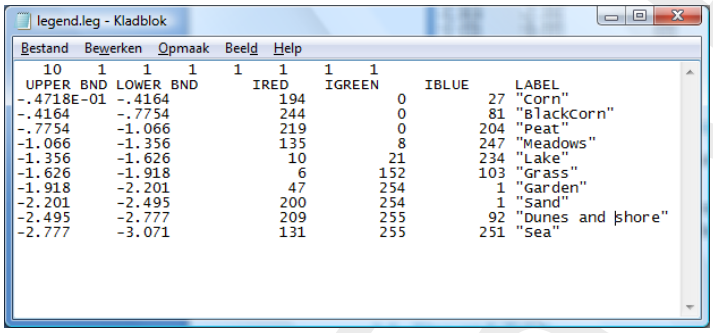
Variable	Format	Description
#	-	Comments
Ncols	Integer	Numbers of columns
Nrows	Integer	Numbers of rows
Xllcorner	Real	The western (left) x-coordinate
Yllcorner	Real	The southern (bottom) y-coordinate
Xurcorner	Real	The eastern (right) x-coordinate
Yurcorner	Real	The northern (top) y-coordinate
NODATA_ value	Real	The value that is regarded as “missing” or “not applicable”; this line is optional, but highly recommended as iMOD will expect this line to be declared
IEQ	Integer	Identifier to specify a uniform grid (IEQ=0) or a nonuniform grid (IEQ=1)
Cellsize_X	Real	The length of the x-side of a square cell
Cellsize_Y	Real	The length of the y-side of a square cell

9.15 LEG-files

Each map file in iMOD (IDFs, IPFs, ISGs, GENs and IFFs) can be displayed by classes that are defined in a legend. The legend is stored internally, but can be saved to, and loaded from disk. The syntax of a *.LEG-file is as follows:

NClass	Number of classes. Bear in mind that legends that have *NClass<=50*, behave differently than legends that have *50<NClass<=255*, see for more information 0.
ColourMarks	In the *Legend* window (see 3.3.4), these *ColourMarks* (0-1) define whether the *ColourMark* is turned on or off in the colour ramp.
Upper BND	Upper Boundary of the class
Lower BND	Lower Boundary of the class
IRED	Red component in RGB-colour model (0-255)
IGREEN	Green component in RGB-colour model (0-255)
IBLUE	Blue component in RGB-colour model (0-255)
Label	Label, make sure the length of the label is less or equal to 50 characters. For *Labels* that contain more than one word, such as Dunes and Shore, they should be bracketed by quotes, i.e. “Dunes and Shore”.

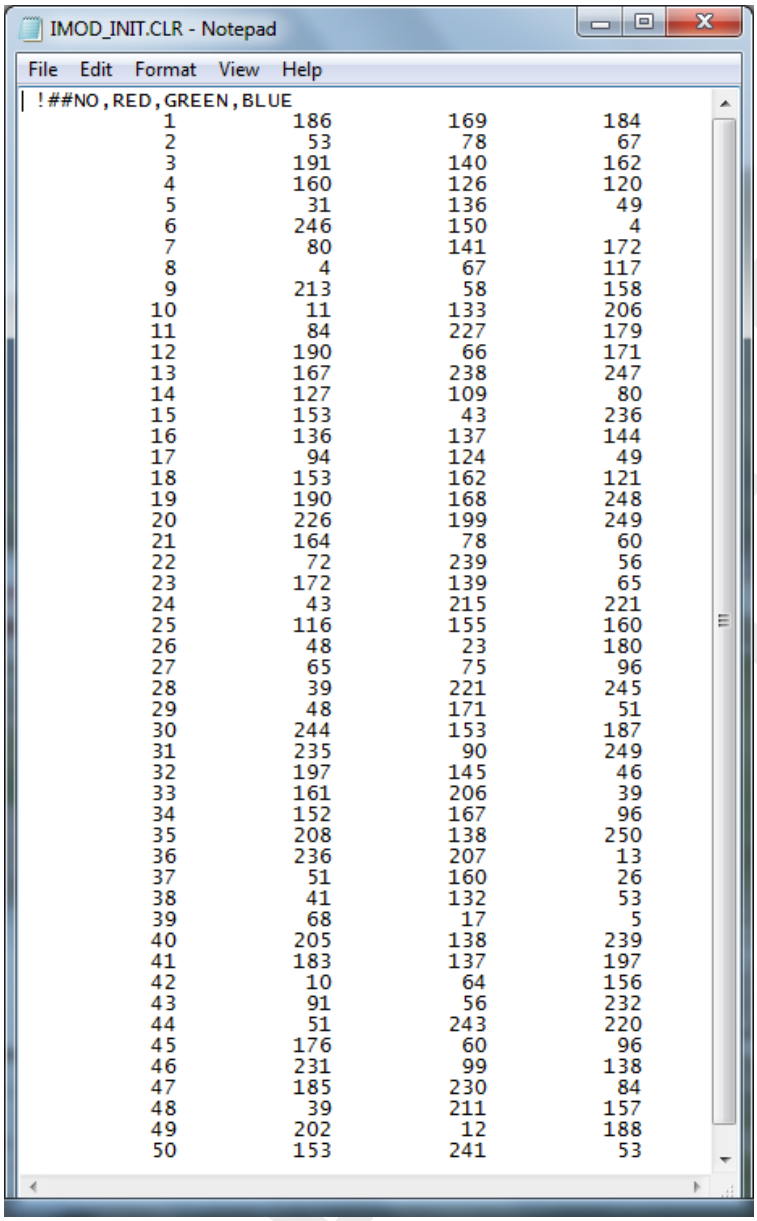
Example of a LEG-file:



Note: Use comma’s and/or space delimiters within LEG-files.

9.16 CLR-files

iMOD supports 50 predefined colours to be used as default in a variety of iMOD functionalities. The colour definitions are stored in the IMOD_INIT.CLR file which is stored in the USER directory.




```
!##NO,RED,GREEN,BLUE
         1        186        169        184
         2         53         78         67
         3        191        140        162
         4        160        126        120
         5         31        136         49
         6        246        150          4
         7         80        141        172
         8          4         67        117
         9        213         58        158
        10         11        133        206
        11         84        227        179
        12        190         66        171
        13        167        238        247
        14        127        109         80
        15        153         43        236
        16        136        137        144
        17         94        124         49
        18        153        162        121
        19        190        168        248
        20        226        199        249
        21        164         78         60
        22         72        239         56
        23        172        139         65
        24         43        215        221
        25        116        155        160
        26         48         23        180
        27         65         75         96
        28         39        221        245
        29         48        171         51
        30        244        153        187
        31        235         90        249
        32        197        145         46
        33        161        206         39
        34        152        167         96
        35        208        138        250
        36        236        207         13
        37         51        160         26
        38         41        132         53
        39         68         17          5
        40        205        138        239
        41        183        137        197
        42         10         64        156
        43         91         56        232
        44         51        243        220
        45        176         60         96
        46        231         99        138
        47        185        230         84
        48         39        211        157
        49        202         12        188
        50        153        241         53
```

9.17 DLF-files

The colour information for boreholes will be read from a DLF-file. On default, iMOD tries to read the file: {user}\settings\DRILL.DLF. If this fails, iMOD will use its internal default values. The syntax of a DLF-file is as follows:

LABELS	Use the first line of the file to identify the columns. These are however unchangeable! Repeat the following attributes (CLASS, IRED, IGREEN, IBLUE, LEGEND) for each line.
CLASS	Use the first column to specify the search-string that should match the first column of the associated *.TXT file as described.
IRED	Red saturation (0-255)
IGREEN	Green saturation (0-255)
IBLUE	Blue saturation (0-255)
LEGEND	Label used in the legend.
WIDTH (optional)	Enter the width category (1-10) used in 2D and 3D plotting. This is an optional value and is assigned the value 1 by default.

Example of a DLF-file:

```
CLASS,IRED,IGREEN,IBLUE,LEGEND
FS,255,255,0,"Fine Sand",1.00
C,148,0,211,Clay,0.25
G,218,165,32,Gravel,2.00
"P H",244,164,96,"Peat Holland",0.50
L,238,130,238,Loam,0.75
```

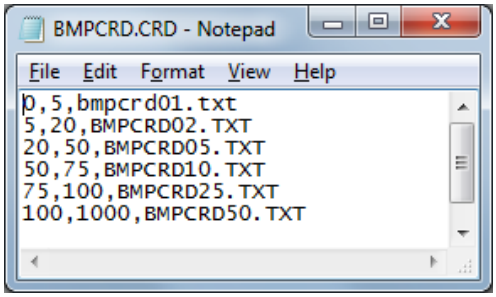
Note: The maximum string length of the *Class*-column is 20 characters. The maximum string length for the LEGEND column is 50 characters. The maximum number of classes is 250 (lines, excluding the header).

9.18 CRD-files

The display of tiled background maps is directed by the iMOD coordinate CRD-file. The CRD-file is defined with the keyword TOP25 in the iMOD preference file.

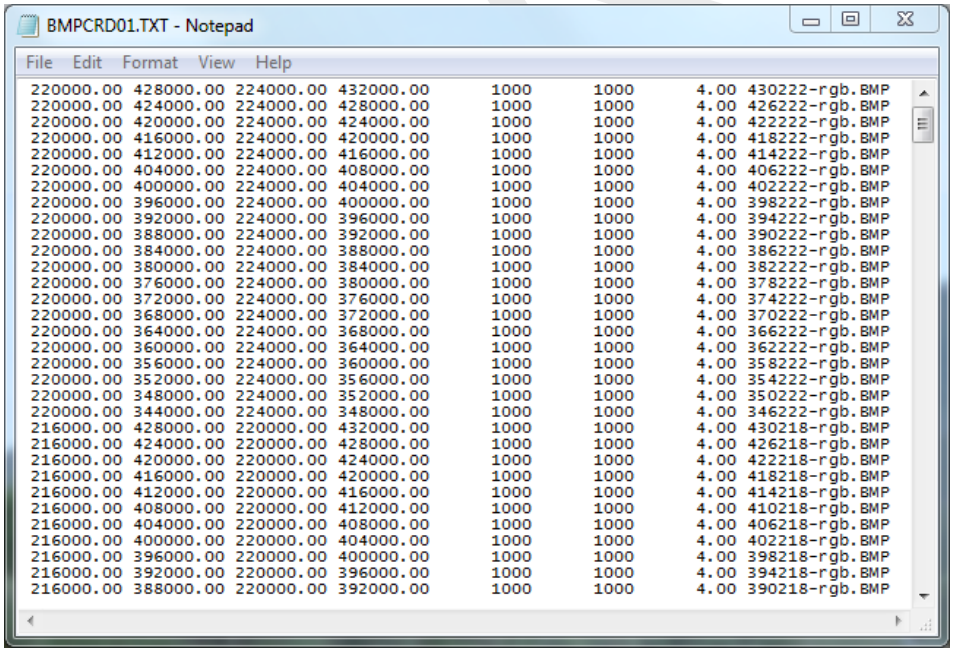
The CRD-file links BMP-files as background maps by specifying the scale (at unit 1000) at which the BMP-files will be displayed and the name of the TXT-files in which the BMP-files are defined.

Example of a CRD-file:



The TXT-files referred to in the CRD-file define the BMP-files used as background maps. For each BMP-file is specified: coordinates LL-corner, coordinates UR-corner, number of pixels in X-direction, number of pixels in Y-direction, cellsize (m), name of the BMP-file.

Example of a TXT-file linked to a CRD-file:

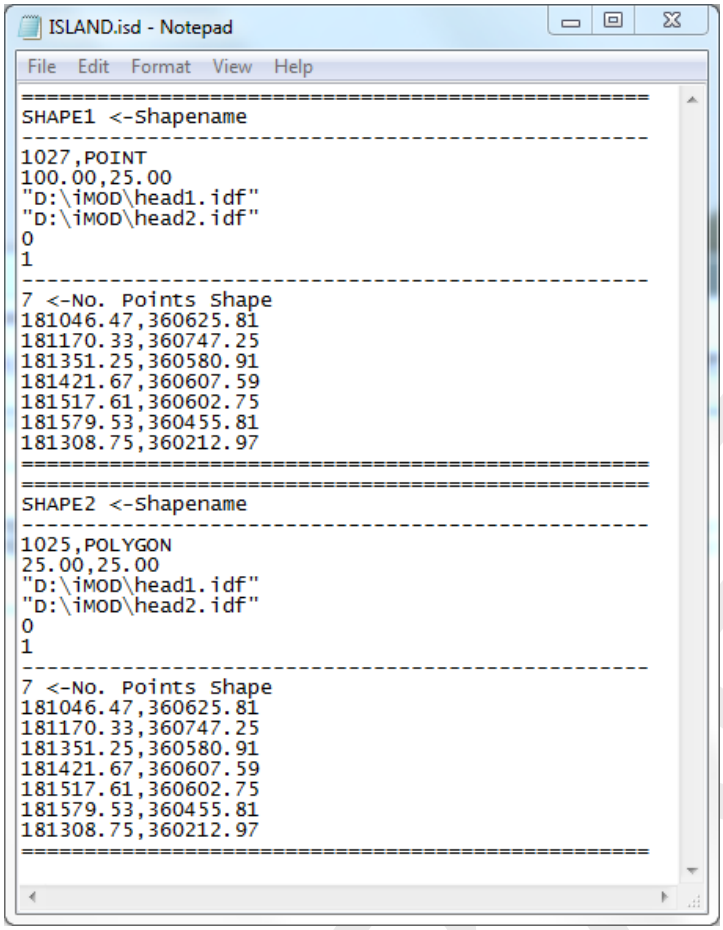


9.19 ISD-files

An ISD-file contains information about the location of startpoints used in the calculation of pathlines from model output. An ISD-file can be created by IMOD using the *Define Startpoints* option from the *Toolbox* option on the main menu. The syntax of an ISD-file is as follows:

Delimiter line	
ShapeName	Name of the shape, use quotes for a name containing more than one word
Delimiter line	
Number, Shape-type	Number of the shape type, type of shape 1027,POINT 1026,RECTANGLE 1025,POLYGON 1024,CIRCLE 1028,LINE 1139,GRID
Number, Number	Depending on the shape type: POINT: radius of the circle around the point, distance between points on the circle POLYGON: distance X, distance Y between points within the polygon CIRCLE: radius of the circle, distance between points on the circle LINE: distance between points along the line (second number not necessary) GRID: vertical offset of point, apply zero to put the points on the values of the grid (top-level file) (second number not necessary)
Top-level file / Numeric value	IDF-file defining the top level at which startpoints are defined or Numeric value defining the top level at a fixed elevation
Bottom-level file / Numeric value	IDF-file defining the bottom level at which startpoints are defined or Numeric value defining the bottom level at a fixed elevation
Number	Number indicating whether a reference level is used 0 = reference level is not used; 1 = reference level is used
Reference level	Reference level used to position the startpoints ;
Vertical interval number	The number of points to be used between the specified top-level and bottom-level
Delimiter line	
No. Points Shape	Number of points defined laterally as startpoints
Xcrd,Ycrd	X- and Y-coordinates of the startpoints
Delimiter line	

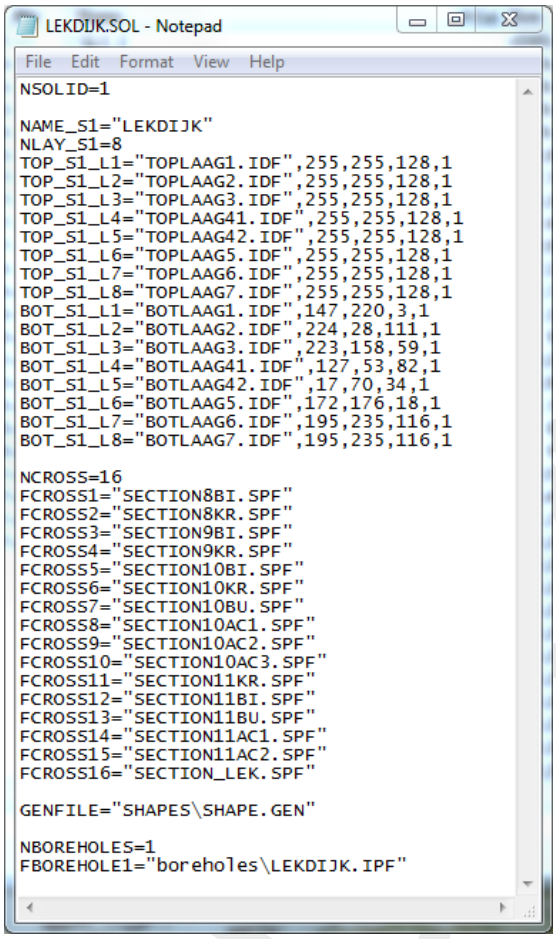
Example of an ISD-file:



9.20 SOL-files

A SOL-file describes the number of modellayers that are within a *Solid*. A solid contains several IDF-files that describe the top and bottom elevation of modellayers. Those files are listed in a SOL-file too. Each modellayer is described by two IDFs, one that stores the top elevation and one for the bottom elevation. Thereafter, a SOL-file includes a number of references to SPF-files that describe how the elevation of those top and bottom elevations vary along a cross-sectional line. A SOL-file will be created and updated by iMOD, however, it can be edited outside iMOD easily. The syntax of a SOL-file is shown below:

Example of a SOL-file with reference to 8 modellayers and 16 cross-sections:

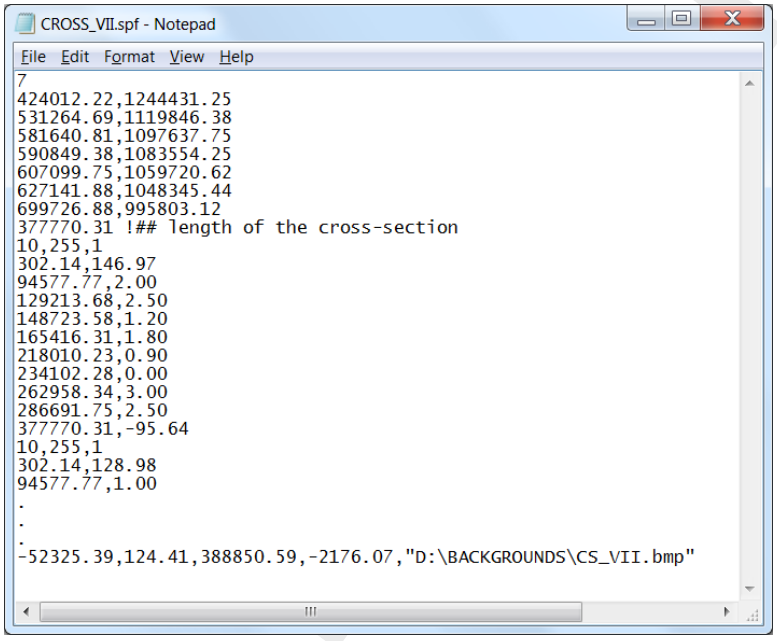


9.21 SPF-files

A SPF-file describes the variation in the top and bottom elevation along a cross-sectional line. Each SPF-file can be alternated individually within iMOD or within regular text editors, such as NotePad. The latter is not recommended however.

NCRD	Number of coordinates that describe the cross-sectional line.
X{i},Y{i}	Enter *NCRD* number of coordinates *X* and *Y*.
TotLength	Total length of the cross-section. This is used (and updated whenever the coordinates *X* and *Y* are adjusted inside and/or outside iMOD) to avoid any adjustments of the elevations outside the outer limit of the cross-sectional line.
NP{i},Clr{i},Lt{i}	Number of alternations of the top (*i* =uneven) or bottom (*i* =even) elevations. *Clr{i}* and *Lt{i}* are the linecolour (colour number for a colourdisplay of 16 million colours; 255=red) and linethickness (between 1 and 10) of the i^{th} top or bottom IDF as mentioned in the SOL-file that directs to this SPF-file.
x{i},z{i}	The distance from the first coordinate (X1,Y1) of the cross-sectional line, followed by the vertical position of the elevation (top or bottom).
x_1,y_1,x_2,y_2,*bitmap*	On the last line of the SPF file it is possible to add a bitmap (*.jpg, *.bmp or *.png) that need to be attached to the cross-section. Specify the position of the bitmap by the x_1,y_1,x_2,y_2 variables. **Note:** Be aware that iMOD uses the upper-left corner as x_1, y_1 and the lower-right corner as x_2, y_2.

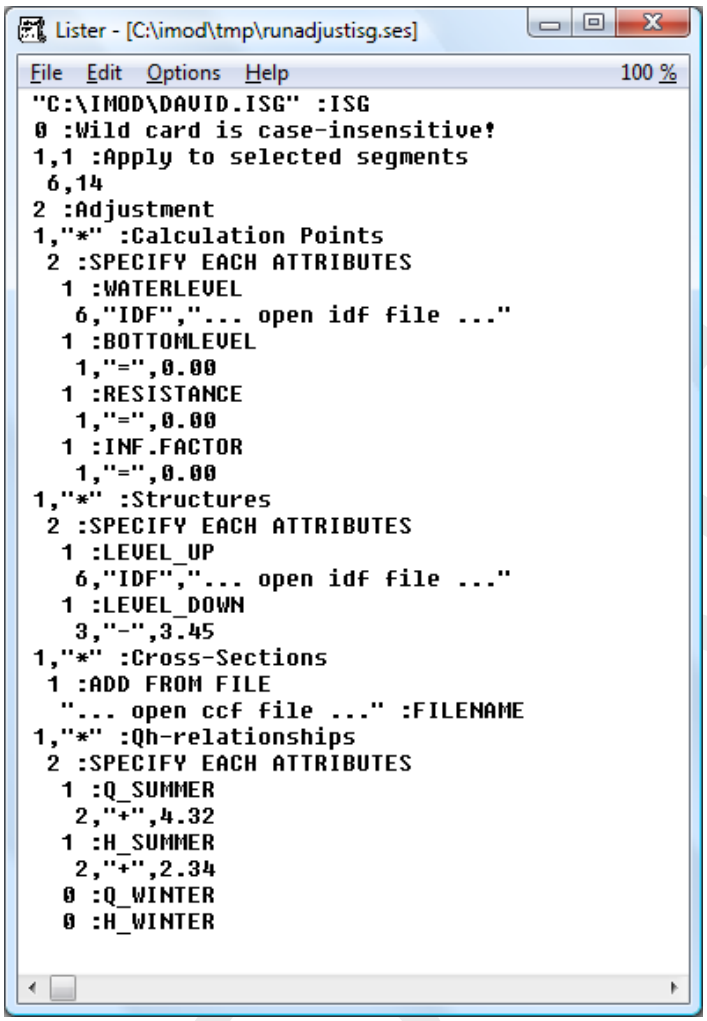
Example:



9.22 SES-files

A SES-file describes the operations that need to be carried out by *ISG Edit*. The file can be loaded into *ISG Edit* in order to recompute identical operations sequentially.

Example of a SES-file:



9.23 GEF-files

9.23.1 CPT GEF-file

A CPT GEF-file usually contains information about the penetration length, measure cone resistance, friction resistance, pore pressure, friction number, inclination, corrected depth, time, corrected cone resistance, net cone resistance, pore ratio, cone resistance number, weight per unit volume, in-situ initial pore pressure, total vertical soil pressure and effective vertical soil pressure.

Keyword	Content
#GEFID=	1,0,0
#PROCEDURECODE=	GEF-CPT-Report, 1,1,0, -
#COMPANYID=	CPT bv, Sondeerburg, 31
#PROJECTID=	CT, 380730
#FILEDATE=	1998,02,18
#TESTID=	C2-265
#FILEOWNER=	W.A. van Buuren
#COLUMN=	2

#LASTSCAN=	1251
#COLUMNINFO=	1, m, penetration length, 1
#COLUMNINFO=	2, MPa, Cone, 2
#MEASUREMENTTEXT=	9, ground level, horizontal reference level
#ZID=	31000, -2.41
#EOH=	
0.12;0.205	
0.14;0.199	
0.16;0.219	
0.18;0.252	
.....	
.....	
25.04 20.261	
25.06 21.368	
25.08 23.121	

See for a more extensive description the TNO manual "GEF-CPT-Report.pdf".

9.23.2 Borehole GEF-file

A Borehole GEF-file usually contains information about the layer depth, the numerical sand fraction ("zandmediaan" in Dutch) and the numerical gravel fraction ("grindmediaan" in Dutch). In addition information can be given on the clay percentage, silt percentage, sand percentage, gravel percentage, organic matter percentage, main soil type (sand, clay, etc.), color intensity, soil color and Carbon carbonate fraction.
Once the IPF-file has been created, this file can be loaded into iMOD and analyzed by making use of e.g. the cross-section visualization tool. Within iMOD you are able to choose the column you prefer to link to the *.DLF file legend in the 2D- and 3D-tool (see iMOD configuration in section 6.8.1).

Note: Before loading the IPF-file into iMOD check the coordinate system (projection) of the data in the IPF-file. This might be different from the default coordinate system of iMOD; WSG84, EPSG:28992 Amersfoort RDnew. A reprojection of your GEF or IPF file might be preferred.

See for a more extensive description the TNO manual "GEF-BORE-Report.pdf".

10 Runfile

To initiate a groundwater flow model simulation using iMODFLOW a *Runfile* is required. This file describes:

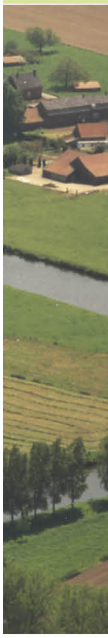
- the location of the (sub)model;
- the simulation time (number of stress periods);
- the current distribution of computational nodes (grid size);
- the current number of model layers to be used;
- the collection of files that describe the model parameters;
- the desired output variables to be saved.

A *Runfile* gives an overview of the entire model configuration and is therefore easy to transfer and reproduce. A major difference between the input files necessary for other conventional simulators (e.g. standard Modflow) is that a *Runfile* does not contain any model data, unless a constant value is assigned to a model parameter. It consists mainly of references to other files (IDFs, IPFs, ISGs, GENs) that contain the actual model data.

The philosophy of using iMODFLOW and the concept of a *Runfile* is that all the referred files are constructed for a REGION of interest rather than for an AREA of interest. A REGION of interest can be extremely large (provinces, countries); an AREA of interest is often smaller (watersheds, water management areas). The central idea is that model data is collected for large regional areas at the finest scale available for that type of model data, e.g. land use observations at a resolution of 100 x 100 meter and precipitation distribution at a scale of 1000 x 1000 meter. These files are referred to in the *Runfile* and iMOD combines the different data to the desired AREA of interest and resolution at the moment a simulation will start. Simulations for different AREA's of interest can be performed using identical model data that need to be maintained at a single location.

This chapter covers the *Runfile*-related topics:

- section 10.1: the general structure of a *Runfile*.
- section 10.2 - section 10.19: a one-by-one description of *Data Sets* 1 to 19.
- section 10.20: *Runfile*-history: any differences between *Runfile*-structures per iMOD release.
- section 10.21: all options of how a *Runfile* can be used to start a MODFLOW simulation.
- section 10.22: an example of printed MODFLOW output.
- section 10.23: a table containing the names and content of the output folders.



10.1 Runfile Description

A *Runfile* lists the configuration necessary to start a simulation. It consists of several *Data Sets* that are briefly commented in the following table.

Repeat	Data Set	Description
	1	Result folder
	2	e.g. number of model layers, stress periods, type of simulation, spatial network methodology
	3	File to monitor time-series during a simulation
	4	e.g. flags that specify number of sub models, computation mode (simulation, debug, export modes), well positioning and usage of scenarios
	5	Solver configuration (e.g. closure criterion)
	6	Location of the (sub)model and the chosen raster size
	7	*Not supported in iMOD 4.x releases.*
	8	Activated modules(9)/packages(9) and their corresponding output
	9	File that describes maximal extension of model domain
		MODULES FOR EACH LAYER
Each package (9)	**10**	Number of files
	11	Actual file and association to particular model layer
		PACKAGES FOR EACH LAYER AND STRESS PERIOD
Each stress period	**12**	Length of current stress period, name convention assigned to result files and whether results should be saved
Each package (9)	**10**	Number of files
	11	Actual file and association to particular model layer

In the following sections each of these *Data Sets* are described in more detail. Bear in mind that all input variables are free format!

10.2 Data Set 1: Output Folder

Data Set 1	**OUTPUT-FOLDER**
OUTPUT-FOLDER	Notification of the folder name in which the model results will emerge. Within this folder an output file will be created that summarizes the model simulation, such a file is called: log_[version-tag].txt. A non-existing folder will be created automatically. It is compulsory to fill in a complete path, e.g.: c:\model\test rather than a relative path, and not a relative path, e.g.: ..\..\test1

10.3 Data Set 2: Configuration

Data Set 2	**NLAY,MXNLAY,NPER,SDATE,NSCL,IFTEST,ICONCHK,IIPF, IUNCONF,IFVDL,IARMWP**
NLAY	The number of model layers that the current simulation will take into account. There is no need to adjust the Runfile as long as NLAY will be smaller or equally to MXNLAY. Whenever NLAY becomes less than MXNLAY all active model cells, within model layer NLAY, operate as a Dirichlet boundary condition and simulate a vertical flux over the edge stressed by the starting head condition (SHD, see Data Set 8) or last given heads from the constant head package (CHD, see Data Set 8) for that particular model layer.
MXNLAY	*This runfile-option is scheduled for a next iMOD release; currently MXNLAY always equals NLAY.* MXNLAY is used to determine whether the lowest model layer NLAY should be modeled as an open model boundary, which is the case whenever NLAY $<$ MXNLAY.

NPER	The number of time periods (conform the stress periods within standard MODFLOW) to be simulated. For each time period it is compulsory to define at least appropriate model input. iMODFLOW does not support the concept of time steps within time periods as standard MODFLOW does.	
SDATE	This option has become obsolete as it is now steered via the keyword ISAVEENDDATE in the iMODBatch function RUNFILE, see section section 8.6.5. For the sake of backward compatibility with earlier runfiles SDATE is still a keyword in the runfile, but acts as a dummy.	
NSCL	Flag that determines the methodology of constructing the modeling network:	
	0	The modeling network will be identical to the network that is described in the IDF file (or entered xy coordinates) given by de keyword BNDFILE, see Data Set 9. In this manner, it is possible to compute with networks that distorted spatially on different locations.
	1	The modeling network will be defined by Data Set 6. The buffer around the area of interest will be equal to the chosen grid sizes within that area of interest.
	2	The modeling network will be defined by Data Set 6 whereby the grid sizes within the buffer around the area of interest will increase up to a given maximum size.
IFTEST	Flag to indicate the mode in which iMODFLOW will perform the simulation:	
	0 (default)	Start the simulation without testing the existence of files. Any none existing file will terminate the simulation and after recovering such file the simulation need to be restarted from the beginning.
	1	*This option 1 is scheduled for a next iMOD release.* Test the entire *Runfile* on the existence of files and correct content of IPF files. The model will not be computed and at the end of the evaluation a summary is written in a file called: imodflow_[version-tag].log.
ICONCHK	Within a model configuration it can occur that within identical model cells, the drainage level from the Drainage package (DRN, see Data Set 8) and/or the drainage level from the Overlandflow package (OLF, see Data Set 8), or even other elements from a River package (RIV, see Data Set 8) are positioned below a specified water level of the River package (RIV) itself. To avoid such undesired situation, e.g. within flooded areas, these drainage systems (DRN, OLF and RIV) can be inactivated for this situation, automatically.	
	0	No adjustments of the DRN and OLF packages
	1	The DRN, OLF and/or RIV systems are turned off whenever their drainage levels are below existing river water levels (RIV-package) in corresponding model cells.
IIPF	This flag indicates whether time series need to be computed during the simulation.	
	0	No time series included
	n	Include *n*-number of IPF files with location of time series to be computed for each stress period throughout the simulation, See Data Set 3.
	-n	Apply an interpolation of computed heads toward the location of the measurement.
IUNCONF *(optional)*	This flag indicates the usage of unconfinedness for aquifers.	
	0	All aquifers are confined, unconfinedness is supported via the NAM-file export mode only, see section section 5.5 and section 8.6.5.
IFVDL *(optional)*	This flag indicates the methodology to be used computing the conductance for streams given in the ISG package. The default value is IFVDL=0.	
	0	River conductance is a function of river length (m), wetted perimeter (m) and river resistance (d), solely.
	1	River conductance is a function of river length (m), wetted perimeter (m), river resistance (d), permeability and thickness of appropriate aquifer, resistance of first aquitard.
	Whenever IFVDL=1, it is obliged to have the keywords TOP and BOT activated or the package SFT should be active	

IARMWP	Use this to specify artificial recharge to be read from an IPF file

10.4 Data Set 3: Timeseries (optional)

Data Set 3	Apply only whenever IIPF<>0 from Data Set 2 **IPF_TS,IPFTYPE,IXCOL,IYCOL,ILCOL,IMCOL,IVCOL** (all compulsory)		
IPF_TS	Notification of an IPF filename that stores the location of measures to be monitored as time series throughout the simulation.		
IPFTYPE	Type of the IPF file		
	1	For a steady-state simulation the 5 attributes within the given IPF should be (in this order):	
		X	x coordinate
		Y	y coordinate
		ILAY	Model layer identification
		OBS	Observed value (OBS=0.0 if no observation is available)
		VAR	variance of the measurement $\sigma^2 = \mathrm{Var}(X)$ (VAR=1.0 if no observation is available)
		After the simulation the given IPF is copied to the folder [OUTPUT-FOLDER]\[ipfname] and contains the additional records:	
		COMP	Computed values
		DIFF	Difference (COMP-OBS)
		DIFFW	Weighed difference (COMP-OBS)*$\left(\sqrt{\sigma^2}\right)^{-1}$
	2	For the transient simulation the attributes within the given IPF should be (in this order):	
		X	x coordinate
		Y	y coordinate
		ILAY	Model layer identification
		After the simulation the given IPF is copied to the folder [OUTPUT-FOLDER]\[ipfname] and contains the additional record:	
		ID	Reference ID to the individual time series which are stored within the file [OUTPUT-FOLDER]\timeseries\location_[i].txt.
	Whenever the IPF_TS consists of time series initially, those will be included in the final time series. Time series within the buffer zone (see Data Set 6) will be left out.		
IXCOL	Specify the column number in the IPF file (**IPF_TS**) that is representative for the X-coordinate of the measurement.		
IYCOL	Specify the column number in the IPF file (**IPF_TS**) that is representative for the Y-coordinate of the measurement.		
ILCOL	Specify the column number in the IPF file (**IPF_TS**) that is representative for the layer identification of the measurement.		
IMCOL	Specify the column number in the IPF file (**IPF_TS**) that is representative for the measurement value. Whenever the number of columns in the IPF file be less than 4, **IMCOL=0** and no observations will be used.		
IVCOL	Whenever the number of columns in the IPF file be less than 5, **IMCOL=0** and no observations will be used.		
	>0	Specify the column number in the IPF file (**IPF_TS**) that is representative for the variance of the measurement $\mathrm{Var}(X)$. The variance of a random variable X is its second central moment, the expected value of the squared deviation from the mean $\mu = \mathrm{E}[X]$, thus: $\mathrm{Var}(X) = \mathrm{E}\left[(X-\mu)^2\right]$. This definition encompasses random variables that are discrete, continuous, neither, or mixed. The variance (σ^2) can also be thought of as the covariance of a random variable with itself: $\mathrm{Var}(X) = \mathrm{Cov}(X,X) = \sigma^2$.	
	<0	Whenever **IVCOL** is negative (e.g. **IVCOL=-5**), it means that instead of variances (σ^2), weight values w are entered. Weight values are computed as $w = \left(\sqrt{\sigma^2}\right)^{-1}$.	

10.5 Data Set 4: Simulation mode

Data Set 4	**NMULT,IDEBUG,IDOUBLE,IPOSWEL,ISCEN,IBDG,MINKD,MINC**	
NMULT	The number of areas to be computed sequentially. The areas need to be defined in Data Set 6. NMULT need to be 0 in case NSCL=0.	
IDEBUG	A flag indicating the output frequency during a simulation:	
	0	Default configuration whereby a solution of the simulation is saved on disk after convergence, or whenever the number of iteration exceeds MXITER (see Data Set 5) and the simulation terminates thereafter.
	1	Debug configuration whereby both input for the activated packages and intermediate solutions during the iteration process are saved on disk.
IDOUBLE	This option replaces the previous IEXPORT option which became obsolete since v3-series. By default IDOUBLE=0, which means that the results of the simulation will be single precision values. If IDOUBLE=1, the results will be double precision, keep in mind that in that case, all files will be doubles in size as well.	
IPOSWEL	Flag indicating the assignment of well onto the model network:	
	0	Assignment of the well onto the single model grid cell is based on the x and y coordinates of the well.
	1	*Not supported in iMOD 4.x releases.*
ISCEN	Flag indicating the usage of scenario definitions:	
	0	No usage of scenarios
	1	*Not supported in iMOD 4.x releases.*
IBDG *(optional)*	Flag indicating the definition of budget computation for packages:	
	0	*Not supported in iMOD 4.x releases.*
	1 (default)	Fluxes from packages (except CHD) within one single model cell are saved separately. As a result the output file name convention (see ILSAVE in Data Set 8) will include a *system* number, e.g. *_sys1_*; *_sys12_*.
MINKD *(optional)*	Minimal value for transmissivity (m^2/day). Use MINKD > 0 for reasons of stability in combination with horizontal anisotropy.	
MINC *(optional)*	Minimal value for the vertical resistance (days). Use MINC > 1 for reasons of stability and to avoid large contrasts between horizontal en vertical conductances. Especially whenever the cell size (CELLSIZE) increases, it could be advisable to specify MINC > 1.	

10.6 Data Set 5: Solver configuration

Data Set 5 *(optional PCG)* *(optional PKS)*	**OUTER,INNER,HCLOSE,QCLOSE,RELAX** **,NPCOND,MAXWBALERROR** **,PARTOPT,IDFMERGE**
OUTER	Maximal number of "outer" iteration loops. The iterative procedure used in MODFLOW for solving nonlinear problems is commonly referred to as Picard iteration. It splits the solving process into an outer iteration loop, the equation that needs to be solved in the "inner" is (re)formulated.
	If OUTER < 0, the Parallel Krylov Solver (PKS) is activated instead of PCG, and for this solver OUTER = -OUTER is taken as the number of outer iterations. For PKS, the supported preconditioning is Incomplete LU-factorization only, which is automatically being set. Hence, you are not allowed to set NPCOND when PKS is activated.
INNER	Maximal number of "inner" iteration loops. Within each inner iteration loop, the equation that was formulated in the outer iteration loop is (partly) solved. In common, it is more expensive to have much inner iteration though each iteration loop represents a temporary formulation of the equation. True linear systems can speed up drastically by increasing the number of inner iterations and RELAX $= 1$, however, most groundwater models are a mixture of linear and non-linear elements and therefore a fair trade-off between robustness and speed seems to be NITER $= 20$.

HCLOSE	**m**	Closure criterion for the hydraulic head (state variable). Commonly it is practice to choose HCLOSE to be 2 orders of magnitude smaller than the desired accuracy to be obtained.
QCLOSE	**m**^{3}	Closure criterion for the mass balance. This criterion depends on the grid size, since large grid cells produce larger errors in mass balances than smaller ones does.
RELAX	Relaxation factor that quantifies the amount of confidence for each solution obtained after an inner iteration loop, default value is 0.98. It influences the robustness and efficiency of convergence. For purely linear systems it can/must be 1.0, though non-linear system prefers lower values (e.g. 0.50-0.97). It is difficult to know the optimal value for RELAX beforehand. Use the adaptive damping (IDAMPING) instead.	
NPCOND *(optional, PCG only)*	Pre-conditioning method. If the Preconditioning Method is set to Cholesky, the Relaxation parameter can be set. Although the default is 1, in some cases a value of 0.97-0.99 may reduce the number of iterations required for convergence.	
	1	Modified Incomplete Cholesky (for use on scalar computers)
	2	Polynomial matrix conditioning method (for use on vector computers or to conserve computer memory)
MAXW-BALERROR *(optional, PCG only)*	Maximal overall acceptable error for the water balance in percentage (the default value = 0.0% and the solver stops whenever the criterion of QCLOSE cannot be met). Whenever the external-iteration does not converge (due to numerical instability), the simulation will continue whenever the overall error in de the water balance is less than the given MAXWBALERROR criterion.	
PARTOPT *(optional, PKS only)*	Subdomain partition option. There are two methods supported: PARTOPT $= 0$ (default) for uniform partitioning in lateral x and y-direction; PARTOPT $= 1$ that enables the Recursive Coordinate Bisection partitioning method computes the subdomain dimensions according to a load pointer IDF grid. Note that each subdomain always includes all model layers.	
	0	Uniform subdomain partitioning (default)
	1	Recursive Coordinate Bisection (RCB) subdomain partitioning
IDFMERGE *(optional, PKS only)*	Flag for merging parallel subdomain IDF output files. The default is IDFMERGE $= 0$, corresponding to no merging. In this case, each subdomain writes its output IDF files like "<*IDFNAME*>_p<*MPIRANK*>.idf", where "<*IDFNAME*>" is the iMOD output variable name, e.g. "head_steady-state_l1" and "<*MPIRANK*>" the three-digit subdomain MPI rank identifier. Note that enabling this option could slow down overall parallel computations.	
	0	No merging of subdomain IDF output files (default)
	1	Merging of subdomain IDF output files

10.7 Data Set 5a: RCB load pointer grid (optional)

Data Set 5a	**LOADFILE**
LOADFILE	IDF file that represents with weights to be used when PKS is enabled (Data Set 5: OUTER < 0) and the RCB subdomain partitioning option is enabled (Data Set 5: PARTOPT $= 1$). Only the absolute weights are being used for the subdomain partitioning.

10.8 Data Set 6: Simulation window (optional)

Data Set 6	Apply whenever NMULT=1 and NSCL=1 **XMIN,YMIN,XMAX,YMAX,CSIZE,BUFFER** Apply whenever NMULT$>$1 and NSCL=1 **IACT,XMIN,YMIN,XMAX,YMAX,CSIZE,BUFFER,CSUB** Apply whenever NMULT=1 and NSCL=2 **XMIN,YMIN,XMAX,YMAX,CSIZE,MAXCSIZE,BUFFER,CSUB** Apply whenever NMULT$>$1 and NSCL=2 **IACT,XMIN,YMIN,XMAX,YMAX,CSIZE,MAXCSIZE,BUFFER,CSUB**
IACT	Flag that determines the whether a sub model need to be computed:

	-1	Sub model will be computed only whenever the result folder does not exist
	0	Sub model will not be computed
	1	Sub model will be computed and overwrite existing results if available
XMIN	**m**	Lower left X-coordinate of the area of interest
YMIN	**m**	Lower left Y-coordinate of the area of interest
XMAX	**m**	Upper right X-coordinate of the area of interest
YMAX	**m**	Upper right Y-left coordinate of the area of interest
CSIZE	**m**	Grid cell size within the area of interest and within the buffer.
MAXCSIZE	**m**	This is the maximum grid cell size within the buffer. Within the buffer the entered grid cell size CSIZE, will increase gradually up to MAXCSIZE. Apply whenever NMULT$>$1 and NSCL=2.
BUFFER	**m**	This represents the size of the buffer around the area of interest. The total simulation model will have a total width of (XMAX-XMIN)+2*BUFFER and a total height of (YMAX-YMIN)+2*BUFFER.
CSUB (*optional*)	This is the name of the result folder for the current sub model, yielding [OUTPUT-FOLDER]\[CSUB]\as result folder. Whenever no name is given, the default folder name will be submodel[i], where i represents the ith sub model within NMULT.	

10.9 Data Set 8: Active packages

Data Set 8	**IPM,NLSAVE,ILSAVE(NLSAVE),KEY**

IPM This represents whether a specific time independent module (mod) / time dependent package (pck) is active in the current simulation. The following package are supported:

	Key	**Act**	**Description**
(mod)	**CAP**	**0**/**1**	Usage of the unsaturated zone package
(mod)	**BND**	**0**/**1**	(compulsory) Usage of boundary conditions
(mod)	**SHD**	**0**/**1**	(compulsory) Usage of starting heads
(mod)	**KDW**	**0**/**1**	Usage of hydraulic conductance
(mod)	**VCW**	**0**/**1**	Usage of vertical resistances
(mod)	**KHV**	**0**/**1**	Usage of horizontal permeabilities
(mod)	**KVA**	**0**/**1**	Usage of vertical anisotropy for aquifers
(mod)	**KVV**	**0**/**1**	Usage of vertical permeabilities
(mod)	**STO**	**0**/**1**	Usage of storage coefficients
(mod)	**TOP**	**0**/**1**	Usage of top of aquifers
(mod)	**BOT**	**0**/**1**	Usage of bottom of aquifers
(mod)	**PST**	**0**/**1**	Usage of parameter estimation
(mod)	**PWT**	**0**/**1**	Usage of the purge-water table package
(mod)	**ANI**	**0**/**1**	Usage of the horizontal anisotropy package
(mod)	**HFB**	**0**/**1**	Usage of the horizontal flow barrier package
(mod)	**IBS**	**0**/**1**	Usage of interbed storage/subsidence
(mod)	**SFT**	**0**/**1**	Usage of streamflow thickness
(pck)	**WEL**	**0**/**1**	Usage of the well package
(pck)	**DRN**	**0**/**1**	Usage of the drainage package
(pck)	**RIV**	**0**/**1**	Usage of the river package
(pck)	**EVT**	**0**/**1**	Usage of the evapotranspiration package
(pck)	**GHB**	**0**/**1**	Usage of the general-head-bound. Package
(pck)	**RCH**	**0**/**1**	Usage of the recharge package
(pck)	**OLF**	**0**/**1**	Usage of the overland flow package
(pck)	**CHD**	**0**/**1**	Usage of the constant-head package
(pck)	**ISG**	**0**/**1**	Usage of the segment package

It is easy to turn an IPM on or off by assigning 0 (off) or 1 (on). There is no need to adjust the *Runfile* for these adjustments. It is not necessary to include all the packages and/or packages in a runfile and the order you specify is irrelevant. Of course, the packages BND, SHD, (KDW or KHV), (VCW or KVV) are obliged for any (multi-layered) model!

NLSAVE Determines the number of model layers for which output need to be saved on disk. NLSAVE may be larger than NLAY, however all layers that exceed the current simulation will be neglected.
Important Note: the NLSAVE settings for BND/bdgbnd or STO/bdgsto determine the layers to be saved for all fluxes (bdgbnd, bdgsto, bdgflf, bdgfrf and bdgfff). This means that settings for only KHV or only KVV are always neglected and will be overruled by the settings given for BND or STO.
Important Note 2: the NLSAVE settings for ISG/bdgisg determine the layers to be saved for the river fluxes (bdgriv). The NLSAVE settings for the RIV-package are overruled by the settings given for ISG.

ILSAVE This parameter stores the model layers for each IPM keyword for ILSAVE model layers. The model layers may be given in any order, e.g. 4, 5, 1. Any layer that exceeds the current NLAY is neglected.
Important Note: the ILSAVE settings for BND/bdgbnd or STO/bdgsto determine the layers to be saved for all fluxes (bdgbnd, bdgsto, bdgflf, bdgfrf and bdgfff). This means that settings for only KHV or only KVV are always neglected and will be overruled by the settings given for BND or STO.
Important Note 2: the ILSAVE settings for ISG/bdgisg determine the layers to be saved for the river fluxes (bdgriv). In case ILSAVE settings are defined for RIV, these are overruled by the settings given for ISG.

=0	Identifies all modellayers
>0	Modellayer identification

The following result will be saved:

Key	NAME	Unit	Description
PST	-	-	No output available
CAP	BDGCAP	m^3/day	Flux between the unsaturated zone and the saturated zone
BND	BDGBND	m^3/day	Flux for constant head boundaries
SHD	HEAD	m	Hydraulic head
KDW	BDGFRF	m^3/day	Flux over the eastern model faces
	BDGFFF	m^3/day	Flux over the southern model faces
VCW	BDGFLF	m^3/day	Flux over the lower model faces
KHV	BDGFRF	m^3/day	Flux over the eastern model faces
	BDGFFF	m^3/day	Flux over the southern model faces
KVV	BDGFLF	m^3/day	Flux over the lower model faces
STO	BDGSTO	m^3/day	Fluxes for storage
TOP	-	-	No output available
BOT	-	-	No output available
PST	\PEST folder	-	No output available, other than *.txt and *.ipf files that write performance and residuals of optimalisation
KVA	-	-	No output available
PWT	-	-	No output available
ANI	BDGANI	m^3/day	Fluxes caused by anisotropy
HFB	-	-	No output available
IBS	BDGIBS	m^3/day	Fluxes for interbed storage
SFT	-	-	No output available
WEL	BDGWEL	m^3/day	Fluxes for wells
DRN	BDGDRN	m^3/day	Fluxes out drainage
RIV	BDGRIV	m^3/day	Fluxes for rivers
EVT	BDGEVT	m^3/day	Fluxes out evapotranspiration
GHB	BDGGHB	m^3/day	Fluxes for general head boundaries
RCH	BDGRCH	m^3/day	Fluxes for recharge
OLF	BDGDRN	m^3/day	Fluxes out overland flow.
CHD	BDGBND	m^3/day	Flux over constant head boundaries

ISG	BDGISG	m^3/day	Fluxes for rivers (*fast SOBEK*)

All fluxes that extract water from a specific model cell are negative. Therefore, seepage water values are negative, within the file BDGFLF. Identical elements within one single model cell are lumped together in the output file, e.g. fluxes from different drainage systems in a single model cell add together. The naming convention for all files is:

DELT=0	[NAME]_[SNAME]_l[ILAY].idf; e.g. head_steady-state_l1.idf or bdgflf_quarter_l10.idf
DELT>0	[NAME]_[yyyymmdd]_l[ILAY].idf; e.g. head_20101231_l1.idf or bdgdrn_20110814_l1.idf

Whenever the option IBDG=1 (see Data Set 4), the naming convention will be including the system number of the package, e.g.: bdgriv_20110814_l1_**sys1**.idf The number of system(s) is defined by the NFILES parameter in Data Set 10.

KEY	Packages should be specified between brackets: "(" and ")" so iMODFLOW can recognize the keyword. It will be used to compare this with the variable KEY in Data Set 10.

10.10 Data Set 9: Boundary file

Data Set 9	**BNDFILE** or **XMIN,YMIN,XMAX,YMAX**
XMIN,YMIN, XMAX,YMAX	Enter the coordinates of the entire model at maximum extension (area of regional interest). Beyond these limits the model will give an error and below these limits constant head boundary conditions will be applied, automatically whenever the area of interest is smaller.
BNDFILE	IDF file that represents the entire model at maximum extension (area of regional interest). This file will be used differently for the following flag values for NSCL (see Data Set 2):

NSCL=0	This file will be used to determine whether it extends the given area of local interest defined in Data Set 6. If so the area of local interest will be trimmed to fit the area of regional interest. On the other hand, whenever the area of local interest is smaller (in many cases), the boundary nodes along the cutting edges are transformed into "open"-boundaries. The starting heads (SHD) or constant-head (CHD) values fixate these boundaries.
NSCL>0	The network as described in the given IDF file is used for the modeling simulation. Any network schematization is accepted as long as it will not exceed the maximum extension of the area of regional interest.

10.11 Data Set 10: Number of files

Data Set 10	**NFILES,KEY**
NFILES	This expresses the number of entries that will follow, zero entries can be defined by NFILES=0. It is possible the reuse the entries obtained in the previous stress period by assigning the value NFILES=-1. **For several package a single entry consist of multiple files (*parts*).** Moreover: each individual part of a package should be repeated NFILES times before entering the next part of a package; whenever a single nodata value is read for one of the individual parts of a package for a particular location, the package on that particular location will be turned off!; See table below for the individual parts (No.) of each package (they should be entered in this order!):

Key	No.	Unit	
PST	1	-	Number of parameters to be estimated, see Data Set 14, 15, 16 and 17 for more specific input information.
CAP	n		Number of input files (this is needed to determine the number of files to be copied, which equals this number – the 22 (or 21 whenever IARMWP=1) compulsory IDF/IPF files)

1	-	**BND**	Boundary setting, used to specify active MetaSWAP elements
2	-	**LGN**	Landuse code, should be referred to by the file luse_svat.inp
3	cm	**RTZ**	Rootzone thickness (min. value is 10 centimeter).
4	-	**SFU**	Soil Physical Unit should be referred to by fact_svat.inp.
5	-	**MET**	Meteo Station number should be referred to by mete_svat.inp.
6	m+MSL	**SEV**	Surface Elevation.
7a	-	**ART**	(if IARMWP=0) Artificial Recharge (= Irrigation) Type, 0=no occurrence, ART$>$0 means present at current location whereby ART=1: from groundwater, ART=2: from surface water extraction
7b	-	**ART**	(if IARMWP=1) Location of the actual Artificial Recharge, the value of each location refers to the attribute ID in the IPF-file (see dataset 8b)
8a	-	**ARL**	(if IARMWP=0) Artificial Recharge (= Irrigation) Location, number of model layer from which water is extracted.
8b	-	**ARL**	(if IARMWP=1) IPF with locations (X,Y) for Artificial Recharge (= Irrigation), the number of model layer (ILAY) from which water is extracted, the identification (ID) of the area on which the recharge is applied and the capacity (CAP) in mm/day. All of those as separate columns in the IPF file, thus the following fields: X,Y,ILAY,ID,CAP. Here, the source of the artificial recharge is always groundwater.
9	mm/d	**ARC**	(if IARMWP=0) Artificial Recharge (= Irrigation) Capacity. The capacity of the irrigation installation. The applied capacity depends on the duration of the irrigation (part of a day) as specified in the file luse_svat.inp
10	m2	**WTA**	Wetted Area specifies the total area occupied by surface water elements. Value will be truncated by maximum cellsize.
11	m2	**UBA**	Urban Area, specifies the total area occupied by urban area. Value will be truncated by maximum cellsize.
12	m	**PDU**	Ponding Depth Urban Area, specifying the acceptable depth of the ponding of water on the surface in the urban area before surface runoff occurs
13	m	**PDR**	Ponding Depth Rural Area. Same as above but for rural area.
14	day	**OFU**	Runoff Resistance Urban Area, specifying the resistance surface flow encounters in the urban area. The minimum value is equal to the model time period.
15	day	**OFR**	Runoff Resistance Rural Area. Same as above but for rural area.

	16	day	**ONU**	Runon Resistance Urban Area, specifying the resistance surface flow encounters to a model cell from an adjacent cell in the urban area. The minimum value is equal to the model time period.
	17	day	**ONR**	Runon Resistance Rural Area. Same as above but for rural area.
	18	m/d	**QIU**	QINFBASIC Urban Area, specifying the infiltration capacity of the soil surface in the urban area. The range is 0-1000 m/d. The NoDataValue -9999 indicates unlimited infiltration is possible.
	19	m/d	**QIR**	QINFBASIC Rural Area. Same as above but for rural area.
	20	m+MSL	**PWT**	Level of the Perched Water Table level. When groundwater falls below this level then the capillary rise becomes zero.
	21	-	**SFC**	Soil Moisture Factor to adjust the soil moisture coefficient. This factor may be used during calibration. Default value is 1.0.
	22	-	**CFC**	Conductivity Factor to adjust the vertical conductivity. This factor may be used during calibration. Default value is 1.0.
	..n			Remaining files will be copied to the simulation folder as set by OUTPUTNAME (Data Set 1)
BND	1	-		IDF with boundary settings; 0 = inactive, $>$0 = active, $<$0 = fixed for each model layer
SHD	1	m+MSL		IDF with starting heads for each model layer. Inactive cells will be transformed to nodata value -999.99
KDW	1	m^2/day		Transmissivity for each model layer (trimmed internally to be minimal 0 m^2/day)
VCW	1	days		Vertical Resistance between model layers (trimmed internally for minimal 0.001 days). For reasons of scaling, it is important to assign the nodata value for VCW to be zero!
KHV	1	m/day		Horizontal Permeability for each model layer.
KVV	1	m/day		Vertical Permeability for each aquitard (in between modellayers!). KVV is assumed to be 1/3*KHV for the modellayers!
STO	1	-		Storage coefficient for each model layer, i.e. the specific storage coefficient multiplied with the thickness of the model layer, for the first unconfined model layer, enter the specific storage coefficient instead, e.g. 0.15.
TOP	1	m+MSL		Top of the aquifer.
BOT	1	m+MSL		Bottom of the aquifer.
KVA	1	-		Vertical anisotropy for aquifers
PWT	6	-		Layer identification of the PWT unit; elements with values $<$= 0 will be removed.
		-		Storage coefficient of the phreatic part underneath the PWT layer
		m+MSL		Top of the PWT layer
		m		Thickness of the PWT layer

		m	Thickness of layer of the aquifer above the PWT layer in which the transmissivity will be adjusted.
		day	Vertical resistance of the clay underlying the PWT unit. This should be larger or equal to the given C value of the PWT layer, otherwise the C value will be used given by the module VCW.
ANI	2	-	The anisotropic factor **perpendicular** to the main principal axis (axis of highest permeability). Factor between 0.0 (full anisotropic) and 1.0 (full isotropic). **Do not use a nodata value of 0.0 since this will deactivate the package!**
		degrees	The angle along the main principal axis (highest permeability) measured in degrees from north (0), east (90), south (180) and west (270). **Do not use a nodata value of 0.0 since this will deactivate the package!**
HFB	1	-	GEN file (*.gen) defining the location and FCT values of faults/horizontal barriers. When GEN files are assigned to layer number 0, iMOD will assign the fault to the appropriate model layers automatically; in that case the GEN file needs to be a 3D GEN (see section 9.10). When the TOP and BOT of the aquifer are defined in the runfile, the FCT value is assigned to a resistance r; otherwise FCT is assigned to a factor f that is used to multiply the conductances between cells. When FCT (so factor f or the resistance r) is zero the barrier is impermeable! See section 12.15 for details on how the conductance between cells is calculated.
IBS	4	m+MSL	Preconsolidation head or preconsolidation stress in terms of head in the aquifer. Preconsolidation head is the previous minimum head value in the aquifer. For any model cells in which specified HC is greater than the corresponding value of starting head, value of HC will be set to that of starting head.
		-	The dimensionless elastic storage factor for interbeds present in model layer. The storage factor may be estimated as the sum of the products of elastic skeletal specific storage and thickness of all interbeds in a model layer.
		-	The dimensionless inelastic storage factor for interbeds present in model layer. The storage factor may be estimated as the sum of the products of inelastic skeletal specific storage and thickness of all interbeds in a model layer.
		m	The starting compaction in each layer with interbed storage. Compaction values computed by the package are added to values in this array so that printed or stored values of compaction and land subsidence may include previous components. Values in this array do not affect calculations of storage changes or resulting compaction. For simulations in which output values are to reflect compaction and subsidence since the start of the simulation, enter zero values for all elements of this array.
SFT	2	m+MSL	Stream Flow Thickness
		m/d	Permeability
WEL	1	m^3/day	An IPF file with:

			1	Three columns representing the x,y coordinate and the rate, e.g.: *x,y,q,{z1,z2}*
			2	Two columns representing the x,y coordinate and a third column referring to associated files with time-variant rates, e.g. x,y,[id],*{z1,z2}*
			The parameters z1 and z2 express the screen of the well and are optional. Use these parameters in combination with ILAY=0 (see Data Set 11)	
	DRN	2	m^2/day	Conductance of the drainage system within a single model cell; elements with values $<=$ 0 will be removed.
			m+MSL	Elevation of the drainage system.
	RIV	4	m^2/day	Conductance of the drainage system within a single model cell; values need to be $\geq$ 0.
			m+MSL	Elevation of the water level.
			m+MSL	Elevation of the bottom level. Whenever the elevation of the bottom level is higher than the entered elevation of the water level, iMODFLOW will adjust internally the elevation of the bottom level to be equal to the elevation of the water level. Any corrections made are listed by a negative system number in the *.LST file whenever the IDEBUG flag is set on 1.
			-	Infiltration factor:
				=0 No infiltration is allowed
				$>$0 Infiltration is allowed whenever the head is below the stage up to a maximum (stage minus bottom pressure) whenever the head is less than the bottom. Infiltration conductance is calculated as: river conductance * infiltration factor.
	EVT	3	mm/day	*This option can only be used in combination with ILAY=1* Evapotranspiration strength.
			m+MSL	Top elevation for maximal evapotranspi-ration strength.
			m	Thickness in which evapotranspiration strength reduced to zero.
	GHB	2	m^2/day	Conductance of the general head system within a single model cell; elements with values $<=$ 0 will be removed.
			m+MSL	Elevation at the general head boundary.
	RCH	1	mm/day	*This option can only be used in combination with ILAY=1* Recharge strength.
	OLF	1	m+MSL	Surface elevation where above overland flow takes place; elements with values equal to the no-data value will be removed.
	CHD	1	m+MSL	Elevation of constant heads at the location where BND $<$ 0 only.
	ISG	1	-	Specific segment file for the simulation of water systems directly from vectors.
KEY	Text string that identifies the Key of the module/package listed by Data Set 10.			

10.12 Data Set 11: Input file assignment

Data Set 11	**ILAY,FCT,IMP,FNAME**	
ILAY	This represents the model layer to which the input data assigns to. For the package: BND, SHD, KDW, VCW, STO, TOP, BOT and ANI, it is not sustained to assign more input data to identical model layers. However, multiple assignments to identical model layers are sustained for the other packages: PWT, HFB, WEL, DRN, RIV, EVT, GHB, RCH, OLF, CHD and ISG. ILAY can be used as follows:	
	>0	Expresses the model layer number to which the module and/or package is assigned to
	=0	Automatic allocation of model layers to packages. It is compulsory to have included the TOP and BOT package (see Data Set 8). Only the following packages are affected by ILAY=0:
		HFB The given elevation in a 3-D GEN file (see section 9.10) will be used to determine the actual model layer, use the iMODBatch function GEN2GEN3D to construct those 3-D GEN files (see section 8.4.2)
		WEL Given IPF should contain the records, X,Y,Q,Z1,Z2. Z1 and Z2 will be used to assign well strength to the appropriate model layer(s)
		DRN The given elevation will be used to determine the actual model layer
		RIV Both, the given stage and bottom elevation will be used to determine the appropriate model layer(s)
		GHB The given elevation will be used to determine the model layer.
		OLF The given elevation will be used to determine the model layer.
		ISG Both, the given stage and bottom elevation will be used to determine the appropriate model layer(s). Moreover, the specified wetted perimeter will be used to adjust conductances.
	<0	Assign package to the highest active model cell with a BND-value > 0 (see Data Set 10, Key=BND). Every package (except the CHD package) can be affected by ILAY<0; package however, are not supported.
FCT	The multiplication factor for the input data (nodata values excluded). It is possible to use the FCT parameter to assign mean values, e.g. apply FCT=0.5 for conductance's for rivers in summer and winter periods. Moreover, it is possible to compute a weighed mean of two periods within the package assigned to the identical model layer.	
IMP	Addition for the input data (nodata values excluded). Mathematical order is that multiplication (FCT) comes before the addition (IMP).	
FNAME	The name of the input file, although it is sustained, it is preferable to note the FNAME with an absolute path (e.g. c:\fname) rather than a relative path (..\fname). The following format of these files are assigned to the existing package:	
	Format	**Module/package**
	IDF	CAP,BND,SHD,KDW,VCW,KHV,KVV,STO,PWT,ANI,CHD, DRN,RIV,EVT,GHB,RCH,OLF,IBS,TOP,BOT,KVA
	IPF	WEL
	GEN	HFB
	ISG	ISG
	In case the input parameters are constant over the entire modeling domain, a constant value can be given. An exception to this is made for the packages WEL,HFB and ISG packages	

10.13 Data Set 12: Time discretisation

Data Set 12	**KPER,DELT,SNAME,ISAVE,ISUMSAVE**
KPER	The number of the stress period. It will be used solely to verify whether the current stress period matches the stress period read. If not a warning appears in the log file.
DELT	**day** The length of the current stress period.
SNAME	The date or name for the current stress period.

	DELT=0	For steady-state simulations, it should state “steady-state”, mainly for reasons of compatibility with iMOD.
	DELT>0	For transient simulations it should state the date notated as: yyyymmdd; e.g. 20101231 to express the 31^{th} of December 2010. Usage of these format is recommended strongly for compatibility with iMOD (e.g. time-series plotting).
	As a consequence, all result files will show the given date/name in their names, e.g. head_[yyyymmdd]_l1.idf or bdgflf_steady-state_l8.idf.	
ISAVE	This parameter defines whether output (as defined by Data Set 8) is generated for the current stress period.	
	-1	Result will be saved with the buffer excluded
	0	No results will be saved
	1	Results will be save with the buffer included
	In case NMULT>1 (see Data Set 4), ISAVE will become abs(ISAVE) because the proper merging procedure will use the results in the buffer area.	
ISUMSAVE *(optional)*	This optional parameter allows to sum all fluxes for each package per model layer. If more package entries are defined per model layers, those will be lumped into a single budget quantity.	
	0	Explicitly save all budget per package entry as a separate results file.
	1	Sum all individual fluxes per model layer for the package entries.

10.14 Data Set 14: Parameter Estimation – Main settings

Data Set 14	**PE_MXITER,PE_STOP,PE_SENS,PE_NPERIOD,PE_NBATCH, PE_TARGET(.),PE_SCALING,PE_PADJ,PE_DRES,PE_KTYPE**	
PE_MXITER	MXITER can have different meanings:	
	<0	iMODFLOW will be run a single run and adjust all parameters accordingly and than stop.
	=0	If PE_MXITER is equal to zero, a sensitivity matrix will be computed yielding Jacobian values (finite difference between the change in head and the parameter update) for the entire zones. Those values will be written to disk in .\head\head_{date}_l{i}_sens_{param}_ils{ils}.idf. Those values can be helpful to estimate the adjustment to a parameter to yield a desired improvement of the head and/or flux (assuming the model act linearly). The process will stop whenever all parameters are perturbed.
	>0	Maximum number of iterations.
PE_STOP	Stop criterion whenever decrease of objective function J becomes less or equal to the ratio J_i/J_{i-1}. Entering a value of 0.1 means than the optimization stops whenever the objective function value J_i for the current optimization step i, is reduced less than 10% of the last objective function value J_{i-1}.	
PE_SENS	Enter the acceptable sensitivity for parameters to be included in the parameter upgrade vector, e.g. PE_SENS=0.5 mean that parameters that have less than 0.5% sensitivity will be left out until they achieve a higher sensitivity.	
PE_NPERIOD	Enter the number of periods. If PE_NPERIOD $>$ 0, than repeat Data Set 15 for each period.	
PE_NBATCH	Enter the number of batch files to be included during the parameter estimation. Each batch file can have its own fraction that determines the weigh for the total objective function value.	
PE_TARGET(.)	Enter a fraction for each target:	
	(1)	The difference for each stress period between an available measurement and its corresponding observation
	(2)	The difference between the measurement dynamics and the observational dynamics

	The entered fraction should be entered relative to each other since iMODFLOW will recomputed the normalized values for the fraction. e.g. entering 1.0 and 2.0 will yield the fraction values 0.33 and 0.66, they will be summed equal to one. Whenever **PE_NBATCH>0** (see Data Set 16), the entered weigh values for each batch file will be included in the final normalization of the fractions.	
PE_SCALING *(optional)*	Enter a scaling option:	
	0	No use of scaling/Eigenvalue decomposition (SVD)
	1	Only use of scaling
	2	Use of scaling and Eigenvalue decomposition (SVD)
	3	Only use of Eigenvalue decomposition (SVD)
	In case a SVD decomposition is used (PE_SCALING=2 and PE_SCALING=3), eigenvalues that explain at least 99% of variance are included.	
PE_PADJ *(optional)*	Enter the stopping criteria for Parameter ADJustment, e.g. **PE_PADJ=0.05** means than whenever the parameter adjustment vector is less than 0.05, the optimization will stop. By default PE_PADJ=0.0 which means that the optimization will stop only whenever to parameters adjustment is applied.	
PE_DRES *(optional)*	Enter the minimal acceptable absolute residual used for the objective function. Absolute residuals smaller that PE_DRES will not be included in the objective function and therefore not influence any parameter adjustment. By default PE_DRES=0.0 which means that all residuals will be included.	
PE_KTYPE *(optional)*	Enter the type of Kriging to be used (whenever the PilotPoint concept is used). By default Simple Kriging is applied (PE_KTYPE=1), select PE_KTYPE=2 for Ordinary Kriging. The latter is used whenever a trend exists in the PilotPoints.	

10.15 Data Set 15: Parameter Estimation – Period Settings

Data Set 15	Apply PE_NPERIOD times (see Date Set 14) **S_PERIOD,E_PERIOD**
S_PERIOD [yyyymmdd]	Enter the start date for the period for which observations from the entered IPF file (IPF_TS IN Data Set 3), need to be included, e.g. S_PERIOD=19890101 to express the 1^{th} of January 1989.
E_PERIOD [yyyymmdd]	Enter the end date for the period for which observations from the entered IPF file (IPF_TS IN Data Set 3), need to be included, e.g. S_PERIOD=20120321 to express the 21^{th} of March 2012.

10.16 Data Set 16: Parameter Estimation – Batch Settings

Data Set 16	Apply PE_NBATCH times (see Date Set 14) **B_FRACTION,B_BATCHFILE,B_OUTFILE**	
B_FRACTION	Enter the fraction for the results from the current batch files. The entered fraction will be normalized together with the entered fraction for PE_TARGET(.) (see Data Set 14).	
B_BATCHFILE	Enter the name of the batch file to be executed after each simulation, e.g. C:\BATCHFILES\FLOWLINES.BAT	
B_OUTFILE	Enter the name of the output file from the batch file (B_BATCHFILE), e.g. C:\BATCHFILES\OUTPUT\FLOWLINES.OUT. The syntax of the file should be as follows:	
	N	Enter the number of records, e.g. N=2.
	Z,V,H	Enter for each record *i* to N the measurement (Z), variance (V) and computed value (H). They can be entered in “free”-format. These values will be added to the total objective function value and included in the determination of the gradient.

10.17 Data Set 17: Parameter Estimation - Parameters

Data Set 17	**PACT,PPARAM,PILS,PIZONE,PINI,PDELTA,PMIN,PMAX,PINCREASE, PIGROUP,PLOG**	
PACT	Activation of the current parameter.	
	0	Parameter is not adjusted, initial parameter value PINI remains unchanged during the estimation

	1	Parameter is part of the estimation process	
PPARAM	Parameter type, choose from:		
	Type	Transf.	
	KD	LOG	Transmissivity, equal to KDW
	KH	LOG	Horizontal permeability, equal to KHV
	KV	LOG	Vertical permeability, equal to KVV
	VC	LOG	Vertical resistance, equal to VCW
	SC	LOG	Storage coefficient equal to STO
	RC	LOG	River conductance as mentioned in RIV
	RI	LOG	River infiltration factor as mentioned in RIV
	DC	LOG	Drainage conductance as mentioned in DRN
	IC	LOG	River conductance as mentioned in ISG file
	II	LOG	River infiltration factor as mentioned in ISG file
	AH	-	Angle of Anisotropy
	AF	LOG	Factor of Anisotropy
	VA	LOG	Vertical Anisotropy
	HF	LOG	Horizontal Barrier Factor
	MS	LOG	MetaSWAP storage coefficient (Theta)
	MC	LOG	MetaSWAP conductance (k)
	RE	-	Recharge
	EX	LOG	External parameter, specify on the next line the batch file that need to be executed to modify any parameter.
	EP	LOG	Corey-Epsilon parameter for the UZF package.
PILS	Enter the layer number or system number for the parameter PPARAM. In case KD,KH,KV,C,S,AH,AF,VA,EP are used apply a model layer number, for the other parameters apply the system number.		
PIZONE	Enter the zone number (integer value) for which the parameter PPARAM need to be adjusted. For the parameter type HF this is irrelevant since all lines from the HF module will be optimized together, not differentiation can be made along the line within the same system. You should enter a value but it will not be used!		
PINI	Enter the initial multiplication factor for the parameter PPARAM.		
PDELTA	Enter the step size to be used for the sensitivity computation. PDELTA should be larger than 1.0		
PMIN	Enter the minimum multiplication factor for the parameter PPARAM that might be applied during the optimization.		
PMAX	Enter the maximum multiplication factor for the parameter PPARAM that might be applied during the optimization.		
PINCREASE	Enter the maximum increase of the parameter factor.		
PIGROUP *(optional)*	Enter the group number to which the parameters belongs, parameters within the same group will be estimated simultaneously.		
PLOG *(optional)*	Enter whether the parameter need to be log transformed, e.g. set **PLOG**=1 to log transform the parameter, set **PLOG**=0 to use a linear relation. By default the settings will be applied as described by the **PPARAM** keyword.		

10.18 Data Set 18: Parameter Estimation – Zones

Data Set 18	**NZONES**
NZONES	Enter the number of zones to be used.

10.19 Data Set 19: Parameter Estimation – Zone Definition

Data Set 19	**IDF,IPF or CONSTANT**
IDF	Enter for **NZONES** an IDF file that contains the position of zones. The zone numbering should be equal to the value PIZONE from Data Set 16. You can specify **PIZONE** to be specified in more than one IDF.
IPF	Enter an IPF file that contains the location of a pilot point. The content of the IPF should be x, y and $zone$. Within a single IPF more zones per point or more points per zone can be defined.

CONSTANT	Enter a constant value to specify one **PIZONE** for the entire model area, e.g. CONSTANT=2
A fraction can be added to specify a fraction that that parameter will be used for the parameter optimization, e.g. the value within the IDF will be 2.34, meaning that the parameter belong to zone number 2 and taken for 34% part of the optimization of that parameter.	

10.20 Runfile history

10.20.1 Upcoming additional runfile options

We are implementing additional runfile-options scheduled for a next iMOD release;
in section 10.2 to section 10.18 these additional options are denoted as
"This runfile-option is scheduled for a next iMOD release".

10.20.2 Updating from iMOD 4.2 to iMOD 4.2.1

You can re-use your existing iMOD 4.2-runfile in iMOD 4.2.1 without any changes.

10.20.3 Updating from iMOD 4.1.1 to iMOD 4.2

You can re-use your existing iMOD 4.1.1-runfile in iMOD 4.2 without any changes.

10.20.4 Updating from iMOD 4.1 to iMOD 4.1.1

You can re-use your existing iMOD 4.1-runfile in iMOD 4.1.1 without any changes.

10.20.5 Updating from iMOD 4.0 to iMOD 4.1

You can re-use your existing iMOD 4.0-runfile in iMOD 4.1 without any changes, except for the following issues:

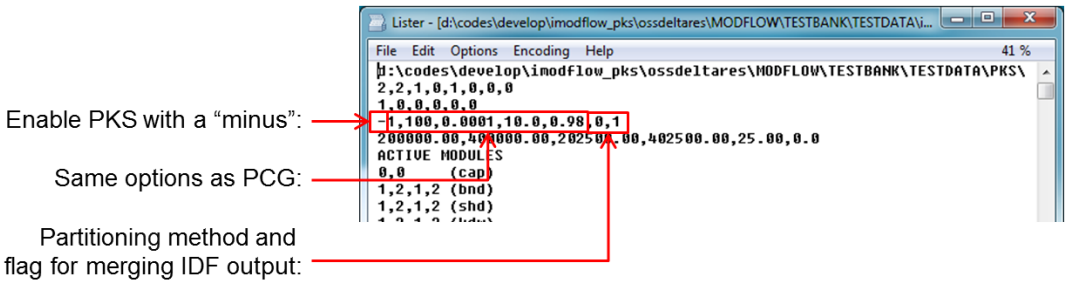
- ISG definitions now must include FCT and IMP parameter specifications.
- The RCH and EVT package cannot be defined for ILAY=0 anymore.

Note: The processing of FCT and IMP has been corrected, possibly yielding different results.

Note: The use of the SFR-, LAK-, UZF- and MNW-packages in a runfile is not supported; these packages have to be configured in a so-called project (*.PRJ) file.

10.20.6 Updating from iMOD 3.6 to iMOD 4.0

You can re-use your existing iMOD 3.6-runfile in iMOD 4.0 without any changes using the default PCG solver. iMOD 4.0 contains the Parallel Krylov Solver package too, however, it is recommended always trying to run your model with the default PCG solver first.
To run your model using the PKS-package - after installing the third party MPI-software (see iMOD installation instructions, section 2.3) - change the Solver Settings in Data Set 5 accordingly; see section 10.6 for details. An example of a runfile including PKS-solver settings is given in the figure below.



To define or change the PKS-solver settings via the iMOD-GUI see section 7.9. For instructions on how to run a model including the PKS-package, see section 10.21.

10.20.7 Updating from iMOD 3.4 to iMOD 3.6

You can re-use your existing iMOD 3.4-runfile in iMOD 3.6 without any changes.

Note: The use of the SFR-, LAK-, UZF- and MNW-packages in a runfile is not supported; these packages have to be configured in a so-called project (*.PRJ) file.

10.20.8 Updating from iMOD 3.3 to iMOD 3.4

You can re-use your existing iMOD 3.3-runfile in iMOD 3.4 without any changes.

10.20.9 Updating from iMOD 3.2.1 to iMOD 3.3

You can re-use your existing iMOD 3.2.1-runfile in iMOD 3.3 without any changes.

10.20.10 Updating from iMOD 3.2 to iMOD 3.2.1

The only difference between iMOD 3.2 and iMOD 3.2.1 is that in iMOD 3.2.1 it is possible to use folder- and filenames including spaces. As a consequence, to be able to re-use your iMOD 3.2-runfile in iMOD 3.2.1 utilizing spaces in folder- and/or filenames, the folder- and/or filenames have to be between double quotes.

10.20.11 Runfiles prior to iMOD 3.x

iMOD became open source mid 2014 resulting in the release of **iMOD 3.0**. Compared to iMOD 2.x some runfile options are not supported in iMOD 3.x releases; in the tables of section 10.2 to section 10.18) these options are denoted as: *"Not supported in iMOD 3.x releases"*.

10.21 Starting a Model Simulation

There are 2 ways to start a model simulation:

1 Inside the iMOD-GUI:
In section 7.9 detailed instructions are given on how to interactively configure the model location, model grid size and model output, including a description of how to specify necessary settings for either the PCG-solver or the Parallel Krylov Solver in the 'Solver settings'-tab of the 'Start Model Simulation' window.

2 Outside the iMOD-GUI:

◇ By entering the appropriate command manually at the DOS-prompt in a 'Window Command Processor'-box; an example of such a command is:

```
d:\iMOD\iMODFLOW_V4_3_METASWAP_SVN1233_X64R.exe model.run
```

- By specifying and storing such a command in a batch-file, e.g. `run.bat`, and double-clicking this `run.bat` file or typing `run.bat` at the DOS-prompt and pressing Enter.
  The content of such a batch file should have the following structure:

  On line one: `[name of iMODFLOW-executable] [name of the model runfile]`
  On line two: `pause`

  The `Pause`-statement causes the command tool to remain visible after the simulation has finished; this is handy for reasons of inspection in cases you invoke the `run.bat` file by double-clicking it; omitting the 'Pause'-statement causes the 'Windows Commander Processor'-box to close immediately as soon as the model simulation has finished.

- When using the **PKS-package**:

  Prior to using the PKS-package MPI software has to be installed, see section 2.3. Here's an example of how to start a multi-core model simulation from outside the iMOD-GUI by entering the following command in a 'Windows Command Processer'-box:

```
"C:\Program Files\MPICH2\bin\mpiexec.exe" -localonly 2 iMODFLOW.exe iMODFLOW.run
```

  In this example MPI launches two processes of iMODFLOW.exe instances on two computational cores, meaning that the model runs using two subdomains. The `-localonly` option ensures that you should not necessarily have to be connected to your network for running with MPI.

10.22 Example Output file

Example output written by iMODFLOW in the iMODFLOW.list-file:

```
                              MODFLOW-2005
     U.S. GEOLOGICAL SURVEY MODULAR FINITE-DIFFERENCE GROUND-WATER FLOW MODEL
                          VERSION 1.8.00 12/18/2009

 LIST FILE: IMODFLOW.list
                       UNIT   10

 OPENING IMODFLOW.dis
 FILE TYPE:DIS   UNIT   11   STATUS:OLD
 FORMAT:FORMATTED             ACCESS:SEQUENTIAL

 OPENING IMODFLOW.bas6
 FILE TYPE:BAS6   UNIT   12   STATUS:OLD
 FORMAT:FORMATTED             ACCESS:SEQUENTIAL

 OPENING IMODFLOW.lpf
 FILE TYPE:LPF   UNIT   13   STATUS:OLD
 FORMAT:FORMATTED             ACCESS:SEQUENTIAL

 OPENING IMODFLOW.oc
 FILE TYPE:OC   UNIT   14   STATUS:OLD
 FORMAT:FORMATTED             ACCESS:SEQUENTIAL

 OPENING IMODFLOW.met
 FILE TYPE:MET   UNIT   15   STATUS:OLD
 FORMAT:FORMATTED             ACCESS:SEQUENTIAL

 OPENING IMODFLOW.wel
 FILE TYPE:WEL   UNIT   19   STATUS:OLD
 FORMAT:FORMATTED             ACCESS:SEQUENTIAL

 OPENING IMODFLOW.pcg
 FILE TYPE:PCG   UNIT   22   STATUS:OLD
 FORMAT:FORMATTED             ACCESS:SEQUENTIAL

 OPENING IMODFLOW.rch
 FILE TYPE:RCH   UNIT   23   STATUS:OLD
 FORMAT:FORMATTED             ACCESS:SEQUENTIAL
# data:

 BAS -- BASIC PACKAGE, VERSION 7, 5/2/2005 INPUT READ FROM UNIT   12

 DISCRETIZATION INPUT DATA READ FROM UNIT   11
 # Discretization file
    3 LAYERS       142 ROWS       160 COLUMNS
    1 STRESS PERIOD(S) IN SIMULATION
 MODEL TIME UNIT IS DAYS
 MODEL LENGTH UNIT IS METERS
  Confining bed flag for each layer:
   1   1   0

                    DELR =   25.0000

                    DELC =   25.0000
```

Example output written by iMODFLOW in the iMODFLOW.list-file (continued):

```
OPENING FILE ON UNIT   99:
D:\IMOD\TUTORIALS\TUT4_INITIAL_MODELING\SURFACE_LEVEL.IDF
 FILE IDENTIFIED AS IDF

OPENING FILE ON UNIT   99:
D:\IMOD\TUTORIALS\TUT4_INITIAL_MODELING\SURFACE_LEVEL.IDF
 FILE IDENTIFIED AS IDF

OPENING FILE ON UNIT   99:
D:\IMOD\TUTORIALS\TUT4_INITIAL_MODELING\SURFACE_LEVEL.IDF
 FILE IDENTIFIED AS IDF

  MODEL LAYER BOTTOM EL. =  -15.0000     FOR LAYER   2

BOT. EL. OF QUASI-3D BED =  -15.0000     FOR LAYER   2

  MODEL LAYER BOTTOM EL. =  -20.0000     FOR LAYER   3


STRESS PERIOD     LENGTH       TIME STEPS     MULTIPLIER FOR DELT    SS FLAG
-----------------------------------------------------------------------------
       1         0.000000          1                    1.000         SS

STEADY-STATE SIMULATION



# Basic Package file
THE FREE FORMAT OPTION HAS BEEN SELECTED

OPENING FILE ON UNIT   99:
D:\IMOD\TUTORIALS\TUT4_INITIAL_MODELING\BOUNDARY.IDF
 FILE IDENTIFIED AS IDF

          BOUNDARY ARRAY =              1 FOR LAYER   2

          BOUNDARY ARRAY =              1 FOR LAYER   3

AQUIFER HEAD WILL BE SET TO  -999.99     AT ALL NO-FLOW NODES (IBOUND=0).

            INITIAL HEAD =   0.00000     FOR LAYER   1

            INITIAL HEAD =   0.00000     FOR LAYER   2

            INITIAL HEAD =   0.00000     FOR LAYER   3

OUTPUT CONTROL IS SPECIFIED ONLY AT TIME STEPS FOR WHICH OUTPUT IS DESIRED
HEAD PRINT FORMAT CODE IS   0    DRAWDOWN PRINT FORMAT CODE IS   0
HEADS WILL BE SAVED ON UNIT   28    DRAWDOWNS WILL BE SAVED ON UNIT    0
```

Example output written by iMODFLOW in the iMODFLOW.list-file (continued):

```
LPF -- LAYER-PROPERTY FLOW PACKAGE, VERSION 7, 5/2/2005
        INPUT READ FROM UNIT   13
CELL-BY-CELL FLOWS WILL BE SAVED ON UNIT   29
HEAD AT CELLS THAT CONVERT TO DRY=  -9999.0
No named parameters
STORAGECOEFFICIENT OPTION:
Read storage coefficient rather than specific storage

  LAYER FLAGS:
LAYER       LAYTYP        LAYAVG         CHANI        LAYVKA        LAYWET
--------------------------------------------------------------------------
   1            0             0     1.000E+00             1             0
   2            0             0     1.000E+00             1             0
   3            0             0     1.000E+00             1             0

  INTERPRETATION OF LAYER FLAGS:
                     INTERBLOCK     HORIZONTAL    DATA IN
       LAYER TYPE   TRANSMISSIVITY   ANISOTROPY   ARRAY VKA   WETTABILITY
LAYER      (LAYTYP)      (LAYAVG)       (CHANI)      (LAYVKA)      (LAYWET)
-------------------------------------------------------------------------------
   1      CONFINED      HARMONIC     1.000E+00    ANISOTROPY  NON-WETTABLE
   2      CONFINED      HARMONIC     1.000E+00    ANISOTROPY  NON-WETTABLE
   3      CONFINED      HARMONIC     1.000E+00    ANISOTROPY  NON-WETTABLE

WETTING CAPABILITY IS NOT ACTIVE IN ANY LAYER

   HYD. COND. ALONG ROWS =   25.0000     FOR LAYER   1

 HORIZ. TO VERTICAL ANI. =   1.00000     FOR LAYER   1

QUASI3D VERT. HYD. COND. =   25.0000     FOR LAYER   1

   HYD. COND. ALONG ROWS =   25.0000     FOR LAYER   2

 HORIZ. TO VERTICAL ANI. =   1.00000     FOR LAYER   2

QUASI3D VERT. HYD. COND. =   25.0000     FOR LAYER   2

   HYD. COND. ALONG ROWS =   25.0000     FOR LAYER   3

 HORIZ. TO VERTICAL ANI. =   1.00000     FOR LAYER   3

WEL -- WELL PACKAGE, VERSION 7, 5/2/2005 INPUT READ FROM UNIT   19
# Well Package file
No named parameters
MAXIMUM OF      1 ACTIVE WELLS AT ONE TIME
CELL-BY-CELL FLOWS WILL BE SAVED ON UNIT   33
LISTS OF WELL CELLS WILL NOT BE PRINTED


    0 Well parameters

RCH -- RECHARGE PACKAGE, VERSION 7, 5/2/2005 INPUT READ FROM UNIT   23
# Recharge Package file
No named parameters
OPTION 1 -- RECHARGE TO TOP LAYER
CELL-BY-CELL FLOWS WILL BE SAVED ON UNIT   34
```

Example output written by iMODFLOW in the iMODFLOW.list-file (continued):

```
    0 Recharge parameters

 PCG -- CONJUGATE-GRADIENT SOLUTION PACKAGE, VERSION 7, 5/2/2005
 # Preconditioned Conjugate-Gradient Package
 MAXIMUM OF    500 CALLS OF SOLUTION ROUTINE
 MAXIMUM OF     50 INTERNAL ITERATIONS PER CALL TO SOLUTION ROUTINE
 MATRIX PRECONDITIONING TYPE :    1


                                SOLUTION BY THE CONJUGATE-GRADIENT METHOD
                               -----------------------------------------
                  MAXIMUM NUMBER OF CALLS TO PCG ROUTINE =      500
                     MAXIMUM ITERATIONS PER CALL TO PCG =       50
                                MATRIX PRECONDITIONING TYPE =        1
       RELAXATION FACTOR (ONLY USED WITH PRECOND. TYPE 1) =    0.98000E+00
 PARAMETER OF POLYNOMIAL PRECOND. = 2 (2) OR IS CALCULATED :        1
                          HEAD CHANGE CRITERION FOR CLOSURE =    0.10000E-03
                    RESIDUAL CHANGE CRITERION FOR CLOSURE =    0.10000E-01
           PCG HEAD AND RESIDUAL CHANGE PRINTOUT INTERVAL =        1
    PRINTING FROM SOLVER IS LIMITED(1) OR SUPPRESSED (>1) =        0
                           STEADY-STATE DAMPING PARAMETER =    0.10000E+01
                              TRANSIENT DAMPING PARAMETER =    0.10000E+01
1
                           STRESS PERIOD NO.    1, LENGTH =   0.000000
                           -----------------------------------------------

                             NUMBER OF TIME STEPS =     1

                              MULTIPLIER FOR DELT =     1.000

                           INITIAL TIME STEP SIZE =   0.000000
  DATA IDENTIFIED AS GCD

 OPENING FILE ON UNIT   99:
 D:\IMOD\TUTORIALS\TUT4_INITIAL_MODELING\WELL.IPF
  FILE IDENTIFIED AS IPF

      1 WELL

                RECHARGE =  1.000000E-03
```

Example output written by iMODFLOW in the iMODFLOW.list-file (continued):

```
SOLVING FOR HEAD

SOLVING FOR HEAD


    2 CALLS TO PCG ROUTINE FOR TIME STEP   1 IN STRESS PERIOD    1
   40 TOTAL ITERATIONS

MAXIMUM HEAD CHANGE FOR EACH ITERATION (1 INDICATES THE FIRST INNER ITERATION):

   HEAD CHANGE    HEAD CHANGE    HEAD CHANGE    HEAD CHANGE    HEAD CHANGE
  LAYER,ROW,COL  LAYER,ROW,COL  LAYER,ROW,COL  LAYER,ROW,COL  LAYER,ROW,COL
-------------------------------------------------------------------------------
 1 -0.5874      0 -0.1771      0 -0.1761      0 -0.7192E-01  0  0.7406E-01
  (  3, 66, 85)  (  3, 64, 86)  (  1, 67, 86)  (  2, 66, 86)  (  2, 67, 86)
 0  0.9079E-01  0  0.6515E-01  0  0.8769E-01  0  0.1096      0  0.8734E-01
  (  1, 67, 85)  (  3, 68, 84)  (  3, 71, 80)  (  3, 69, 85)  (  3, 68, 85)
 0  0.6230E-01  0  0.4693E-01  0  0.3349E-01  0  0.2092E-01  0  0.1545E-01
  (  3, 70, 86)  (  3, 75, 94)  (  2, 74, 97)  (  2, 80, 91)  (  1, 69, 84)
 0  0.1377E-01  0  0.9860E-02  0  0.6983E-02  0  0.4455E-02  0  0.2620E-02
  (  1, 70, 83)  (  3, 62, 89)  (  3, 60, 84)  (  3, 57, 93)  (  3, 54, 97)
 0  0.1524E-02  0  0.7823E-03  0  0.4961E-03  0  0.3366E-03  0  0.2882E-03
  (  3, 52, 98)  (  1, 56,100)  (  3, 55,102)  (  3, 60,103)  (  3, 57,104)
 0  0.2953E-03  0  0.2745E-03  0  0.2177E-03  0  0.1764E-03  0  0.1264E-03
  (  3, 55,105)  (  3, 59,106)  (  3, 57,108)  (  3, 48,105)  (  3, 51,114)
 0 -0.8075E-04  0 -0.4793E-04  0  0.3236E-04  0  0.3095E-04  0 -0.3215E-04
  (  3, 95, 86)  (  3, 94, 88)  (  3, 80,113)  (  3, 83,113)  (  3, 67, 46)
 0 -0.2928E-04  0 -0.2481E-04  0 -0.1933E-04  0 -0.1490E-04  1 -0.4253E-05
  (  3, 65, 47)  (  3, 66, 45)  (  3, 68, 43)  (  3, 81, 41)  (  3, 50, 65)

MAXIMUM RESIDUAL FOR EACH ITERATION (1 INDICATES THE FIRST INNER ITERATION):

   RESIDUAL       RESIDUAL       RESIDUAL       RESIDUAL       RESIDUAL
  LAYER,ROW,COL  LAYER,ROW,COL  LAYER,ROW,COL  LAYER,ROW,COL  LAYER,ROW,COL
-------------------------------------------------------------------------------
 1  -152.8      0  -84.27      0  -148.5      0   201.4      0   120.0
  (  2, 66, 85)  (  2, 66, 85)  (  3, 66, 85)  (  2, 66, 85)  (  2, 66, 85)
 0  -41.51      0  -101.4      0  -69.33      0   29.29      0   48.91
  (  2, 66, 85)  (  2, 66, 85)  (  2, 66, 85)  (  2, 66, 86)  (  2, 66, 85)
 0  -38.63      0  -21.28      0  -14.11      0   15.20      0   12.37
  (  3, 66, 85)  (  3, 66, 85)  (  3, 67, 84)  (  3, 66, 85)  (  3, 66, 85)
 0   9.477      0   7.380      0   4.691      0   3.219      0  -2.036
  (  2, 67, 85)  (  2, 67, 85)  (  2, 66, 85)  (  2, 66, 85)  (  2, 67, 85)
 0  -1.218      0 -0.6710      0 -0.3769      0  0.1990      0  0.2378
  (  2, 66, 86)  (  3, 65, 86)  (  2, 66, 85)  (  2, 66, 86)  (  3, 67, 84)
 0  0.1558      0  0.1361      0  0.1027      0  0.1180      0  0.7172E-01
  (  3, 67, 84)  (  2, 65, 87)  (  2, 64, 86)  (  2, 67, 84)  (  2, 67, 84)
 0 -0.5025E-01  0 -0.2430E-01  0 -0.2890E-01  0  0.2159E-01  0  0.1591E-01
  (  2, 65, 87)  (  2, 65, 86)  (  2, 65, 86)  (  2, 65, 87)  (  2, 65, 87)
 0  0.1436E-01  0 -0.1247E-01  0 -0.1108E-01  0 -0.8894E-02  1 -0.5670E-02
  (  2, 65, 86)  (  3, 67, 84)  (  2, 65, 87)  (  2, 65, 87)  (  2, 65, 87)
```

Example output written by iMODFLOW in the iMODFLOW.list-file (continued):

```
 OUTPUT CONTROL FOR STRESS PERIOD    1   TIME STEP   1
    SAVE HEAD FOR ALL LAYERS
    SAVE BUDGET
    SAVE BUDGET
    SAVE BUDGET
 INFO. Sign of cbc-flux FLOW RIGHT FACE is swapped!
 INFO. Sign of cbc-flux FLOW FRONT FACE is swapped!
 INFO. Sign of cbc-flux FLOW LOWER FACE is swapped!

 HEAD WILL BE SAVED ON UNIT   28 AT END OF TIME STEP   1, STRESS PERIOD    1
1
  VOLUMETRIC BUDGET FOR ENTIRE MODEL AT END OF TIME STEP  1 IN STRESS PERIOD   1
  ---------------------------------------------------------------------------

     CUMULATIVE VOLUMES      L**3       RATES FOR THIS TIME STEP      L**3/T
     ------------------                 ------------------------

           IN:                                  IN:
           ---                                  ---
             STORAGE =           0.0000             STORAGE =           0.0000
      CONSTANT HEAD =            0.0000      CONSTANT HEAD =       7.7523E-10
               WELLS =           0.0000               WELLS =           0.0000
            RECHARGE =           0.0000            RECHARGE =        4581.2500

            TOTAL IN =           0.0000            TOTAL IN =        4581.2500

          OUT:                                 OUT:
          ----                                 ----
             STORAGE =           0.0000             STORAGE =           0.0000
      CONSTANT HEAD =            0.0000      CONSTANT HEAD =        4081.3162
               WELLS =           0.0000               WELLS =         500.0000
            RECHARGE =           0.0000            RECHARGE =           0.0000

           TOTAL OUT =           0.0000            TOTAL OUT =        4581.3164

            IN - OUT =           0.0000             IN - OUT =      -6.6406E-02

 PERCENT DISCREPANCY =           0.00     PERCENT DISCREPANCY =           0.00




          TIME SUMMARY AT END OF TIME STEP   1 IN STRESS PERIOD    1
                     SECONDS     MINUTES      HOURS       DAYS        YEARS
                     -----------------------------------------------------------
    TIME STEP LENGTH  0.0000      0.0000      0.0000      0.0000      0.0000
 STRESS PERIOD TIME  0.0000      0.0000      0.0000      0.0000      0.0000
         TOTAL TIME  0.0000      0.0000      0.0000      0.0000      0.0000
1

 Run end date and time (yyyy/mm/dd hh:mm:ss): 2016/03/25  8:44:27
 Elapsed run time:  2.340 Seconds
```

10.23 Example Output Folders

The output folder (Data Set 1) is created during a model simulation and all selected results (Data Set 8) are stored in subfolders:

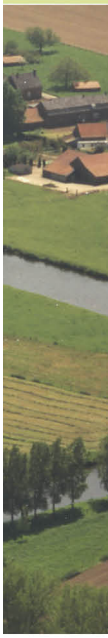
Folder	Subfolder*	File	Content
OUTPUT-FOLDER		imodflow.list	Log file of the entire model simulation
	HEAD	head_steady-state_l[ilay].idf	Steady-state Head
		head_[yyyymmdd]_l[ilay].idf	Transient Head
	BDGFLF	bdg_steady-state_[ilay].idf	Steady-state flux
		bdg_[yyyymmdd]_[ilay].idf	Transient flux
		bdg_sys[i]_[yyyymmdd]_[ilay].idf	
	BDG[pck]	[pck]_steady-state_[ilay].idf	Steady state [pck]-information
		[pck]_[yyyymmdd]_[ilay].idf	Transient [pck]-information

* see for further details Data Set 8

11 iMOD tutorials

This chapter contains the following tutorials:

1. Tutorial 1: **Map Display** (section 11.1) with exercises on:
   - ◇ Displaying an IDF-file and manipulate its associated legend;
   - ◇ Displaying an IPF file and configure its presentation;
   - ◇ Using the 3-D Tool;
   - ◇ Saving your display configuration.
2. Tutorial 2: **Map Operations** (section 11.2) with exercises on:
   - ◇ Calculate differences between two IDF-files;
   - ◇ Assign values to an IDF-file, conditionally;
   - ◇ Perform an up- and or downscaling of the cellsize for an IDF-file.
3. Tutorial 3: **Map Analyse** (section 11.3) with exercises on:
   - ◇ Creating cross-sections over several IDF-files (combined with an IPF file) and manipulate the configuration;
   - ◇ Computing timeseries out of IDF-files (combined with an IPF file);
   - ◇ Using the 3-D Tool.
4. Tutorial 4: **Create your First Groundwater Flow Model** (section 11.4) with exercises on:
   - ◇ Creating the basic input files for a simple groundwater flow model;
   - ◇ Enhancing the model with an extraction well to compute the drawdown caused by the well;
   - ◇ Simulating flowlines that describe the catchment area of the well;
   - ◇ Experiment with extraction rates to compute the maximum sustainable yield without extracting water from the sea.
5. Tutorial 5: **Solid Tool** (section 11.5) with exercises on:
   - ◇ Visualizing the boreholes in 3D;
   - ◇ Enhancing the subsoil characteristics based on the boreholes using the Solid Tool;
   - ◇ Simulating the updated model to observe the consequences of an aquitard in-between two aquifers;
   - ◇ Simulating flow of particles.
6. Tutorial 6: **Model Simulation** (section 11.6) with exercises on:
   - ◇ Understanding the content of a model configuration file, i.e. a runfile;
   - ◇ Simulating a groundwater flow model for different cell sizes and areas of interest;
   - ◇ Understanding the resulting folder structure with results;
   - ◇ Defining a simple model scenario and include such a configuration to an original model configuration.
7. Tutorial 7: **Interactive Pathline Simulation (IPS)** (section 11.7) with exercises on:
   - ◇ Define the starting points interactively;
   - ◇ Change the appearance of the particles;
   - ◇ Start (and stop) the pathline simulation;
   - ◇ Practice the interactive functionalities.
8. Tutorial 8: **Surface Flow Routing (SFR)** (section 11.8) with exercises on:
   - ◇ Define the model and head- and flux boundaries using the FHB package;
   - ◇ Define the outline of the stream network;
   - ◇ Set the characteristics of each stream and define the connections within the stream network;
   - ◇ Start the SFR simulation and examine the outcome.
9. Tutorial 9: **Lake Package (LAK)** (section 11.9) with exercises on:
   - ◇ Interpolate a gradually declining interface for the first model layer;
   - ◇ Define a simple, five layered, transient model and constant head boundaries along the model;
   - ◇ Define the input for the LAK package;
   - ◇ Start the model simulation and examine the outcome;
   - ◇ Combine the LAK package with the SFR package.



10 Tutorial 10: **Multi-Node Well- (MNW) and Horizontal Barrier Flow (HFB) Package** (section 11.10) with exercises on:

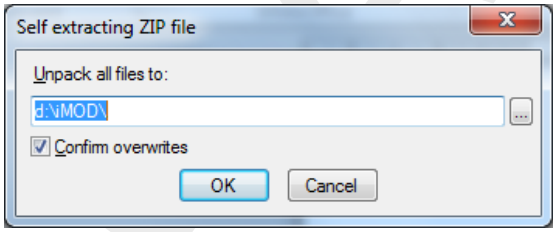
- ◇ Load an existing modelling project and display the model in 3-D;
- ◇ Construct a quick and simple modelling project with the WEL package;
- ◇ Define the model as an unconfined model and simulate the model;
- ◇ Modify the modelling project with the MNW package and simulate the model;
- ◇ Inspect both results;
- ◇ Change some parameters in the MNW package to simulate well losses.
- ◇ Include the horizontal barrier flow package (HFB) and simulate the results for that configuration.

11 Tutorial 11: **Unsaturated Zone Package (UZF)** (section 11.11) with exercises on:

- ◇ Create a transient PRJ file with TOP, BOT, KHV, RCH and EVT package;
- ◇ Simulate the RCH and EVT package for an unconfined model and examine the results;
- ◇ Modify the PRJ file with the UZF package;
- ◇ Simulate the UZF package and examine the results and compare it with the conventional RCH and EVT model;
- ◇ Modify the parameters of the UZF package to see the impact of parameters;

The tutorials come with a Tutorial Data Set located in {*installfolder*} \tutorials; in this manual {*installfolder*} \tutorials refers to the full path of the sub-folder . \tutorials, see section 2.2. If the Tutorial Data Set subfolder . \tutorials is not present (anymore) on your computer, download it from `oss.deltares.nl`. and perform the following steps:

1 Locate the self-extracting archive **iMOD_Tutorial_Data_Set_V4_3.exe** on your system. If not available, please download it from `http://oss.deltares.nl/web/imod/tutorials`.
2 Double-click the archive **iMOD_Tutorial_Data_Set_V4_3.exe**, the following pop-up window appears:



3 In the pop-up window choose the destination-folder where you want to unzip the Tutorial Data Set: you can accept the default {path of installfolder} (e.g. D: \iMOD) by clicking the OK-button, or choose another location first; after clicking the OK-button the archive will be unzipped.

After the archive has finished self-extracting (it may take a while to extract more than 6600 files...) a new sub-folder tutorials has been created in the above chosen destination-folder.

Note: In this user manual {*installfolder*} \tutorials refers to the full path of the newly created tutorials-sub-folder, e.g. to D: \iMOD \tutorials.

The folder {*installfolder*} \tutorials contains a sub-folder for each individual tutorial:

- ◇ . \TUT_Map_Display;
- ◇ . \TUT_Data_Map_Oper;
- ◇ . \TUT_Map_Analyse;
- ◇ . \TUT_Initial_Modeling;
- ◇ . \TUT_Solid_Building;
- ◇ . \TUT_Model_Simulation;
- ◇ . \TUT_IPS;
- ◇ . \TUT_SFR;
- ◇ . \TUT_LAK;
- ◇ . \TUT_MNW;
- ◇ . \TUT_UZF.

11.1 Tutorial 1: Map Display

This tutorial gives a brief introduction to several display options for IDF (rasters) and IPF (points) files. See for more detailed references chapter 6 and subsections.

Outline

This is what you will do:

- ◇ Displaying an IDF-file and manipulate its associated legend;
- ◇ Displaying an IPF file and configure its presentation;
- ◇ Using the 3D Tool;
- ◇ Saving your display configuration.

Required Data

For this tutorial you need the following iMOD Data Files (IDF), iMOD Point Files (IPF) and TXT-files to which the IPF-files are directing:

- ◇ KR_TCC.IDF;
- ◇ KR_BCC.IDF;
- ◇ NAWO_TCC.IDF;
- ◇ NAWO_BCC.IDF;
- ◇ BOREHOLE.IPF;
- ◇ OBSERVATION.IPF;
- ◇ Folder BOREHOLE that contains seven folders called SUBSET{i} containing files called B{i}.TXT that represents borehole-logs;
- ◇ Folder OBSERVATIONS chat contains files called B{i}.TXT that contains values of the timeseries.

All these files are located in the folder: {path of installfolder} \TUTORIALS \TUT_MAP_DISPLAY.

Getting Started

1 If iMOD is not yet installed, please follow the instructions as described in section 2.2.
2 Launch iMOD by double click on the {path of installfolder} \iMOD__V4_3_X64R.exe or {path of installfolder} \iMOD__V4_3_X32R.exe in the Windows Explorer.

The IMOD_INIT.PRF is the only file that iMOD needs at the initial startup. If it does not exist, iMOD will create one. The file contains several keywords that are needed by a variety of functionalities in iMOD, however, the keyword USER is the only one that is obligatory. In the coming up tutorial you'll notice that the content of the IMOD_INIT.PRF file will change. Let us examine the current content.

3 Click on the *Preferences* button.

This displays the *Preferences* window. On default the keyword [USER] is selected and the path that is assigned to that keyword is displayed underneath the list box. Probably it shows the folder {path of installfolder} \IMOD_USER. Several folders will be created in the USER folder. Those folders might be used by iMOD for different purposes, moreover, during your iMOD sessions new folders could be created. However, the most important thing you need to remember about the USER folder is that it stores data **created** by iMOD, e.g. temporary files, model results and drawings. In this case you might interpret a USER folder as a project folder as well, e.g. USER D: \IMOD \PROJECT_X.

Okay, let us continue with iMOD.

4 Click on the *Apply* button.
5 Select the option *Create a new iMOD Project* from the *Start iMOD* window and click the *Start* button.

An empty graphical window will appear surrounded by a default axes and a scale bar. The initial

position of the graphical window is (-1,-1) by (1,1). It is possible to turn off the axes and scale bar, so:

6 Go to *View* and then choose the option *Layout* and turn the options *Show Scalebar* and/or *Show Axes* on and off and observe what is happening.

Display of an IDF-file

An IDF-file stores rasterized data, let us open an IDF-file:

7 Go to *View* in the menu bar and select the *iMOD Manager* (or use the shortcut *Ctrl+M*). Select the *Open Map* button ( ) and select the IDF-file KR_TCC.IDF and click the button *Open*. If the file is not showing up, you might need to change the folder to the appropriate tutorial folder, {path of installfolder} \tutorials \TUT_MAP_DISPLAY.

Observe that the loaded IDF-file emerges in the *iMOD Manager*. iMOD will adjust the zoomlevel automatically to display the entire IDF.

8 Use the zoom buttons on the toolbar ( ) to familiarize with their behaviour.

Note: Please remember that a right-click of the mouse button is necessary to stop moving the map around ( ).

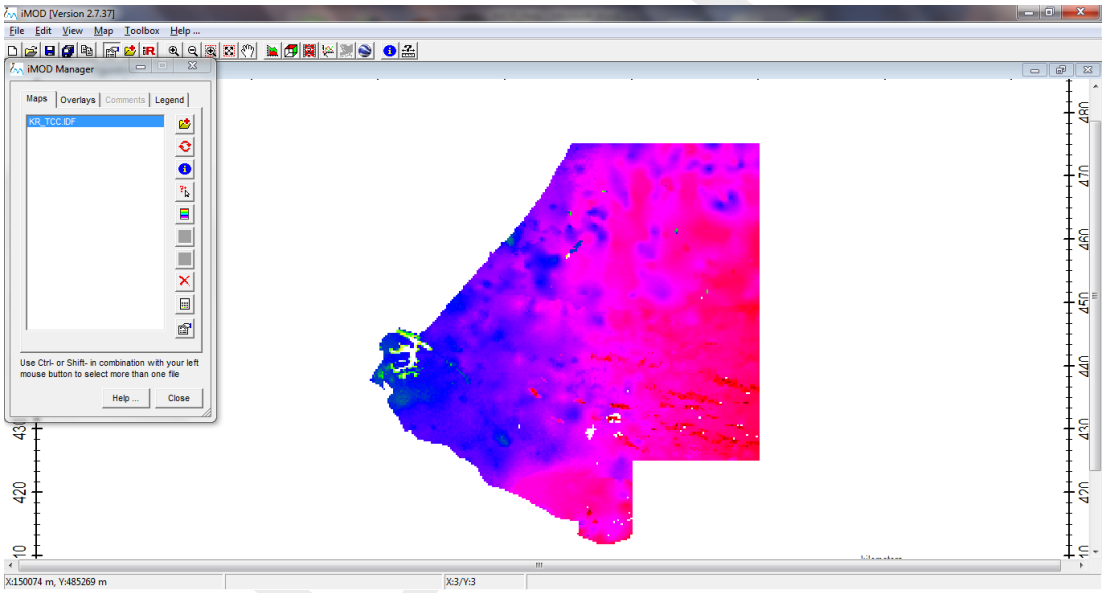


Figure 11.1: *Example of a 2D IDF-view.*

Adjust the legend

Each IDF-file that has been loaded into the *iMOD Manager* will be displayed by a legend with values that decline linearly between the maximum and minimum values of the IDF-file. A legend is connected to the IDF internally and can be changed easily.

9 Select the *Map* option from the main menu, choose the option *Current Zoom Level* and then choose the option *Percentiles*.

By selecting the percentile option, iMOD will compute classes for a legend based on the distribution of the IDF values, like a duration curve. Since the option *Current Zoom Level* has been chosen, the legend will be computed for those values that are inside the current zoom level only.

10 Adjust the legend for the other options (*Linear*, *Percentile* and *Unique Values*) and observe their

differences in combination with the options *Current Zoom Level* and *Entire Zoom Extent*.

11 Click the *Legend* tab on the *iMOD Manager* to display the current legend colours and values.

Adjusting a legend like this automatically, is extremely useful whenever the content of an IDF-file needs to be explored. However, legends can be constructed manually and/or loaded from disk.

12 Click the *Legend* button ( ) on the *Legend* tab of the *iMOD Manager* to display the *Legend* window (see section 6.6.1). Make sure you've selected the IDF on the *Map* tab to gain access to this particular *Legend* tab.

13 Deselect the numbered buttons on the left that indicate *2*,*3*,*4*,*5* and*6* to turn off their appearance in the colours used by the legend. Click the *Apply* button and observe the renewed legend ranging linearly from dark brown to cyan (light-blue).

In this way it is easy to change the colour range of the legend.

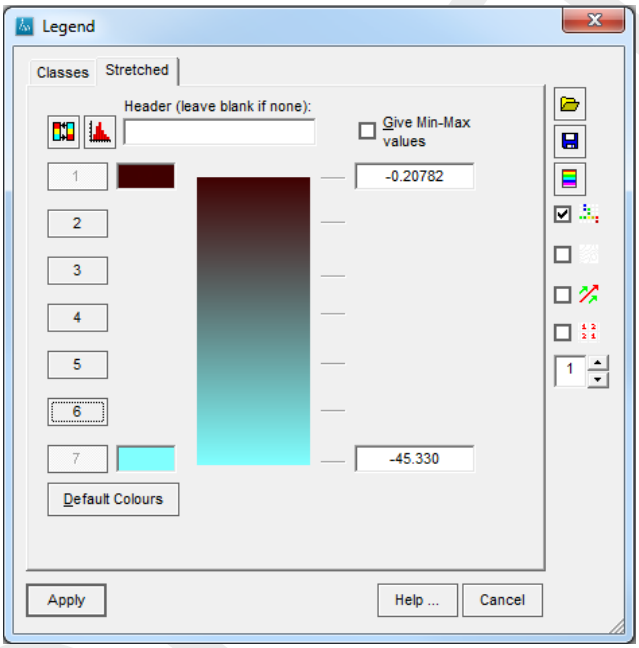


Figure 11.2: *Example of a two-coloured legend.*

Let's use more colours in the legend.

14 Reopen the *Legend* window (step 12) and change the dark brown colour into a red one by clicking on the coloured field. Include more colours in the legend by selecting the buttons that indicate a *2*,*3*,*4*,*5* and/or *6*. See the effects for different legends by clicking the *Apply* button.

iMOD distinguishes two types of legends:

- *Stretched:* a legend that consists of 255 colours and classes that can be specified for 7 levels only;
- *Classes:* a legend that consists of maximal 50 classes and colours that can be specified individually.

Reopen the *Legend* window (step 12) again and let us create a legend with 10 classes:

15 Click the *Classes* tab on the *Legend* window and give in [10] classes in the *Class Definition* window that appears. Check the option *Take classes as-is* and click the *Ok* button.

16 Each row in the table represents a class. Change the values in the first column (*Upper*) for each row into [0.0; -5.0; -10.0; -15.0; -20.0; -25.0; -30.0; -35.0; -40.0; -45.0]. Observe that the second column (*Lower*) will be adjusted automatically, except for row 10. Change the second column for

row 10 into [–50.0] to specify the lowermost limit of the legend.

17 The column *Label* will not be updated automatically, this is the text that will be printed next to the legend. Click the *Update Labels* button to reflect the entered legend value correctly.

Let's look at another way of adjusting the legend, more convenient actually.

18 Click the *Stretched* tab to return to the 255 classes legend and then return back to the Classes tab. Given in [10] classes and deselect the option *Take classes as-is*and click the *Ok* button.

iMOD will try to adjust the number of classes such that a legend is created with nicely legend classes, automatically. Select the *Take classes as-is* option whenever you do not want iMOD to create nice, round classes. Or, alternatively when you do want to have more control on the legends, select the option *Fixed Interval* and specify the interval, minimal and maximal values for the legend classes.

19 Click the *Save* button () to save this legend on disk. Use the *Open* button () to reload the legend (this is not necessary of course).

20 Click the *OK* button to observe the display of the IDF-file with this adjusted legend.

Let us plot a legend on the map

21 Click the *Map* option from the main menu, choose the *Legend* option and then choose *Plot Legend on Map*.

22 Click the left mouse button inside the legend to change the mouse cursor into a  – symbol. Now the legend can be moved to the desired position.

Select the canvas window with your left-mouse button and observe how the cursor changes when the mouse is moved to the boundary of the legend.

23 Drag around the legend and reshape its size.

The text size of the legend will be adjusted automatically to fit the boundary box of the legend. Change the width or height of the legend box in case the label text is not readable.

24 Remove the legend by deselecting the *Plot Legend on Map* option.

Let us open some more IDF-files.

25 Open the IDF-files: KR_BCC.IDF, NAWO_TCC.IDF, NAWO_BCC.IDF.

26 Select all IDF-files in the *Maps* tab of the *iMOD Manager* by dragging the mouse over all files. Or use the combination Ctrl-left mouse button to select the different IDF-files.

Whenever more than one IDF-file is selected in the *iMOD Manager* the *Legend* button will become inactive. However, the following method can be used to adjust all legends simultaneously.

27 Click the *Map* option from the main menu, choose the option *Legend* and then choose the option *Synchronize Legends* to display the *Synchronize Legends by:* window. Select the first IDF (KR_TCC.IDF) and click the *Apply* button.

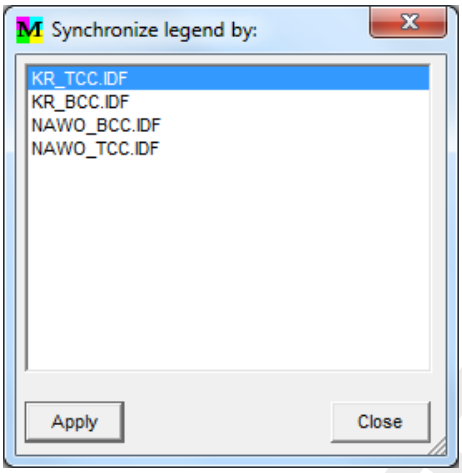


Figure 11.3: *Example of the 'Synchronize legend by:' window.*

28 Observe that all IDF-files have identical legends. Select in the *iMOD Manager* each file sequentially and click the *Redraw* button ( ).

Display of an IPF file

An IPF file stores pointwise information, such as boreholes and/or observation wells. Let us open such a file.

29 Click the *Open Map* button ( ) and select the file BOREHOLE.IPF.

An IPF file is an iMOD-Point-File and see section 9.7 for more detailed information about the content of these IPF files. The IPF file that we've just opened in iMOD does contain the following information:

- X-CRD: X coordinate value of borehole in UTM coordinates;
- Y-CRD: Y coordinate value of borehole in UTM coordinates;
- ID: Identification name for borehole;
- SURFACELEVEL: Altitude of surfacelevel at borehole in m+MSL;
- Z_END: End depth of borehole in m+MSL;
- I_USED: Attribute specifying whether this particular borehole has been used in building the geological model.

Be aware of the fact that all of these attributes do not have any direct meaning in iMOD or whatsoever. In the next steps we will show how these attributes can be used in iMOD.

30 Click the *Zoom Full Extent* button ( ) on the tool bar to adjust the zoom level such that all points are displayed.

All points will be plotted as grey dots initially, however, it is easy to change that.

31 Click the *Map* option from the main menu, choose *IPF-options* and then choose *IPF Configure* to display the *IPF Configure* window.

iMOD will use the first column of the IPF file (label is X-CRD) for the X coordinate (*X-Crd.:*) and the second column (label is Y-CRD) for the Y coordinate (*Y-Crd.:*). On default, the Z coordinate will be assigned to the first column, too, which is incorrect.

32 Select the label [SURFACELEVEL] from the dropdown menu at the menu field *Z-Crd.:*.

iMOD is able to position points in 3D and/or in cross-sections when this *Z-Crd.:* is assigned properly.

33 Select the option *Highlight* and select the label [I_USED] in the dropdown menu to the right.

iMOD will increase the symbol and applies a different colour to *highlight* each point that has values for the chosen label [I_USED] not equal to zero. This feature can be useful to emphasize specific points on a map.

34 Click the *Pick Colour* button to open the standard Windows *Colour* window.
35 Select the colour cyan (light-blue) from the *Custom Colors* field.
36 Click the *Ok* button.

All points will be coloured as cyan (light-blue) in this manner, however, a legend can be used to colour the points as well, so:

37 Select the option *Apply to* and choose the label [Z_END] in the dropdown menu.

iMOD will create a legend initially, based on the minimum and maximum values of the label [Z_END]. The legend functionalities as described by step 12-18 can be applied to IPF files too.

38 Click the *Colouring and Styles* to change the *Symbol No.* to [21] and the *Thickness* to [2].
39 Click the *Apply* button to apply the entered configuration.
40 Click the *Close* button to close the configuration window.

iMOD will colour all points by their values for the label [Z_END] and highlight those that have values for [I_USED] that are not equal zero. iMOD will use an inversed colour (i.e. black becomes white, red becomes light-blue) of the colour used to emphasize the point by a disc around the original point.

Let us adjust the zoomlevel such that we enter coordinates that are the centre of the current zoomlevel.

41 Zoom in onto a particular area by selecting the *View* option from the main menu and then choose the option *Goto XY*. Enter the coordinates [111000.0] and [456000.0] for the *X- and Y coordinate*, respectively.

As we used *Zoom (m)* as [500.0], the zoomlevel will have a minimum width and/or height of about 2 x 500 meter. Let us measure that.

42 Click the *Measurement* tool () from the tool bar to measure the distances of the current display. The measured distance can be found at the bottom of the screen in the grey-coloured bar. Break-off with your right mouse button.

The *Measurement* tool can be used to identify distances between objects on the map, use the left mouse button to include more points during the measuring of the distance.

Since we've zoomed in, let us place some labels to the points to see the actual values for [Z_END].

43 Click the *Labels* button on the *IPF Configure* window (see step 31) to open the *Define Label to be Plotted* window. Select the label [Z_END] in the list and turn off the option *Use different colouring for each field*. Select a *Textsize* of 6 and select the option *Use Labelname*.

Notice that some labels will overlap other labels. iMOD does not support (yet) any advanced labeling to avoid overlapping. Use the zoom functionalities to avoid overlapping.

44 Try to add more labels, remember that it could be handy, in that case, to display the column names too, select therefore the option *Use Labelname*.

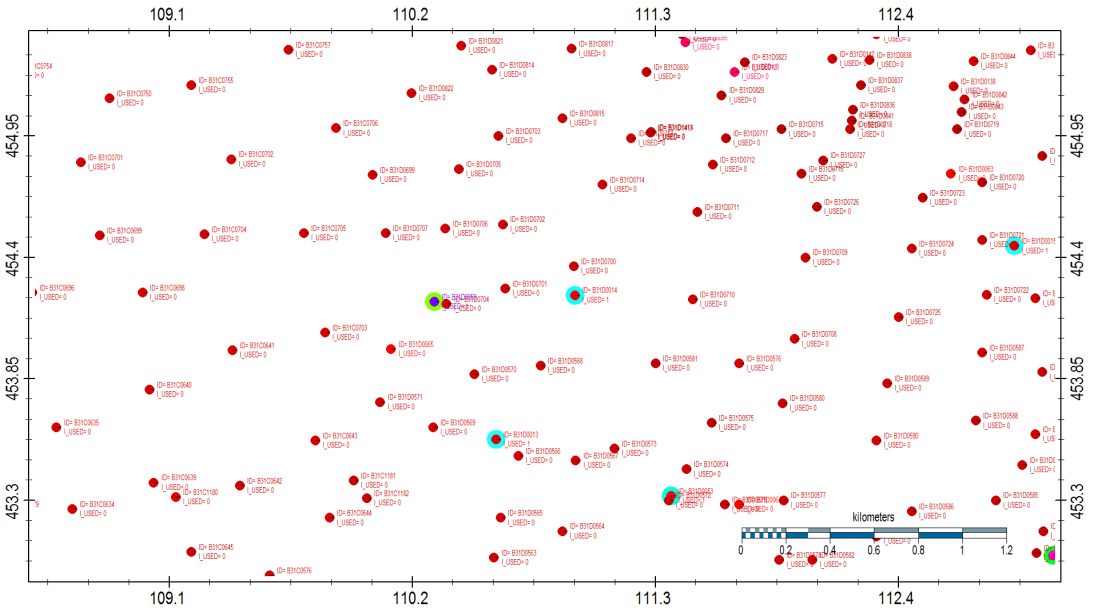


Figure 11.4: *Example of plotted labels using the 'Labels' button of the IPF Configure window.*

45 Turn off all labeling. Use the combination Ctrl-left mouse button to deselect the labels from the *Define Label to be Plotted* window (see step 43). You should select the BOREHOLE.IPF in the *iMOD Manager* solely to outgrey the *IPF Configure* option. (Note: before going to step 46 turn on all labeling again. This is needed for step 51.)

46 Click the *3D Tool* ( ) from the tool bar to enter the 3D environment to observe the boreholes.

By default the coloring used to display the boreholes is different than used in this dataset, so we will load the proper legend file used for displaying the lithology of the boreholes.

47 Click the *Load* ( ) button on the IPF's tab on the *3D IDF Settings* window and select the [BOREHOLES.DLF] from the . \TUT_Map_Display folder. iMOD will reload the IPF file and displays the boreholes according to the legend read from the DLF file. See section 9.17 for more detailed information about a DLF file.

48 Use your left mouse button to rotate the image and your right mouse button to zoom.

The 3D Tool is simulated by OpenGL libraries and is very powerful; however, the display of borehole data can take a while to load since all boreholes are stored in individual text files that need to be processed sequentially. The associated IO consumes most of the time.

49 Check the options *Boundary Box* and *Axes* from the *Miscellaneous* tab in the *3D Plot Settings* window to display the axes and a boundary box.

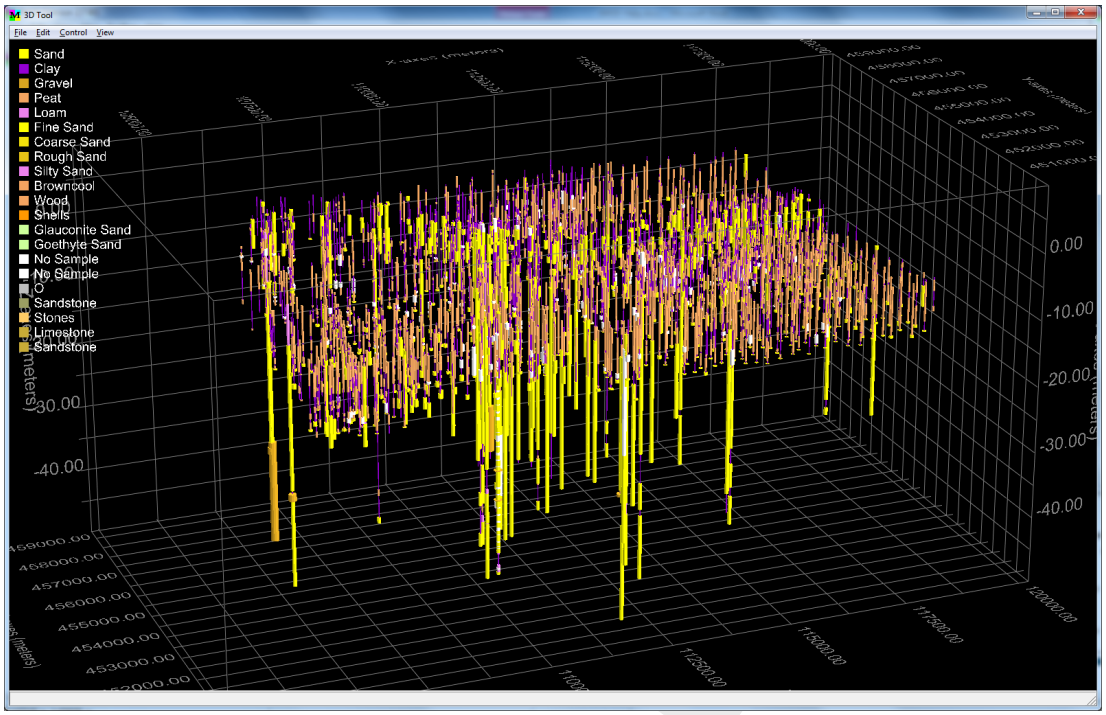


Figure 11.5: *Example of a 3D-display of boreholes.*

Let us reshape the representation of the boreholes. As you can notice each borehole represents a lithology as displayed in the *Legend for Boreholes* table. This legend can be created inside and/or outside iMOD; however, the last column expresses the width that will be used to present the corresponding lithology. So, [Clay] is displayed by a smaller width (with=0.25) than [Sand] that has a width of 1.0.

50 Change the width for different lithologies and even change colours by clicking in the appropriate column(s). Click the redraw button ( ) to update the 3D image for your inserted changes. For example you might increase the width for Sand to 2.0 to distinguish the difference between clay and sand more. Use the *Load* ( ) button to restore the legend setting to the original values by selecting the . \TUT_Map_Display folder \boreholes.dlf file.

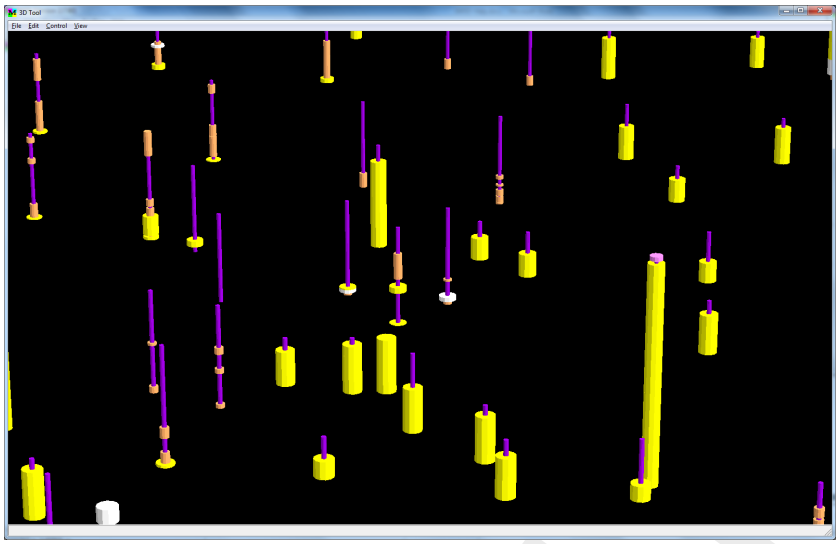


Figure 11.6: *Example of using different thickness's when displaying lithology of boreholes in 3D.*

Alternatively we can change the 3D representation of the borehole.

51 Click the option *Plot labels, use following colour* to add to the boreholes the label selected in the*Define Labels to be plotted* window.

52 Select the properties button ( ) to enter the *Define Labels to be plotted* window, turn off the checkbox *Fancy* and click the *OK* button. Observe what has happened. Check the option *Fancy* again and see the effects of the options *Size*, *Number of Subdivisions* and the option *Shade*.

Often the number of boreholes is large and therefore we would like to concentrate on those with a particular bore depth. Let us select only those with a penetration depth of more than 100 meter.

53 In '3D Plot Settings' in the IPF's-tab click the ( ) icon to start the *3D IPF settings* dialog. Check the option *Hide boreholes with less penetration depth* and enter the value of [100]. Click the *Apply* button and observe what happened.
Extra:

54 Select the *Identify* tab and select the *Map Value* button ( ) and click on the borehole of interest in the *3D tool* window. The *Point Information* tab gives an overview of the basic point characteristics of the selected borehole. The *Borehole Information* tab displays the specific drill information (including Lithology and sand-fraction) for each individual layer. Repeat this procedure for different boreholes by selecting the *Map Value* button again.

55 Close the 3D Tool by clicking the *File* option from the main menu and then choose *Quit 3D Tool*, or alternatively use the close button ( ).

Let us combine in 3D the boreholes with the top and bottom IDF's we've loaded into iMOD previously.

56 Select in the *iMOD Manager* all IDF-files together with the BOREHOLES.IPF and enter the *3D Tool*.

You'll notice that prior to the 3D tool the *3D IDF Settings* dialog appears. In this dialog the appearance of the IDF-files can be configured. For example, an IDF can be represented by planes (quads between mids of gridcells giving a smooth surface) and/or cubes (representing the gridcells as flat surfaces, like Lego-blocks). However, any adjustments in this dialog can be made while in the 3D environment as well, so let us accept the dialog as it is.

57 Click the *Apply* button.

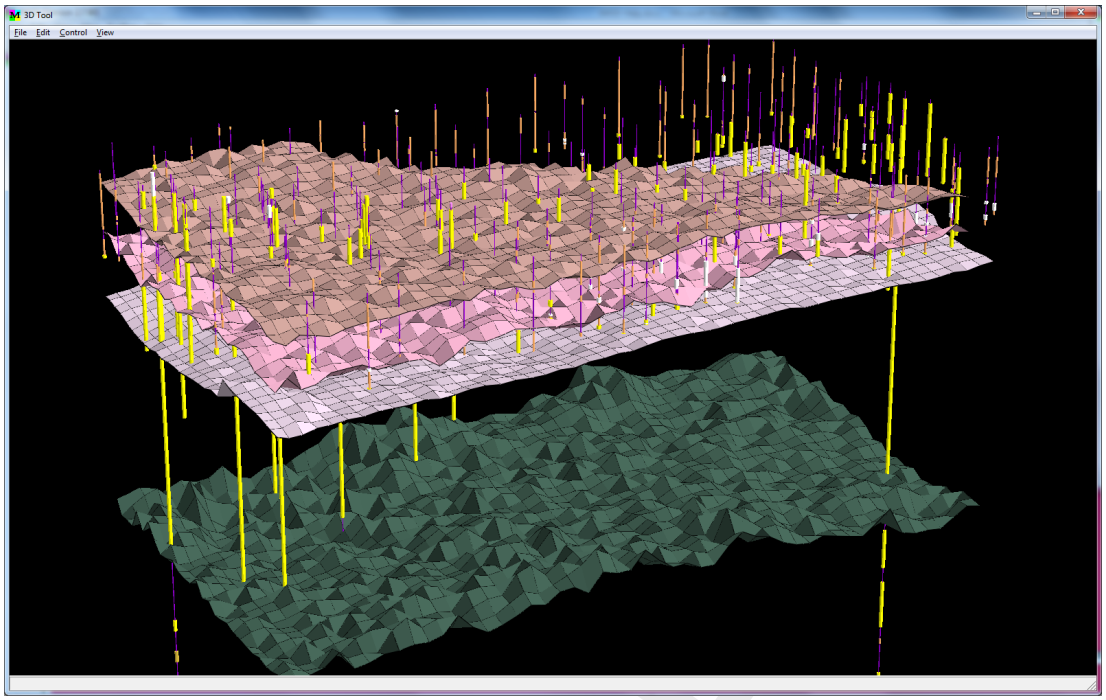


Figure 11.7: *Example of 3D image of a set of planes and boreholes; display depends on options chosen in the 3D IDF Settings-window.*

You'll see the graphical representation of the surface for the different IDF's. Another way to do that is by means of a cross-section (section 11.3). Since the IDF-files represent a clay-body, it is nice to draw them as solids.

58 Click the properties button ( ) to change the settings used to display the IDF-files.

Each row defines how that particular IDF will be displayed. To make a solid of two IDF-files you should combine an IDF with another one. The next image shows how the settings should be configured. For example we combined the IDF-file KR_TCC.IDF (top of the KR-formation) with KR_BCC.IDF (bottom of the KR-formation) by selecting that file from the dropbox in the third column. Also we changed to *Off* the *Type* in the second column of the KR_BCC.IDF-file. Similarly we adjusted this for the NAWO formation.

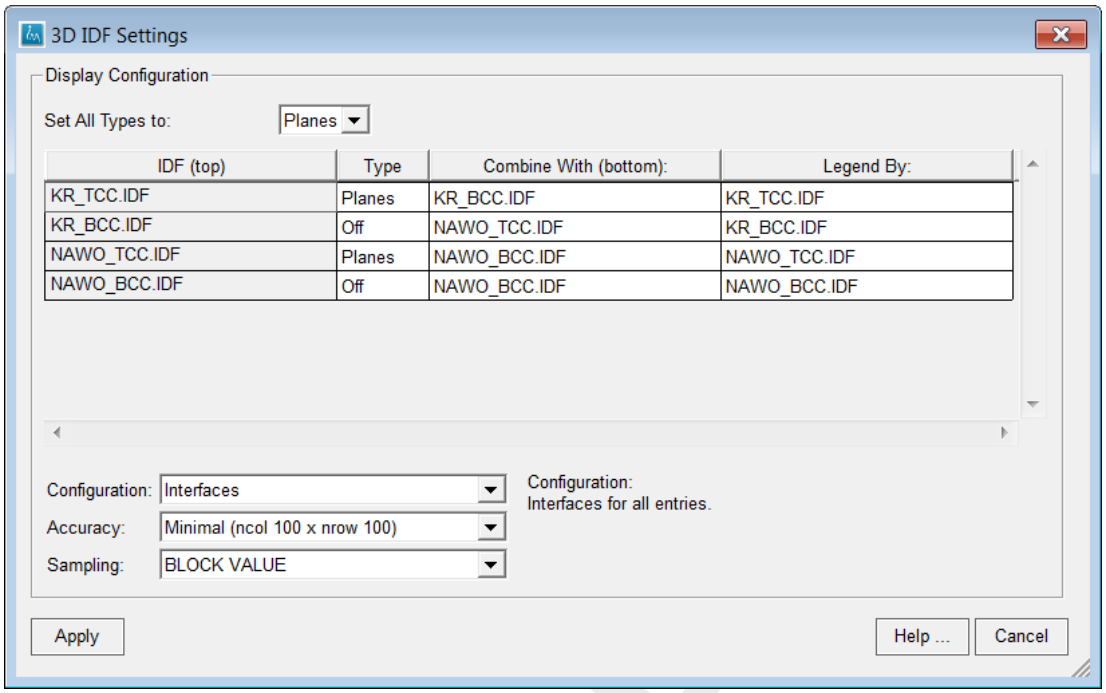


Figure 11.8: *Example of a 3D IDF Settings window for displaying pairs of IDF's as solids.*

59 Adjust the 3D IDF Settings window as above, keep in mind that your order of files might be different yielding a slightly different configuration. Click the *Apply* button.

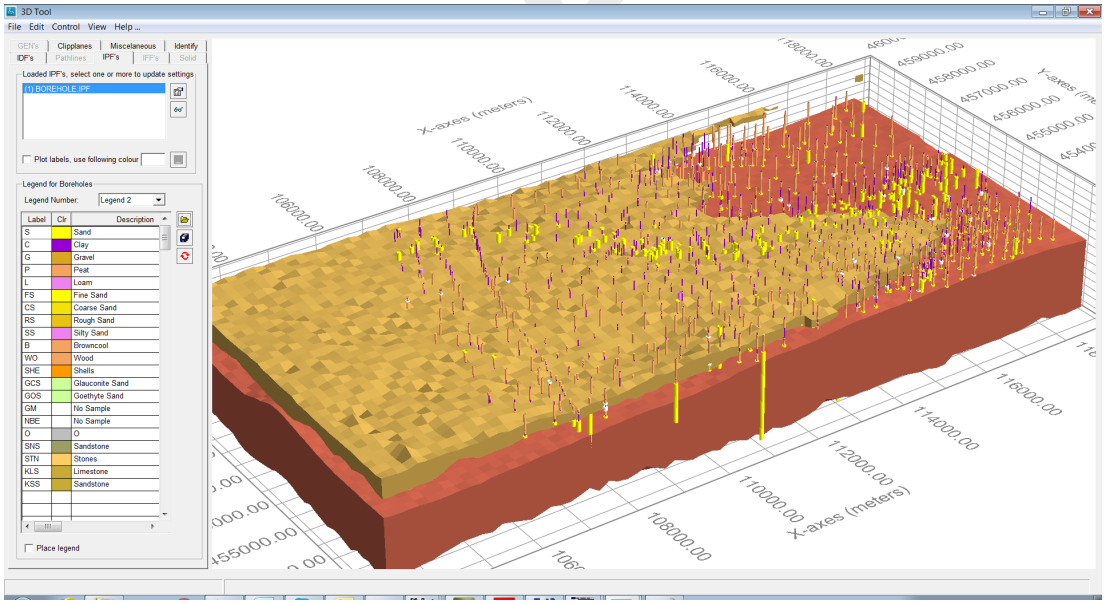


Figure 11.9: *Example of 3D-image of displaying pairs of IDF's as solids.*

60 Activate and deactivate files from the *IDF's* tab on the *3D Plot Settings* window. Experiment with the options *Filled*, *Wireframes* and *Filled+Wireframes* to see the effects and finally place a legend by checking the *Legend* checkbox.

61 Experiment with the functionalities on the *3D Plot Settings* window. See what you could do with the *Plot Original Window* options from the *Miscellaneous* tab.

62 Close the *3D Tool* window (see step 55)

Let us open another IPF file.

63 Open the IPF file [OBSERVATION.IPF] and adjust the zoom level to display all points.

All observation points are displayed by a grey circular dot, however, these points have timeseries associated. Let us look at these associated timeseries.

64 Select the *Map* option from the main menu, choose *IPF-options* and then *IPF Configure* to start the *IPF Configure* window.
65 Select the option *Labels* to start the *Define Labels to be plotted* window.
66 Place a label named ID at each point (see step 43) by selecting the attribute [ID] from the *Select one or more labels* from the menu field and select a *Textsize* of 6.

By default any " \" string will be deleted from the ID field, so the ID-string will shorten whenever it will be displayed on the graphical canvas.

67 Click the *OK* button to close the *Define Labels to be plotted* window and click the *Close* button to close the *IPF Configure* window. Observe the results. You might want to change the number of labels by repeating steps 64 to 66 again.
68 Select the *Map* option from the main menu, choose *IPF-options* and then choose the *IPF Analyse* option to display the *IPF Analyse* window.
69 Click the option *Select For* in the dropdown menu when you right click your mouse button on the graphical window (see figure below). In the *IPF Find* window, select the label [ID] next to the menu field *Attrib.:,* check the*Use following character expression* button and enter the *Search String:* [*B31D011*].

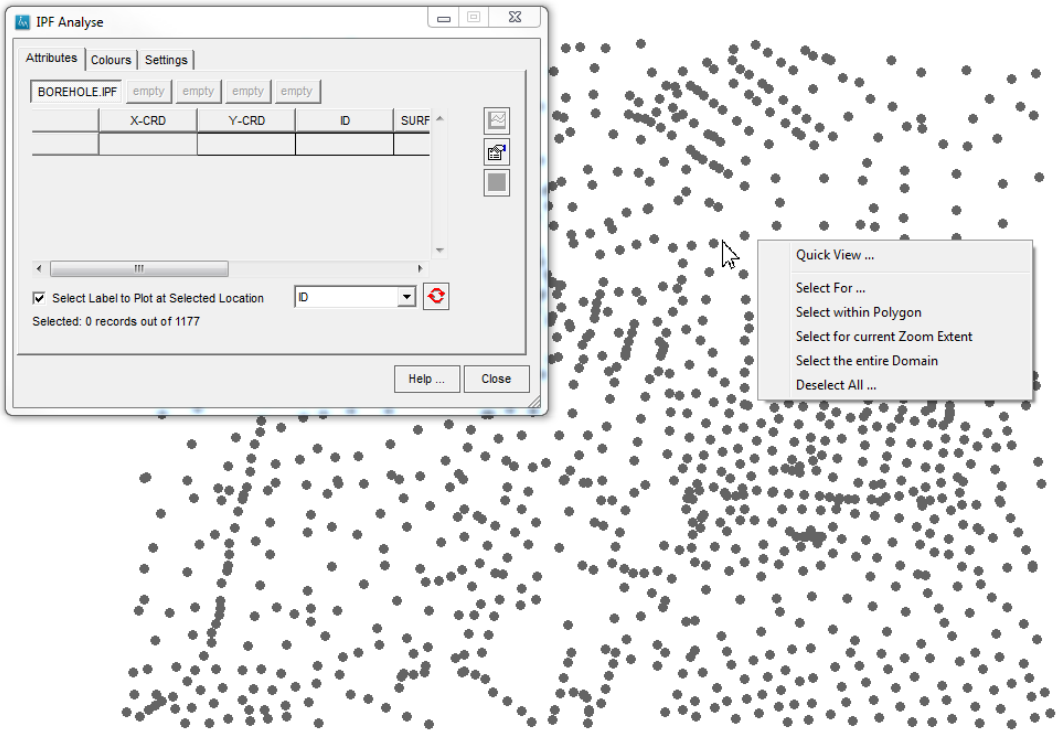


Figure 11.10: *Pop-up window with 'Select For' option when right-clicking on canvas when IPF Analyse window is active.*

iMOD will find any point that satisfies this search string. Notice that the wildcard is necessary at the first portion of the search string, since all label IDs start with "observation \". As a result 2 points will be selected and displayed in the table on the *IPF Analyse* window. Both points represent two different observation screens. Let us display the associated time series.

70 Click the *IPF Figure* button ( ) in the *IPF Analyse* window to open the *IPF Analyse Figure*

window (see section 6.8 for more information).

Two figures are displayed. Whenever one figure is selected in the *Select one/more to plot* list, a table is presented with the actual values for the time series.

71 Select one of the items in the list *Select one/more to plot* and analyse the content of the table.
72 Select the checkbox *Plot all figures in one frame* and select both items in the list*Select one/more to plot*. Use the zoom functionalities to analyse the figure in more detail ( ).
73 Quit the *IPF Analyse Figure* window by selecting the option *File* and then choose *Quit*.

Let us look at another way of adding/deleting points from the selection table.

74 Move the mouse over the points and observe that the mouse symbol changes to . It indicates that when clicking the mouse the particular point will be added to the selection. If the mouse symbol changes to , it indicates that the particular point will be deleted from the selection.
75 Explore the dropdown menu at your right mouse button to experiment with more options to (de)select points.

Additionally to the display options of timeseries in the *IPF Analyse Figure* window, let us plot timeseries on the map.

76 Select the *Settings* tab on the *IPF Analyse* window and select the option [Simple] from the *Graph* dropdown menu and click the *Apply* button.

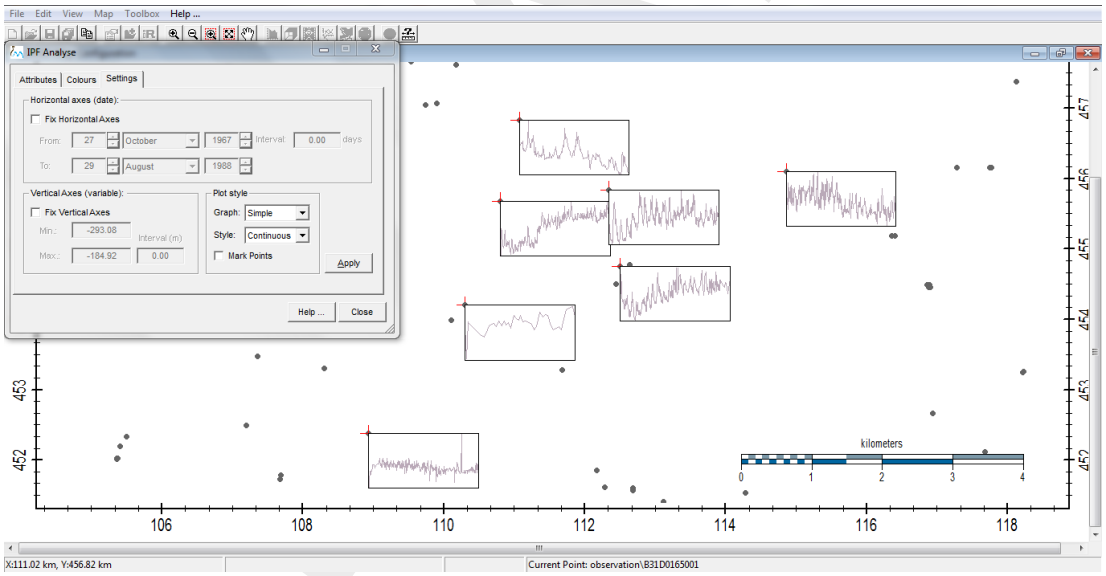


Figure 11.11: *Example of plotted timeseries next to selected points using the option 'Simple' from the Graph dropdown menu in the Setting tab of IPF Analyse.*

For those points in the selection table on the *Attributes* tab, their associated timeseries will be plotted on the map. Each time another point is added or deleted the display is updated.

77 Use your left mouse button on the map to add and/or delete points from the selection table.

Whenever a small crossed-out rectangle is displayed, it means that the associated timeseries for that point is missing.

78 Click the *Close* button to stop the *IPF Analyse* window.

Whenever the *IPF Analyse* window is closed, timeseries cannot be plotted on the map anymore.

Show a background image

One of the first things one would like to display is an image of the underlying topography. Let's do that.

79 Select the option *View* from the main menu and then select *Add Background Image* .. from the dropdown menu. This will start the *Add Background Image* dialog.
80 Select the option *Add* from the dialog and select the file {path of installfolder} \tutorials \TUT_Map_Display \wsrl.bmp from the Windows Explorer, see section 5.3 for more information about this dialog.
81 Select the *Apply* button that closes the dialog.
82 Click the *Show Background Image* ( ) on the main menu whenever the image does not appear.

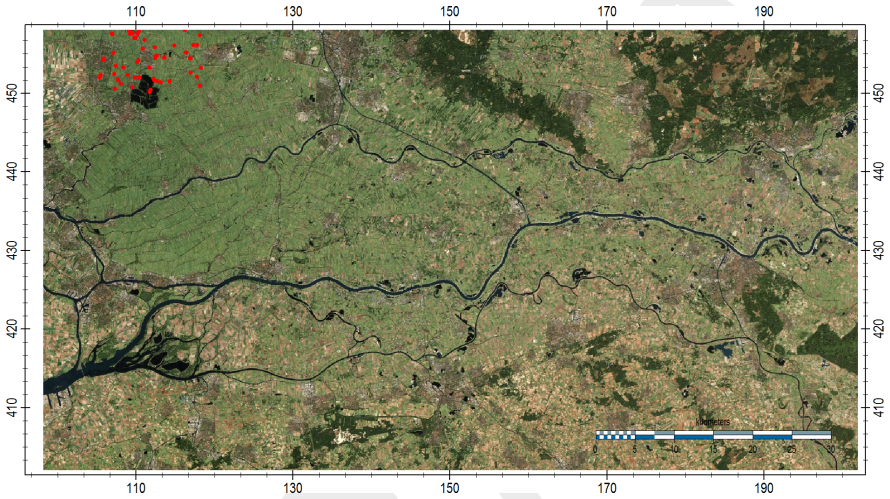


Figure 11.12: *Example of showing a topographical map (full extent, red dots represent the observation.ipf).*

Save /Open a Display Configuration

The entire configuration of legends and settings for the files that are loaded in the *iMOD Manager* can be saved on disk. Whenever iMOD will be restarted, this file can be loaded to recover the iMOD session again.

83 Click the *Save As Current Project* button ( ) on the tool bar and enter a name for the file, e.g. TUTORIAL1.IMF.

The filename entered will be saved in the {USER} \IMFILES folder on default, however, another location can be entered too. For reasons of efficiency and transferability, it is advisable to store these IMF files in that particular folder.

Let us quit iMOD now.

84 Click the *File* option from the main menu and choose the option *Quit* and confirm this action.

Let us restart iMOD.

85 Repeat step 2 in the beginning of this tutorial to launch iMOD.
86 Select TUTORIAL1 from the display list and select the *Start* button from the *Start iMOD* window.

As expected, the original iMOD session has been restored.

11.2 Tutorial 2: Map Operations

This tutorial gives an introduction to several map operations using IDF-files. See for more detailed references section 6.7.

Outline

This is what you will do:

- ◇ Calculate differences between two IDF-files;
- ◇ Assign values to an IDF-file, conditionally;
- ◇ Perform an up- and or downscaling of the cellsize for an IDF-file.

Required Data

For this tutorial you need the following iMOD Data Files (IDF):

- ◇ TOP_LAYER3.IDF;
- ◇ BOTTOM_LAYER3.IDF;
- ◇ KD-VALUE_LAYER3.IDF.

All these files are located in the folder:{path of installfolder} \tutorials \TUT_DATA_MAP_OPER.

Getting Started

1 Launch iMOD by double clicking the iMOD executable in the Windows Explorer, and start with *Create a new iMOD Project*.
2 Go to *View* in the menu bar and select the *iMOD Manager* (or use the shortcut *Ctrl+M*).

Calculate Layer Thickness

Quite often it is necessary to compute the difference between two maps. In this example we compute the thickness of a particular model layer. We start by opening the files: TOP_LAYER3.IDF, BOTTOM_LAYER3.IDF from disk.

3 Click the *Open IDF* button () from the *Maps* tabs on the *iMOD Manager*. Select the above mentioned files in the *Open File* window and click the *Open* button. Go to the folder where the tutorial material has been installed.

After the files have been opened, those files will be added to the list of opened iMOD files in the *iMOD Manager*. iMOD will draw the first IDF from the list and will zoom to the full extent of that IDF automatically. The latter will occur only whenever no maps were available in the *iMOD Manager*.

Let us compute the thickness of model layer 3.

4 Click on the maps TOP_LAYER3.IDF and BOT_LAYER3.IDF as they appear in the *iMOD Manager*, while pressing the *Ctrl* button.

Alternatively, you can select those files by left click your mouse and drag your mouse position over the files. If necessary, deselect those files that are undesired by clicking your left mouse button in combination with the *Ctrl* button.

5 Click on the *iMOD Calculator* button () on the *Maps* tab of the *iMOD Manager* to enter the *Map Operations* window.

The selected IDF-files (inputfiles) will be filled in, as well as the outputfile. On default the output file will be saved in the folder: {USER} \TMP \DIFF.IDF. The default equation (*Formulae* is [C=A-B]) subtracts the first IDF minus the second IDF, in the order in which those IDF-files will appear in the

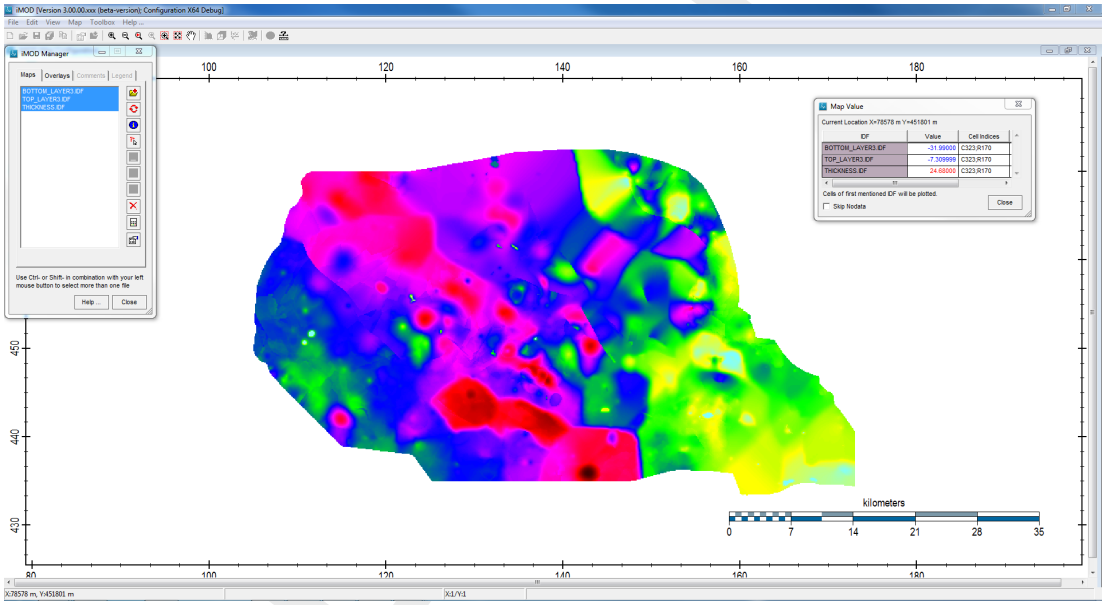
iMOD Manager. Whenever your BOT_LAYER3.IDF is mentioned before the TOP_LAYER3.IDF in the iMOD Manager, you have to change the *Formulae* into [C=B-A].

6 Change the output file DIFF.IDF into THICKNESS3.IDF, so enter this name in the field specified behind *Map C*.
7 Select the option *Map A*. This allows the size of the THICKNESS3.IDF to be exactly the size of the first mentioned IDF. Whenever you need results for the current zoom extent only, click the option *Window* instead, this will speed up the calculation since only a part of the selected maps (*Map A* and *Map B*) will be subtracted.
8 Click the *Compute* button.

The computed difference between the files TOP_LAYER3.IDF and BOT_LAYER3.IDF will be saved in the TMP folder of your USER environment by the name THICKNESS3.IDF. iMOD has added that file to the *iMOD Manager*, automatically.

Let us check the result.

9 Select the maps TOP_LAYER3.IDF, BOTTOM_LAYER3.IDF and THICKNESS3.IDF from the *iMOD Manager*.
10 Select the *Map Value* button ( ) on the *Map* tabs on the *iMOD Manager* window to start the *Map Value* window. Check the results by moving your mouse around the graphical display. Stop this inspection by rightclicking your mouse somewhere on the graphical display.



Example of the use of the 'Map Value' button when moving the mouse over the canvas.

Since this kind of visual inspection is rather easy to use, it is recommended to use it frequently to check any computations.

11 Select the *Map Info* button ( ) to inspect some simple statistics for THICKNESS3.IDF. Observe that the history of the THICKNESS3.IDF is saved and that the content is shown in the *Additional Information* box. Moreover click the Statistical button ( ) to get some statistical characteristics of the data.

Calculating Permeability

Each resulting IDF can be used subsequently for other map operations.

12 Open the T-VALUE_LAYER3.IDF (or KD-VALUE_LAYER3.IDF in older Tutorial sets) by clicking the Open IDF button ( ) and select this file from the {path of installfolder}\tutorials \TUT_DATA_MAP_OPER.
13 Select the maps T-VALUE_LAYER3.IDF (or KD-VALUE_LAYER3.IDF in older Tutorial sets) and the THICKNESS3.IDF from the *iMOD Manager*.
14 Select the *IDF Calculator* ( ) and change for *Map C* the IDF-file DIFF.IDF into K-VALUE_LAYER3.IDF.
15 Enter the formula: [C=A/B].

All values in Map A (will be KD-VALUE_LAYER3.IDF) will be divided by the values of Map B (THICKNESS3.IDF). If map A and map B are reversed, the equation can be entered as [C=B/A] without interchanging the IDF name next to the field *Map A* and *Map B*.

16 Select the option *Window* and click the *Compute* button; in this manner we will compute the permeability only for the current zoom extent.

To decrease computational times, map operations can be computed for the current zoomlevel on the graphical display only. The resulting IDF will have dimensions equal to the zoom level, however, cellsizes will be copied from the first mentioned IDF in the equation.

17 Check the result again with *Map Value* ( ).

Bear in mind that the computed legend classes are initially based on the minimum and maximum values of the IDF-file(s). Whenever you do not see much detail on the map, those minimum and maximum values might be far apart from each other. Use the *Percentile* legend (by clicking on: in the *Stretched* tab) for more detail on the values of the data. Looking at this graph can provide you with more insight in the color distribution of your legend.

Conditioned Map Operation

IDF Edit is a tool in which map operations can be applied for a particular selection of cells. In this tutorial a simple example is demonstrated. Suppose a map is required that shows all areas of the third model layer that have a thickness of more than 25 meter.

First we make an empty copy of THICKNESS3.IDF and name it THICKNESS3_25.IDF.

18 Enter the *Map Calculator* with THICKNESS3.IDF, enter the equation [C=0.0*A] and enter a filename for *Map C* to be THICKNESS3_25.IDF.

By means of the *Map Calculator* it is easier to make copies of IDF-files, rather than using the Windows Explorer, since the content can be blanked out and/or the resulting IDF can be resized (use the option *Window* or the option*x1,y1,x2,y2* where you can specify specific coordinates yourself).

Next step is to enter *IDF Edit*.

19 Select the option *IDF Edit* from the *IDF Options* menu from the *Map* menu.

It is not relevant what IDF is (de)selected in the *iMOD Manager*, since all IDF-files that are inside the *iMOD Manager* can be manipulated in *IDF Edit*.

Important is to specify an IDF that operates as a template. All mids of raster cells inside that particular IDF will be used to store any selection. Please be aware of the fact that a coarse IDF used as a template, will not make adjustments to a finer IDF.

20 Select THICKNESS3_25.IDF from the dropdown menu *Use selected IDF to store selected cells*.
21 Click the *Select* button to open the *IDF Edit Select* window.
22 Select the IDF-file THICKNESS3.IDF from the dropdown menu by *Evaluate IDF A:* and specify the *Logic* operator to be [>] and enter a *Value* to be [25].
23 Click the *Get Selection* button and observe that 97088 cells are selected.

The current selection will be displayed as filled rectangles. Especially whenever a large selection need to be displayed it can take a while.

24 Click the option *Show Selection* ( ) to turn the selection on or off.

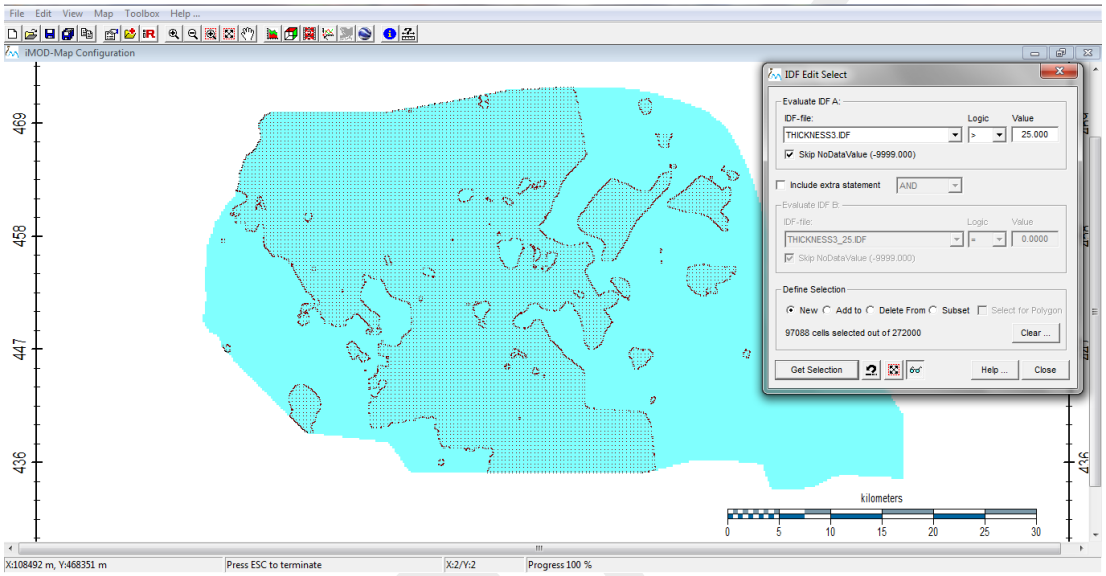


Figure 11.13: *Example of displaying the selected grid cells using the 'Show Selection' button in the 'IDF Edit Select' window.*

A selection is stored on disk in the {USER} \tmp folder. For iMOD-user a specific filename will be used, {USERNAME}_selected{i}.dat. These files will store the current (i) and previous selections (i-?). As long as these files exist, iMOD can undo a selection.

25 Click the *Undo Selection* ( ) button to reset the selection. Repeat step 20 to 22 again to restore the selection.
26 Close the *IDF Edit Select* window and click the *Calculate* button in the *IDF Edit* window to open the*IDF Edit Calculation* window.

This window offers several functions to adjust values in IDF-files of the current selection.

27 Select the option *Take From* and select THICKNESS3.IDF from the dropdown menu.
28 Select the THICKNESS3_25.IDF from the dropdown menu at the menu field *Assign Value TO*.
29 Click the *Calculate* button.

The *iMOD Manager* can be used whenever the *IDF Edit Calculate* window is active. Let us check whether the computation has been carried out correctly.

30 Select the maps THICKNESS3.IDF and THICKNESS3_25.IDF in the *iMOD Manager*, click the *redraw* button ( ) and click the *Map Value* button ( ). Inspect the values.

As long as the *IDF Edit Calculation* window is active, any computation to any IDF can be undone.

31 Click the *Undo Calculation* button ( ) in the *IDF Edit Calculation* window to undo the computation. Repeat step 29 to compute the values again.
32 Click the *Close* button and confirm the next window.

Let us see what other method can be used to make a selection and/or calculation.

33 Clear the current selection by clicking the *Clear* button in the *IDF Edit* window. Accept the following window stating whether you're sure to delete the selection.
34 Click the *Trace* button and select the option [Greater than selected value] in the *Values should be* option. Leave the option *Search Criterium* selected for [5 Point] which means that iMOD will search connecting cells that are connected on a five-point stencil. Use the option [9 Points] to use diagonal connected cells too.

This option allows you to make a selection that is determined by the location that you will select on the graphical window for all cells that have values greater than the value at the selected point. The selection should be connected to each other which makes it quite different from the previous selection method. Let us do that.

35 Select a cell from which a selection should be made that have greater values. Click your left mouse button. Please note that it may take a while to get the selection.

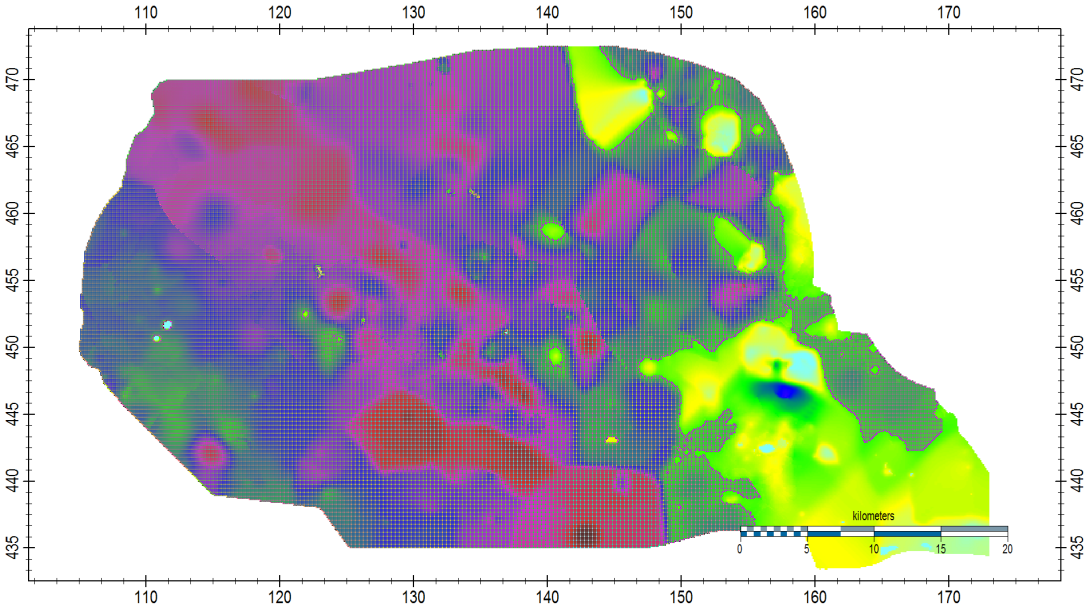


Figure 11.14: *Example of displaying selected cells using the Trace option.*

Finally we can adjust the current selection by means of a drawing functionality that allows you to interactively draw regions to add to the selection and/or remove from the current selection.

36 Select the *Draw* option in the *IDF Edit* window and choose option *Remove Cells* from the *IDF Edit Draw* window. Move over the cells you want to remove by holding down your left mouse button. Observe what is happening, try to add cells to the selection too.
37 Click the *Close* button and finally close the *IDF Edit* window by clicking its *Close* button too.

Map Scaling

One of the great options of iMOD is its ability to rescale data files. A variety of up- and downscaling algorithms have been implemented. In this example we will rescale the top elevation of a model layer from a cell size of 100 x 100 meter into a cell size of 1000 x 1000 meter.

38 Select the map TOP_LAYER3.IDF from the iMOD Manager.

39 Enter the *Map Calculator* ( ) and select the *Scale/Size* tab.

On this particular tab, a variety of up- and downscaling options are available by the menu fields *Upscale Formulae* and *Downscale Formulae*.

40 The resulting IDF will be saved in the same folder as the TOP_LAYER3.IDF and will be called TOP_LAYER3_SCALED.IDF. We accept this default output name.
41 Enter a gridsize of 1000 meter in the *Scale* field.

As a consequence, all resulting IDF-files from up- or downscaling will become IDF-files with equidistant cellsizes.

42 Select the option *Arithmetic Mean* from the *Upscale Formulae* menu. Click the *Compute...* button.

This formula takes the arithmetic mean for all values that lie inside a coarsened raster of the resulting IDF-file.

43 Observe the values of the TOP_LAYER3.IDF in relation to the scaled version TOP_LAYER3_SCALED.IDF. Use *Map Value* ( ) and inspect the Additional Information in *Map Info* ( ).

Experiment with different Formulae for up- and downscaling.

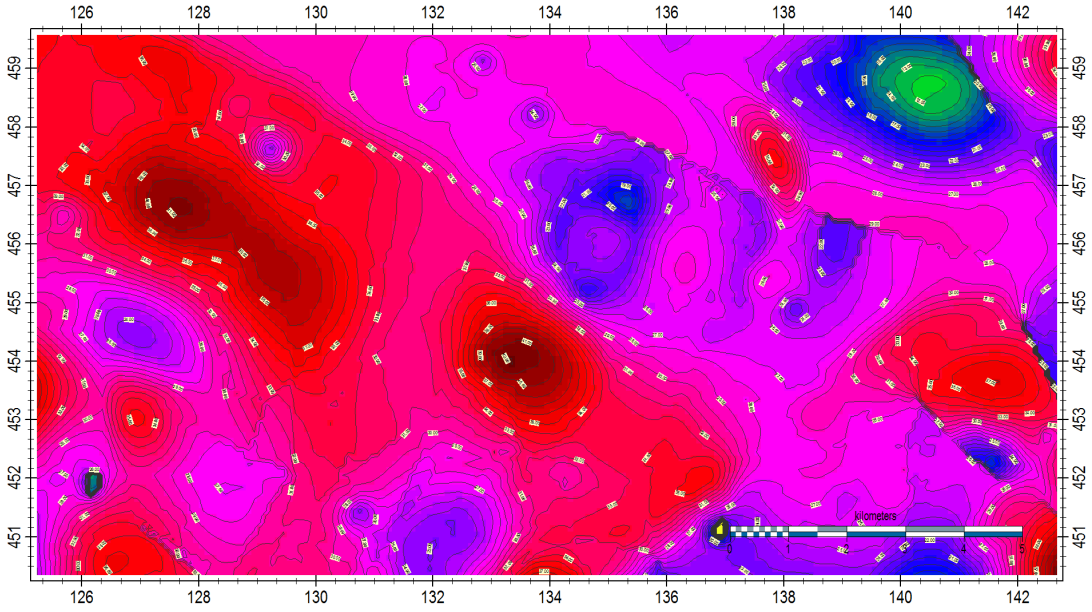


Figure 11.15: *Contour map of the original THICKNESS3.IDF-file (cell size 100x100 meter).*

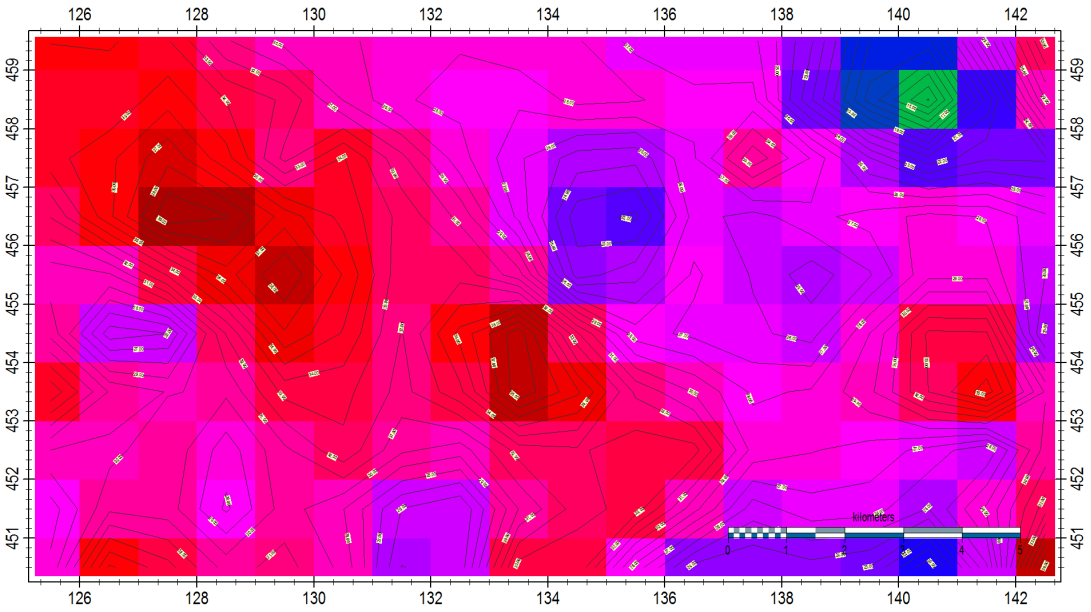


Figure 11.16: *Contour map of the upscaled THICKNESS3_SCALED.IDF-file (cell size 1000x1000 meter).*

11.3 Tutorial 3: Map Analyse

This tutorial gives a brief introduction to several options to visualize and analyse the content of IDF (raster) files. See for more detailed references section 7.1 (Cross-Section Tool), section 7.2 (Time-series Tool) and section 7.3 (3D Tool).

Outline

This is what you will do:

- ◇ Creating cross-sections over several IDF-files (combined with an IPF file) and manipulate the configuration;
- ◇ Computing timeseries out of IDF-files (combined with an IPF file);
- ◇ Using the 3D Tool.

Required Data

For this tutorial you need the following iMOD Data Files (IDF):

- ◇ Folder HEAD that contains HEAD_*_L1.IDF-files that represent transient model results;
- ◇ Folder SUBSOILSYSTEM that contains SURFACE_LEVEL.IDF and HYDROLOGICAL_BASE.IDF that represent the top and bottom elevation of the modeled hydrological system. In-between there are 6 aquitards distinguished that are described by their top and bottom elevations, called TOP_SDL{i} and BOT_SDL{i}, respectively for each of the 6 aquitards. Moreover, a subfolder called BOREHOLES containing borehole information stored in the file BOREHOLES.IPF
- ◇ Folder OBSERVATION that contains the file OBSERVATION.IPF representing several synthetic measurements.

All these files are located in the folder: {path of installfolder} \tutorials \TUT_MAP_ANALYSE.

Getting Started

1 Launch iMOD by double click on the iMOD executable in the Windows Explorer, and start with Create a new iMOD Project.
2 Go to View in the menu bar and select the iMOD Manager (or use the shortcut *Ctrl+M*).

Cross-Section

Let us start by creating a cross-section that visualizes the subsoil system as described by the data stored in the folder SUBSOILSYSTEM.

3 Open all IDF-files that are located in the SUBSOILSYSTEM subfolder of the folder {path of installfolder} \tutorials \TUT_Map_Analyse. Click the *Open IDF* button () from the *Maps* tabs on the *iMOD Manager*. Select the IDF-files in the *Open File* window and click the *Open* button.

All files will appear in the *iMOD Manager*, they will be ordered similar to the order in which they appeared in the Windows *Open File* window. Whenever the *Cross-Section Tool* is used to visualize the subsoil system it is important that IDF-files are arranged such that internal values are higher for IDF-files that appear higher in the list. Let us change the order of the files in the *iMOD Manager*.

4 Select the file HYDROLOGICALBASE.IDF from the *iMOD Manager* and click the button () sequentially until the file is at the bottom of the list. Select the file SURFACE_LEVEL.IDF and click the button () sequentially to put the file on top of the list.

Now we should arrange the TOP_SLD{i} and BOT_SLD{i} properly.

5 Select all TOP_SLD{i} files simultaneously in the *iMOD Manager* and press the button () to move them all together direct below the SURFACE_LEVEL.IDF. Now deselect the file TOP_SLD1.IDF

by clicking the Ctrl-key and your left mouse button simultaneously. Now press the button ( ) to move all files directly below the file BOT_SLD1.IDF.

It is important to place the IDF-files in the right order and to put together the top and bottom of each layer. Also, you can arrange files simultaneously by selecting them. It is faster to select multiple files and move them downwards to move the file underneath upwards.

6 Use the *Map Value* ( ) to inspect whether all files are arranged in the proper sequence.

Let us now make a cross section of the subsoil.

7 Select all IDF-files from the *iMOD Manager* and select the option *Toolbox* and select the option *Cross-Section Tool*, or, click alternatively, the *Cross-Section Tool* button ( ) from the main toolbar.

iMOD will display an empty graphical canvas, called the *iMOD Cross-Section CHILD* window, since no cross-section has defined yet. Let's start drawing the location of the cross-section.

8 Click the *Draw Line of the Cross-Section* button ( ) and left click your mouse button somewhere in the *Draw Cross-Section* window. Now move your mouse and you'll notice that the cross section will be built up automatically and will be refreshed each time you move your mouse. Right click somewhere else on the *Draw Cross-Section* window to store the line.

Whenever you move your mouse in the *iMOD Cross-Section CHILD* window, you'll notice a circle on the line of the cross-section that directs to the current location in the cross-section. Once a cross-section has been drawn, you can adjust/manipulate it, let's do that.

9 Move your mouse in the neighbourhood of the cross-section line in the *Draw Cross-Section* window. Click your left mouse button whenever the mouse symbol changes to . You can drag the current location of the cross-section.
10 Release the left mouse button and move towards one of the ends of the cross-sectional line. Press the left mouse button as soon as the symbol changes to . Move the mouse and watch that you can change the start- and/or end-location of the line.

Let's change the configuration of the cross-section, so our aquitards will be filled by different colours and our aquifers will become yellow.

11 Click the *Cross-Section Properties* button ( ) on the *Draw Cross-Section* window to display the *Cross-Section Properties* window for a more detailed description of its functionalities.
12 Click the option *Block Lines* to display the cross-section with lines that represent the true values and extent of the grid cells, rather than connecting grid cell mids. Click the *OK* button to observe the effects.
13 Re-open the *Cross-Section Properties* window.
14 Click the checkbox of the fifth column (*Line*) in the first row (Label is *Adjust all*) twice. Once to select it and then to deselect it. All rows underneath will become unchecked.
15 Click the checkbox of the seventh column (*Fill*) in the first row to check all rows underneath.
16 Click the first inputfield of the fourth column (*Colour*) and select a yellow colour from the default *Colour* window. All rows become yellow in this way.
17 Click the inputfield for the fourth column (*Colour*) for the third row (*Label*=TOP_SLD1.IDF) and change the colour into, let's say, green. Repeat this for TOP_SLD2.IDF, TOP_SLD3.IDF, TOP_SLD4.IDF, TOP_SLD5.IDF and TOP_SLD6.IDF. Give a grey colour to the HYDROLOGICALBASE.IDF. Click the *OK* button and observe the result.

Since this can be quite laborious, iMOD facilitates several display configurations that configure the table assuming the IDF-files are ordered in a particular manner (see section 7.3.1). For this set of IDF files you might use the one below.

18 Choose the configuration *Quasi 3D model (aquitard)* from the dropdown menu. Turn off the *Block Fills* to have a smoother surface. Check the differences with or without this feature.
19 Click the *OK* button to close the *Properties* window.

As you might observe, the cross-section gives a clear image of the subsurface. Moreover, the settings we just applied in the steps 11 until 19 are stored internally. Whenever you leave the *Cross-Section Tool* and re-enter it, these settings remain intact, except for the coordinates of the cross-section. Whenever you would like to re-use the same coordinates save them in the Miscellaneous tab on the *Cross-Section Properties* window, or alternatively save the last drawn cross-sections as a Demo-IMF (see section 7.1 for more information on this).

Let us try it.

20 Leave the Cross-Section Tool by clicking the *Close* button on the *Draw Cross-Section* window, or press the symbol () on the top-right of the*iMOD Cross-Section CHILD* window.

Let's us include some boreholes in the cross-section of the subsoil.

21 Open the file BOREHOLES.IPF from the folder {path of installfolder} \tutorials \TUT_MAP_ ANALYSE \SUBSOILSYSTEMS \BOREHOLES
22 Select this file together with all other files before entering the *Cross-Section Tool* (step 7).
23 Draw a cross-section (step 8) and insert some extra points by clicking your left mouse button, while drawing the cross-sectional line. Observe that all settings are still intact and a vertical dashed line is drawn at the intermediate points. Moreover, all boreholes that are within a close range to the cross-sectional line, are projected perpendicular on the cross-section.

We need to tell iMOD to use a different colour legend for plotting the boreholes, just like we did in 11.1.

24 Click the *Properties* button () on the *Draw Cross-Section* window.
25 Select the tab *Colouring* on the *Cross-Section Properties* window and click the *Open DLF* button (). Select the [BOREHOLES.DLF] from the . \TUT_MAP_ANALYSE folder and click the *Open* button.

iMOD will redraw the cross-section using this renewed legend and will use this legend during your iMOD session.

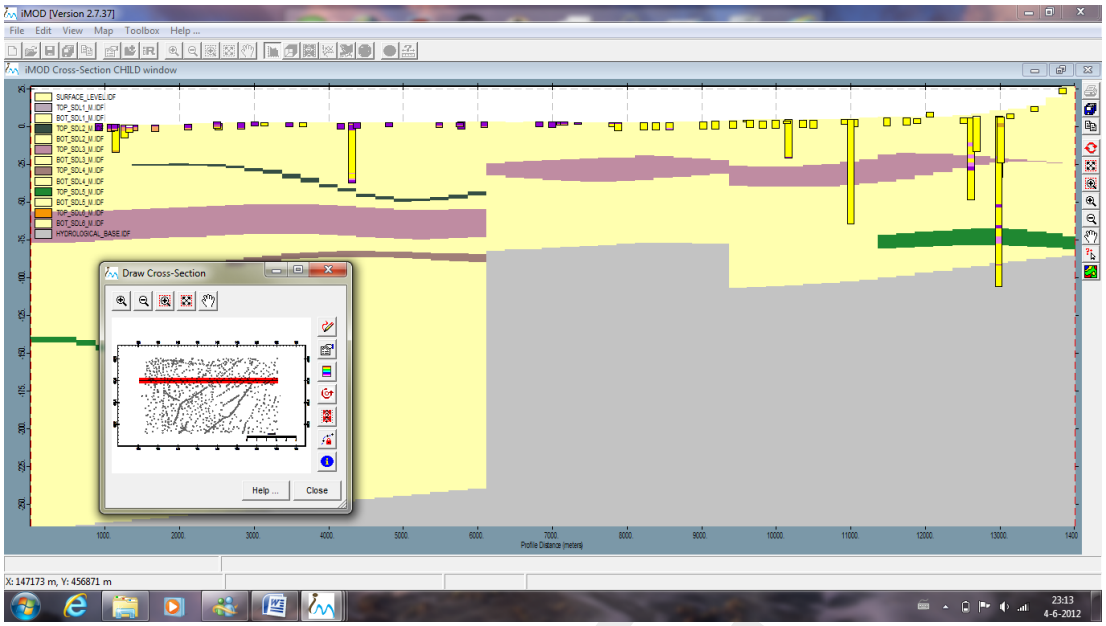


Figure 11.17: *Example of interactively generating a vertical cross-section of a 3D subsurface including boreholes.*

Note: Boreholes within a user-specified horizontal distance from the cross-section are projected on the vertical cross-section; this specified distance is visualised by the thickness of the trajectory of the cross-section (red line) in the 'Draw Cross-section' window. While drawing the trajectory of the cross-section in the 'Draw Cross-section' window, the vertical cross-section through the subsurface is updated in the 'Cross-Section CHILD' window simultaneously; when dragging an existing trajectory, the cross-section is updated as soon as the left mouse button is released.

26 Try to apply the zoom functionalities on the right of the *iMOD Cross-Section CHILD* window, and those located on the *Draw Cross-Section* window.
27 Close the *Cross-Section Tool* (see step 20).

You can use the *flip* button ( ) to rotate your cross-section clock-wise. Another option is to analyse the cross-section with the *Movie* button ( ). This option might be helpful if you would like to analyse e.g. the spatial variability of your vertical cross-section.

3D Tool

The next step is to analyse these data in the *3D tool*.

28 Click the *3D Tool* icon in the menu bar.
29 Now you're in the *3D IDF Settings* dialog. Select the same display configuration from the configuration dropdown menu as you did in the *Cross-Section tool* (see step 18).
30 Observe the contents of the *Display Configuration* and try to understand what happened. Click the *Apply* button. You should see an image similar to the figure below. You can, of course, change all settings in the *3D IDF Settings* window.

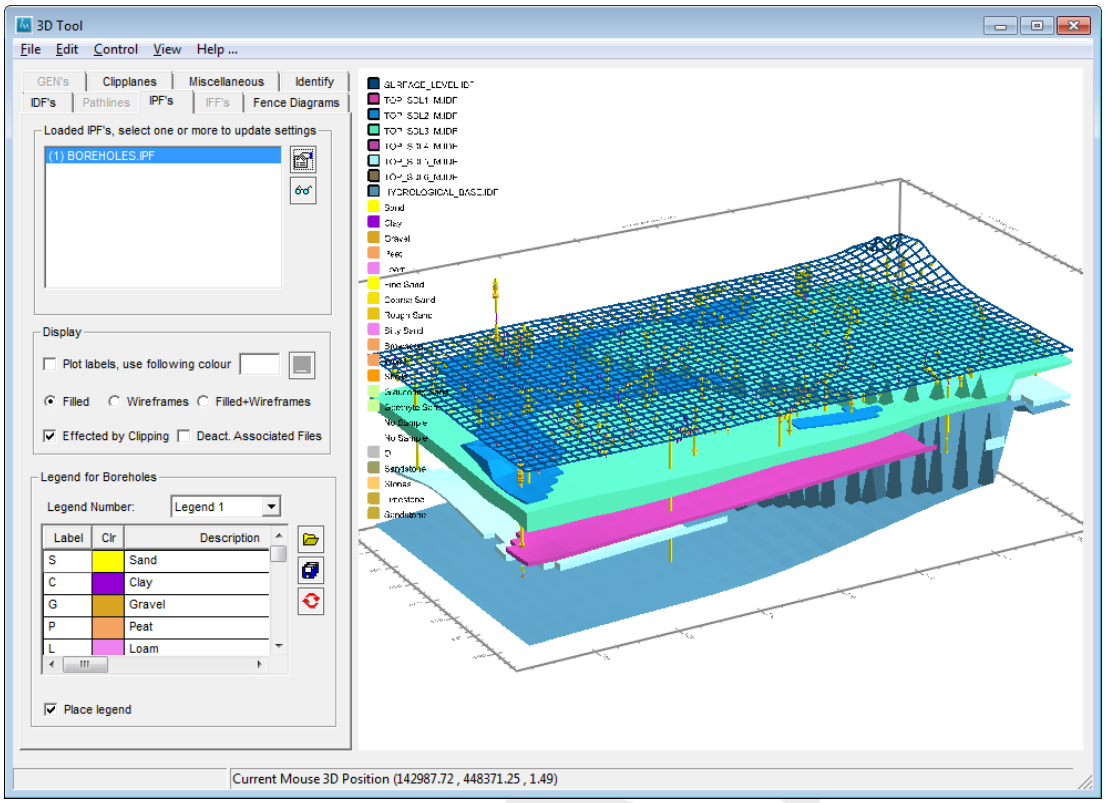


Figure 11.18: *3D Tool view of the subsurface and borehole data used in the previous 2D cross-section exercise.*

Let's experiment with the fence-diagrams and clipping planes functionalities. First we switch to a 3D solid model.

31 Select the *IDF's* tab from the *3D Tool* window.
32 Select the properties button ( ) to open the *3D IDF Settings* window.
33 Select the option [3D Model] from the dropdown menu at the field *Configuration*.
34 Click the button to close the *3D IDF Settings* window.
35 Select the lower most IDF file called HYDROLOGICALBASE.IDF.
36 Select the *Fence Diagram* tab from the *3D Tool* window.
37 Select the option *Draw* button ( ).

In this mode, you're able to actually draw a line on the 3D graphical canvas. If you move your mouse into that area, a red vertical line appear and a red dot. The red dot is the actual position of your mouse on the 3D image (actually the HYDROLOGICALBASE.IDF) and the x-,y- and z-coordinates are presented in the window status bar.

38 Click your left mouse button somewhere left, in the deeper part of the HYDROLOGICALBASE.IDF.
39 Now move your mouse button to the right and observe that a red rectangle will be drawn showing the actual position of the fence diagram.
40 Click your left mouse button to insert an additional knick-point somewhere half-way the left- and right utmost boundaries of the image.
41 Click your left mouse button somewhere in the right.
42 Click your right mouse button to stop drawing, the fence diagram will be computed as displayed directly.

Well, that's no too bad. Try to add another fence-diagram, the following figure shows what the result might be.

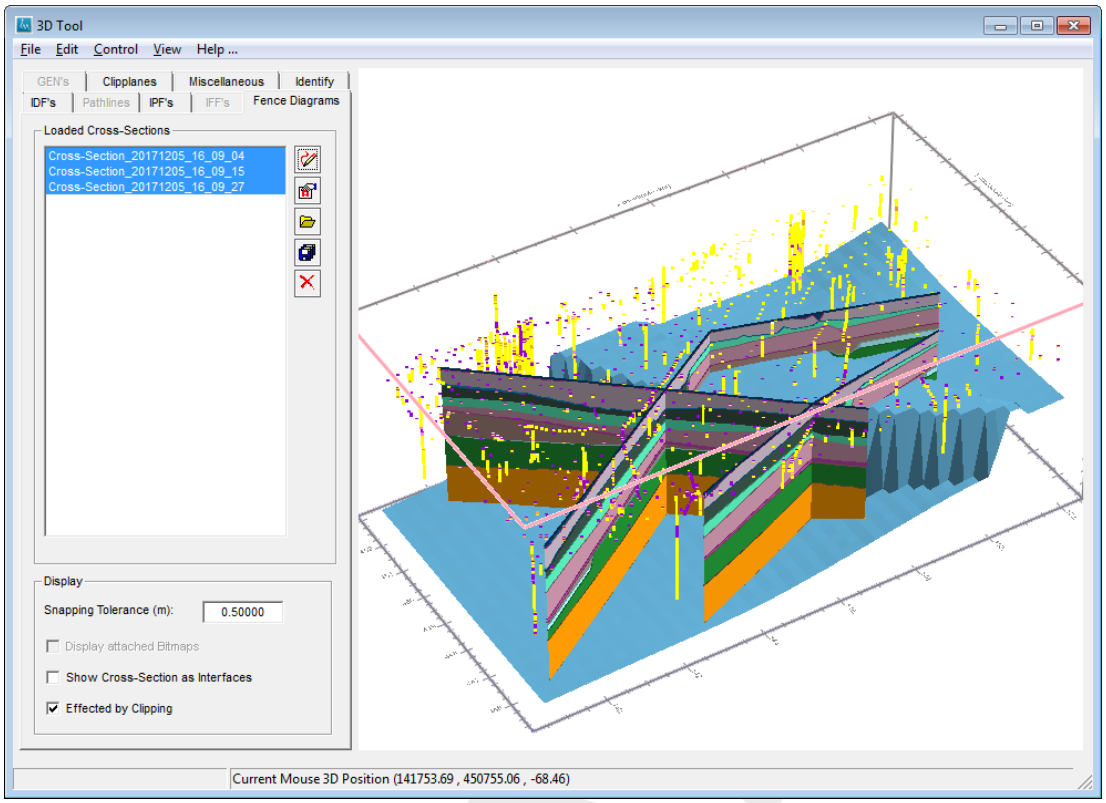


Figure 11.19: *3D Tool view of the subsurface and borehole data after drawing a fence diagram interactively.*

Note: The colours in the fence diagram are the colours assigned to the individual IDF files; you can each of them individually via the *IDF's* tab.

43 Save all cross-section via the option *Save As* ( )

44 Delete all cross-section via the option *Delete* ( )

45 Click *Yes* to confirm to remove the cross-sections.

46 Load in again some of the cross-sections via the open *Open Cross-section* ( ).

Remember that these cross-section file can be (re) used by the Solid Tool (see Tutorial 5, section 11.5). Let's apply some clipping planes.

47 Deselect the option *Effected by Clipping* on the *Fence Diagram* tab from the *3D Tool* window.

48 Select the *IDF's* tab again from the *3D Tool* window.

49 Select all the listed IDF files, except the last one called HYDROLOGICALBASE.IDF.

50 Select the option *Filled+Wireframes* to emphasize the top of the IDF files.

51 Select the *Clipplanes* tab from the *3D Tool* window.

52 Select the entry [ClippingPlane West] from the *Available Clipplanes* field.

53 Use the slider at the entry field *West* to move the clipping plane from the western border to the eastern boundary.

Observe that once the clipping is active the subsurface model is torn open and it is possible to look into it. To avoid that it is possible to fill those openings with a solid colour. This is called capping.

54 Select the option *Capping* from the *Clipplanes* tab.

For each entry in the 3D Tool, it is possible to exclude clipping. Let us do that.

55 Select the *IDF's* tab one more time from the *3D Tool* window.
56 Select the IDF files 7 up to 13.
57 Deselect the *Clipping* checkbox.
58 Select all entries again (except the last one).

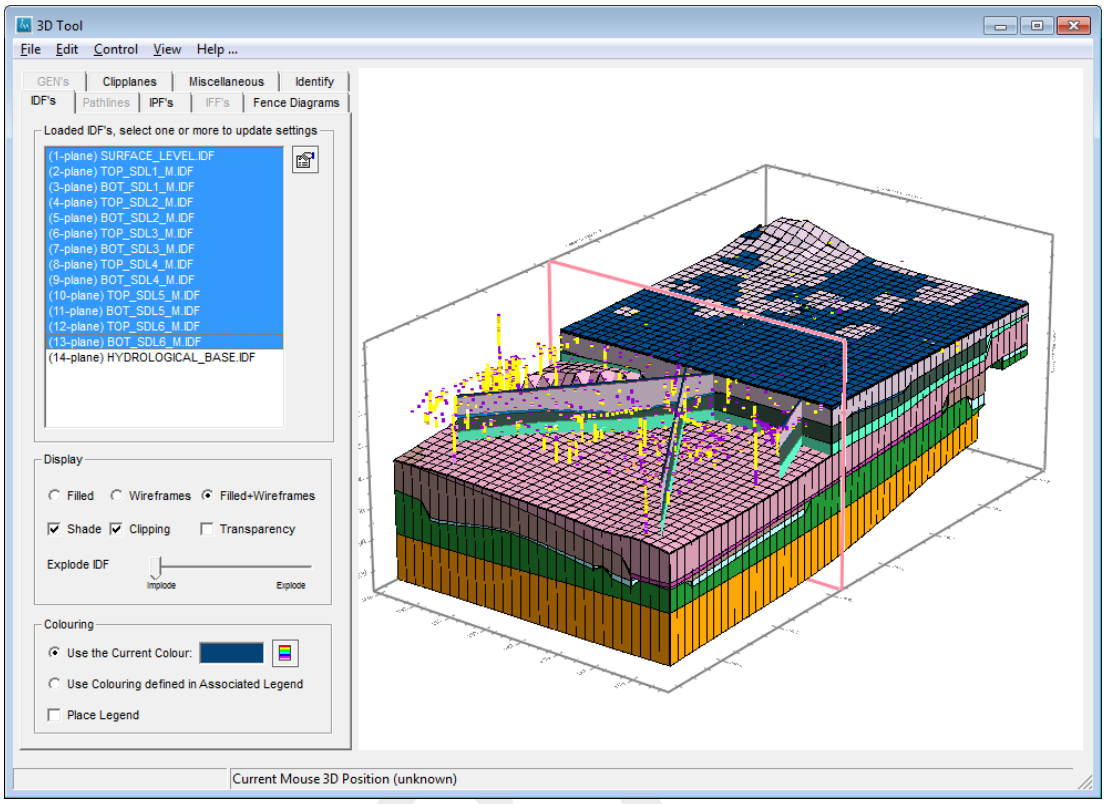


Figure 11.20: *3D Tool view of the subsurface and borehole data after drawing a fence diagram interactively.*

That's pretty fancy, so we can exclude the boreholes from the clipping as well.

59 Select the *IPF's* tab from the *3D Tool* window.
60 Deselect the *Effected by Clipping* checkbox.
61 Select the *Clipplanes* tab from the *3D Tool* window.
62 Move the *West* slider to see what happens.

Note: iMOD uses a technique to count how many time pixels are drawn in order to decide whether that position need to be capped. If the number of IDF files is uneven, this might cause some undesired interferences. Also if you activate more than one clipping plane in combination with capping. Try that to see what happens.

Okay, that is enough in 3D, let's go back.

63 Close the *3D Tool* by selecting *File* from the main menu and then *Close 3D Tool*.
64 Click the *Save As Current Project* button ( ) on the Map Menu bar and enter a name for the file, e.g. TUTORIAL3.IMF. All settings for the cross-section will be saved into the TUTORIAL3.IMF for later use.

Timeseries

Let's draw some timeseries interactively. In iMOD you need to open just one IDF-file that contains specific information about a date in its name notation, such as *_20101231_* to express the 31^{th} of

December 2010. Without having to open other files for other dates, iMOD searches for equivalent files, instead. Just as easy!

65 Open a single IDF-file located in the HEAD subfolder of the folder

{path of installfolder} \tutorials \TUT_MAP_ANALYSE. Click the *Open IDF* button( ) from the *Maps* tabs on the *iMOD Manager*.

66 Click the option *View* from the main menu, choose *Show IDF Features* and select *IDF Raster Lines*.

Choose to see the selected IDF. Observe that the current IDF has a non-equidistant network.

67 Zoom in on a particular area in the highly detailed area to observe the network layout even better.

68 Click the option *Toolbox* from the main menu and the option *Timeserie Tool* to start drawing time-series, interactively. Accept the *Available Dates* window for now. You could have specified a selection of the available IDF-files.

iMOD will read/open all available IDF-files from the same folder as the IDF-files that you've opened in step 65. This could take several seconds, watch the progress in the status bar. Once this has finished the *Draw Timeseries* window will be displayed.

69 Move your mouse over the screen and watch how the timeseries will be updated for the adjusted location.

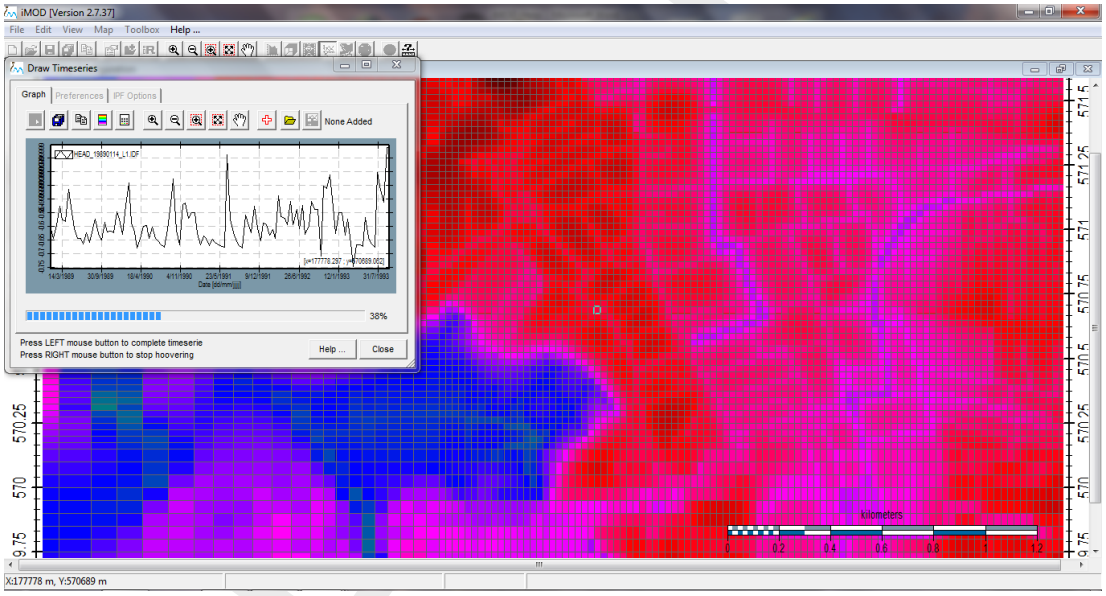


Figure 11.21: *Screen shot of the 'Draw Timeseries'- and 'Timeseries Tool'-windows while hovering with the mouse over a map of a series of IDF-files.*

Since iMOD will draw a timeserie that can be computed within one second only, you'll notice that not the entire timeserie will be plotted. This can be seen in the progress bar on the bottom of the *Draw Timeseries* window.

70 Click your right mouse button to compute the entire timeserie and stop the hoover mode.

As soon as the hovering has stopped you can examine the entire drawn timeserie.

71 Use the regular zoom buttons to navigate on the timeserie.

72 Explore the tab *Preferences*, to see what you can do and how it works.

73 Change the appearance of the timeserie by clicking the *Legend* button ( ) on the *Graph* tab.

Experiment with the options that are available in the *Individual Colouring* window.

11.4 Tutorial 4: Create your First Groundwater Flow Model

This tutorial gives a short introduction in creating a groundwater flow model from scratch. It yields a preliminary model that will be enhanced even more in section 11.5.

All steps in this tutorial were demonstrated during the first live iMOD webinar, recorded on May 2016. You can watch the recordings via the webinar page on the iMOD website (http://oss.deltares.nl/web/imod/webinars). *(Be aware that some parts of the tutorial might be improved or edited in the mean time)*

Outline

This is what you will do:

- ◇ Create the basic input files that are necessary to simulate a simple groundwater flow model;
- ◇ Enhance the model with an extraction well to compute the drawdown caused by the well;
- ◇ Simulate flowlines that describe the catchment area of the well;
- ◇ Experiment with extraction rates to compute the maximum sustainable yield without extracting water from the sea.

Required Data

For this tutorial you need the following iMOD Data Folders:

- ◇ island.png/island.pngw: this image sketches the outlines of the island

This file is located in/below the folder: {path of installfolder} \tutorials \TUT_INITIAL_MODELING.

Beside this data you will need the iMODFLOW executable to make the final model computations.

Getting Started

1 Place the iMODFLOW executable somewhere on your disk (for instance next to the iMOD-GUI executable) and define the keyword MODFLOW in the IMOD_INIT.PRF.

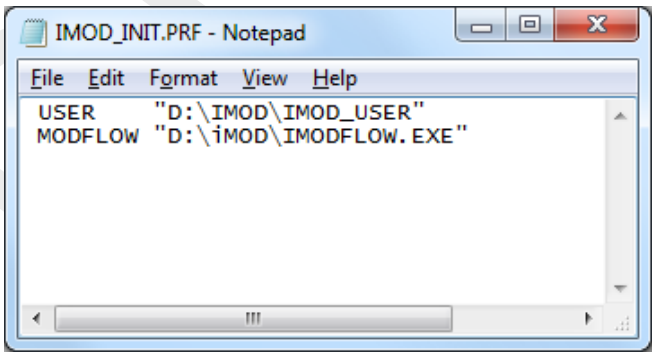


Figure 11.22: *Example of a content of an iMOD_INIT.PRF file.*

See section 11.1 and chapter 9 for more information about the folder structure in iMOD and a description of IMOD_INIT.PRF (see section 9.1 for more information about this PRF file). Please restart iMOD after changing the IMOD_INIT.PRF file.

2 Launch iMOD by double clicking the iMOD executable in the Windows Explorer, and start by selecting the option *Create a new iMOD Project*. Click the *Start* button.

Background Image

One of the first things one would like to display is an image of the outline of our island that we're going to model. Let's do that.

3 Select the option *View* from the main menu and then select *Add Background Image* ... from the dropdown menu. This will start the *Add Background Image* dialog.
4 Select the option *Add* from the dialog and select the file {path of installfolder} \tutorials \TUT_INITIAL_MODELING \island.png from the Windows Explorer, see section 5.3 for more information about this window.
5 Select the *Apply* button that closes the window.
6 Click the *Show Background Image* ( ) on the main menu whenever the image does not appear.

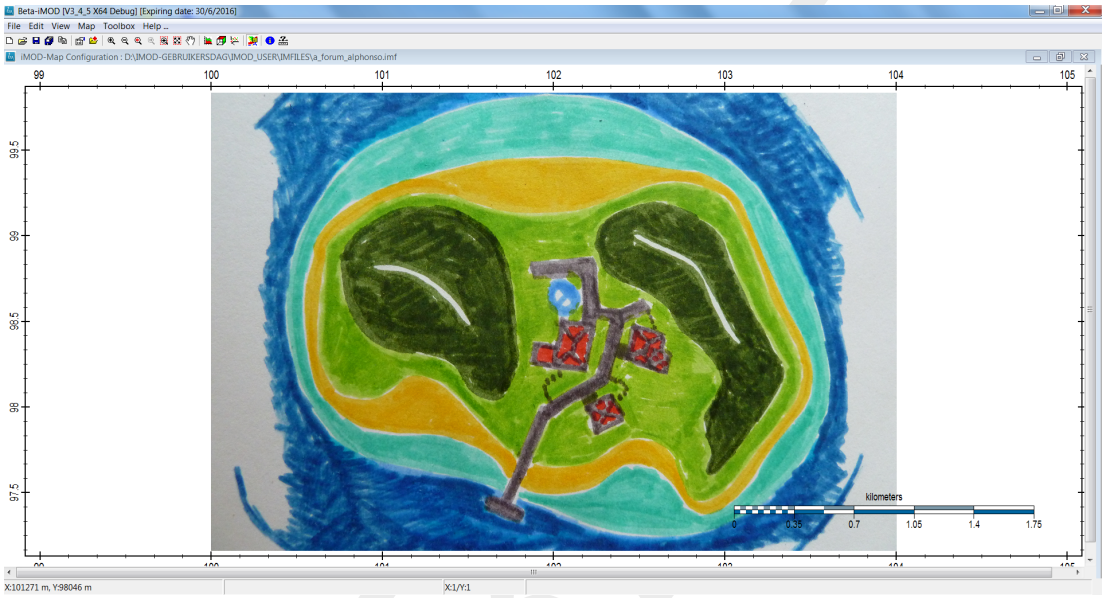


Figure 11.23: *Example of showing a topographical map using the main menu 'View', 'Show Background Image(s)' option.*

This island represents a small tropical island (3 x 3 km) somewhere in the Pacific. It is surrounded by shallow waters with crystal clear water (light blue), white beaches all around (yellow) and meadows (light green) with some bush areas (dark green). In the centre of the island there exists a small settlement with a few houses that use groundwater for watering their fields and cattle and use it as primary source for drinking water. They extract groundwater at the centre of the island (blue circle).

Many people have discovered the beauty of this island and plans arise to build a resort on the island. This will increase the pressure on the natural water resource and the question to be answered is: "*How much water can be sustainably extracted from the subsoil, without attracting seawater in the near future to the pumps?*".

With this very simple example we will use iMOD to build a hypothetical model of this island. By means of this example we will illustrate the methodology in iMOD to create a groundwater flow model. At this stage we will ignore any effects of density-driven components caused by salt water. The following steps will be undertaken:

- ◇ build an IDF-file for the surface level of the island which will be our uppermost boundary of the system modeled;
- ◇ create an IDF-file that defines the boundary conditions of the model, for which part the groundwater head needs to be computed and for which part this is known beforehand;
- ◇ create an IPF file that describes the location and rate of the pumping well;
- ◇ create a runfile that describes the necessary files and values for the simulation;
- ◇ simulate the model using the runfile;

- ◇ create startpoints for the particle tracking simulation and carry out this simulation;
- ◇ modify the extraction rate in order to search for the maximum sustainable yield.

Okay, a lot of work needs to be done, so let's go!

Creating the topography

Our model will describe the groundwater flow between the surface level and the bedrock in the subsoil. Our first task is to get a digital representation of the surface elevation. Often this is available in the form of a Digital Elevation Model (DEM), unfortunately, we're lacking this DEM for our island, so we have to sketch it ourselves.

7 Select the option *Edit*, *Create Feature* from the main menu, then select the option *IDF's from...* and finally the option *Polygons/Lines [GEN]* to start the *Create IDF* window.

With this functionality in iMOD we're able to create simple features in the format that iMOD needs to perform a model simulation. In this case we would like to create the outline of the surface level and therefore we need to draw the contours of the surface level and assign appropriate levels to it. After that we can tell iMOD to interpolate from the contours. Okay, let's start this by digitizing the shore of the island (i.e. 0.0 m+MSL contour).

8 Make sure you've shown the topographical image of the island (repeat step 3 upto 6 whenever you don't see the image);

9 Click the *Draw* button (), this will start the *Select* window;

10 Select the option *Polygon* from the *Shape types*;

11 Click the *Ok* button.

Your cursor has been changed into the following cursor symbol  which means that you can start drawing a polygon.

12 Click your left mouse button on the graphical canvas at the location of the first point of the polygon to be drawn. Repeat clicking your left mouse button to insert more point of the polygon. Whenever you are satisfied click your right mouse button to stop this process. Note: if you are a left-handed person and you converted your mouse button settings, 'left mouse button' should be 'right mouse button' and vice versa in these tutorials.

See section 4.5 for more details how to modify the polygon once you've created it.

Figure 11.24: *Example of the polygon that you might have created.*

Now we have to assign a surface level of 0.0 m+MSL to the drawn polygon.

13 Click the *Information* button ( ) to start the *Content of File* window. Click the *Yes* button whenever iMOD asks "*Do you want to add additional data to the shapes?*".

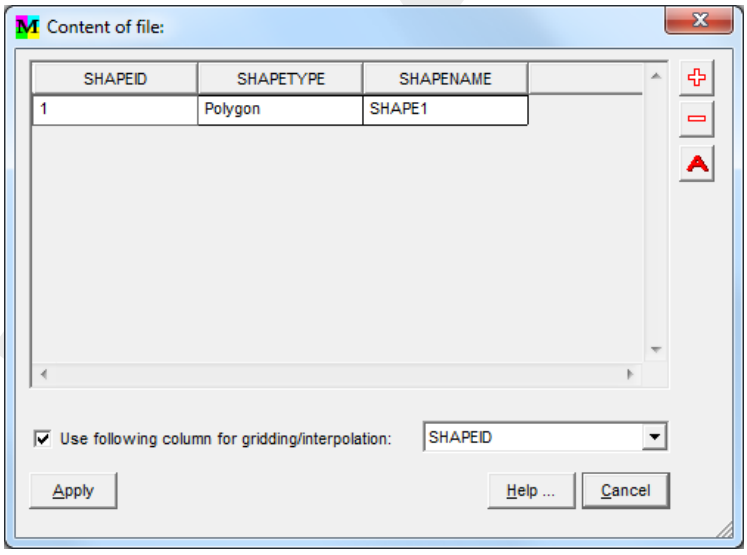


Figure 11.25: *Example of the 'Content of file:' window.*

With this *Content of file* window we can observe/change the attributes that can be added to the shapes (polygons, lines). We have to add a new attribute to the data to store the contour values of the surface level. Let's do that.

14 Click the *Add Attribute* button ( ) and enter the label [Level] in the Input window that arises.

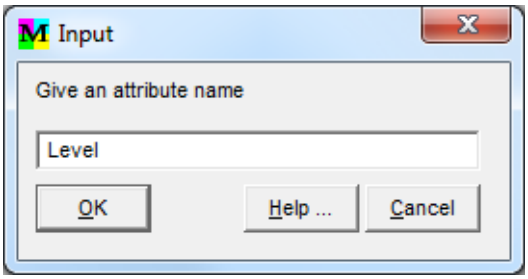


Figure 11.26: *Example of the 'Input' window to add an attribute.*

15 Click the *OK* button to return to the *Content of file* window. Observe that an extra column has been added to the table.
16 Enter the value [0.0] to the input field Level.
17 Select the option *Use following column for gridding/interpolation* and select the attribute [Level] from the dropdown menu. iMOD will use the values from this column during the interpolation.

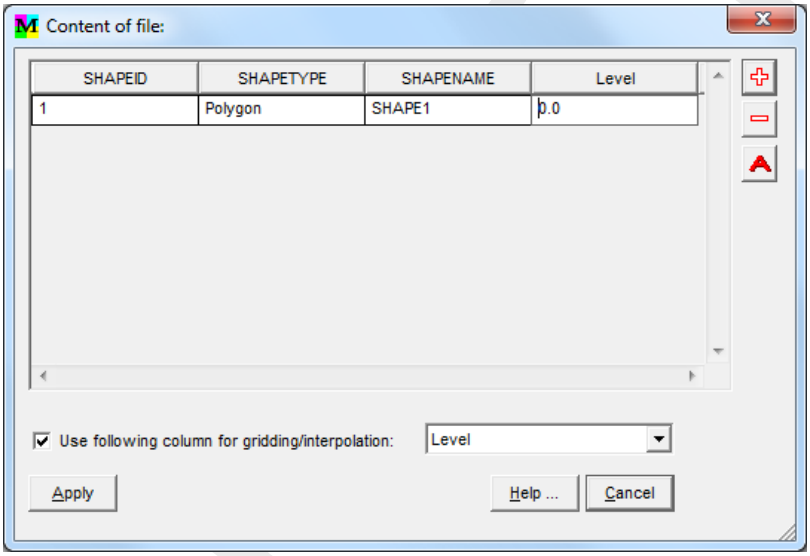


Figure 11.27: *Example of the 'Content of file:' window.*

18 Click the *Apply* button.

Similarly repeat steps 12 to 18 to sketch the other polygons and lines of the surface elevation of the island. Choose the corresponding elevation values from the table below. Please note that it is not necessary to add the attribute column named [Level] for each shape since this will be applicable for all shapes that are entered. Also use a [Line] feature to express the watershed on the more elevated parts of the island.

Table 11.1: *Elevation of the island elements*

Island level description	**elevation** [m]
Hills ridge (west and east)	15
Hills feet (west and east)	5
Border green area	2
Island Boundary	0
Shallow water	-1
Deep water	-5

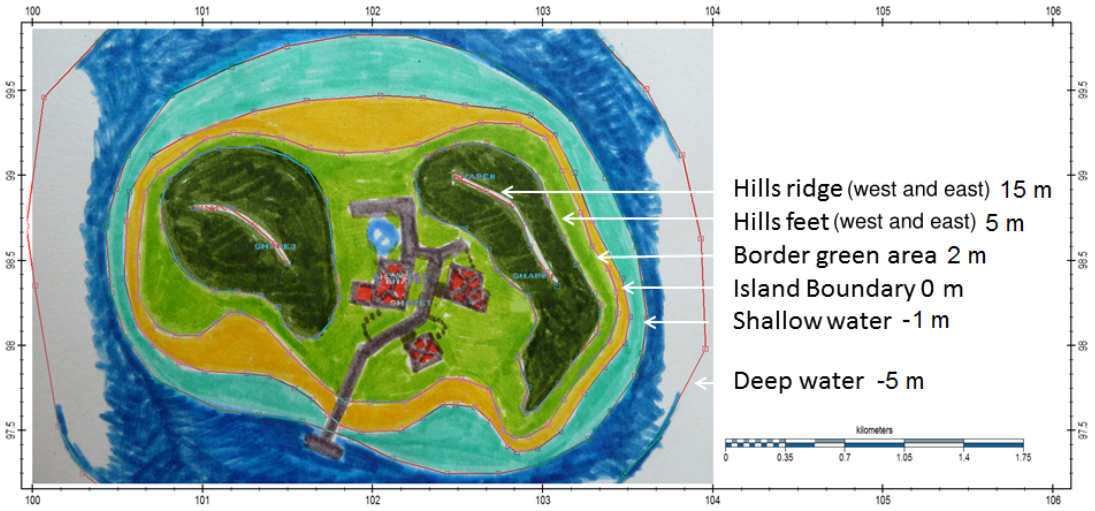


Figure 11.28: *Example of a final result sketching the surface level for the island.*

19 To save the set of shapes listed in the 'GENs-tab' window click on the *Save As* button ( ) and save as ISLAND.GEN in the folder iMOD_USER \SHAPES. Please note that it does not matter whether only some of the shapes were selected, all shapes listed in the 'GENs'-tab will be saved as one set of shapes.

The associated labels are saved to the file ISLAND.DAT at the same location. Both files (ISLAND.GEN and ISLAND.DAT) may be modified outside iMOD using a text-editor. Please note that the first column in the DAT file will be used to reference between the GEN and DAT file. Make sure that this connection remains intact!

Once we've outlined the surface level, we will interpolate the contours to a grid (IDF) with rastersize of 10 meter. This will be accurate enough for our simulation. However, gridsizes at this stage will not be determined for the final simulation scale. See section 11.6 for more information on scaling issues for model simulations.

20 Click the button *GEN-Extent* to adjust the coordinate settings for the IDF to be created such that the entire GEN will be included in the gridding.
21 In the boxes for the Lower Left and Upper Right coordinates of the extent *(XLLC, XURC etc)*, please remove the centimeter values and round down or up to meters.
22 Enter a cell size of 10 meter in the field *Cellsize (m)*.
23 Select the option [PCG (Preconditioned Conjugate Gradient)] from the *Method* dropdown menu.

This interpolation method will follow the given contours accurately giving a smooth representation of the entered contours.

24 Click the *Apply* button and save the gridded IDF-file in the folder iMOD_USER \DBASE as SURFACE_LEVEL.IDF. Probably you need to create this folder first!

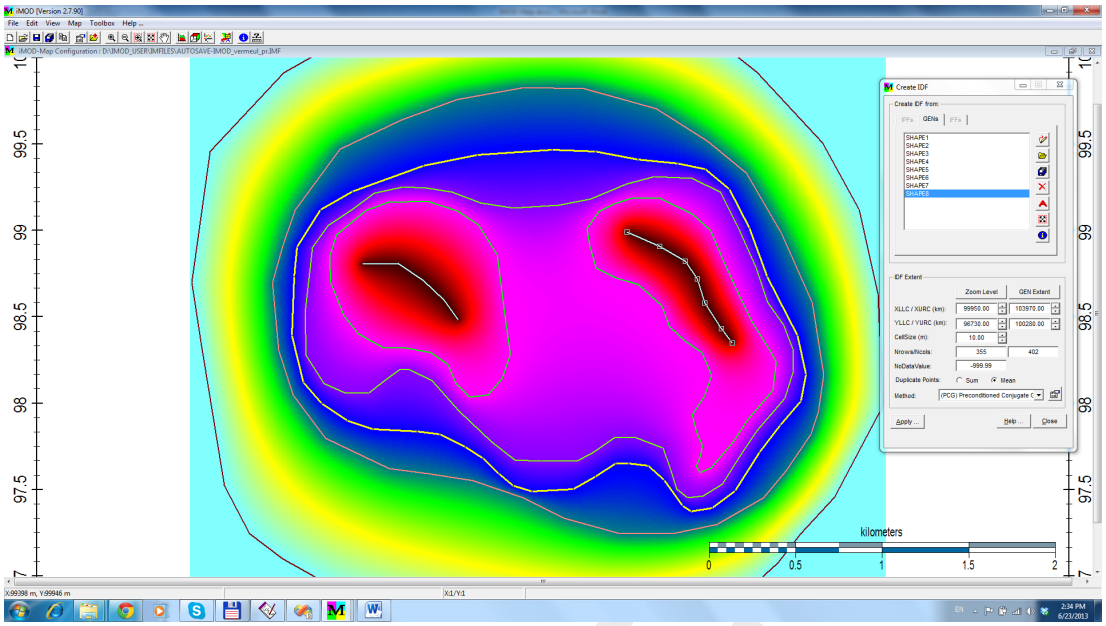


Figure 11.29: *Example of a resulting topography of the island.*

It is completely irrelevant where files are saved actually; however, in order to keep your project organized well, it is advisable to create a clear structure in which you save all files that are related to the model. **Note:** Commonly, we use the foldername DBASE to store all model files. So whenever we refer to the folder DBASE in the coming parts of this tutorial we actually denote the IMOD_USER \DBASE folder.

25 Use your experience from the Tutorials 1, 2 and 3 to create a 3D image of the topography we just created.

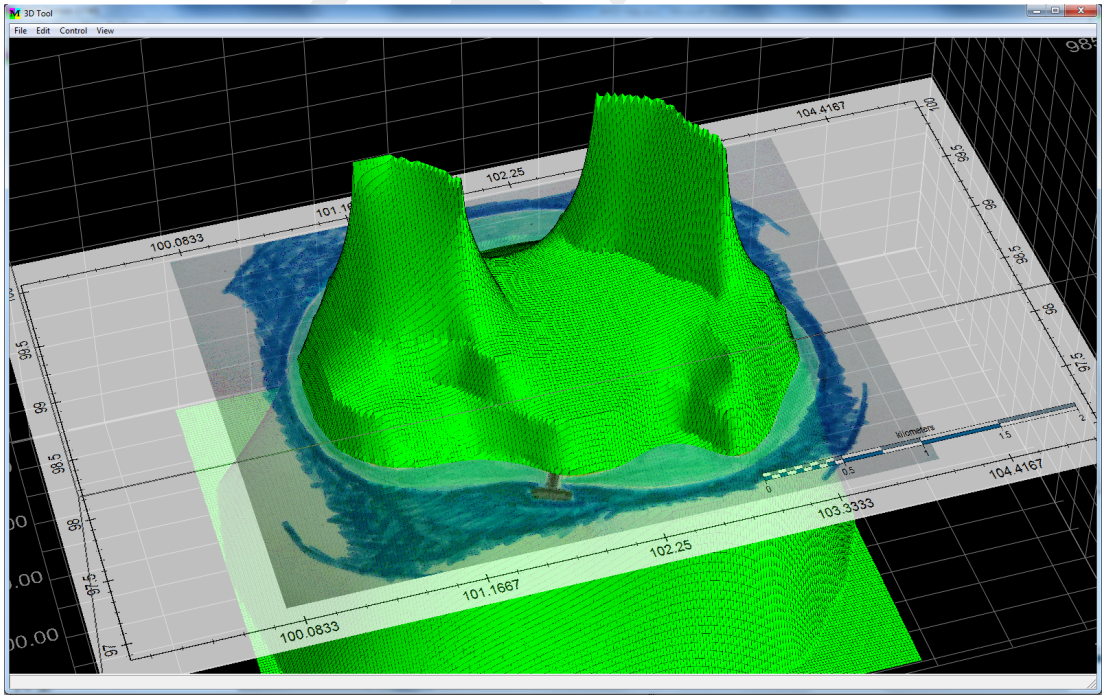


Figure 11.30: *Example of a 3D image of your created island.*

Creating the boundary conditions

Okay, now we've outlined the uppermost boundary of our model, we will specify those areas that are part of the model simulation (*active areas*) and areas that have fixed values (*fixed or non-active areas*) for the hydraulic heads. We will use IDF-Calc and IDF-Edit that you both have used in section 11.2 (we assume you finished section 11.2 before you moved on to section 11.4). Okay, we need to copy the SURFACE_LEVEL.IDF to an IDF that we can use for the definition of boundary conditions.

26 Select the [SURFACE_LEVEL.IDF] from the *iMOD Manager* and Click the *IDF Calculator* button.
27 Enter the IDF-file [. \DBASE \BOUNDARY.IDF] in the Map C field on the *Algebra* tab on the *Map Operations* window. Make sure you use the same folder name as the one used for . \DBASE \SURFACE_LEVEL.IDF.
28 Make sure the entered formulae is [C=A].
29 Select the option *Map A* to create an IDF that has the same dimensions as the IDF-file mentioned by *Map A* (i.e. SURFACE_LEVEL.IDF).
30 Click the *Compute* button.

Please note that when we created the BOUNDARY.IDF and it is drawn and listed in the iMOD Manager, it is a copy of the SURFACE_LEVEL.IDF. Now we are going to determine the *active areas* of the simulation by selecting the area with surface level values above zero. Areas with values less than zero will be *fixed areas*. So let's continue with that.

31 Select the [BOUNDARY.IDF] from the *iMOD Manager* and click the *Map* option from the main menu, select *IDF Options* and then *IDF Edit* option to start the *IDF Edit* window.
32 Click the *Select* button to start the *IDF Edit Select* window.
33 Select the option [>=] from the dropdown menu *Logic* in the groupbox *Evaluate IDF A*.
34 Click the *Get Selection* button to get a selection of all cells that have values greater and equal zero.

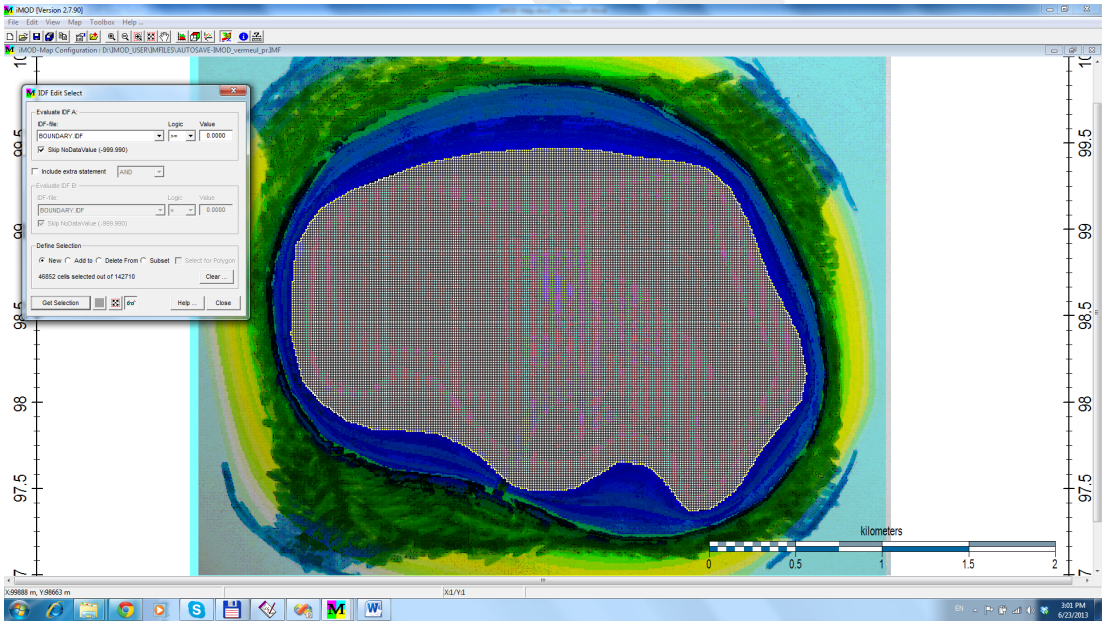


Figure 11.31: *Example of the selection of cells with values greater or equal to zero.*

35 Click the option *Close* to return to the *IDF Edit* window again.
36 Click the *Calculate* option from the *IDF Edit* window to start the *IDF Edit Calculation* window.
37 Select the option *New Value* in the group *Define Values by* and enter the value [1] so it says *New Value* [=] [1.0].
38 Click the *Calculate* button.
39 Click the *Close* button and click *Yes* on the appearing window asking you to be sure to leave this Edit environment.

Repeat steps 32 upto 39 to adjust all values that have values less than zero and calculate those values

to become -1.0.

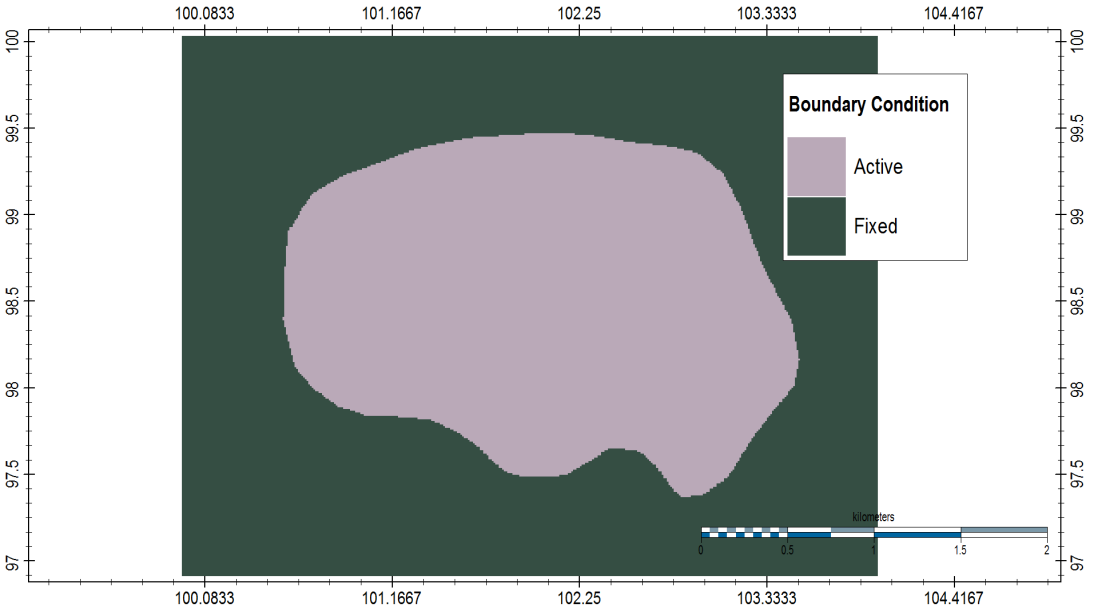


Figure 11.32: *Example of assigned active and fixed head cells.*

Last thing we need to do is to create an IPF file (iMOD Point File) to represent the well in our model. We have a single well situated in the centre of the island, in the following steps we will create this simple file inside iMOD; however, it can be easily modified/created outside iMOD. For large datasets it is often more convenient to process these types of data in another program.

40 Deselect the BOUNDARY.IDF in the iMOD Manager.

41 Make sure you've switched the background image on, if not press *Show Background Image* ( ) again.

42 Select the option *Show Transparent IDFs* from the *View* menu, to be able to either see the background image and the IDF files to be analysed.

43 Select the option *Edit* from the main menu, then select the option *Create Feature* and then *IPF's* to start the *Create IPF's* window.

44 Click the *Draw* button ( ) and click your left mouse button when the cursor is on the location of the well.

45 Click your right mouse button to return to the *Create IPF's* window.

46 Click the *Information* ( ) button and click the *Yes* button on the question "*Do you want to add additional data to the shapes?".*

47 Select the third column and press the *Rename* button ( ).

48 Enter the name [Q] in the field *Rename Attribute* and click the *OK* button.

49 Enter a value of [-500.0] for the extraction rate in the first row, third column. Rates are entered in m^3/day and you cannot perturb the first two columns since these are used by iMOD for internal processes.

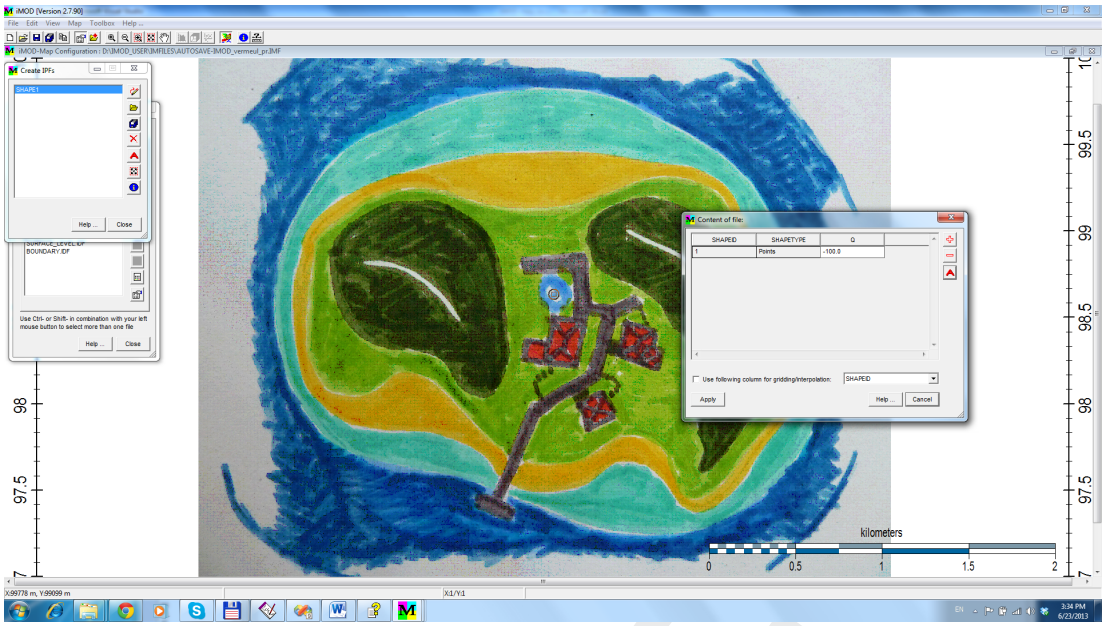


Figure 11.33: *Example of a screen layout at the current step.*

50 Click the *Apply* button to close this window and return to the *Create IPFs* window.

51 Click the *Save As* button ( ) to save this file in . \DBASE \WELL.IPF.

For this hypothetical model, these three IDF-files (. \DBASE \SURFACE_LEVEL.IDF, . \ DBASE \BOUNDARY.IDF and . \DBASE \WELL.IPF) are the only spatial varying data sources. The other model input are constant values throughout the model domain and we will assign those values by creating a modeling project in the following section. In the next figure we've given a sketch of the subsoil and flow patterns that might occur.

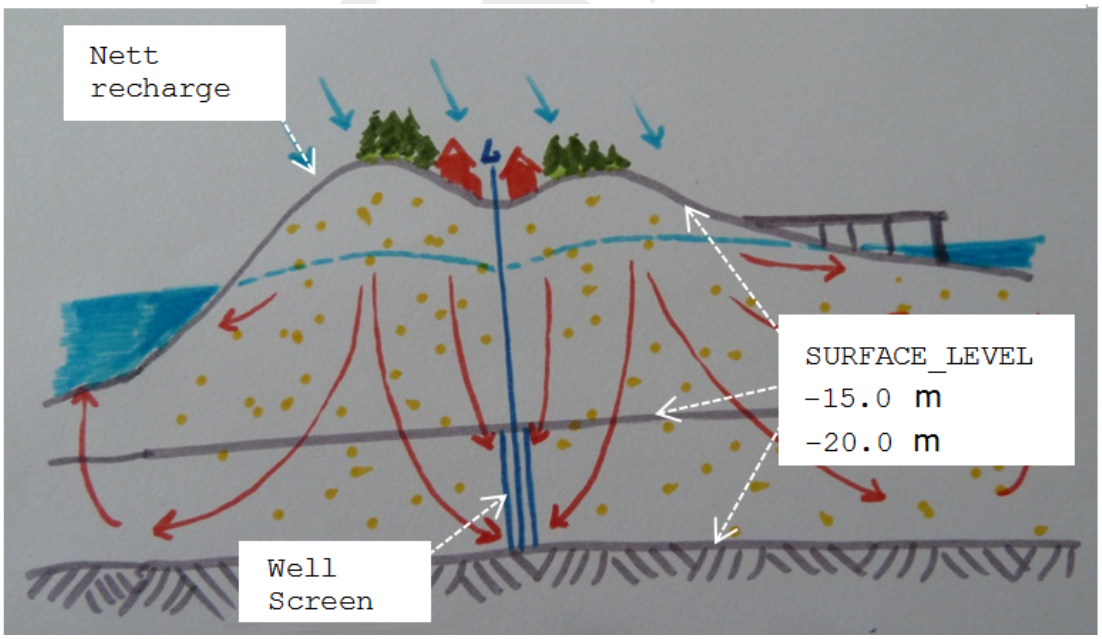


Figure 11.34: *Sketch of a estimated flow pattern that might occur in our island model.*

Creating a Modeling Project and defining a runfile

iMOD arranges a model project by a project file, a so called PRJ file. This file stores all files that are assigned to particular phenomena in the model. From a project file (*.PRJ) you can generate a runfile (*.RUN) that will be used eventually to simulate groundwater heads for a specific configuration. We can imagine that you'll need to simulate different scenarios (e.g. steady-state simulation versus transient simulations) that can be initiated by the *Project-Manager*. Well, probably it is better just to start with it.

52 Select the option *View* from the main menu and select the option *Project Manager* to start the *Project Manager* window.

This window shows all available packages that are supported by iMOD. Still many will come in future though. Okay, we have to fill in this project manager with our model configuration. In the table, shown below, we have outlined the requirements for this particular three-layered model. Note: the "Porosity Aquifer" and "Porosity Aquitard" values are only needed for the pathlines simulation from step 95 onwards.

Table 11.2: *Model requirements for a confined, steady-state three layered model.*

Parameter	model layer	IDF/Constant Value
BND Boundary	1	. \DBASE \BOUNDARY.IDF
	2,3	1
SHD Starting Heads	1,2,3	0.0 m+MSL
TOP Top Elevation	1	. \DBASE \SURFACE_LEVEL.IDF
	2	. \DBASE \SURFACE_LEVEL.IDF - 1.0 m
	3	-15.0 m+MSL
BOT Bottom Elevation	1	. \DBASE \SURFACE_LEVEL.IDF - 1.0 m
	2	-15.0 m+MSL
	3	-20.0 m+MSL
KHV Horizontal Permeability	1,2,3	25.0 m/day
KVA Vertical Anisotropy	1,2,3	1.0
KVV Vertical Permeability	1,2	25.0 m/day
WEL Wells	3	. \DBASE \WELL.IPF
RCH Net Recharge	1	1.0 mm/day

Okay let us fill in the boundary conditions in the *Project Manager*.

53 Select the option [(BND) Boundary Conditions] in the *Project Definition* list.

54 Click the *Properties* button () to start the *Define Characteristics for* window.

In the current window you can specify how the package (in this case the Boundary Condition) needs to be configured. Let us fill this dialog for the boundary condition for model layer 1.

55 Enter the value [1] in the *Assign Parameter to model layer* . . . field, if this is not the case by default.

56 Specify a *Parameter Multiplication Factor* of [1.0], if that is not the case by default. Any parameter can be multiplied with the associated factor during runtime. You can use this factor to easily perform some sensitivity analyses on parameters and their effect on the distribution of the groundwater head.

57 Specify a *Parameter Addition Value* of [0.0], if that is not the case by default. Any value can be added to or subtracted from a parameter.

58 Select the option *Add File* and click the *Open File* button. Select the file . \DBASE \BOUNDARY.IDF from the appropriate folder. This file we've created in step 39, remember?

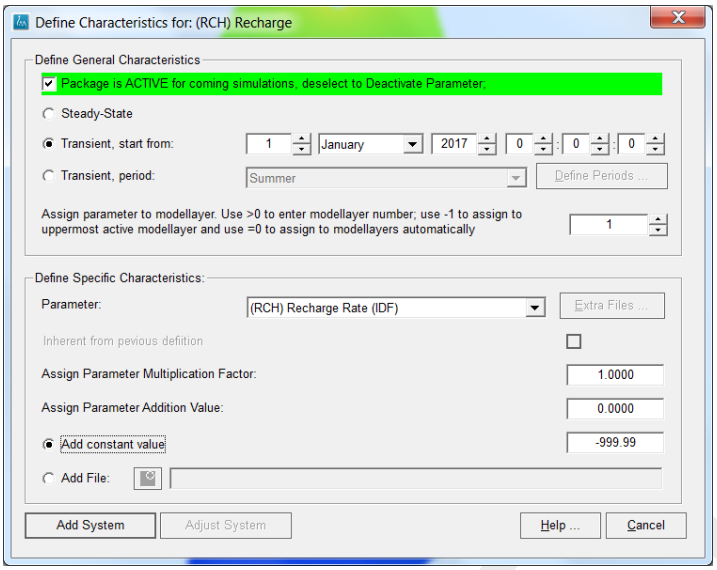


Figure 11.35: *Example of the 'Define Characteristics for:' window, filled in for Recharge (RCH).*

59 Click the *Add New System* button and this will return you to the *Project Manager* window. You'll notice that the option [(BND) Boundary Conditions] has been altered. You can select the "plus" sign to expand the tree view. You'll notice the entered fields in the presented string.

Now let us fill in the remaining parameters from the table given.

60 Repeat the steps 53 upto 59 for the remaining parameters. Take care to select the parameter name in the *Project Definition* list each time you want to open the *Define Characteristics for* window to enter NEW parameters.

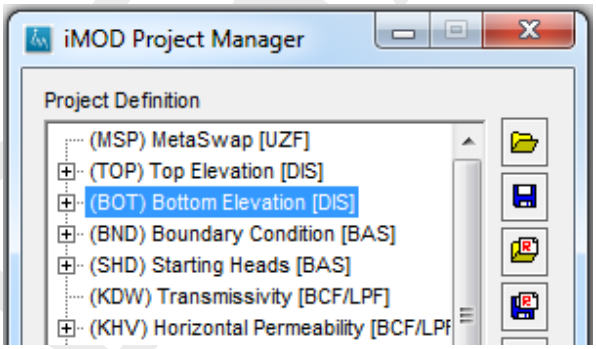


Figure 11.36: *Example of selecting a parameter in the 'Project Definition' window: in this example firsts '(BOT) Bottom Elevation' is selected to expand the tree view by clicking the '+'-sign.*

Whenever you select the expression under an expanded branch in the treeview in the *Project Definition* list, you'll be able to edit an existing entered parameter; see the example below.

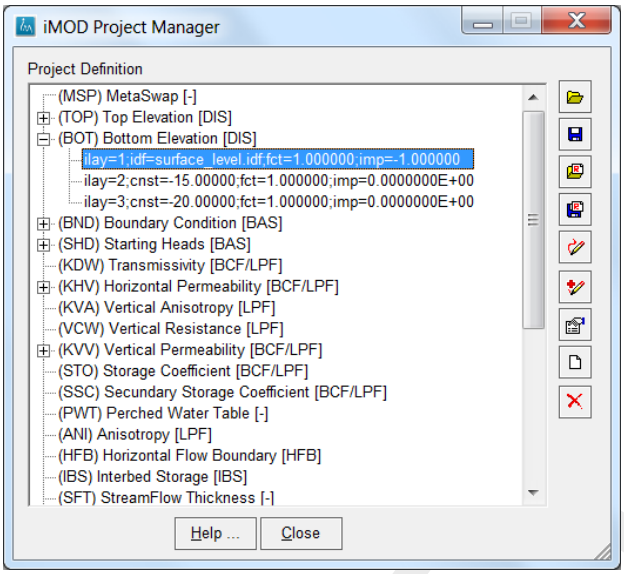


Figure 11.37: *In this example the exisiting (BOT) parameter set of layer 1 is selected. Click on the 'Properties' button to open the 'Define Characteristics for:' window to edit the Bottom Elevation parameters.*

As you may have noticed, we simulate this model with three model layers. The first model layer has a thickness of 1.0 meter (almost no horizontal flow in that model layer) to intercept the recharge. From there water will migrate to the deeper layers 2 and 3. The third model layer is the actual aquifer from which water is extracted via the well screen.

Schematic, the model can be represented by the following figure:

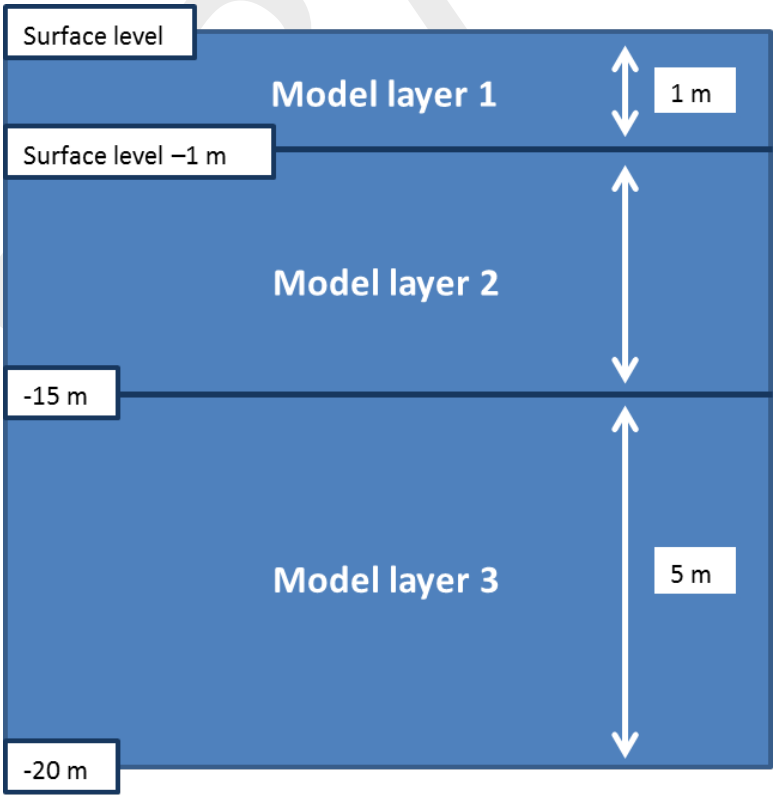


Figure 11.38: *Schematic representation of the model.*

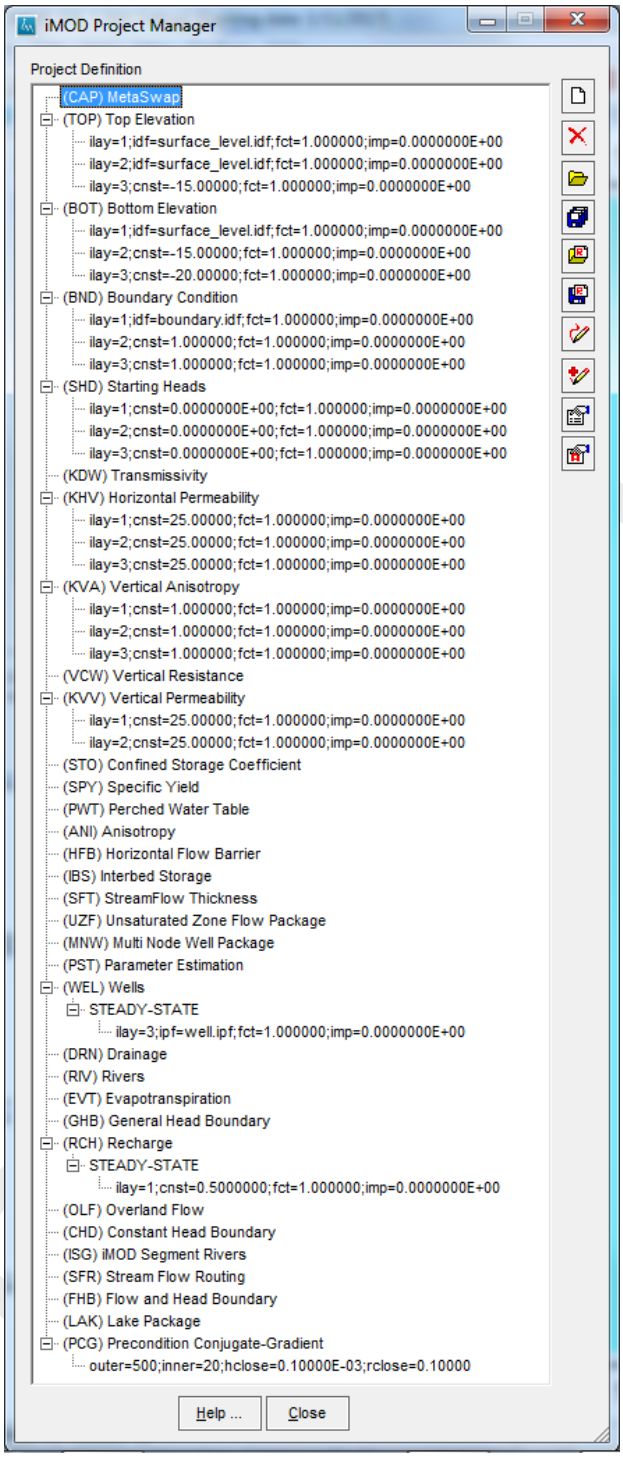


Figure 11.39: *Example of Project Manager window after filling in a model configuration.*

For the meaning and explanation of the available buttons on the Project Manager window go to section section 5.5.

61 Click the *Save* button () to save this model configuration in a PRJ file. This file may be loaded again whenever we need to modify this project.

The next step will be to create a runfile than can be used for a model simulation.

62 Click the *Save Runfile* button ( ) to start the *Define Simulation Configuration* window.

iMOD will fill this dialog depending on the definitions in the *Project Manager*. We are not able to create a transient runfile since we do not have any transient data. We will generate a runfile for a three-layered model.

63 Click on *Activate Packages...* to select all packages in de pop-up window if this is not already the case.
64 Click the *Apply* button to accept the selection and close the *Packages* window.

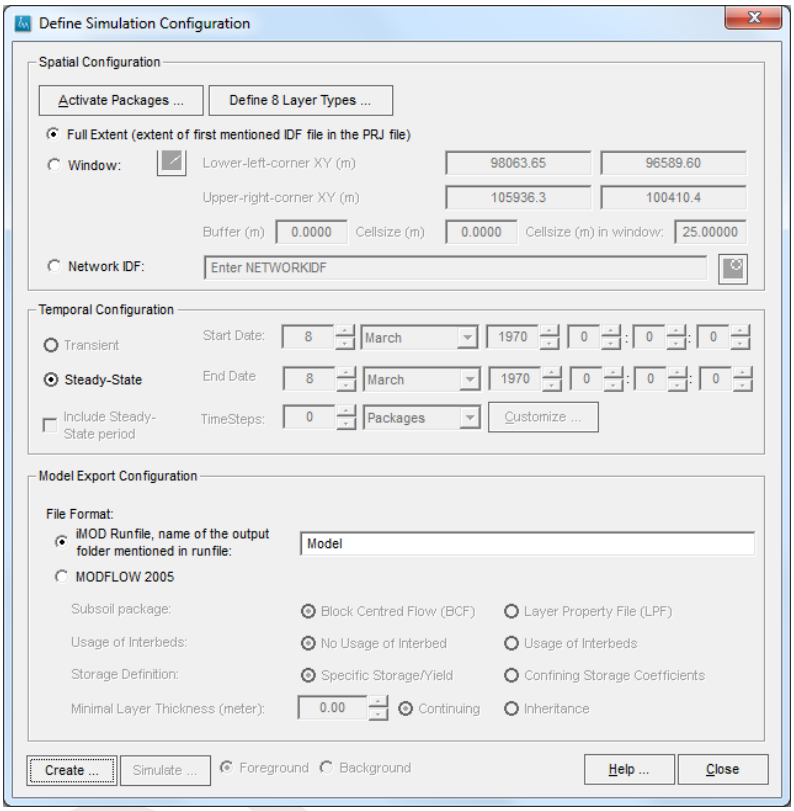


Figure 11.40: *The Define Simulation Configuration window after entering the value '3' for the 'Number of layers'.*

65 Click the *Create ...* button and save the runfile in the folder . \IMOD_USER \RUNFILES and call it ISLAND.RUN.

iMOD will create a runfile for a steady-state simulation taking into account all active packages in the *Project Manager*. This runfile can be used to start the model simulation.

66 Click the *OK* button that says that the runfile has been written successfully. This will return you to the *Project Manager*. Click the *Close* button to close the *Project Manager* window. It remains active in the background and can be re-opened by selecting the option *Project Manager* from the *View* menu at the main menu.

Running the Model

67 Select the option *Toolbox* from the main menu and then the option *Start Model Simulation* to start the *Model Simulation* window.

68 Select the [ISLAND.RUN] from the *Runfiles (*.run)* list.

iMOD will draw a hatched rectangle showing the maximum extent of the model described in the runfile. In section 11.6 we will demonstrate more functionalities in scaling and creating a submodel using the Model Simulation Tool. For now we will just skip most of the functionalities on this window and start running the model.

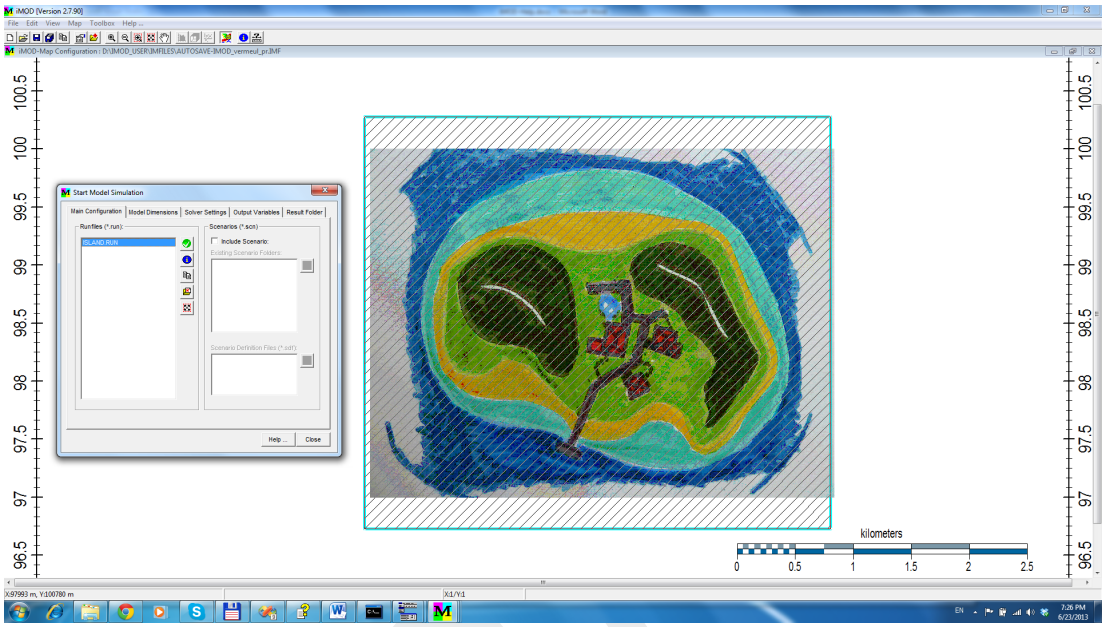


Figure 11.41: *Example of the 'Start Model Simulation' window.*

69 Select the *Result Folder* tab and enter the name [ISLANDQ500] in the *Enter or Select Output Folder* field.

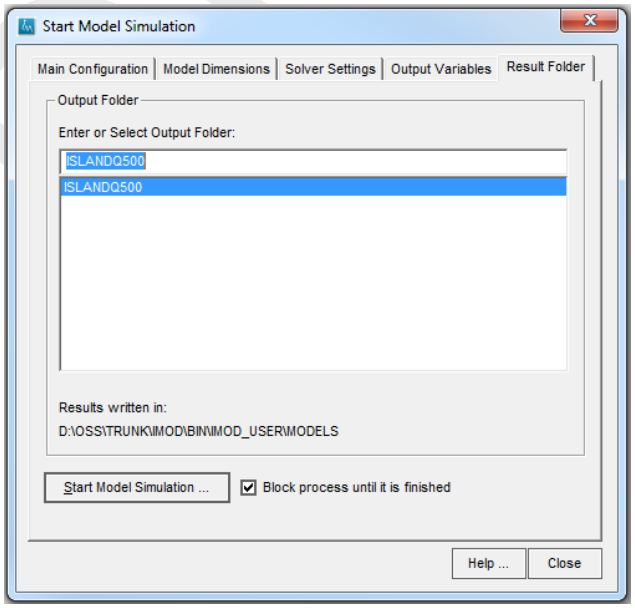


Figure 11.42: *Example of the 'Result Folder' tab in the 'Start Model Simulation' window.*

70 Click the *Start Model Simulation* button to start the model simulation.

The actual simulation will be carried out by the iMODFLOW executable and will run in the DOS box attached to iMOD. Please check whether you can find this window and examine the results, it will look more or like as follows:

iMOD will create folder \IMOD_USER \MODELS \ISLANDQ500 in which the results of the model simulation will be saved. Moreover, a complete copy of the runfile, the used executable for the simulator (e.g. iMOD__V4_3_X64R.exe) and a batch script will be saved too. Double clicking this batch script (RUN.BAT) from Windows Explorer or Total Commander will re-run this model outside iMOD. This can be very convenient whenever some trial-and-error computations have to be carried out. For now, we will work purely within iMOD.

iMODFLOW produces standard output in the file *iMODFLOW.list* (always in the subfolder mf2005_tmp of the Result Folder you specified (in this example . \IMOD_USER \MODELS \ISLANDQ500 \mf2005_tmp); at the end of this file the overall volumetric budget is printed and can be checked for the resulting water balance error (IN - OUT):

```
 VOLUMETRIC BUDGET FOR ENTIRE MODEL AT END OF TIME STEP   1 IN STRESS PERIOD    1
 ----------------------------------------------------------------------------

    CUMULATIVE VOLUMES      L**3       RATES FOR THIS TIME STEP      L**3/T
    ------------------                 ------------------------

          IN:                                IN:
          ---                                ---
            STORAGE =           0.0000            STORAGE =           0.0000
      CONSTANT HEAD =           0.0000      CONSTANT HEAD =       7.7523E-10
              WELLS =           0.0000              WELLS =           0.0000
           RECHARGE =           0.0000           RECHARGE =        4581.2500

           TOTAL IN =           0.0000           TOTAL IN =        4581.2500

         OUT:                               OUT:
         ----                               ----
            STORAGE =           0.0000            STORAGE =           0.0000
      CONSTANT HEAD =           0.0000      CONSTANT HEAD =        4081.3162
              WELLS =           0.0000              WELLS =         500.0000
           RECHARGE =           0.0000           RECHARGE =           0.0000

          TOTAL OUT =           0.0000           TOTAL OUT =        4581.3164

           IN - OUT =           0.0000           IN - OUT =      -6.6406E-02

 PERCENT DISCREPANCY =          0.00     PERCENT DISCREPANCY =           0.00
```

Figure 11.43: *Example of the volumetric water balance as printed by MODFLOW in the iMODFLOW.list-file.*

So, in short, the iMODFLOW standard output file *iMODFLOW.list* contains info on:

- ◇ the model discretization
- ◇ the model time and length units
- ◇ the processed input packages
- ◇ the solver used and how the iteration process progressed
- ◇ the volumetric budget for the entire model, including the percent discrepancy
- ◇ elapsed run time

Let's have a look at some more results.

71 Close the *Model Simulation* window by selecting the *Close* button.
72 Select the option *Map* and then the option *Quick Open* to start the *Quick Open* window, see section 6.2. With this window it is easy to open and view results from a model simulation.
73 Select the option [HEAD] from the *Topic* dropdown menu.
74 Select the options [1], [2] and [3] from the *Layer* dropdown menu.
75 Click the *Open* button.

iMOD will load the selected results files (HEAD for model layers 1, 2 and 3) into the *iMOD Manager* and displays the result on the graphical canvas. Use your experience learned from the previous Tutorials to display the computed heads as shown in the example on the next page. To show the IDF by contourlines, open the *Legend* window and click the *Contourlines* button ( ).

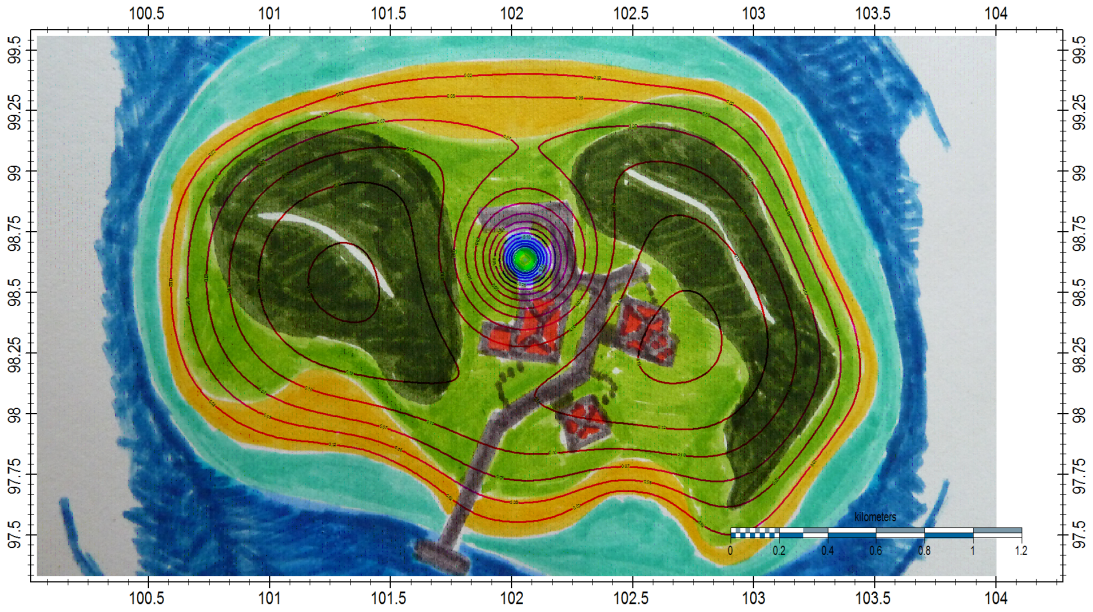


Figure 11.44: *Isolines of the computed hydraulic heads of the island.*

As we can see by the computed hydraulic head, the gradient towards the well in the centre of the island tends to be such that no water is extracted from the ocean. To illustrate this even more we can compute pathlines that show the actual path through the subsoil that groundwater follows from the location of infiltration towards the location of extraction. We call that particle tracking.

Create Startpoints for a Pathline Simulation

76 Select the option *Toolbox* from the main menu and then the option *Define Startpoints* to open the *Open/Create a Startpoint Definition* window, see section 7.13 for more detailed information.
77 Enter the name [ISLAND] in the input field.
78 Click the *Open and Continue* button to open the *Start Point Definition* window.

iMOD offers the possibility to define startpoints for any particle tracking independently of a model, modelsize and or cellsize. Startpoints will be defined by means of a polygon a line and/or points and startpoints are distributed within the limits of that/those polygon(s)/lines. So, let us define startpoints on the island.

79 Click the *Draw* button ( ) and start drawing a polygon. We've done this before (see step 8) so you'll manage to get this done. Make a polygon wide around the island, so we can observe whether seawater is flowing to the well too.
80 Select the right mouse button to stop drawing a polygon.
81 Select the *Definition* tab and enter a cellsize of [50] meter for the *Distance X (m)* and *Distance Y (m)*, so each 50 meter we will have a particle starting.
82 Select the *Open IDF* button ( ) to select the computed hydraulic head of layer 1 to be used as *Top Level*, use the same IDF for the *Bottom Level*.

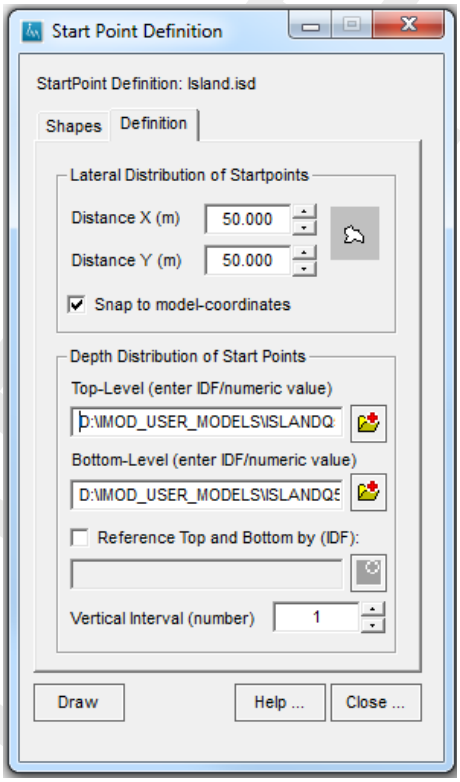


Figure 11.45: *The 'Start Points Definition' window.*

83 Click the *Draw* button to get an idea of the density of the particles to be started.
84 Click the *Close* button to close this *Start Point Definition* window. Whenever iMOD asks to overwrite the current [ISLAND.ISD], do so.

So now we're finished creating startpoints, let's use them in the pathline simulator.

Running the Pathline Simulator

85 Select the option *Toolbox* from the main menu and then the option *Start Pathline Simulation*. This will start the *Pathlines Simulation* window; see section 7.14 for more detailed information.

86 Select the model [ISLANDQ500] in the list at *Existing Results under Models*.

iMOD will search in the appropriate folders to see whether all necessary files are available. For a particle tracking you need at least the budget terms in the x, y and z direction, these files are stored in the . \BDGFRF, . \BDGFFF and . \BDGFLF, respectively. iMOD will display the availability of those files whenever you select a model result.

The example shown here will only highlight the most important steps to perform the particle simulation; however, it is difficult to explain the results whenever one cannot fully understand the technique behind it. So please, read some documentation on particle tracking, done by D.W. Pollock (USGS OpenFile Report 94-464).

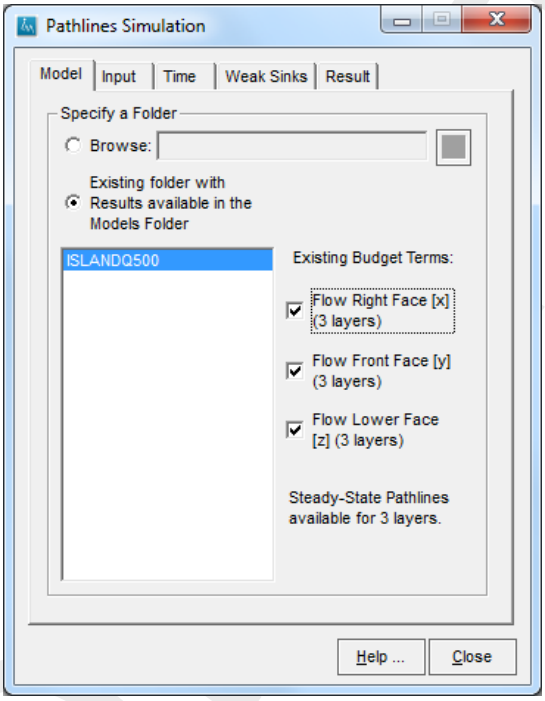


Figure 11.46: *The 'Pathline Simulation' window.*

87 Click the *Input* tab on the *Pathlines Simulation* window.

On this tab we need to tell iMOD the specific information that is needed for the particle simulation. Most important are the top- and bottom interfaces for the two model layers, see section 7.14 for more detailed information on this topic. Let's fill it in quickly.

88 Click the *Properties* button ( ) next to the *Boundary Settings* to display the *Input Properties* window.

89 Enter the [. \DBASE \BOUNDARY.IDF] in the first row and enter [1] for the second and third row. The particle tracking algorithm will use this information to exclude areas where the values in the IDF-files are less or equal to zero.

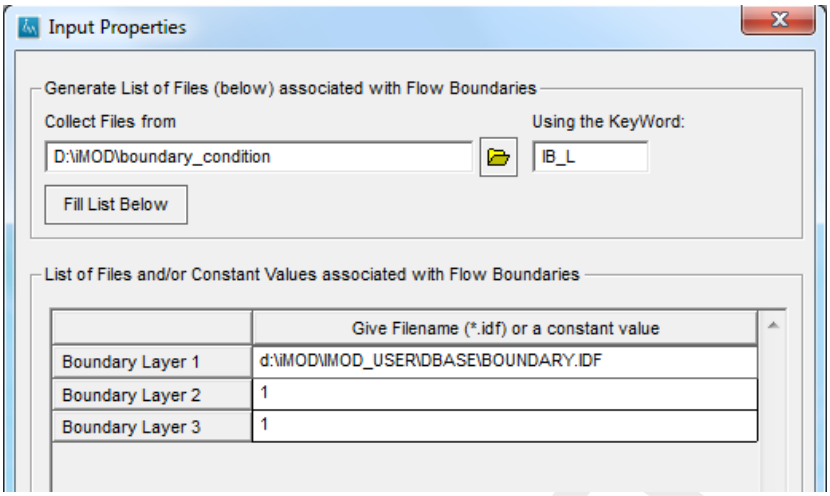


Figure 11.47: *The 'Input Properties' window for the Boundary Conditions.*

90 Click the *Apply* button to use the entered values as input.

91 Click the *Properties* button ( ) next to the *Top- and bottom Files* to display the *Input Properties* window.

92 Specify the tops and bottoms according to the table on page 635; the specification for the top- and bottom files could look similar to the figure below. Please note that for the bottom of layer 1 (and the top of layer 2) we use an IDF which has an elevation 1 meter below the surface level IDF; you can create such an IDF e.g. by selecting the DBASE\SURFACE_LEVEL.IDF in the iMOD manager, and click on the *IDF Calculator* ( ) button, type the Formulae 'C=-1+A' and assign e.g. the name DBASE\SURFACE_LEVEL_MINUS_ONE.IDF to Map C.

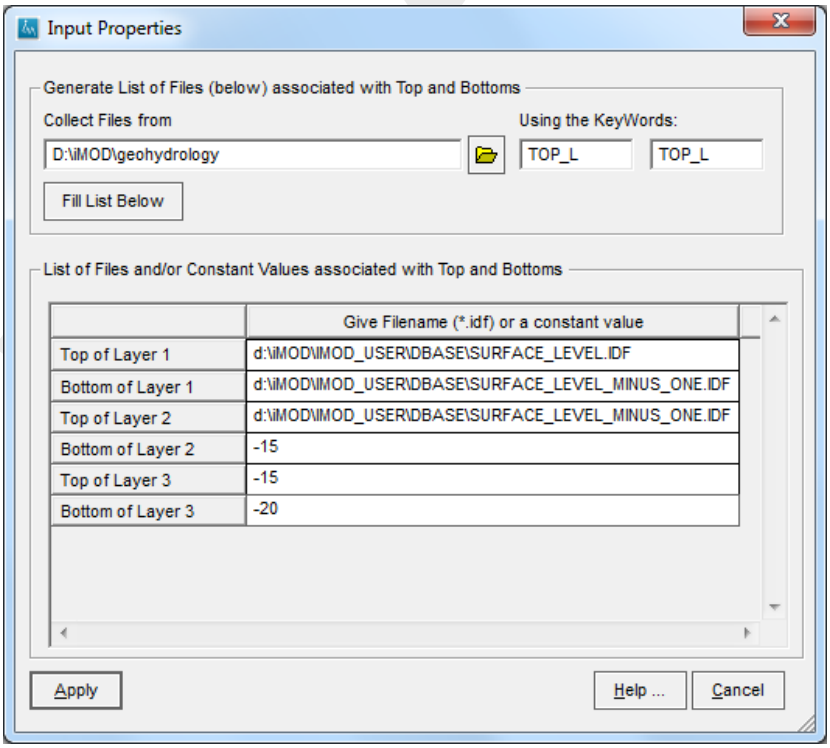


Figure 11.48: *The 'Input Properties' window for the Top- and bottom Files.*

93 Click the *Apply* button to use the entered values; iMOD will check the existence of the specified files.

94 Select the *Properties* button for the other input fields and fill in the *Input Properties* appropriately. Use the configuration as mentioned in the table on page 635 and adjust the following 2 parameters.

Table 11.3: *Adjust the following 2 parameters*

Parameter	model layer	IDF/Constant Value
Porosity Aquifer	1,2,3	0.3
Porosity Aquitard	1,2,3	0.1

95 Click the *Save As* button () to save the input properties to an IPS file. Next time you can use the Load button () to read the input properties from disk. Whenever you save the settings as [. \IMOD_USER \SETTINGS \IMODPATH.IPS], iMOD will read this file automatically each time you start the *Particle Simulation* window.

For now we will skip most of the configuration setting in the other tabs, but feel free to have a look in more detail at section 7.14.

96 Click the *Result* tab and make sure the *Trace Direction* is [Forward]. We compute the particles from the groundwater elevation upto the discharge location (a well and/or the ocean).
97 Select the option *Save Entire Flowpath (*.IFF)* so we will get a file that describes the entire flowpath of each particle.
98 Enter a name for the yielding flowfile, e.g. [ISLAND.IFF].
99 Click the *Start* button. This can take a while, especially whenever you have a lot of particles.

Bear in mind that whenever you have a lot of particles to examine, but you're not actually interested in their paths but only in their age at interception, consider the IPF files as alternative to flowlines. Those files are much-much smaller and can be examined quicker.

100 Click the *OK* button on the *Information* summary that will be displayed after the particle tracking is completed.

iMOD will load the computed IFF file and presents it like black lines. So let's color it by their age, which makes more sense.

101 Click the *Map* option from the main menu, select the option *IFF Options* and then the option *IFF Configure* to display the *IFF Configure* window. See section 6.9.1 for more detailed information about the functionalities on this window.
102 Select the option *Apply Legend to* and select the [TIME (YEARS)] from the dropdown menu.

One of the other items to be plotted is the velocity, that is computed as the flux (m^3/day) divided by the porosity (-) divided by the area (m^2; width*model layer thickness). A change in porosity will change the velocity linearly and therefore the age of the flowline, but will, however, not affect the shape of the pathline.

103 Click the *Close* button to redraw the IFF with the assigned adjustments.

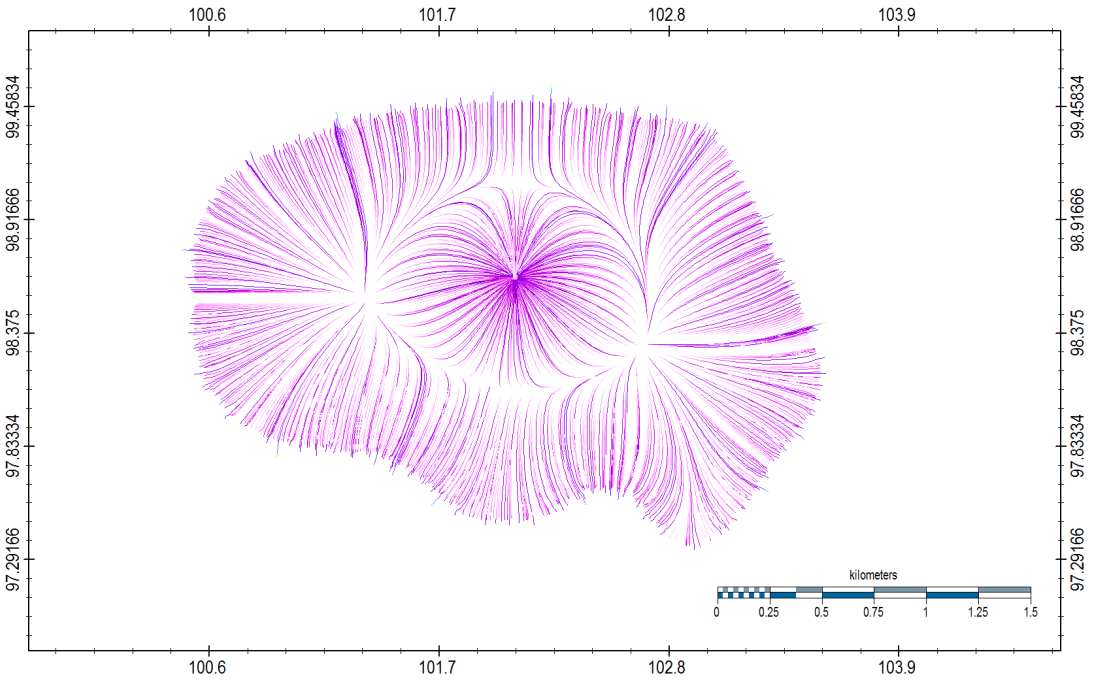


Figure 11.49: *Example of a two-dimensional image of pathlines.*

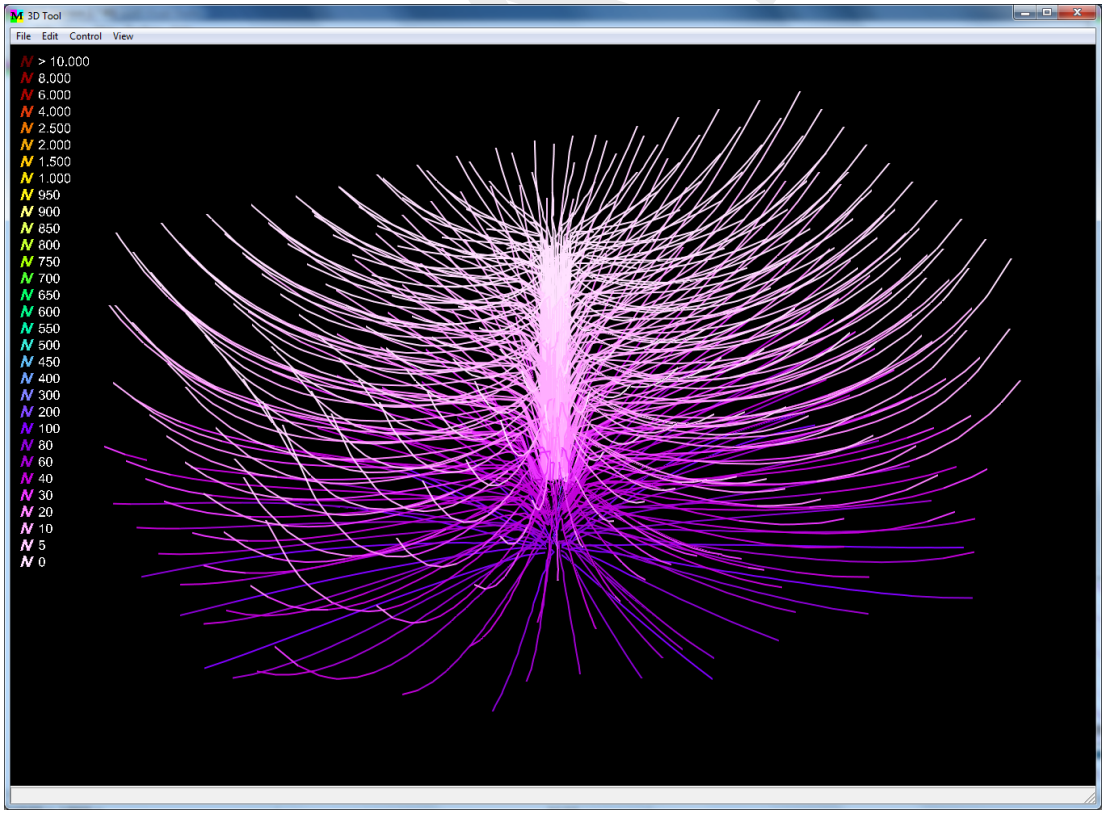


Figure 11.50: *Example of a three-dimensional image of pathlines near the well.*

Please note that the vertical scale is always very much exaggerated! From the 3D image above, it is clear that most of the water penetrates vertically to the deeper subsoil and then flows to the well. Given that the material is highly permeable (25m/day) and homogeneous. In the next section 11.5 we will

enhance this model to include more resistance in the subsoil which will affect the pathline behaviour.

Finally, our major question is still unanswered.

Sensitivities and sustainable yield

So, now we know that an extraction of 500m^3/day will be sustainable, we're still wondering what the maximum will be. It will be your task as hydraulic engineer to give an answer to that question by simulating a variety of model simulations for different extraction rates for the well. Use the next figure to monitor the behavior of the system for different rates. Moreover, you could vary the permeability too, since this parameter is often uncertain. It will be interesting to illustrate the accuracy of your sustainable yield estimation with a bandwidth that expresses the inaccuracy of the permeability too.

As stated above, after each model simulation you should check the total water balance of the model in the iMODFLOW.list file located in the model directory . \MODELS \ISLANDQ500 \mf2005_tmp), it shows the total summary of the model simulation. If you scroll down, you'll see the total water balance for the model.

In the example above shows the quantity of water flowing in from the sea is close to zero while the amount of water flowing out to the sea is 4081m^3/day. You could use these as evaluation criteria as well!

11.5 Tutorial 5: Solid Tool

This tutorial gives a short introduction in enhancing the groundwater flow model from section 11.4 with an aquitard that has been characterized by several boreholes. See for a more detailed description of the Solid Tool section 7.4.

Outline

This is what you will do:

- ◇ Visualizing the boreholes in 3D;
- ◇ Enhancing the subsoil characteristics based on the boreholes using the Solid Tool;
- ◇ Simulate the updated model to observe the consequences of an aquitard in-between two aquifers;
- ◇ Simulate flow of particles.

Required Data

For this tutorial you need the following iMOD Data Folders:

- ◇ SURFACE_LEVEL.IDF: describes the surface level of the model area;
- ◇ BOUNDARY.IDF: describes the boundary conditions;
- ◇ WELL.IPF: ipf file that describes the location and rate of the wells;
- ◇ ISLAND.PRJ: project file that describes the model configuration;
- ◇ BOREHOLES.IPF: ipf file that describes the location and actual borehole data.
- ◇ BEDROCK.IDF: idf file that describes the elevation of the bedrock layer.

All these files are located in/below the folder: {path of installfolder} \tutorials \TUT_ SOLID_BUILDING.

Beside this data you will need the iMODFLOW executable to make the model computations, see step 1 in section 11.4.

Getting Started

1 Launch iMOD by double click on the iMOD executable in the Windows Explorer, and start by selecting the option *Create a new iMOD Project*.

2 Open the SURFACE_LEVEL.IDF from by clicking the *Open IDF* () from the main menu. The file is located in . \TUT_SOLID_BUILDING \.

This file shows the surface level of a small island that we've been modeling in section 11.4. If you want you can add a sketch of the outline of the island by following the steps 3 upto 6 from section 11.4.

Now we have an IDF of the uppermost interface of our model (the actual surface-level), we need to have an IDF for our lowermost interface as well (to start with; this can be modified later). So, we will copy the SURFACE_LEVEL.IDF and assign a default value of 20m-MSL and call it BEDROCK.IDF. We have done that for you, so please open this file in iMOD.

3 Open the BEDROCK.IDF from by clicking the *Open IDF* () from the main menu. The file is located in . \TUT_SOLID_BUILDING \.

These two files describe the vertical and horizontal limits of our model. In-between there exists an aquitard that has been identified by several boreholes, when they we're installing the well.

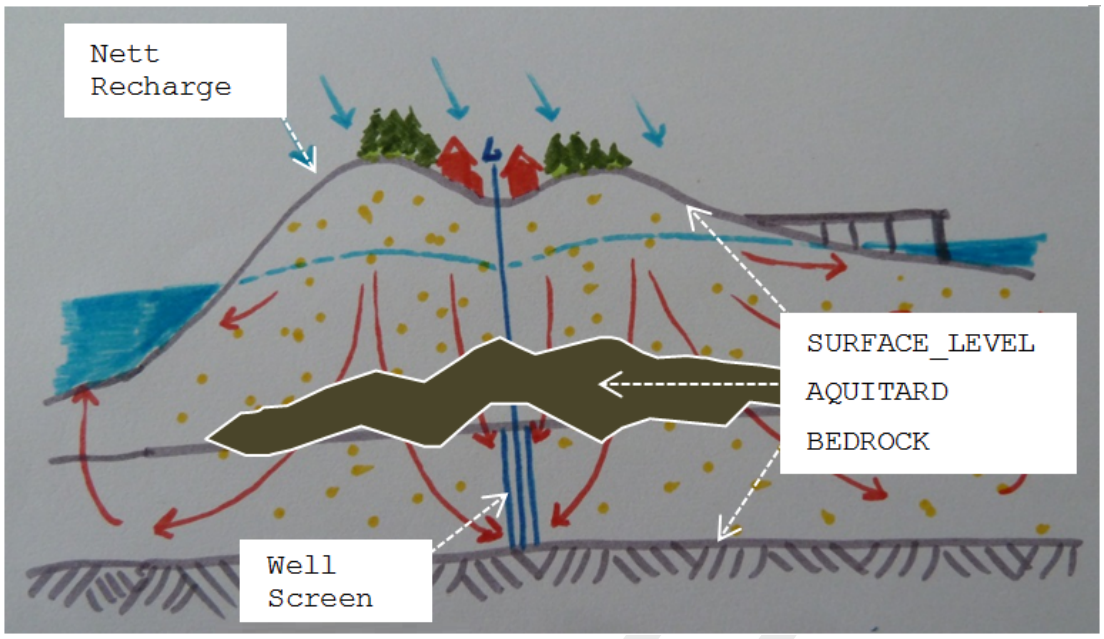


Figure 11.51: *Sketch of a flow pattern that might occur in our island model.*

Boreholes

When they drilled the well in the early 80's, the borehole company found some clays with low permeable material (less than 0.0001 m/day). Probably these are some ancient deposits, but they can interfere with the flowpath to the well screen and therefore might decrease the level of sustainability of the well. To this end, they decided to collect more information about the extent of this clay layer by drilling additional boreholes. A total of 5 boreholes were drilled, let's start by loading these in iMOD.

4 Click the *Open IDF* button ( ) and select the file BOREHOLES.IPF from the . \TUT_ SOLID_BUILDING folder.

The syntax of the file BOREHOLES.IPF is as described in more detail in section 9.7. For now, each location of the boreholes has an x- and y coordinate and a reference to an attached textfile that describes the actual borelog. Let's see how the subsoil should look like whenever we include the boreholes.

5 Use your experience from the previous tutorials to produce the following figure, see steps 46 and further from the first tutorial whenever you need some assistance in this. Note: If not all labels are visible directly, go to the *Clipplanes* tab and select one direction a time until all labels are visible.

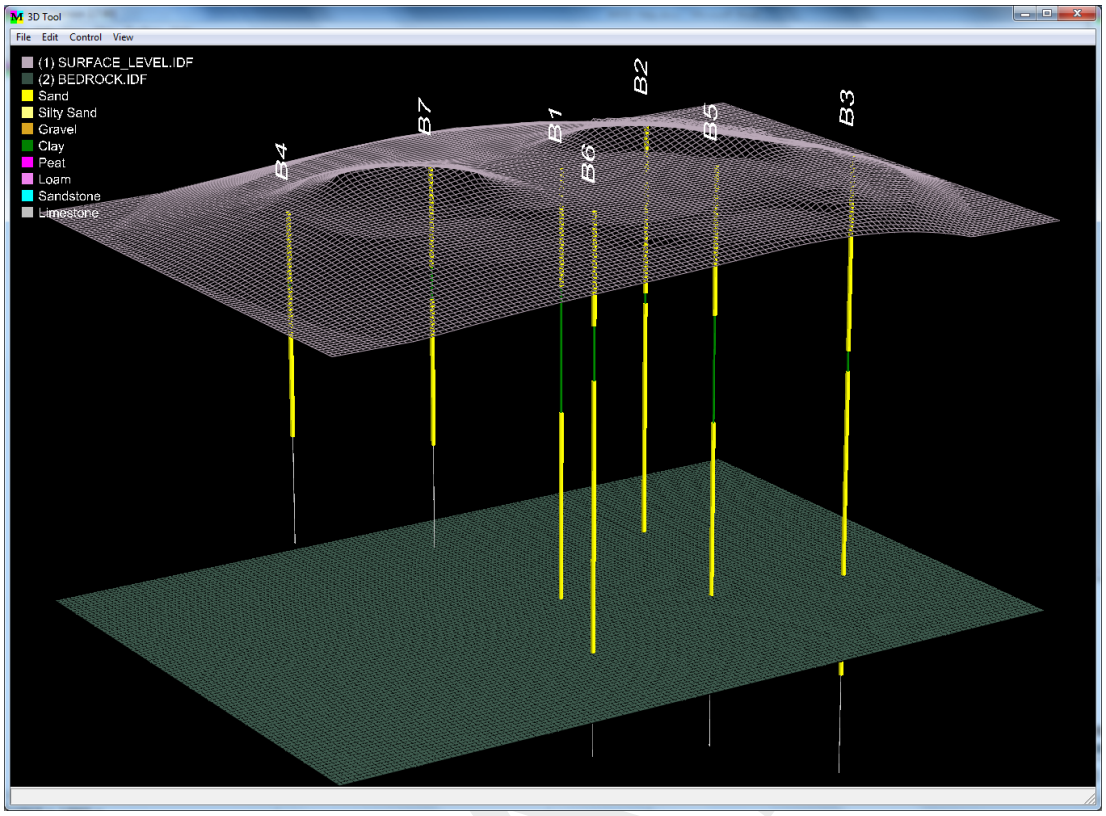


Figure 11.52: *Example of a 3D-image of boreholes of the hypothetical island.*

From the above given figure, it seems that our estimation of the bedrock depth is not accurate as the boreholes show an increasing bedrock depth from the west to the east. Moreover, we can clearly observe a clay layer (green) in-between the aquifer (yellow). This clay layer has its greatest thickness in the centre of the island (off course) and thins out to the side of the island. Probably eroded by some ancient seas. We are going to use the *Solid Tool* to construct the interfaces that describe the top- and bottom elevation of the aquitard, as well as adapting the bedrock level of the limestone.

Create a Solid

A solid is a representation of the subsoil divided into separate interfaces, such as clay and other lower- or higher permeabilities. It contains a set of continuous interfaces that exist throughout the model domain and can be used in a groundwater flow model.

6 Click the *Toolbox* option from the main menu and then select the *Solid Tool* to start the *Solid Tool* window.
7 Click the *New Solid* button ( ) to start the *Create New Solid* window.
8 Select *Enter single TOP and BOTTOM of SOLID*.
9 Select the IDF-file [SURFACE_LEVEL.IDF] from the list mentioned by *TOP-level* and the IDF-file [BEDROCK.IDF] from the list mentioned by *BOT-level*.
10 Enter [ISLAND] in the *Give the name for the Solid* input field.

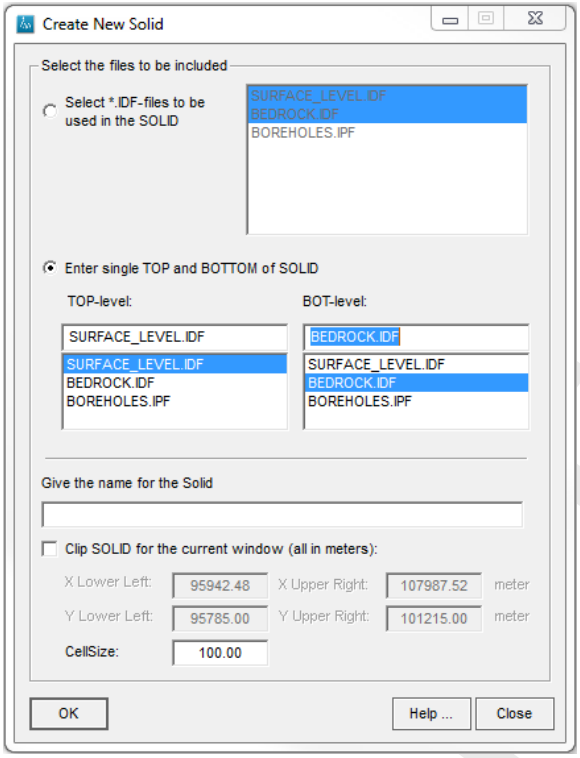


Figure 11.53: *The 'Create New Solid' window.*

iMOD will use the selected IDF-files (SURFACE_LEVEL.IDF and BEDROCK.IDF) as the uppermost and lowermost interfaces of our model. It is important that the files are selected in the right sequence (from the top to the bottom). iMOD can add extra interfaces whenever you specify that, so based on the two selected IDF-files iMOD can create extra interfaces in-between. In our modeling project, we need at least two model layers, one to describe the groundwater head above the aquitard, and one to describe the situation underneath.

11 Click the *OK* button
12 Enter [4] for the *Number of interfaces* .
13 Click the *OK* button
14 Click the *OK* button on the *Information* window that mentions that the solid has been created successfully. This returns you to the *Solid Tool* window again.
15 Click the *Information* button ( ) to display the content of the *.SOL file. This file describes the files to be used for the solid.

As you can see, the names for the top and bottom interfaces are changed into INT_L1.IDF and INT_L4.IDF. These are copied from the SURFACE_LEVEL.IDF and BEDROCK.IDF, respectively. The other interfaces INT_L2.IDF and INT_L3.IDF are interfaces that iMOD created and are by default the mids in-between the surface (L1) and the bedrock level (L4). All these files are located in the folder . \IMOD_USER \SOLIDS \ISLAND.

16 Close the Texteditor as this will return you to the *Solid Tool* window.
17 Select the [ISLAND] in the list on the *Solid Tool* window, if this is not selected yet. Click on *Feed Selected SOL-file to the iMOD Manager*.
18 Open the *iMOD Manager* and observe that there is a file called [ISLAND.MDF]. An MDF-file is a collection of IDF-files into a single file, see section 6.5 for more information about these MDF-files, how to create them and how to dissolve them again.
19 Select the BOREHOLES.IPF and the ISLAND.MDF in the *iMOD Manager*.
20 Click the *3D Tool* button (on the *Solid Tool* window!) to start the 3D Tool (see for more information section 7.3 and step 46 in section 11.1).

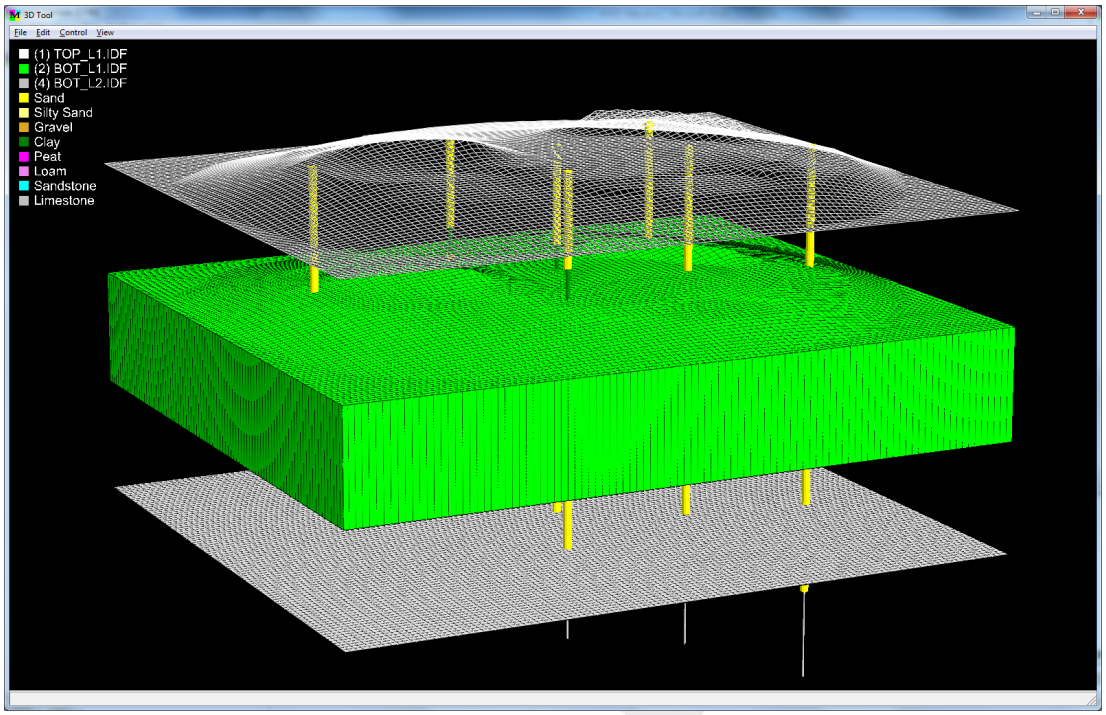


Figure 11.54: *Example of the initial Solid.*

You can see how the current solid looks like, it contains a single aquifer on top of the (green) aquitard, and another aquifer beneath it. It is not very accurate though, compared to the boreholes. We are going to manipulate the green aquitard such that is resembles the boreholes more realistic.

21 Choose the option *File* from the main menu on the *3D Tool* window and then select the *Quit 3D Tool* option, this returns you to the main screen again.

22 Click the *Cross-Section* button ( ) on the *Solid Tool* window (again **not** the one on the main menu!). This will start the *Cross-Section Tool* as you experienced in section 11.3, step 7. Read section 7.1 for more information about this tool.

23 Enlarge the *Draw Cross-Section* window such that you can see the boreholes more clearly.

24 Select the *Snap* option ( ) on the *Draw Cross-Section* window at the *Location* tab (*caution:* do **not** select the same option on the *Cross-Sections* tab!).

25 Click the *Draw Cross-section* button ( ) on the *Draw Cross-Section* window and start drawing a cross-section in the *Draw Cross-Section* window, start from the left borehole B7 via B1, B5 to B3. Click your right mouse button to stop this drawing.

26 Select the tab *Cross-Sections* on the *Draw Cross-Section* window and click the *New Cross-section* button ( ) on the *Cross-Section* window to start the *Fit Interfaces* window.

Here, you can enter a name for the cross-section. We suggest that you enter the name [CROSSB7B1B5B3], so it will be clear, in future, what cross-section this is.

Furthermore, this window offers the possibility to start your initial guess for the cross-section using the current values for those interfaces.

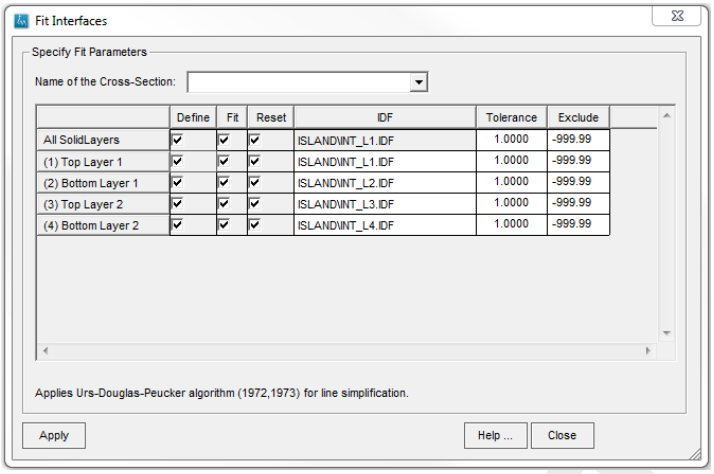


Figure 11.55: *The 'Fit Interfaces' window.*

iMOD will fit the interface along the cross-section on the values read from the appropriate IDF, so in this case iMOD will create a line for the [Top Layer 2] (third row in the table) on the content of the ISLAND \INT_L3.IDF. The accuracy of this fit is determined by the *Tolerance*, which is set to [1.0] meter, which is rather high for this case; however, it is fine for now. Feel free to change the tolerance values to see the impact.

27 Click the *Apply* button.

iMOD will fit each line to the corresponding IDF-files, the result is presented below.

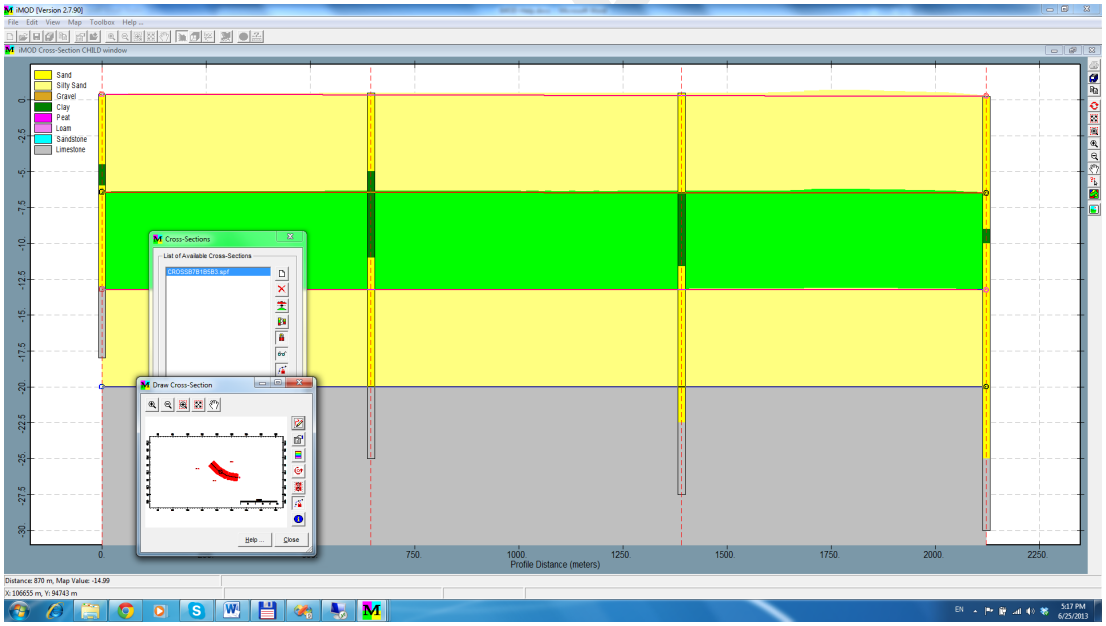


Figure 11.56: *Result of the initial guess for the cross-section based on the values entered in the previous 'Fit Interfaces' window.*

Now we can do two things:

- We can manually edit the line such that it will fit the boreholes. You can easily move your cursor in the neighborhood of the (red) line. Whenever it changes in a red arrow you can click the left mouse button and drag the line to another position. Whenever it becomes a black arrow you can modify the existing node of the line. This behavior is similar to modifying polygons, see section 4.4 for an example.

◇ We can tell iMOD to connect the lines through the boreholes. We will start with this.

Note: iMOD will connect the interfaces through all boreholes in the cross-section. Bear in mind that boreholes might be projected on the cross-section over an undesired distance. To avoid that, decrease the *Fade, view depth* on the *Misc.* tab on *Cross-Section Properties* window.

28 Click the *Fit* button ( ) to adjust the nodes on each line such that the line crosses each borehole at the right position.

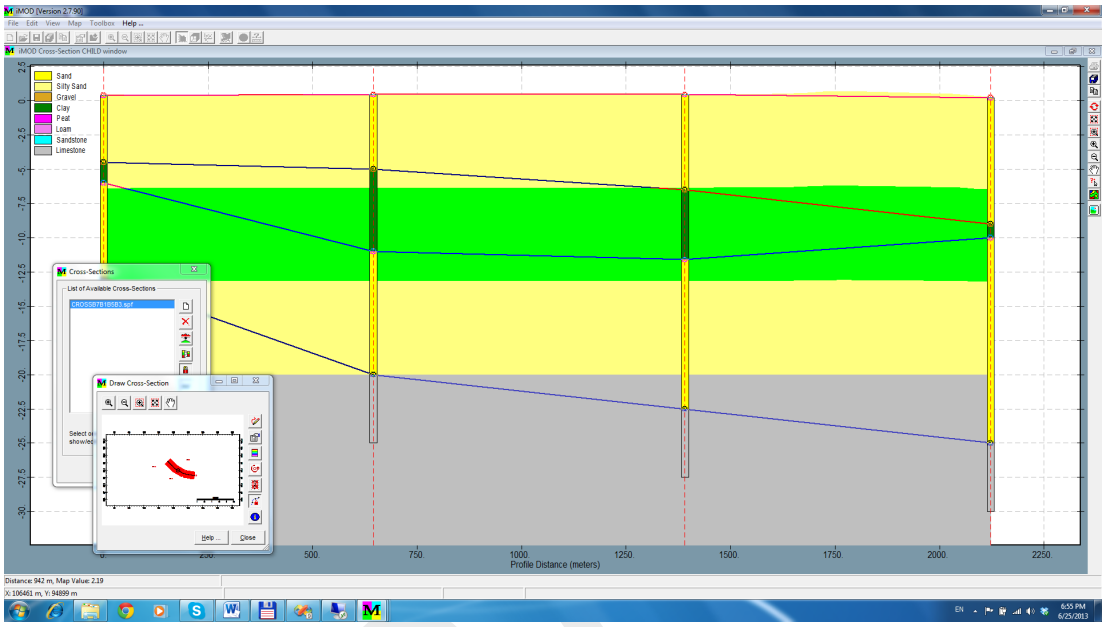


Figure 11.57: *Result of adjusting the nodes on each line such that the line crosses each borehole at the right position using the 'Fit' button.*

For now we will accept this cross-section.

Okay, let us define the other cross-sections. Follow the steps 22 upto 26 for the different cross-sections. Simply press the *New Cross-Section* button ( ) again to start another cross-section. We suggest you draw the following cross-sections (you're free to draw other combinations as well):

[CROSSB7B1B5B3]: B7-B1-B5-B3 (you just did this one!)
[CROSSB6B1B2]: B6-B1-B2
[CROSSB4B7B2B3]: B4-B7-B2-B3
[CROSSB4B6B3]: B4-B6-B3

29 Click the *Close* button on the *Draw Cross-Section* window. You'll be asked to save the current cross-sections, click *Yes*.

iMOD will save the current cross-sections into separate files, e.g. called CROSSB7B1B5B3.SPF in the . \IMOD_USER \SOLIDS \ISLAND folder. Also the ISLAND.SOL will be adjusted such that it includes a reference to this CROSSB7B1B5B3.SPF. Please have a look in the ISLAND.SOL by pressing the *Information* button ( ).

Your result might look like the following example.

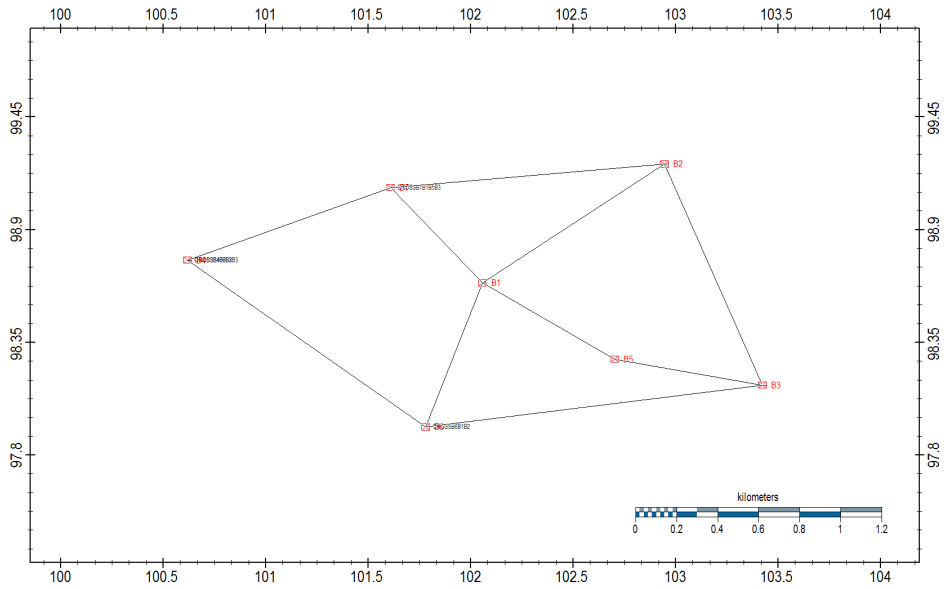


Figure 11.58: *Example of the outline of the cross-sections.*

Bear in mind that the area outside the cross-sections will be extrapolated from the cross-sections. You're allowed to define other cross-sections in those areas too, to direct the interpolation more.

30 Select the [BOREHOLE.IPF] in the *iMOD Manager* solely.

31 Click the *3D Tool* button ( ) on the *Solid Tool* window.

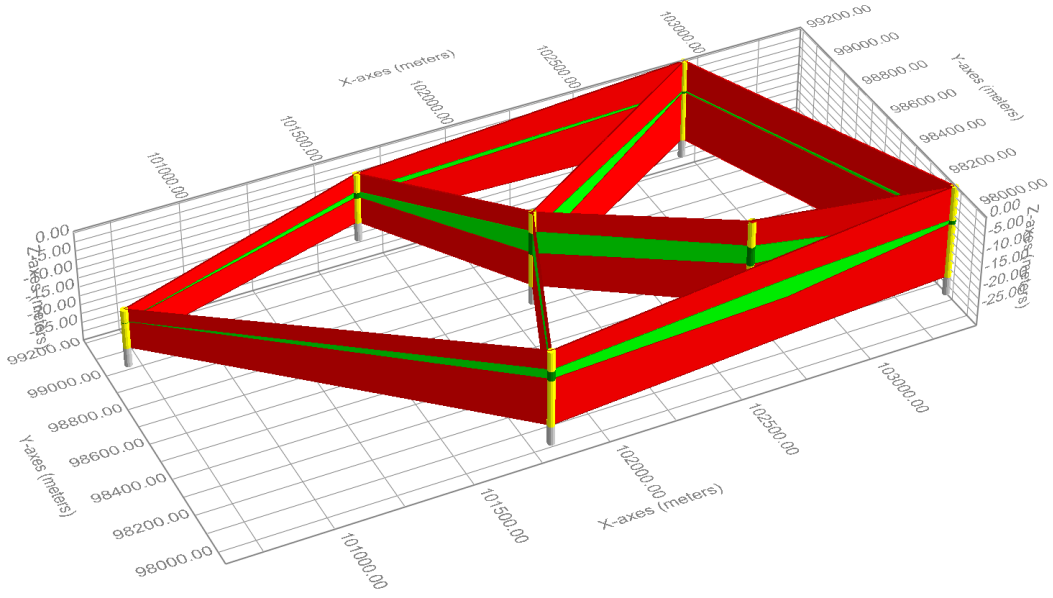


Figure 11.59: *Example of a 3D image of the outline of the cross-sections.*

There is a *Solid* tab active now. On that tab you'll find a list of all the cross-sections, you can select them all or select them individually.

32 Select the cross-section CROSSB7B1B5B3 from the list on the *Solid* tab in the *3D Plot Settings*

window.

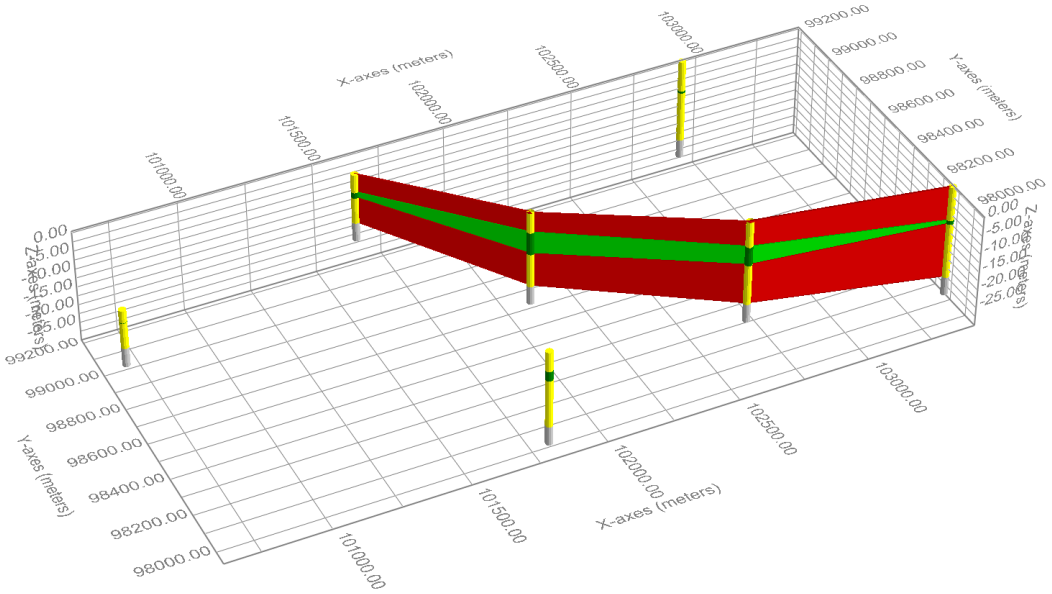


Figure 11.60: *3D image of the individual cross-section [CROSSB7B1B5B3].*

33 Click the *File* option from the *3D Tool* menu and then select the option*Quit 3D Tool* to return to the *Solid Tool* window.

Our next step is to create a fully 3D interpretation of the interfaces by numerical interpolation. The interpolation is based on the cross-sections.

34 Click the *Calculate* button ( ) on the *Solid Tool* window to start the *Compute Interfaces* window.

In this window you'll be allowed to determine what elevations/interfaces need to be computed. Since the top elevation for our first model layer is the SURFACE_LEVEL.IDF (see step 2) we will not recompute that interface, so we turn it off.

35 Deselect the *Calc* option for [(1) Interface 1].

We will overwrite our initial elevations/interfaces since that will increase the performance of our next interpolations. Moreover, we will be able to see any update of our interfaces more easily.

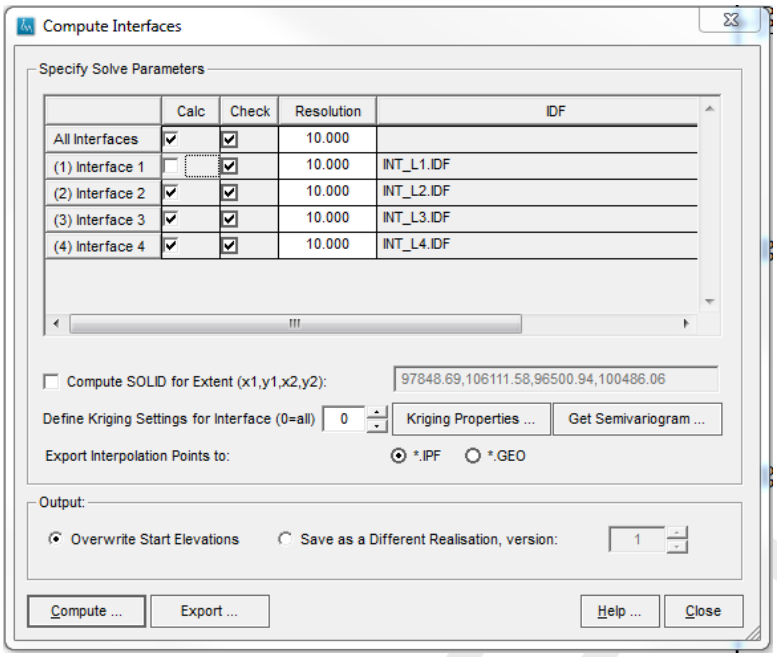


Figure 11.61: *Example of the 'Compute Interfaces' window.*

iMOD uses as default Kriging interpolation. This is far-out the best suitable interpolation method for these interfaces.

36 Click the *Kriging Properties* button to display the settings for the Kriging interpolation, see for more details on the properties on this window Section ...

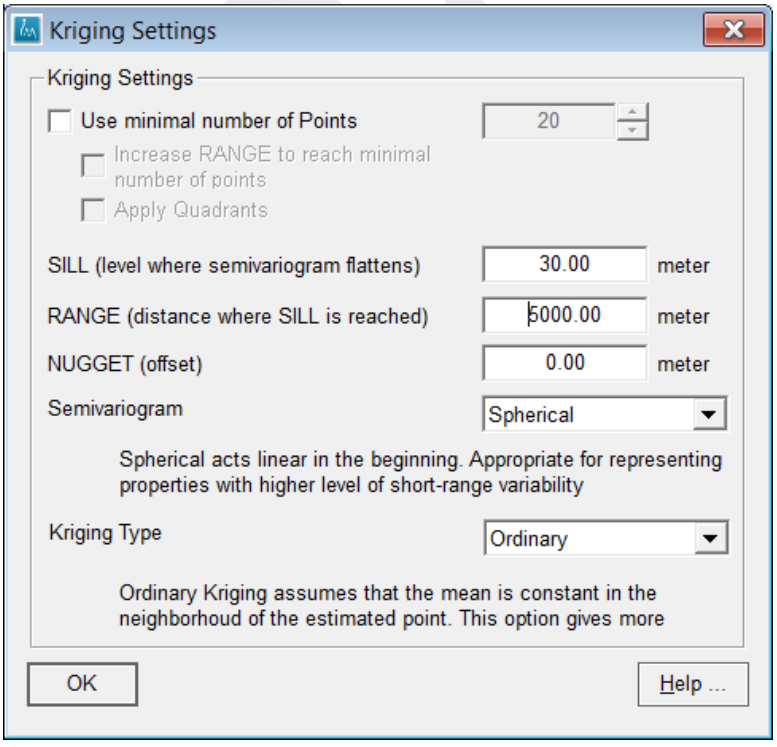


Figure 11.62: *Example of the used Kriging Settings.*

37 Click the *OK* button to accept the default settings of the Kriging parameters.
38 Click the *Compute* button to start the interpolation process.

39 Click the *OK* button whenever the interpolation has been finished.
40 Select ISLAND in the *Solid Tool* and click the *Cross-Section Tool* button on the *Solid Tool* window and select the cross-section [CROSSB7B1B5B3.SPF] from the*List of Available Cross-sections* on the *Cross-Sections* window.

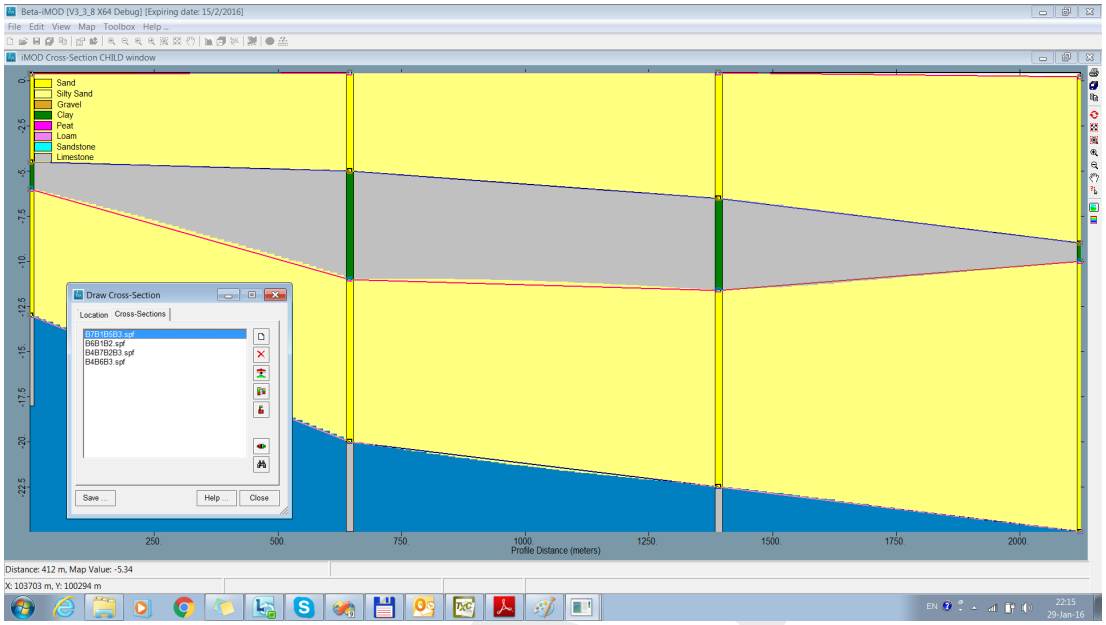


Figure 11.63: *Example of the cross-section CROSSB7B1B5B3 after interpolation.*

Pretty cool, but also a bit unrealistic. We can modify each cross-section easily to become more smooth.

41 Use your mouse cursor to move in the neighbourhood of the a line to be modified and whenever it becomes red, just press your left mouse button and drag the line. Try to create some detail, or even try to create a hole inside the aquitard.

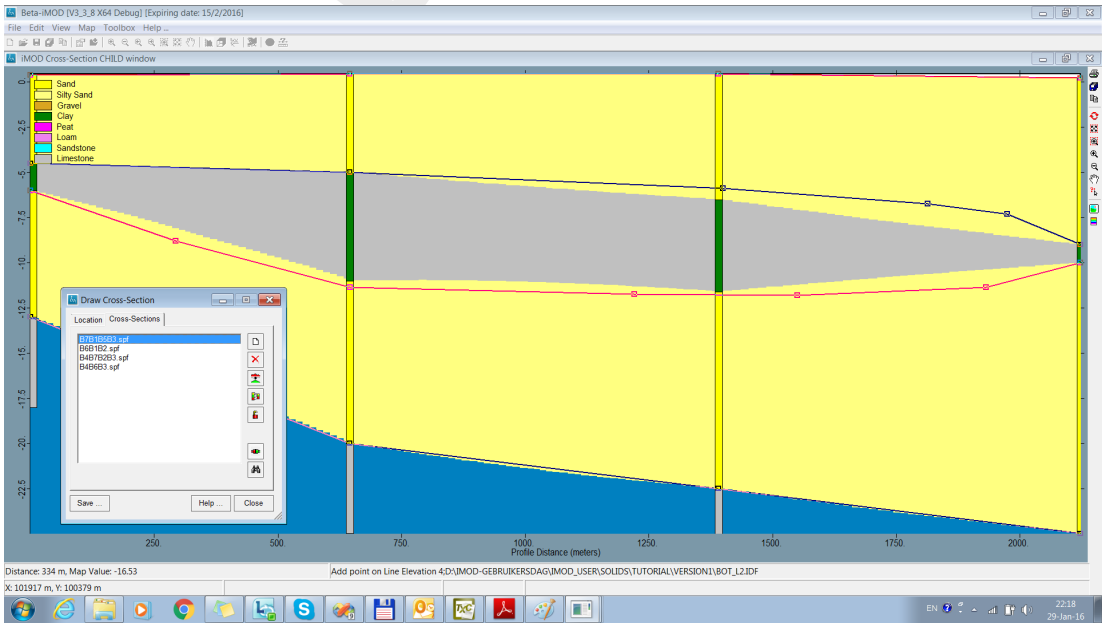


Figure 11.64: *Editing the interfaces of cross-section CROSSB7B1B3B5.*

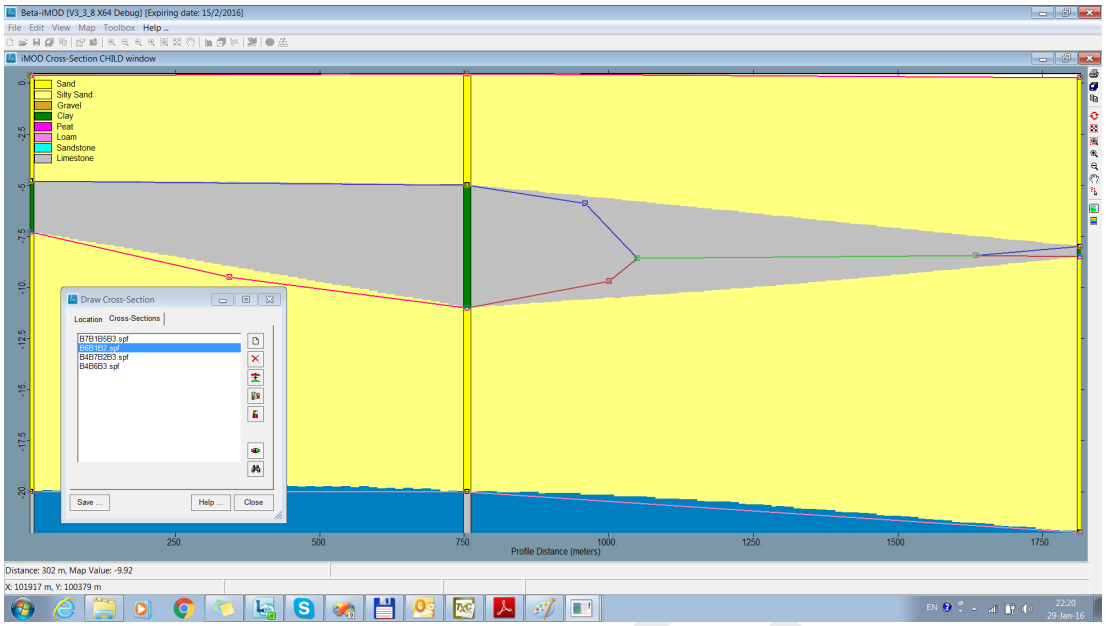


Figure 11.65: *Editing the interfaces of cross-section CROSSB6B1B2.*

Be aware of the fact that whenever you move a node into the neighbourhood of another node from another line, iMOD will try to snap it. Whenever the line turns green, lines will be overlapping each other perfectly, which means that there will be no thickness left for an aquitard. In this way, you can create a hole in the aquitard.

42 Click the *Close* button on the *Cross-Sections* window to close the *Cross-Section Tool*, click the *Yes* button whenever you are asked to save your adjustments.

43 Click the *Calculate* button ( ) and deselect the *Calc* option for [(1) Interface 1] and click the *Compute* button to start a new interpolation.

44 Click the *OK* button whenever the interpolation has finished and re-enter the *Cross-Section Tool* by clicking the *Cross-Section Tool* button ( ) again and select the cross-section [CROSSB6B1B2.SPF] from the list.

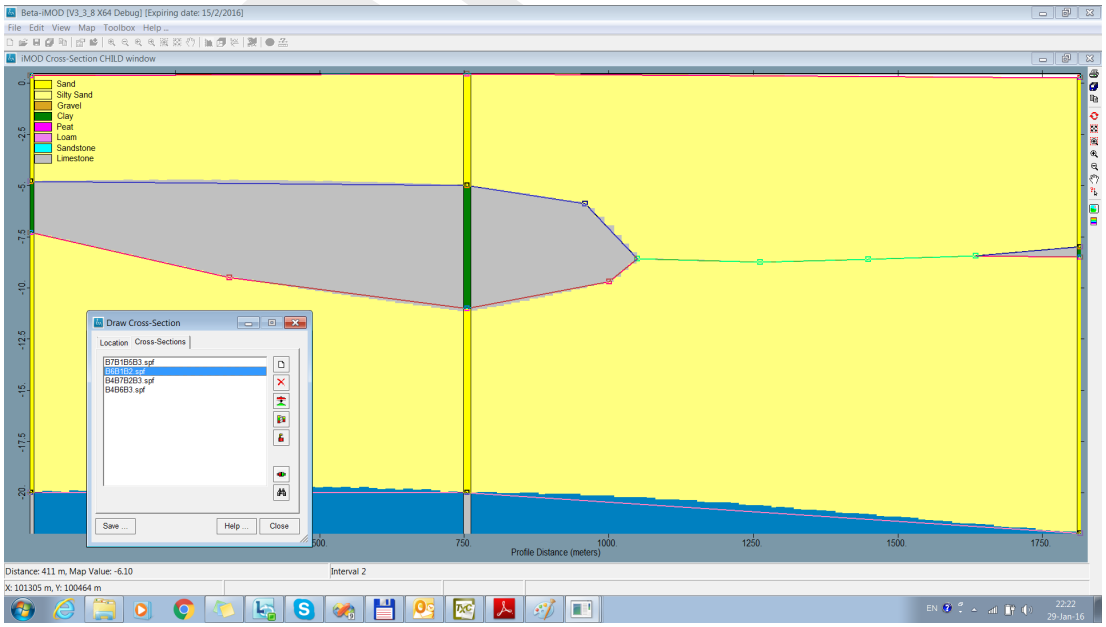


Figure 11.66: *The cross-section CROSSB6B1B2 after manual modification.*

45 Click the *Close* button on the *Cross-Section* window to leave the Cross-Section Tool, click *No* for the question whether you want to save the adjustments (well we did not adjust anything, did we?)

46 Click the *3D Tool* button ( ) and select the *Quasi 3D Model (aquitard)* configuration. This will organize the table for the *Display Configuration* such that iMOD will create a solid representation of aquitards.

47 Click the *Apply* button.

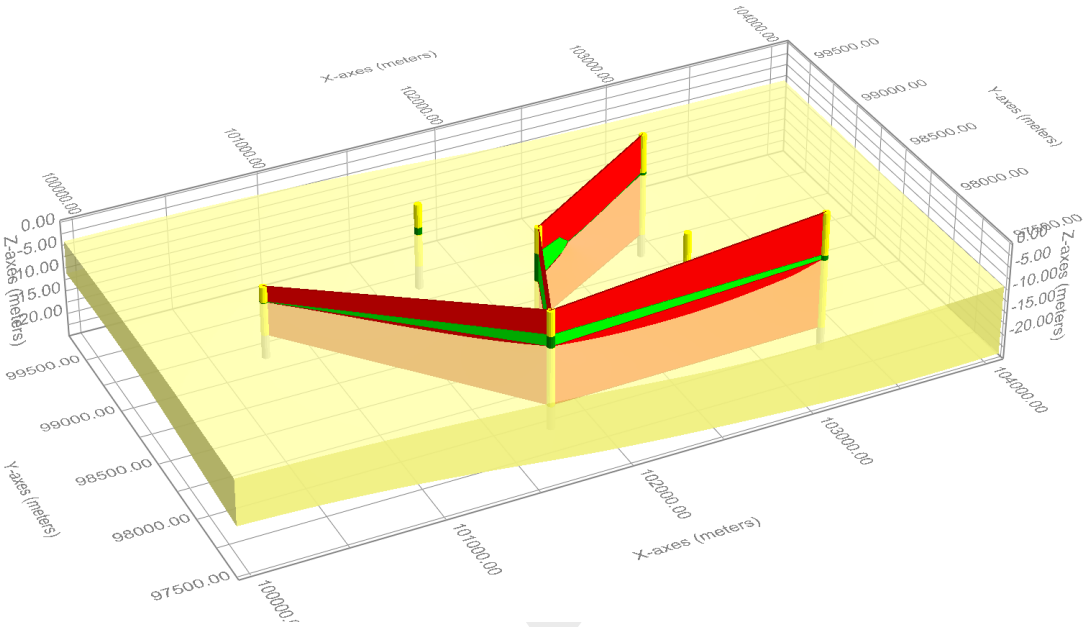


Figure 11.67: *3D image of the computed elevations of cross-section CROSSB6B1B2 and one of the intersecting cross-sections.*

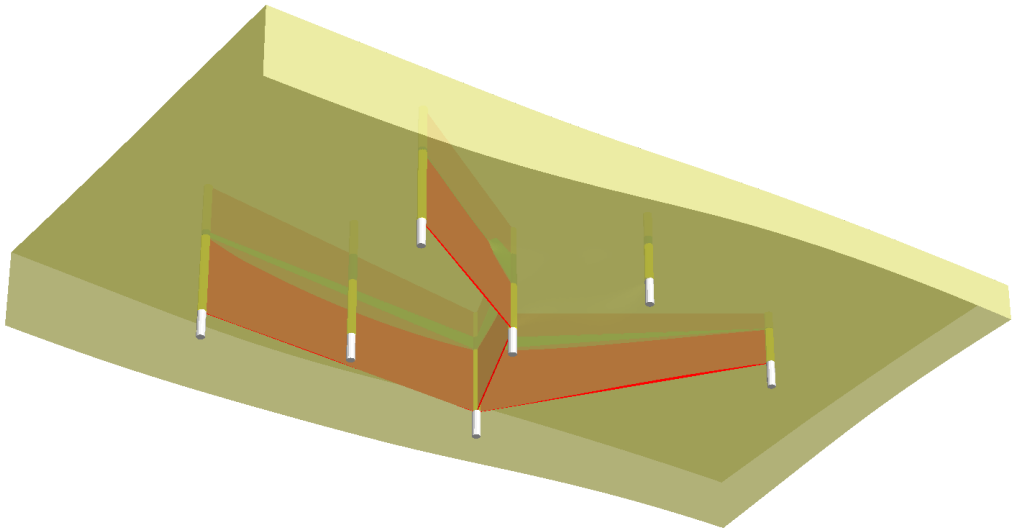


Figure 11.68: *Same cross-sections as previous figure, but now seen from below using transparency view settings.*

The Kriging algorithm generates the uncertainty of the estimate as a standard deviation (m). These files can be visualised per interface, they are save in the same folder as the computed interfaces and included the name _STDEV in the IDF file names.

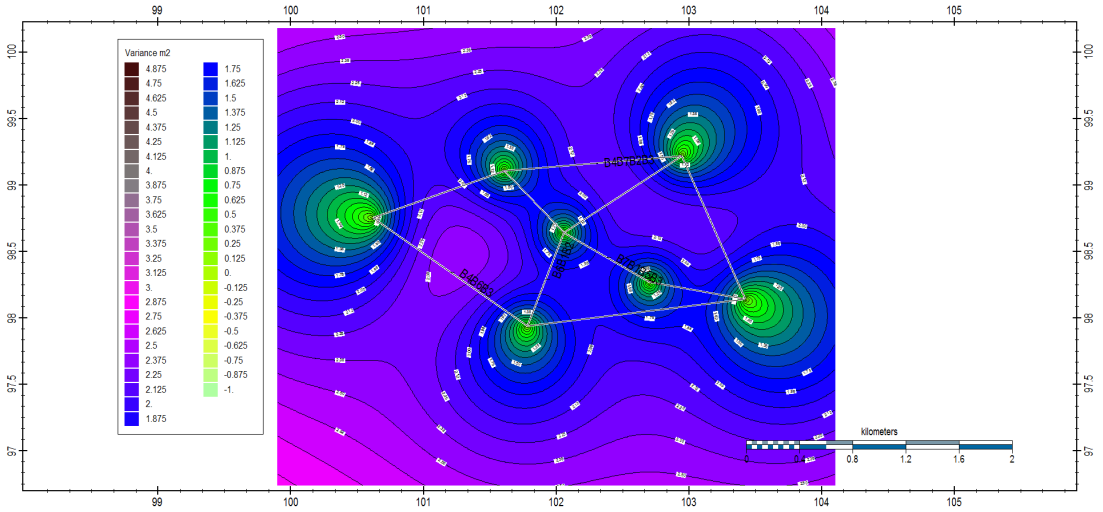


Figure 11.69: *Example of the estimated standard deviation of the estimated interface.*

Okay, for now this looks quite nice, we're done with our solid.

48 Click the option *File* from the *3D Tool* main menu and then select the option *Quit 3D Tool*.
49 Click the *Close* button on the Solid Tool window.

We will examine what the consequences are for the flow paths towards the well. In section 11.4 we've constructed a model from scratch and we will anticipate on your knowledge to do that again. We start with the requirements for this particular three-layered model.

50 Select the option *View* from the main menu and then select the option *Project Manager* to start the *Project Manager* window.

We have used this *Project Manager* in section 11.4 in detail. Please refer to that section for more information. Here, we will create the necessary model configuration as outlined in Table 11.4.

Table 11.4: *Model requirements for a confined, steady-state three layered model.*

Parameter	**model layer**	**IDF/Constant Value**
Boundary	1	. \DBASE \BOUNDARY.IDF
	2,3	1
Starting Heads	1,2,3	0.0 m+MSL
Top Elevation	1,2	. \DBASE \SURFACE_LEVEL.IDF
	3	. \SOLIDS \ISLAND \INT_L3.IDF
Bottom Elevation	1	. \DBASE \SURFACE_LEVEL.IDF
	2	. \SOLIDS \ISLAND \INT_L2.IDF
	3	. \SOLIDS \ISLAND \INT_L4.IDF
Horizontal Permeability	1,2,3	25.0 m/day
Vertical Permeability	1	25.0 m/day
	2	0.0001 m/day (1000 days/m)
Wells	3	. \DBASE \WELL.IPF
Net Recharge	1	0.5 mm/day
Porosity Aquifer	1,2,3	0.3
Porosity Aquitard	1,2,3	0.1

So, the only difference with our previous model in section 11.4 is that we use different values for our Top- and Bottom elevations. Let's start with the Project file we saved in section 11.4.

51 Click the *Open PRJ* button () and select the PRJ file you've saved at step 61 of section 11.4.
52 Adjust the Top for model layer 3 and the Bottom elevations for model layer 2 and 3 by clicking the *Properties* button () and change the parameter in the window accordingly. Don't forget to change the constant value for the Vertical Permeability for model layer 2!
53 Click the *Save As Runfile* button () when you are finished and save the file as [ISLAND_SOLID.RUN] in the . \IMOD_USER \RUNFILES folder.
54 Click the *Close* button to hide the *Project Manager* window.
55 Select the option *Toolbox* from the main menu and then the option *Start Model Simulation* to start the *Start Model Simulation* window.
56 Select the [ISLAND_SOLID.RUN] from the list of available runfile.
57 Select the tab *Results* on the *Model Simulation* window and enter the name [ISLAND_SOLID] as the result name.
58 Click the *Start Model Simulation* button.
59 Close the *Model Simulation* window by selecting the *Close* button.
60 Use the *Quick Open* window to display the computed heads for the first model layer, see step 74 in section 11.4.

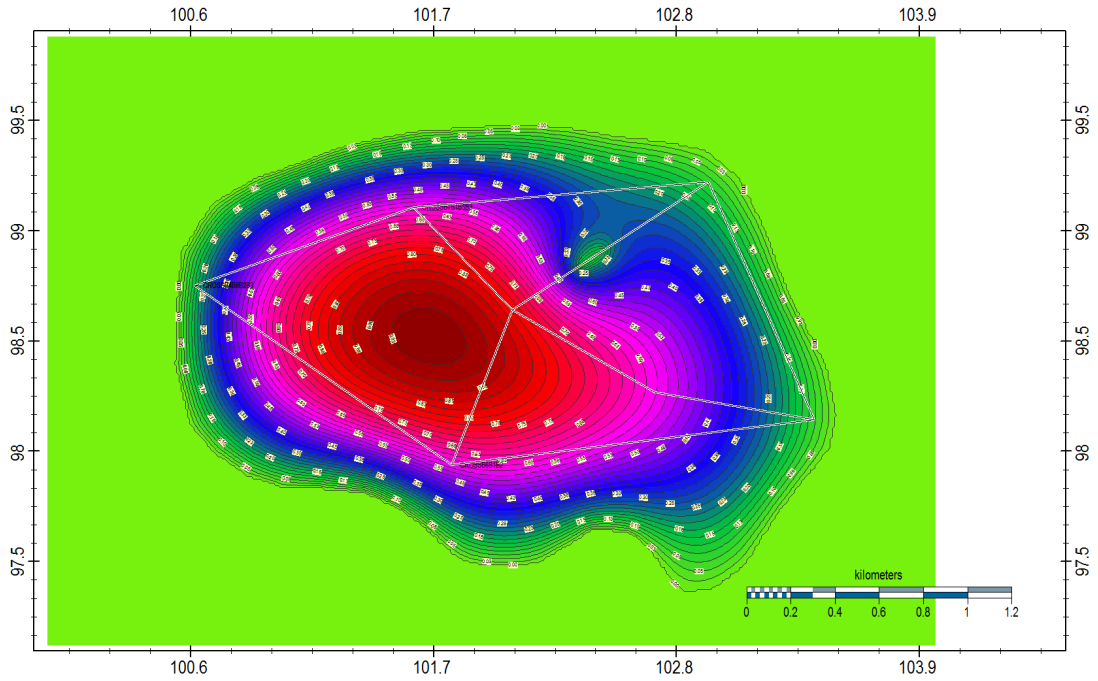


Figure 11.70: *Example of the computed heads using the adjusted subsurface geometry.*

You can see what the effects are from the hole in the aquitard (denoted by the white arrow). So, let's compute the flowlines towards the well. Instead of computing a forward tracing, we will compute a backward trace from the well back to its infiltration areas.

61 Click the *Add Map* button ( ) on the main menu and select the [WELL.IPF] which is situated in the . \TUTORIAL5_SOLID_BUILDING folder. So now we know where the well is actually.
62 Select the option *Toolbox* from the main menu and then choose the option *Define Startpoints*.
63 Enter [ISLAND_SOLID] in the input field and click the *Open and Continue* button.
64 Click the *Draw* button ( ), select the option *Circle* from the Select window and click *OK*. Now locate the well with your mouse cursor and left click your mouse on the well. Press the right mouse button to stop.
65 Select the *Definition* tab on the *Start Point Definition* window.
66 Enter [25] for the *Radius* and [5] for the *Sampling*. We will create startpoints every 5 meter on a circle which has a radius of 25 meter.
67 Click the *Open IDF* button ( ) to select the Top elevation of the second aquifer (actually the third model layer in which the well is located) as *Top Level*, so select the INT_L3.IDF from the . \IMOD_USER \SOLIDS \ISLAND folder. Repeat this for the *Bottom Level* and select the bottom elevation of the second aquifer (INT_L4.IDF).
68 Enter [10] for the *Vertical Interval*. We will have 10 particles starting in-between the top- and bottom elevation of the second aquifer (third model layer).
69 Click the *Draw* button to see the actual location of the start points.

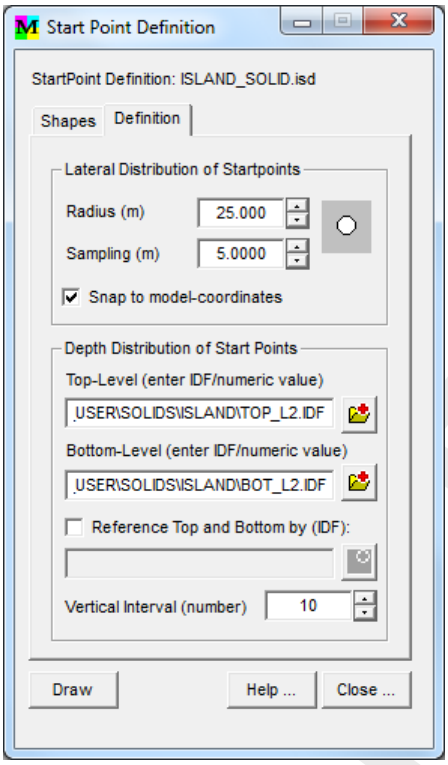


Figure 11.71: *The 'Start Point Definition' window.*

70 Click the *Close* button to save and close the window. Click *Yes* if you will be asked to save the file to [ISLAND_SOLID.ISD].

71 Select the option *Toolbox* from the main menu and the select the option *Start Pathline Simulation...* to start the *Pathlines Simulation* window.

We've have used this functionality before (see section 11.4, steps 85 onwards), so we will be brief this time.

72 Select the model result [ISLAND_SOLID] from the list of *Existing Results*.

73 Select the *Input* tab and click the *Open IPS File* button ( ) and search for the IPF file that you have saved in section 11.4.

If you did not save any IPF file, follow the steps 85 onwards mentioned in section 11.4, to fill in this window. Though we need to do a slight modification too. Since we've changed the interfaces of our model we should change the Pathline settings accordingly. So, . . .

74 Click the Properties button ( ) behind the *Top- and Bottom files* (second dropdown menu) and change the filenames in the list as shown in the next figure (SURFACE_LEVEL.IDF may be used instead of INT_L1.IDF).

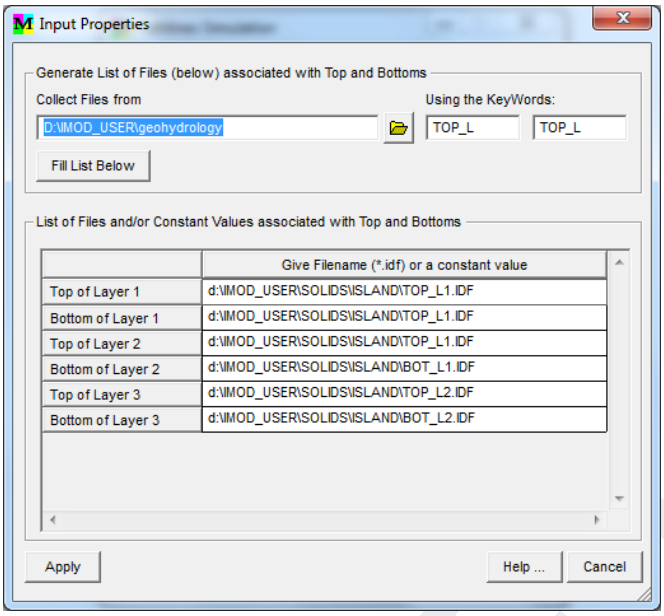


Figure 11.72: *The 'Input Properties' window that appears when choosing 'Start Pathline Simulation...' from the main menu, followed by selecting the 'Input' tab, and clicking the 'Properties' button at the right of 'Top- and Bottom files' field of the 'Pathline Simulation' window.*

75 Click the *Apply* button to return to the *Pathline Simulation* window.
76 Select the [ISLAND_SOLID.ISD] from the list with*Start Point Definition files*.
77 Select the *Results* tab of the *Pathline Simulation* window.
78 Select the *Backward* option from the *Trace Direction*.
79 Select the option *Save Entire Flowpath*.
80 Enter [ISLAND_SOLID.IFF] as the name to save the results to.
81 Click the *Start* button.
82 Click the *OK* button in the Information window that appears after the simulation finished.
83 Click the *Close* button to quit the Pathline Simulation window.

iMOD will display the results directly on screen.

84 Use step 101 upto 102 from section 11.4 to change the visualization such that the total travel times will be displayed.
85 Use the *3D Tool* to visualize the flowlines in combination with the created solid in a single view. You should be able to figure this out by yourself.

Finally, we complete this Tutorial with the results of our well capture zone in a 3D environment.

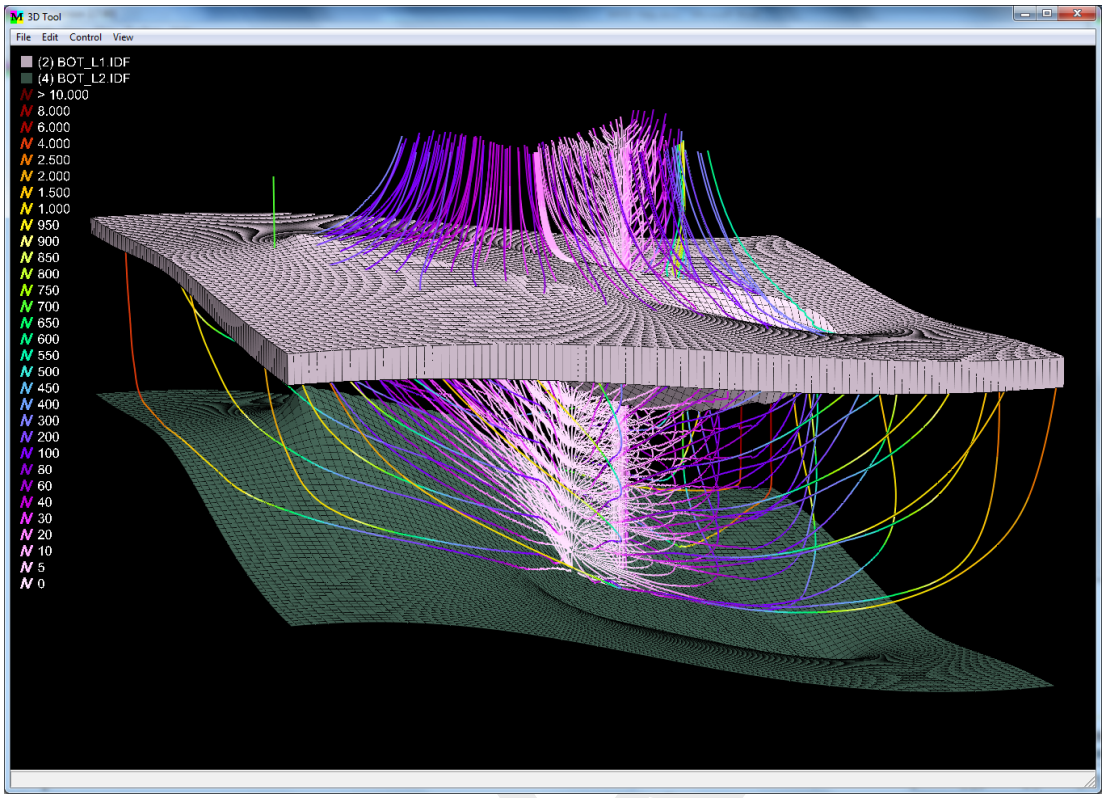


Figure 11.73: *The final pathlines representing the capture zone of the well; capture zone is here defined as that part of the groundwater flow system that contributes water to the pumped well.*

11.6 Tutorial 6: Model Simulation

This tutorial gives a short introduction in starting a groundwater flow model simulation. See for more detailed references *Model Scenarios* and *Model Simulation*.

Outline

This is what you will do:

- ◇ Understand the content of a model configuration file, i.e. a runfile;
- ◇ Simulating a groundwater flow model for different cell sizes and areas of interest;
- ◇ Understand the resulting folder structure with results;
- ◇ Computing and visualizing a waterbalance of the model;
- ◇ Defining a simple model scenario and include such a configuration to an original model configuration.
- ◇ Applying the new PKS-solver to simulated the model parallel.

Required Data

For this tutorial you need the following iMOD Data Folders:

- ◇ BND: IDF-files that describe the boundary conditions;
- ◇ DRN: IDF-files that describe drainage conditions;
- ◇ KDW: IDF-files that describe the horizontal transmissivity;
- ◇ OLF: IDF-files that describe the overland flow conditions;
- ◇ RCH: IDF-files that describe the natural recharge;
- ◇ RIV: IDF-files that describe the river conditions;
- ◇ SHD: IDF-files that describe the starting head conditions;
- ◇ VCW: IDF-files that describe the vertical resistance;
- ◇ WEL: IPF-files that describe the wells;
- ◇ TUT_MODEL.RUN: file that describes the model configuration and refers to the above mentioned folders;
- ◇ SCENARIO.GEN: file that describes the area that needs to be manipulated.

All these files are located in/below the folder: {path of installfolder} \tutorials \TUT_Model_Simulation.

Beside this data you will need the iMODFLOW executable to make the model computations.

Getting Started

1. Copy the TUT_MODEL.RUN into the . \IMOD_USER \RUNFILES folder.
2. Launch iMOD by double click on the iMOD executable in the Windows Explorer, and start by selecting the option *Create a new iMOD Project*.
3. Open the TUT_MODEL.RUN in a text editor (e.g. Notepad++) and observe the $DBASE$ keywords; as soon iMOD uses this runfile to perform a simulation this keyword will be replaced by the string as defined in the IMOD_INIT.PRF file which is in *{installfolder}*. Go to File–>Preferences... and click on ▶ to read the contents of the PRF-file; when you change and save the contents of the IMOD_INIT.PRF file in a text editor, click on ▶ to re-read the keyword settings for the current iMOD session. For more info on *Preferences* see section 2.7, for more info on available keywords see section 9.1. For information on the iMOD folder structure ee section 11.1 and chapter 9.
4. Go to View in the menu bar and select the iMOD Manager (or use the short-key *Ctrl+M*).

Model Parameters

Let us first observe some model parameters and understand what this model might be up to. We use the *Project Manager* for that, so let start that one.

5 Click the option *Project Manager* from the *View* menu.

iMOD simulates a groundwater flow model by means of a runfile. A runfile gives a full description of the use of all files needed for the simulation. The *Project Manager* is able to read the entire runfile and present the content in a treeview field.

6 Click the *Open Runfile* button () and select the TUT_MODEL.RUN file from the {IMOD_USER} \RUNFILE folder.

iMOD presents the content of the runfile in a treeview. Each branch represents a model parameter, and whenever a branch contains more information we can expand the branch to analyse its contents. Let us visualize the starting conditions for this particular model.

7 Make sure the *iMOD Manager* is also active (if not, press CTRL+M).
8 In the *Project Manager* expand the branch called *(SHD) Starting Heads* from the treeview field. Select this branch (not an individual IDF) and click the draw button (). If no image appears you may have to click the *Zoom Full Extent* button ().

As a result iMOD will open all the files of the selected branch and adds them to the *iMOD Manager*. In this manner it is easy to explore the available model parameters for the model. In this model the starting condition of a model simulation is equal to a result of a previous simulation. Since IDF-files are geo-referenced, they can be easily (re)used for different modules and/or packages in a model configuration.

9 Analyse the starting condition by creating several cross-sections (see section 11.3) and compute the difference between the starting condition for model layer 1 and the one for model layer 2 (see section 11.2). This gives you insight in downward and upward fluxes.
10 Click the branch *(RIV) Rivers (Cauchy conditions)* and observe that this model has two river systems. One is connected to model layer 1 and the other one is connected to model layer 2. Furthermore each river system consists of 4 input grids, CONDUCTANCE, RIVER LEVEL, RIVERBOTTOM LEVEL and INFILTRATION FACTOR. Examine the content of these files.
11 Explore the content of the *Project Manager*, e.g. plot the elevation of the existing Rivers in the first system.

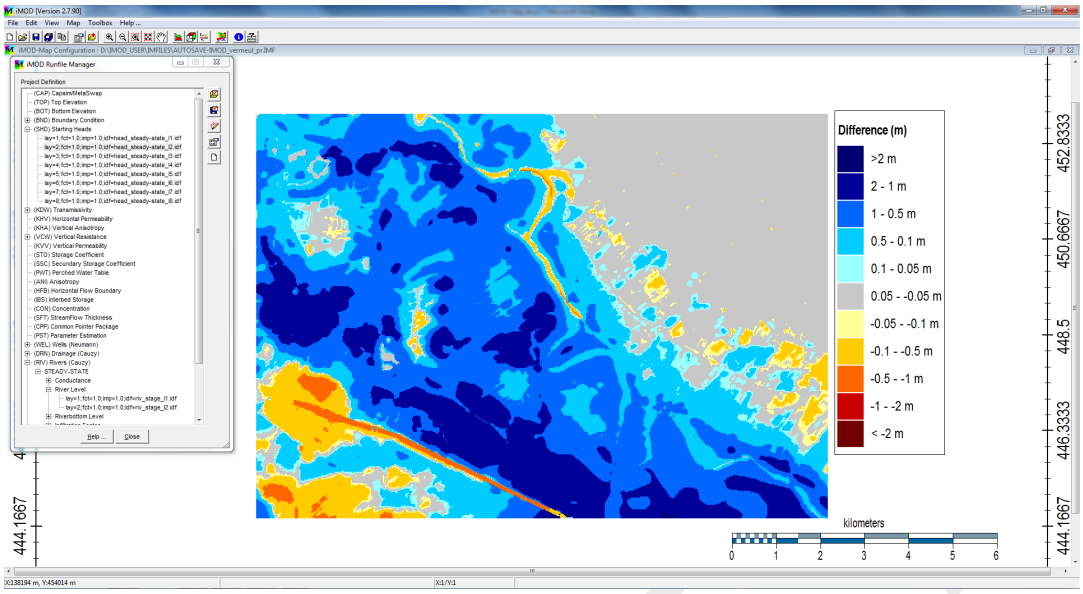


Figure 11.74: *Difference between starting heads of model layers 1 and 2.*

In this particular model a river is discretized for two different model layers , i.e. model layer 1 and model layer 2. The number of river elements is unlimited, however, a single IDF can store one river for each cell, so you should define more IDF files in those cases you want to specify more river elements at the same location. In this particular case we specified river elements for model layer 2 that penetrate the first aquitard and connect to the first aquifer (i.e. the second model layer).

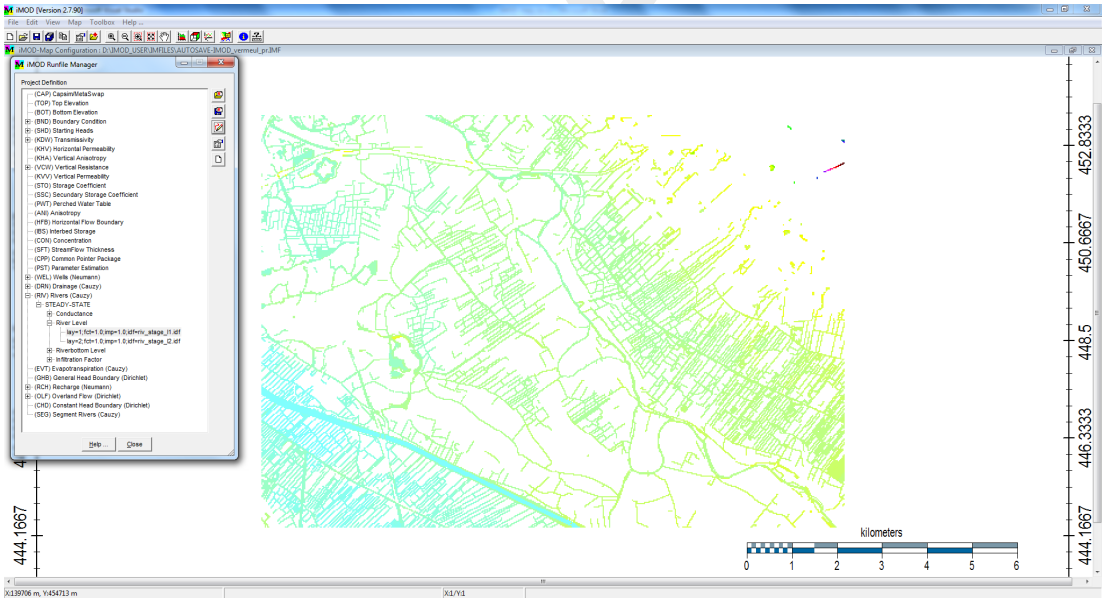


Figure 11.75: *Stages of the rivers of the first system.*

Model Simulation

Let's start the model simulation.

12 Select the option *Start Model Simulation* from the *Toolbox* option on the main menu. Whenever the *Start Model Simulation* window does not appear, check the keyword MODFLOW in the PRF file.

13 Select the TUT_MODEL.RUN from the *Runfiles* list. You should see a hatched area of the maximum extent of the model. If you do not see that, click the *Zoom to Extent* button ( ). Click the

Info () to display the runfile in a texteditor.

The first 15 lines of the runfile can be manipulated by the *Start Model Simulation* window. The rest refers to existing model input data, let's check whether all these data is available.

14 Close the editor and click the *CheckRun* button ().

iMOD will popup a summary file ({USER} \TMP \RUNFILE.LOG) of all files that cannot be found. Use step 6 to open a runfile to change pathnames if needed. If no files are listed, all files can be found and we can proceed.

15 Select the *Result Folder* tab and enter a name for the model results, e.g. MODEL25 and click the *Start Model Simulation* button.

iMOD will copy the selected runfile [TUT_MODEL.RUN] to the {IMOD_USER} \MODELS \
MODEL25 folder and renames it into IMODFLOW.RUN. Thereafter it will copy the simulation executable (e.g. iMODFLOW_V4_3_METASWAP_SVN1233_X64R.exe) to the same folder for archiving purposes, and it will start the simulation by the statement
'*{installfolder}\iMODFLOW_V4_3_METASWAP_SVN1233_X64R.exe IMODFLOW.RUN*'
in this example in the *{installfolder}*\IMOD_USER\MODEL25 folder. A DOS-command tool will open in which the simulation runs. You can proceed with iMOD or wait until the simulation finishes; it will take a very short time since the starting conditions are similar to the results.

The model simulation is always logged in the file IMODFLOW.list located in the subfolder mf2005_tmp, so in this example located in the folder {IMOD_USER} \MODELS \MODEL25 \mf2005_tmp. You should check this file IMODFLOW.list first whenever there is a problem with the simulation; as mentioned above it contains info on:

- ◇ the model discretization
- ◇ the model time and length units
- ◇ the processed input packages
- ◇ the solver used and how the iteration process progressed
- ◇ the volumetric budget for the entire model, including the percent discrepancy
- ◇ elapsed run time

16 When the simulation has finished, choose *Quick Open* from the main menu option *Map*.
17 Select the *Folder* [MODELS], then the variant [MODEL25], then choose the *Topic* [HEAD], and choose *Layer* [1] and then click the *Open* button. Make sure the option *Display* is selected! If everything went well the only option to be selected is [STEADY-STATE] in the dropbox *Time*. For transient simulations, you might be able to select a specific date.
18 Compare the results to the starting conditions. Use tools experienced in section 11.1, 2 and 3 if desired.

Let us simulate this model at a different resolution.

19 Start the *Start Model Simulation* (step 12) again and select the *Model Dimensions* tab and change the cellsize in *Simulate model with cellsizes equal to* = [100]. You can select a cellsize from the dropdown menu and/or enter a different cellsize in the input field to the right of the dropdown menu.
20 Go to the *Result Folder* tab and enter an output foldername [MODEL100] and click the *Start Model Simulation* button.
21 Open the resulting phreatic heads (model layer 1) with *Quick Open* (see step 74) and subtract the head calculated for MODEL100 from the head of MODEL25 using the *Map Calculator* () on the *iMOD Manager* and/or use the *Cross-Section Tool* () on the main toolbar to explore the differences caused by the different simulation cellsizes.

When using the *Cross-Section Tool* with the *Block Line* option (see section 11.3 step 12), you may expect the width of 4 blocks of the MODEL25 line equal to the width of 1 block of the MODEL100 line.

But this is probably not what you observe in step 22 and 25 of this tutorial, because iMOD standard reduces the number of sampling points to speed up the calculations. To get the widths as expect, you have to increase the value of *Maximum number of sampling points* in the *Cross-Section Properties* window (see section 11.3 step 11 or section 7.1 to open this window), e.g. set *Maximum number of sampling points to* [1000].

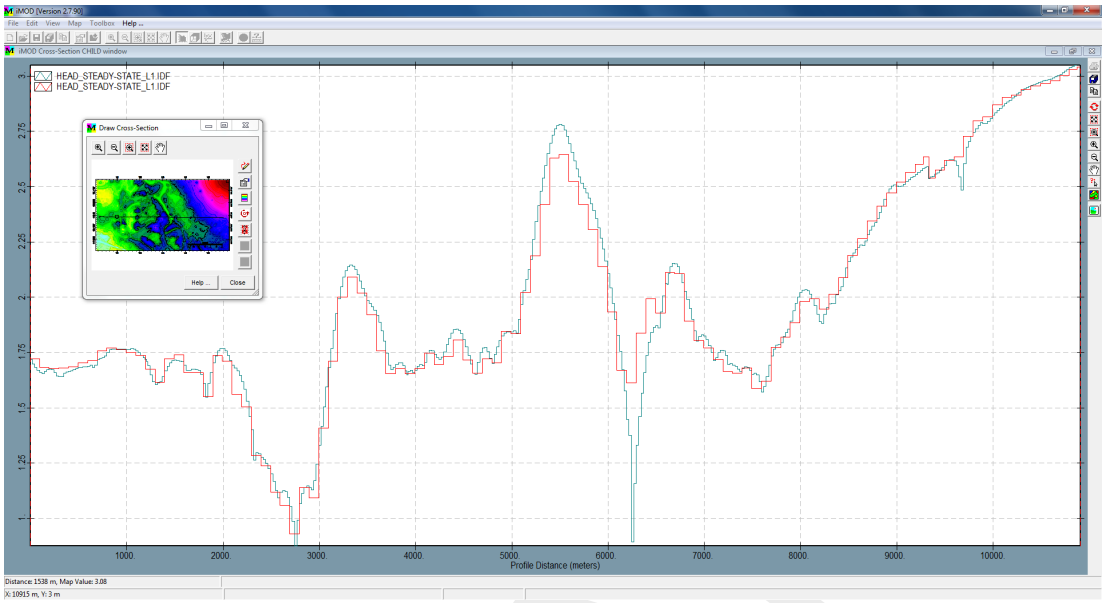


Figure 11.76: *Cross-section of heads of the 25x25 meter model (dark blue) and the corresponding 100x100 meter model (red).*

Let us simulate just a part of the model.

22 Start the *Start Model Simulation* (step 12) again and select the *Model Dimensions* tab and click the *Draw Simulation Area of Interest* button. You can interactively draw the area of interest within the hatched area. Click your left mouse button to set the first corner and give a second left mouse click to specify the opposite corner. You may drag the area of interest interactively by dragging the mouse while your inside the graphical display. Reset the cellsize to 25 meter and include a buffer-zone of 1500 meter.

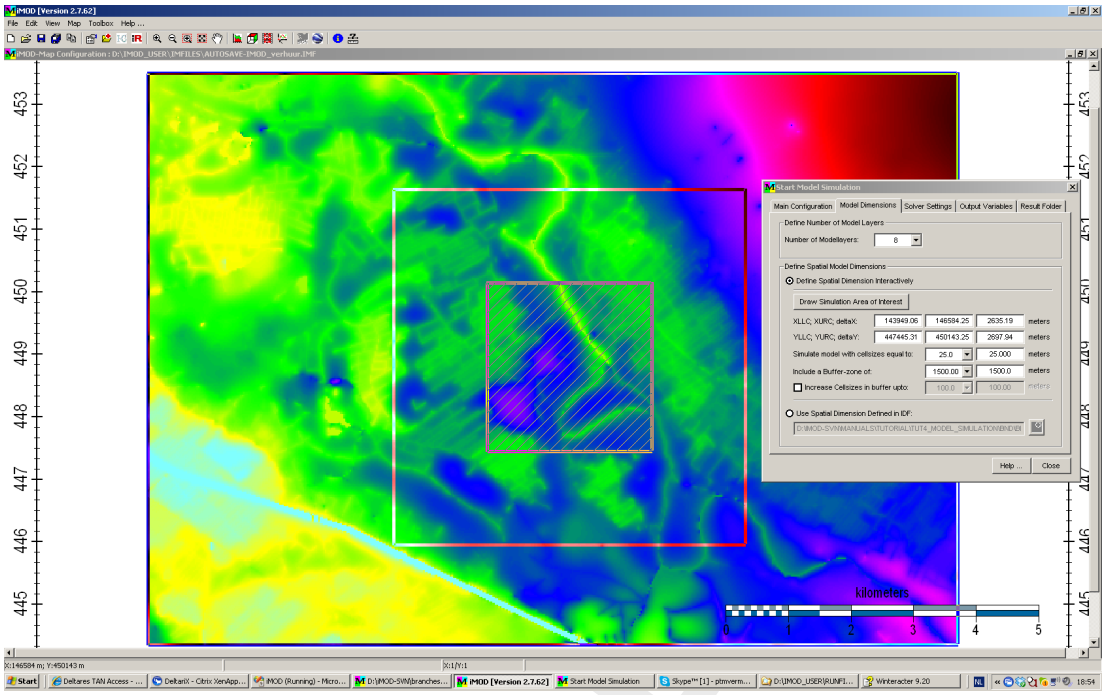


Figure 11.77: *Example of interactively specifying a part of the total model domain (smallest rectangle with hatching-pattern) for a model simulation. Also the size of the surrounding buffer zone can be specified here.*

23 Go to the *Output Variables* tab and select the option *Save Result Variable including the given Buffer Size*.
24 Go to the *Result Folder* tab and enter an output foldername [MODEL25PART] and click the *Start Model Simulation* button.
25 Open the resulting phreatic heads (model layer 1) with *Quick Open* (see step 74) and subtract the results using the *Map Calculator* ( ) on the *iMOD Manager* and/or use the *Cross-Section Tool* ( ) on the main toolbar to explore any differences.

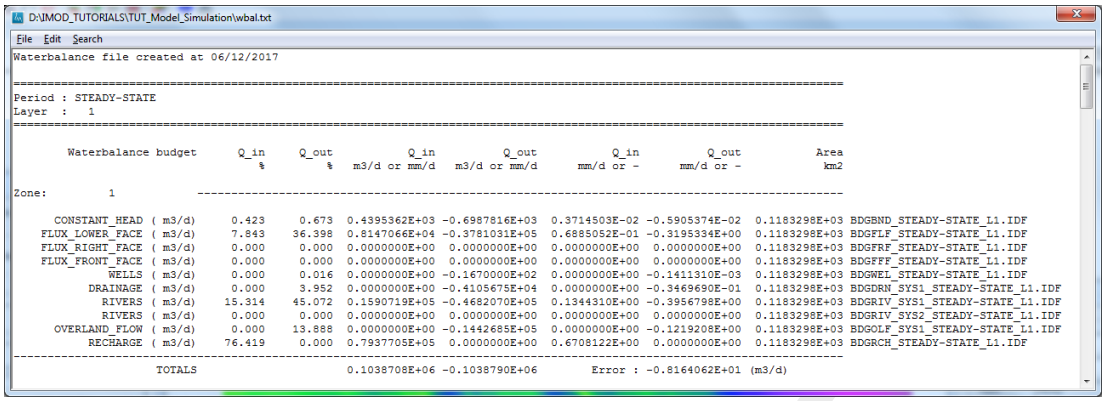
Water Balances

An important aspect of groundwater flow modeling is the ability to compute water balances. In iMOD you can compute these too. It is important that you specify the appropriate output variables prior to your simulation, see tab *Output Variables* in the *Start Model Simulation* window. In this case the defaults for the output variables were used.

26 Select the option *Water Balance* and then *Compute Water Balance* from the *Toolbox* menu.
27 Select the model [MODEL25] from the list at *Existing folder with Results available in the Models Folder*.
28 Click the *Modflow* button to select all water balance term that are relevant to Modflow (saturated groundwater), automatically. The iMOD convention is that all flux related output files start with BDG*. The content of these files is always in m^3/day.
29 Select the *Period and Layers* tab from the *Compute Waterbalance* window.

Here you can specify for what layers, and periods (in case of a transient model) need to be included in the water balance. For now we select all layers (which are selected by default), so we leave it like it is.

30 Click the *Create TXT . . .* button and save the water balance as WBAL.TXT.
31 iMOD will present the content of the WBAL.TXT file. Inspect the terminology and its content; more is explained below.



D:\IMOD_TUTORIALS\TUT_Model_Simulation\wbal.txt

File Edit Search

Waterbalance file created at 06/12/2017

Period : STEADY-STATE
Layer : 1

Waterbalance budget	Q_in %	Q_out %	Q_in m3/d or mm/d	Q_out m3/d or mm/d	Q_in mm/d or -	Q_out mm/d or -	Area km2	
Zone: 1								
CONSTANT_HEAD (m3/d)	0.423	0.673	0.4395362E+03	-0.6987816E+03	0.3714503E-02	-0.5905374E-02	0.1183298E+03	BDGBND_STEADY-STATE_L1.IDF
FLUX_LOWER_FACE (m3/d)	7.843	36.398	0.8147066E+04	-0.3781031E+05	0.6885052E-01	-0.3195334E+00	0.1183298E+03	BDGFLF_STEADY-STATE_L1.IDF
FLUX_RIGHT_FACE (m3/d)	0.000	0.000	0.0000000E+00	0.0000000E+00	0.0000000E+00	0.0000000E+00	0.1183298E+03	BDGFRF_STEADY-STATE_L1.IDF
FLUX_FRONT_FACE (m3/d)	0.000	0.000	0.0000000E+00	0.0000000E+00	0.0000000E+00	0.0000000E+00	0.1183298E+03	BDGFFF_STEADY-STATE_L1.IDF
WELLS (m3/d)	0.000	0.016	0.0000000E+00	-0.1670000E+02	0.0000000E+00	-0.1411310E-03	0.1183298E+03	BDGWEL_STEADY-STATE_L1.IDF
DRAINAGE (m3/d)	0.000	3.952	0.0000000E+00	-0.4105675E+04	0.0000000E+00	-0.3469690E-01	0.1183298E+03	BDGDRN_SYS1_STEADY-STATE_L1.IDF
RIVERS (m3/d)	15.314	45.072	0.1590719E+05	-0.4682070E+05	0.1344310E+00	-0.3956798E+00	0.1183298E+03	BDGRIV_SYS1_STEADY-STATE_L1.IDF
RIVERS (m3/d)	0.000	0.000	0.0000000E+00	0.0000000E+00	0.0000000E+00	0.0000000E+00	0.1183298E+03	BDGRIV_SYS2_STEADY-STATE_L1.IDF
OVERLAND_FLOW (m3/d)	0.000	13.888	0.0000000E+00	-0.1442685E+05	0.0000000E+00	-0.1219208E+00	0.1183298E+03	BDGOLF_SYS1_STEADY-STATE_L1.IDF
RECHARGE (m3/d)	76.419	0.000	0.7937705E+05	0.0000000E+00	0.6708122E+00	0.0000000E+00	0.1183298E+03	BDGRCH_STEADY-STATE_L1.IDF
TOTALS			0.1038708E+06	-0.1038790E+06	Error : -0.8164062E+01 (m3/d)			

Figure 11.78: *Example of a water balance TXT-file.*

In the above given example a water balance is presented for the entire model and for all model layers sequentially. In this case the water balance is given for a steady-state simulation and summed for the entire model area. There is only one zone used and the following terms are organized row wise:

- ◇ **CONSTANT HEAD**
  flux in or out across the boundary according the boundary condition specified around the model.
- ◇ **FLUX LOWER FACE**
  flux over the interface between model layer 1 and 2, along the z-direction downwards.
- ◇ **FLUX RIGHT FACE**
  flux over the interface between column interfaces between cells in model layer 1. along the x-direction eastwards.
- ◇ **FLUX FRONT FACE**
  flux over the interface between adjacent row interfaces in model layer 1, against the y-direction southwards.
- ◇ **WELLS**
  flux in the wells.
- ◇ **DRAINAGE**
  flux out the drainage systems.
- ◇ **RIVERS**
  flux in or out the river systems. There are two river systems presents, so therefore two lines are presented, though the second system is not active in the current model domain.
- ◇ **OVERLAND FLOW**
  flux in the overland flow drainage system.
- ◇ **RECHARGE**
  flux from the recharge.

Especially the (`Q_in` and `Q_out`) percentages are interesting and can be used to observe the relationship between the different water balance terms. In the above example >45% of the groundwater is discharged to the surface water.

32 Close the water balance text file in order to continue. The file can be inspected any time by a regular text editor.

There is another, more interactive manner, to examine water balances, let's try that.

33 Select the *Create CSV . . .* button and save the water balance as WBAL.CSV.
34 iMOD asks to start the *Waterbalance Analyser* after finishing the creation of the WBAL.CSV, click *Yes* to start the *Analyse Waterbalance* window.

Now you can see that iMOD read 8 records, 11 budget terms, 1 period, 8 layers and 1 zone. The *Waterbalance Analyser* can be used to aggregate budget terms and display them graphically.

35 Select the *Budget Terms* tab, here all available budget terms are listed from the WBAL.CSV.

36 Select the option [STEADY-STATE] form the *Timesteps* list.
37 Select the *Aggregation* tab, here all type of aggregation can be performed.
38 Select the *Select All* button from the *Model Layers* list.
39 Select the *Select All* button from the *Zones* list.
40 Select the *Graphics Output* tab, here all types of output can be selected, we leave it for now like it is.
41 Select the *Generate Preview* button to display the current configuration (as set in the previous tabs) in a graphical display.

The following image might appear.

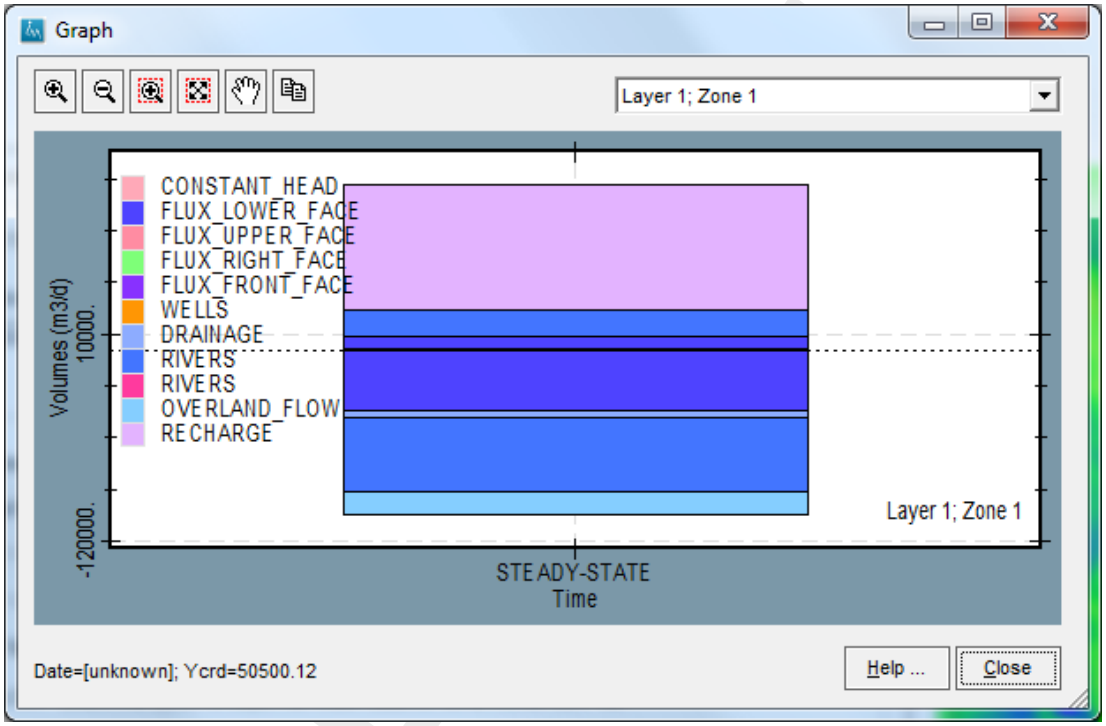


Figure 11.79: *Example of a water balance displayed from a CSV-file.*

From here you can zoom in or zoom out in the image, as well as select a different combination of model layer and zone number. If you have multiply zones nz and multiply layers nl, the list in the drop down menu is as long as $nz \times nl$. Let us display another graphical presentation of the water balance.

42 Click the *Close* button to close the *Graph* window.
43 Select the option *Graphical Representation*.

The water balance is presented as follows:

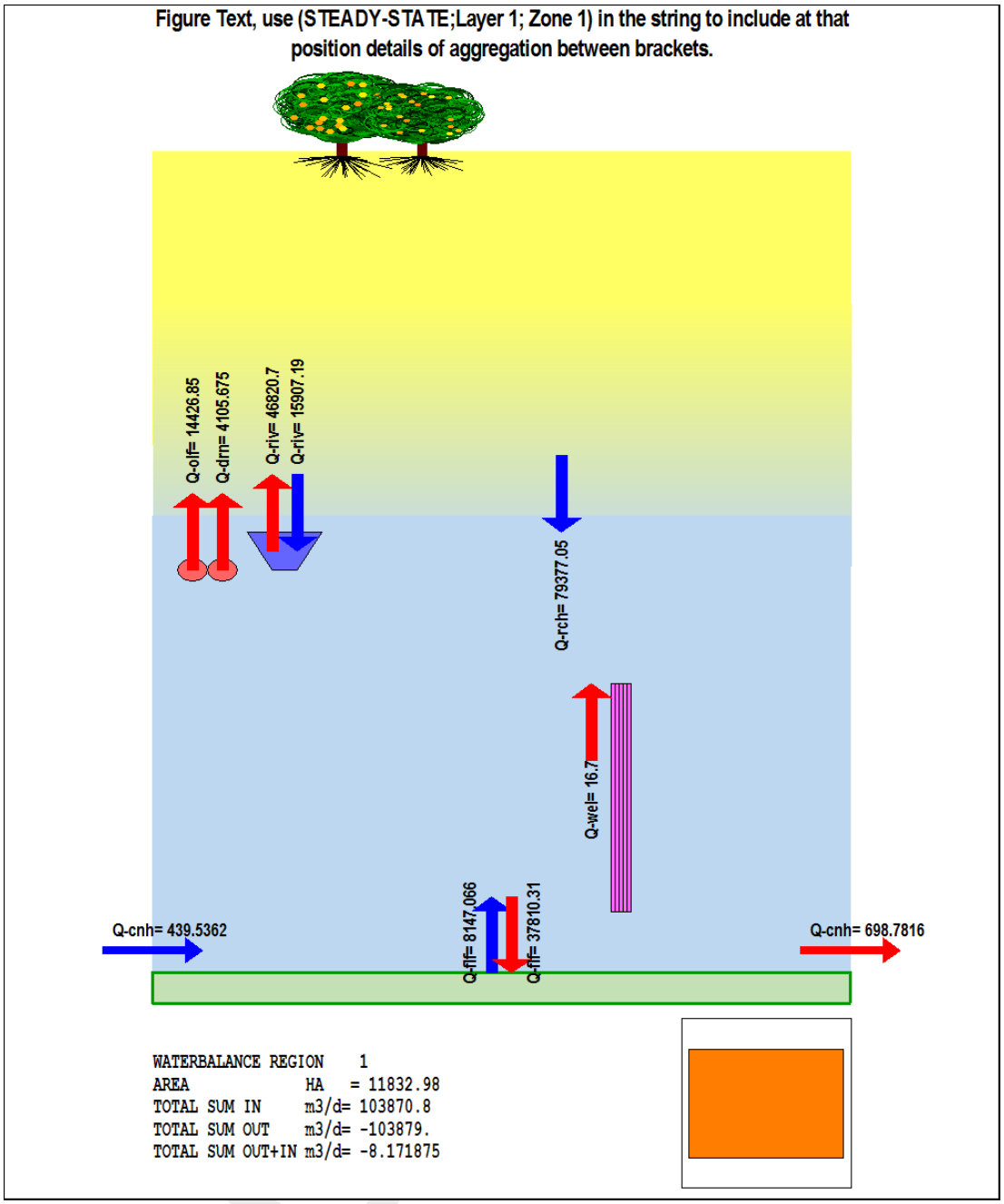


Figure 11.80: *Example of a water balance displayed from a CSV-file.*

There will be a sequence of 8 figures passing by, since we have selected 8 model layers, so repeat the following step 8 times to close all the repeating windows.

44 Click the *Close* button to close the graphical image window.

Let's try a more complicated CSV file in this tool.

45 Click the *Close* button to close the *Graph* window.
46 Select the *CSV-File* tab from the *Analyse Waterbalance* window.
47 Click the *Open CSV-File* button ( ) and select the file {path of installfolder} \tutorials \TUT_Model_Simulation \DELTARES1994.CSV. It's a pretty big water balance file and holds the results of a daily model simulation (365 periods) for 29 budget terms, 19 layers and 17 zones.
48 Select the *Budget Terms* tab.

49 Click the *Select All* button from the *Timesteps* list.
50 Select the *Aggregation* tab.
51 Select the option *Months, a single value per month, starting at the first month of the series*. Although the original data is for a daily base, the *Waterbalance Analyser* can aggregate the budget terms on a monthly base automatically.
52 Click the *Select All* button from the *Model layers* list.
53 Select the option *Sum Selected Layers* from the *Layer Aggregation* input field. In this way, all fluxes as summed over the selected layers.
54 Click the zones 8, 9 and 13 from the *Zones* list. Use the Ctrl-Left mouse button to make that selection.
55 Select the *Graphics Output* tab.
56 Click the *Generate Preview*, make sure the option *Time Series* is still selected.
57 Select the option *Layer [sum]; Zone 13* from the drop down list in the *Graph* window.

The following image appears.

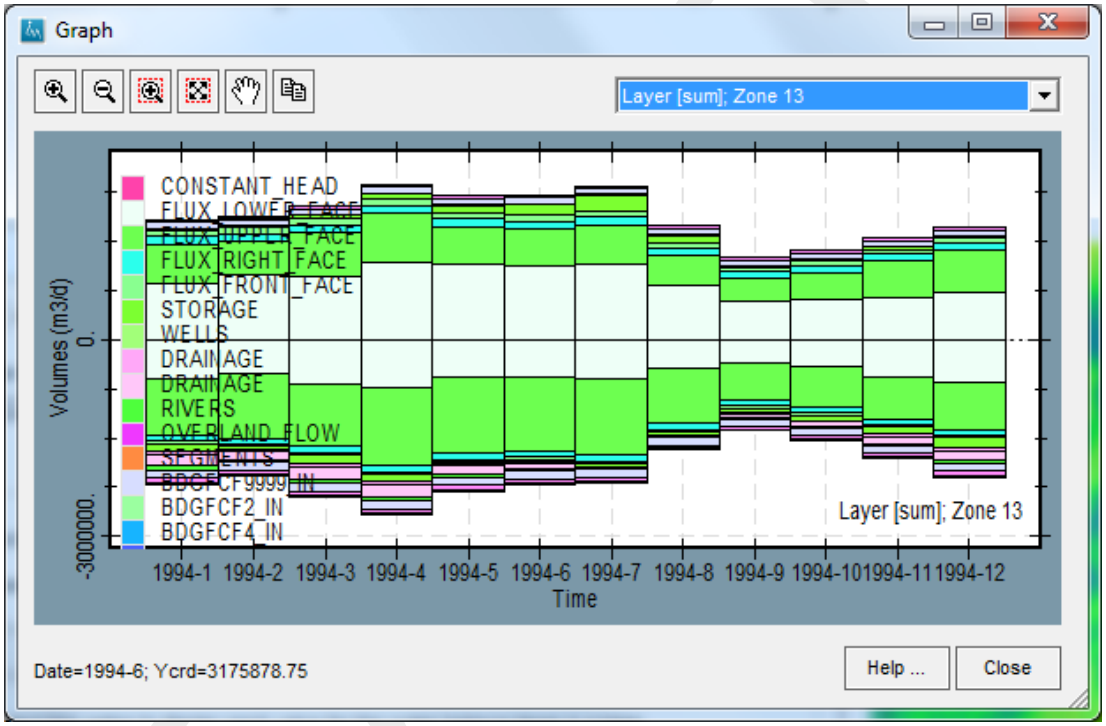


Figure 11.81: *Example of a water balance aggregated on a monthly base from a CSV-file.*

There is a lot to try in this tool, all kind of different settings can be combined, see section section 7.16.2 for more detailed information. Feel free to experiment a bit more with all the possibilities. If you're done, we would like to compute a water balance for a specific region.

58 Click the *Close* button on the *Analyse Waterbalance* window to close it.
59 If the *Compute Waterbalance* window is removed, restart it again by following the steps 26 up to 29.
60 Go to the *Apply To* tab and select the option *Apply* within shapes (*.gen). Select the pencil ( ) and start drawing a polygon on the graphical canvas (see section 4.5 for more information about the specific functionalities that you can use while drawing a polygon).
61 Click the *Create TXT* button and enter the name WBAL_PART.TXT and inspect the resulting water balance file.

In the iMOD Manager you will notice the ⟨NAME OF water balance⟩.IDF. This file reflects the position of the given polygon. You can reuse this file (e.g. after editing) in another water balance computation

(*Apply for NoDataValues in given IDF-file*).

62 Close the water balance text file in order to continue. The file can be inspected any time by a regular text editor.

Scenario Simulation

Let's build a scenario in which we will increase a river stage from the current model configuration. We can do that in two manners. One manner is to adjust the appropriate IDF-files that discretize the river system, e.g. RIV \RIV_STAGE_L1.IDF and RIV \RIV_STAGE_L2.IDF by means of IDF Edit (see section 6.7.4).

63 Click the *Close* button on the *Model Simulation* window, if needed.

64 Select the file RIV \RIV_STAGE_L1.IDF in the iMOD Manager and click the *Redraw* button ( ) to (re)draw it.

65 Make a copy of RIV_STAGE_L1.IDF and RIV_STAGE_L2.IDF by using the Map Operation option. Fill in on tab Algebra in field *Map C* the directory+RIV_STAGE_L1_0.5.IDF, fill in 'C=1.0*A'in the field *Formulae* and select *Map A* and click on *Compute...*. Now a copy of the selected file is made.

66 Repeat the previous step for RIV_STAGE_L2_0.5.IDF

67 Zoom in for the desired river segment at the coordinates [x=145000.0] and [y=448100]. You can use the option *GotoXY* from the *View* menu (see section 5.2) and use *Zoom(m)*=[1500m].

68 Select RIV_STAGE_L1_0.5.IDF in the iMOD Manager and select the option IDF Edit from the Edit menu or with the right mouse click on the map.

69 Click the *Open GEN* button ( ) and open the file SCENARIO.GEN that is located at {path of installfolder} \tutorials \TUT_MODELSIMULATION.

We've created a shape (polygon) to specify the area in which we will change the river stage. Let's assign the measure to be attached to the polygon.

70 Select [SHAPE1] from the list (see figure below).

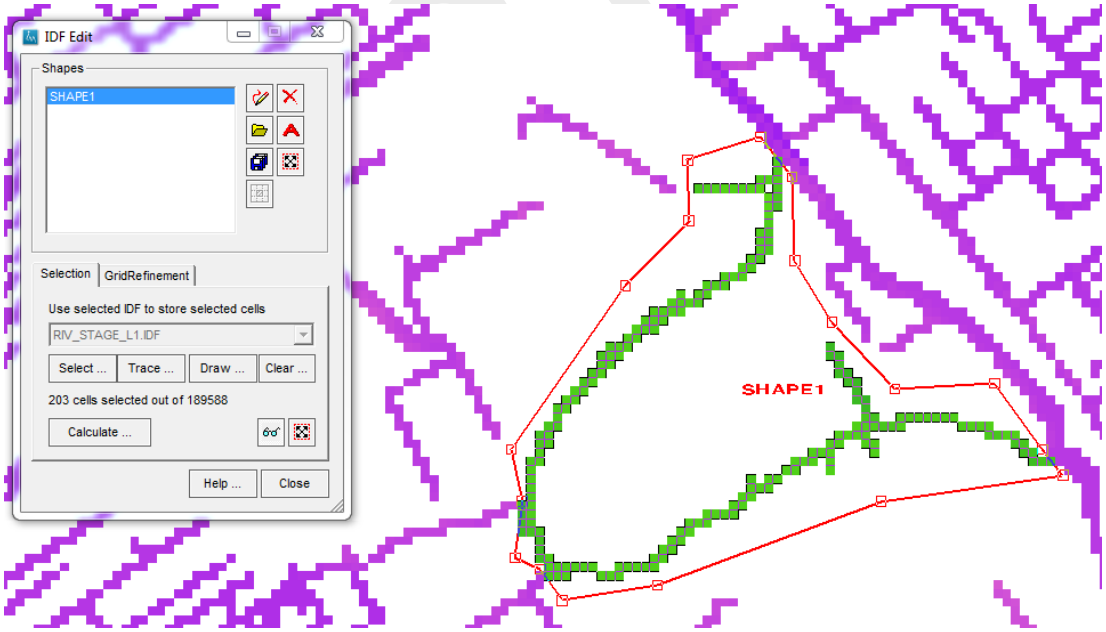


Figure 11.82: *The 'IDF Edit' window in front of the area of interest.*

71 Click on *Select...* at the selection tab of the IDF Edit window.

72 The *IDF Edit Select* window will be opened. Notice that the option *Select for Polygon* is checked!

73 Select 'All'in the *Logic* dropdown menu and click on *Get Selection* at the bottom of the window. Now all river cells in the polygon 'Shape 1'are selected.

74 Click on *Close* to close the *IDF Edit Select* window.

75 Click on *Calculate...* on the *IDF Edit* window.
76 Select the option *New Value* on the *IDF Edit Calculation* window and choose '+'from the dropdown menu.
77 Fill in '0.5'in the field behind the dropdown menu and click on *Calculate*.
78 Click on *Close* and repeat step 68-77 for RIV_STAGE_L2_0.5.IDF.

So, we've created a scenario definition that raises by 0.50m the stage of all river systems that penetrate model layer 1 and 2 inside the current polygon (SHAPE1), by making. use of the iMOD Edit option.

Okay, let's use this scenario definition in a model simulation.

79 Create a new folder in your IMOD_USER_MODELS folder outside of iMOD (in the Windows Explorer), e.g. "C: _iMOD_IMOD_USER_MODELS_RIVER_STAGE". This folder is needed later to store the results of the scenario simulation (step 89).
80 Select the option *Start Model Simulation* from the *Toolbox* option on the main menu. Select the TUT_MODEL.RUN from the *Runfiles* list. (**Note:** if TUT_Model.run is not available in the list copy the runfile from the Tutorial folder to your IMOD_USER/RUNFILE folder.)
81 Click on the *Info* button ( ) to open the runfile in the inbuild texteditor.
82 Find the lines with the filenames of the river stages,]
e.g. '1,1.0,0.0,D: \iMOD \tutorials \TUT_MODEL_SIMULATION \RIV \RIV_STAGE_L1.IDF and 2,1.0,0.0,D: \iMOD \tutorials \TUT_MODEL_SIMULATION \RIV \RIV_STAGE_L2.IDF.
83 Change the filenames into RIV_STAGE_L1_0.5.IDF and RIV_STAGE_L2_0.5.IDF.
84 Click on the save button and close the text editor window.

As you might observe, the area of interest (within the shape) is smaller than the total extent of our model. Let's decrease the size of our model (in order to speed up our simulation).

85 Select the *Model Dimensions* tab and click the *Draw Simulation Area of Interest* button.
86 Left click your mouse approximately 1,000m west of the south west corner of the polygon (SHAPE1) and left click on approximately 1,000m east of the north east corner of the polygon. This will be our area of interest. You can increase or decrease it by moving your mouse in the neighbourhood of the boundaries and drag your mouse as soon as the mouse cursor changes in and .
87 Select or enter a buffersize (*Include a Buffer-zone of*) of [1500m].
88 Select the *Output Variables* tab and select the option *Save Result Variable inclusive the given Buffer Size*.

A buffer zone prevents that model results are affected by boundary conditions on the lateral model boundary. It depends on the scenario configuration, model configuration itself and the geohydrological subsoil what this buffersize should be. It is hard to determine beforehand, so it is wise to analyse the effects near the model boundaries to decide whether your simulation is affected by the lateral boundary conditions too.

89 Go back to iMOD and select the *Result Folder* tab on the *Start Model Simulation* window.
90 Select the folder you created and click the *Start Model Simulation* button to confirm the operation. Results of scenario computations will be stored in the folder you created yourself.
91 After the simulation ended, open the phreatic heads (HEAD_STEADY-STATE_L1.IDF) with *Quick Open* (see step 74).
92 Compute the differences in phreatic heads between the . \MODELS \MODEL25 and this scenario . \MODELS \RIVER_STAGE. Use step 14 and forward from section 11.2.
93 Analyse the differences in head for all model layers to observe whether the chosen buffersize was sufficient. Use *Quick Open* to load all files in the *iMOD Manager*.

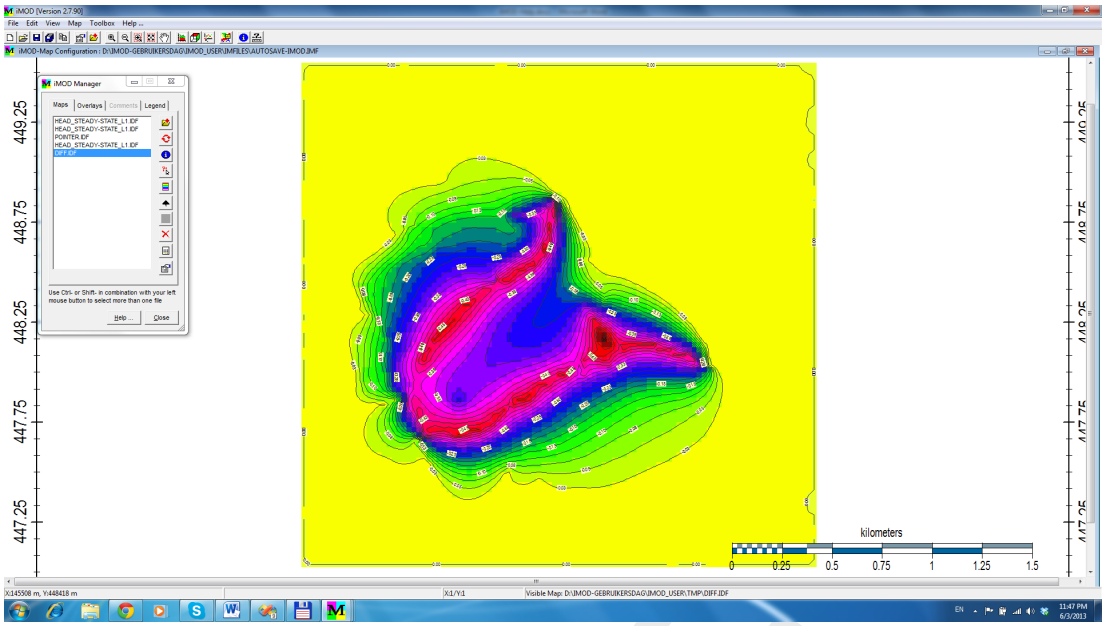


Figure 11.83: *Contour levels of the computed effect of a raised water level.*

In the example above it is clear that the boundaries of our submodel have been chosen appropriately since the change in head is not affected by the model boundary. You can also make a cross-section of the computed effect to judge whether the boundary has been chosen right.

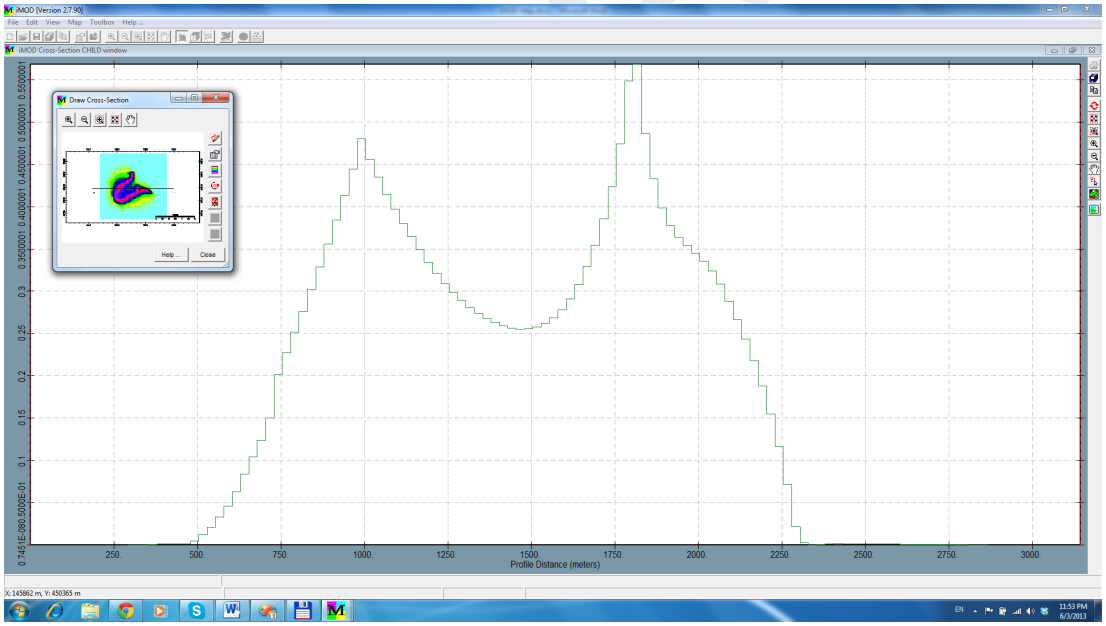


Figure 11.84: *Cross-section of the computed effect of raised water level.*

94 Finally try to answer the question: “Why are the head differences more than 0.50m at some locations (0.55 meter), although river stages are increased by 0.50 meter only?”

Model simulation with the Parallel Krylov Solver (PKS) package

So far we only used the single core PCG solver, now let’s switch to the multi core Parallel Krylov Solver. This requires that you have correctly installed the MPI software, see the iMOD Installation Instructions (section 2.3).

The PKS package solver settings can be configured in two ways:

1 In the iMOD-GUI: in the 'Solver Settings' tab of the 'Model Simulation' window of the 'Tools' main menu option (see the figure below); we will practise this in a minute.
2 Manually: by editing **Data Set 5** of a runfile according the specifications given in section 10.6; an example is given in section 10.20.6.

Note: The PKS package is not yet available in the Project Manager.

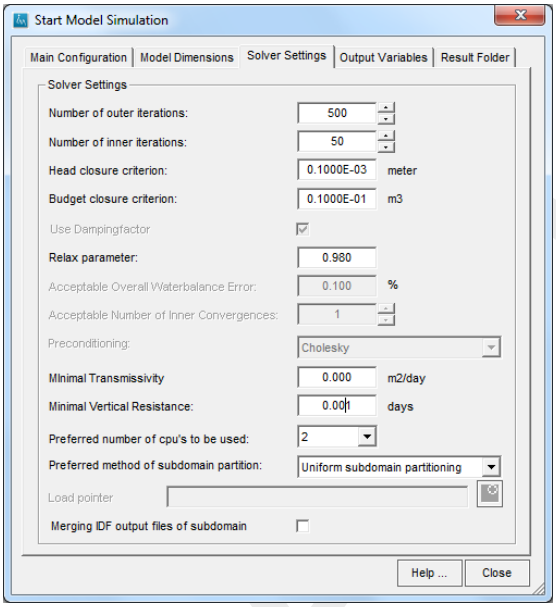


Figure 11.85: *The 'Solver Settings' tab of the 'Model Simulation' window. In this example the user has assigned more than one CPU; as a result the PKS solver is activated.*

When using the PKS package, the model domain is divided in sub-domains *automatically*; the number of sub-domains is always equal to the number of computational cores the user assigns in the 'Solver Settings' window.

The overall computational model performance depends among others on how long it takes to solve each individual sub-domain; an iteration for the whole model domain can only be completed when all individual sub-domains have been solved. This means that *load balancing* is very important for the overall parallel performance. Ideally the actual work/load should be distributed as equally as possible over the multiple computational cores. PKS now supports two methods sub-domain partitioning methods:

1 Uniform sub-domain partitioning in x,y-direction; when e.g. using four CPU's the model domain will be divided into four equally sized sub-domains.
2 The Recursive Coordinate Bisection (RCB) method. The RCB method incrementally partitions the model domain step by step and alternates the partitioning in the x- and y-direction until the number of sub-domains is equal to the number of assigned CPU's. Simultaneously the sub-domains are automatically being re-sized such that ultimately the load of each sub-domain is the same. The load of a sub-domain is the summation of the user-defined weights (or load) of the model cells within the boundaries of that particular sub-domain.

Figure 12.18 in section 12.32.3 shows an example of both methods for the Netherlands Hydrological Model (De Lange *et al.* (2014)) and 128 sub-domains.

When assigning two CPU's and selecting the uniform partitioning method, the model domain will be divided into two equal sub-domains.

When using the RCB method, the user has to specify per model cell a weight representing an estimate of how much each cell contributes to the computational effort to be made to solve the set of equations. There is no partitioning in the z-direction, so ideally the specified weights should also take variations of the total number of active cells per x,y-location (a particular vertical column) into account. One should be aware of the fact that even with the RCB and irregular boundaries, finding an optimal weight distribution can be difficult and subject to trial-and-error. The spatial weight distribution depends on for example differences in the complexity of boundary conditions (stresses) and coupling concepts.

In this tutorial we will exercise the use of the RCB method: you will run the groundwater flow model using two CPU's applying a load balancing grid. This can be done by the following steps.

95 Select the option *Toolbox* from the main menu and then the option *Start Model Simulation* to start the *Start Model Simulation* window.
96 Select the TUT_MODEL.RUN from the *Runfiles* list.
97 Select the *Solver Settings* tab.
98 Within this tab, select *2* for *Preferred number of cpu's to be used*.
99 For *Preferred method of subdomain partition* select *Recursive Coordinate Bisection*.
100 For *Load pointer* select {installfolder}\TUTORIALS\TUT_MODEL_SIMULATION \PKS \LOAD.IDF.

The following figure shows the specified loads of the LOAD.IDF grid: in the left part of the grid all cells have the value 1 and in the right part all cells have a value 2. So in this example we assign twice as much weight to approximately 20% of the model cells (note that this grid is just illustrative since for this model a uniform load of 1 for each computational cell would be most optimal).

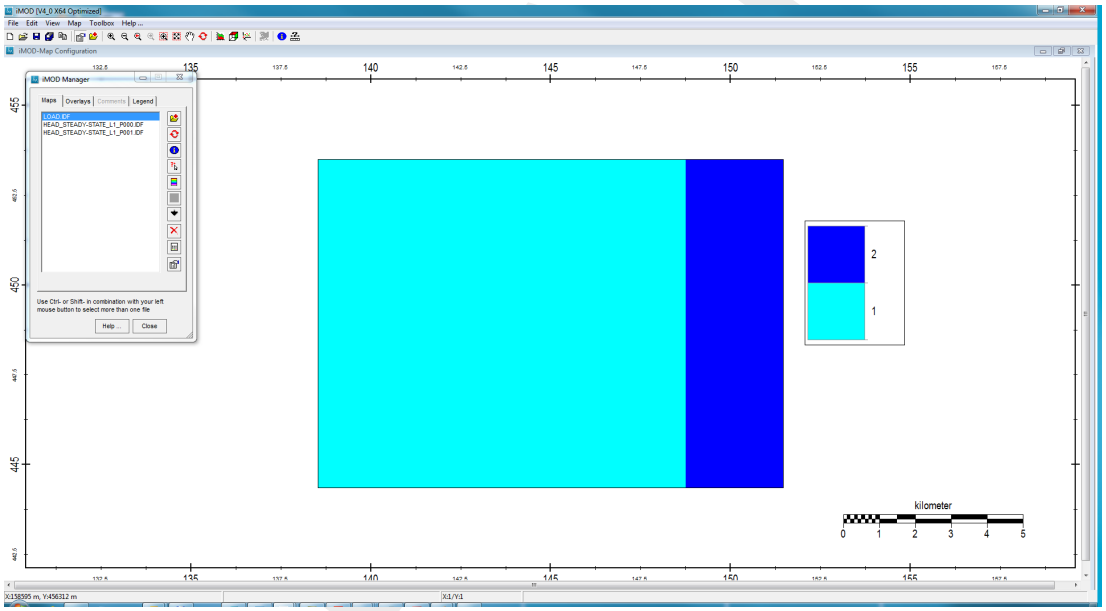


Figure 11.86: *The values of the LOAD.IDF grid used to specify the weights to be used in the Recursive Coordinate Bisection partitioning method; in this example approximately 20% of the model cells were assigned weight values that are two times larger than the rest 80% of the model cells.*

101 Turn on the checkbox for *Merge IDF output files of subdomain*.
102 Select the *Result Folder* tab and enter a name for the model results, e.g. MODEL25_PKS and click the *Start Model Simulation* button; the model will be run using two computational cores.

Similar to a serial computations you can view the head results with *Quick Open* from the main menu option *Map*. We turned on the checkbox *Merge IDF output files of subdomain*: after the model-run the sub-domain-IDF's will be merged to IDF's covering the total model domain and the sub-domain-IDF's are deleted. Of course we are also curious about how the total model domain was partitioned in two sub-domains automatically using our weight distribution grid. To see the partitioning-result of RCB

method we will re-run the model in parallel mode, however, now without turning on the *Merge IDF output files of subdomain*-option:

103 Select the *Solver Settings* tab.
104 Within this tab, turn off the checkbox for *Merge IDF output files of subdomain*.
105 Select the *Result Folder* tab and enter a name for the model results, e.g. MODEL25_PKS2 and click the *Start Model Simulation* button.
106 When the simulation is done, go to *View* in the menu bar and select the *iMOD Manager* (or use the shortcut *Ctrl+M*). Select the *Open Map* button ( ) and click the button *Open*. Navigate to \IMOD_USER \MODELS \MODEL25_PKS2 \head and select the files head_steady-state_l1_p000.idf and head_steady-state_l1_p001.idf for the computed heads for the first model layer. Click the button *Open*.
107 Select the *Map* option from the main menu, choose the option *Current Zoom Level* and then choose the option *Percentiles*.
108 Select the *View* option from the main menu, choose the option *Show IDF Features* and then choose the option *IDF Extent*.

These steps result into the following figure, where the left sub-domain is clearly larger than the right sub-domain due to the specified weights. As you may have noticed the partitioning is not equal to the weight distribution of the LOAD.IDF grid, in other words, by specifying this pointer grid, you are not enforcing a particular partitioning of the model domain. This is caused by the RCB method which results in two sub-domains that each have an equal computational load (based on your estimated weight distribution); as mentioned above, the load of each sub-domain is calculated as the sum of the user-assigned weights of all cells lying within the boundaries of that sub-domain. Suppose we would have taken the LOAD.IDF grid as a basis for partitioning, this would have resulted in a relative load for the left part of '80' and for the right part '2 x 20 = 40'. The RCB automatically shifts the boundary between the sub-domains such that the two resulting sub-domains each have a fifty-fifty (50-50) computational burden; that's why the right sub-domain also contains part of the model domain having cells with weight values equal to 1.

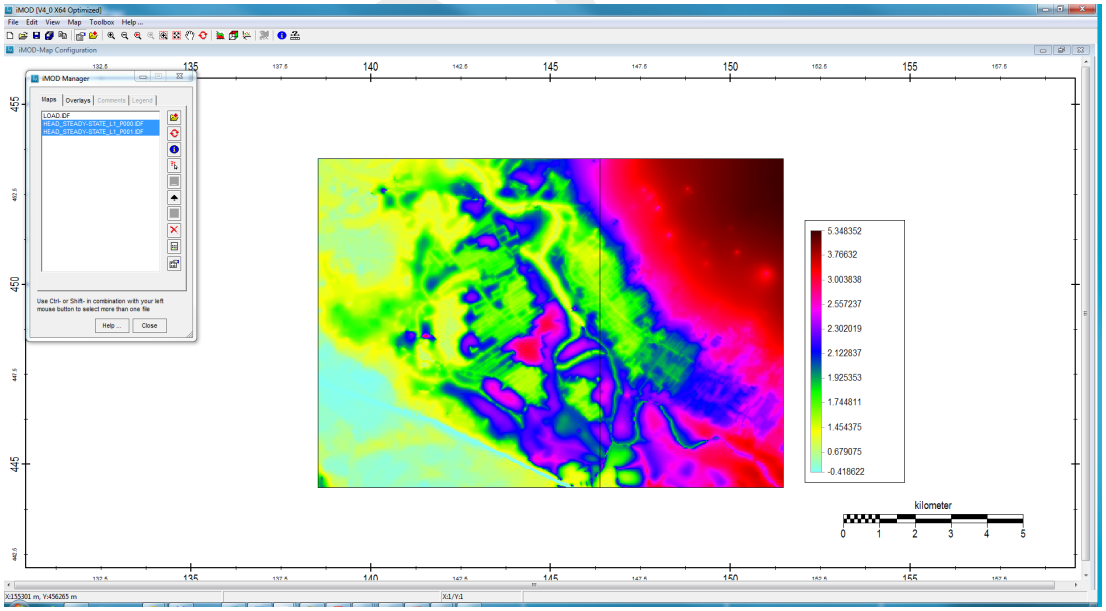


Figure 11.87: *The non-merged head-IDF's of the two sub-domains using the RCB partitioning method. The partitioning is visible when choosing 'View', 'Show IDF features', 'IDF Extent'.*

For your own model, experiment with different weight distributions for finding optimal load balancing. Provided your machine has more than two CPU's available experiment with using (almost) all of them and compare overall performance.

Additional background questions

The following questions are meant for extra training and get more insight in the concept of groundwater modeling.

1 Make a second scenario: increase the stationary groundwater recharge (RCH_L1.IDF) in the entire model area by a factor 1.2 (Note: be sure that you use the equation **C=1.2*A** in the iMOD calculator), thus simulating a possible future climatic change. Follow the procedure analog to increasing the rivers stages. Again, compare computed groundwater levels with those in the default situation ('MODEL25').
2 Explain the spatial pattern of the increase in groundwater levels. In which areas is it more and in which areas less? Examine this by exploring other IDFs e.g. RIV, DRN.
3 Why is a converged model not necessarily a correct model?
4 Consider a drain pipe, ending in a surface water channel. Will the drain pipe keep draining groundwater into the channel if the surface water stage rises above the drain pipe elevation?

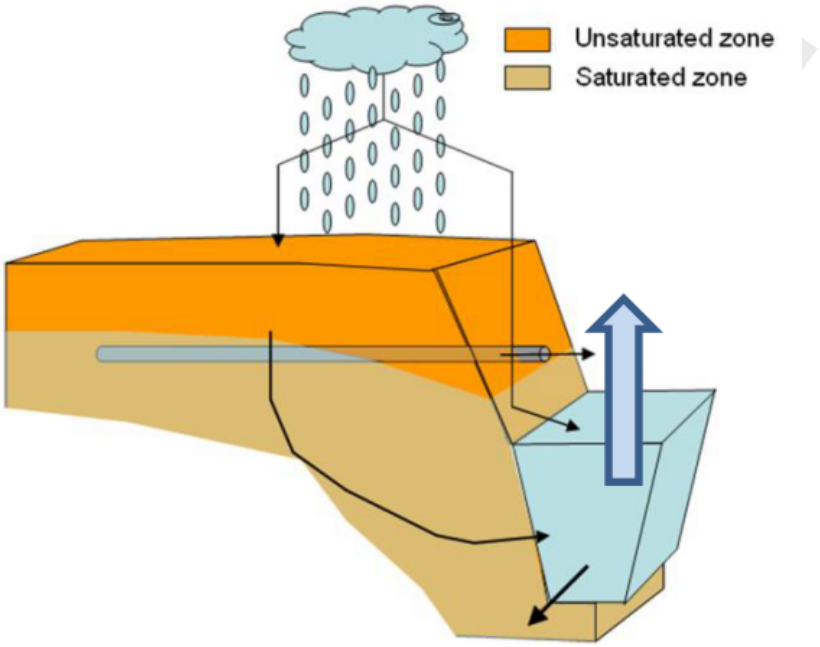


Figure 11.88: *Drain pipe ending in a surface water channel.*

5 In MODFLOW, each package treats an inflow or outflow, resulting from a boundary condition, as an external source or sink (Q_ext). It does not consider possible interactions with other packages / boundary conditions. Knowing this, what will happen in MODFLOW if the surface water stage (RIV) rises above the drain pipe elevation (DRN)?
6 How should DRN_LEVEL_L1.IDF be adjusted to prevent this?
7 OLF_L1.IDF represents the surface elevation, referenced to sea level. Using this OLF file and the simulated heads, calculate the (steady state) groundwater depth in the MODEL25 situation.
8 Subtract the drainage elevation from the surface elevation, and compare the resulting drainage depth map to e.g. Google Maps for the area South East of the city of Utrecht. What is the drainage depth that occurs most often in built-up areas?
9 Compare the drainage elevation in the built-up areas in the north east of the model to the groundwater head. What is the general picture?
10 Compare the drainage elevation in the built-up areas in the (south)west of the model to the groundwater head. What is the general picture?
11 With which of the statements below do you agree most? Motivate.

- ◇ This model is suitable to determine which model cells in built up areas have too high groundwater levels.
- ◇ This model is suitable to determine which towns and villages are dependent on drainage sys-

tems to prevent too high groundwater levels.

11.7 Tutorial 7: Interactive Pathline Simulation

This tutorial gives a brief overview of the capabilities of the Interactive Pathline Simulation Tool (IPS). It allows the user to demonstrate and examine the flow behaviour of the groundwater system in an interactive manner. It is advised to get familiar with description of the IPS first, see section 7.15 that handles all the functionalities, for which a few are outlined in this tutorial.

Outline

This is what you will do:

- ◇ Load an existing model simulation into the IPS tool;
- ◇ Define starting points;
- ◇ Start a pathline simulation and represent them in different ways;
- ◇ Filter pathlines depending on their type of capture;

Required Data

For this tutorial you need the following iMOD Data Files/folders:

- ◇ The entire folder (and subfolders) in {path of tutorialfolder} \TUT_IPS, containing:
  - ▫ BOUNDARY.IDF – model boundary;
  - ▫ SURFACE_LEVEL.IDF – uppermost elevation of the model;
  - ▫ AQUITARD_TOP.IDF – top of the intermediate aquitard;
  - ▫ AQUITARD_BOT.IDF – bottom of the intermediate aquitard;
  - ▫ BEDROCK.IDF – bottom of the underlying aquifer;
  - ▫ . \RESULTS \IMODPATH.RUN – imodpath runfile (see section 8.6.6) referring to all result files that are needed for the particle tracking simulation.

Getting Started

1. Open from {*installfolder*} \tutorials \TUT_IPS \RESULTS \head the file head_steady-state_l1.idf and display heads.
2. Select Interactive *Pathline Simulation* and Start IPS.
3. Select from {*installfolder*} \tutorials \TUT_IPS \RESULTS \the file IMODPATH.RUN.

iMOD is reading the content of the RUN file and starts the 3-D tool.

Define the Starting Points

Position your Starting Points on the level of the calculated groundwater:

4. Select the option *File*.
5. Click the *Open* button and select from {*installfolder*} \tutorials \TUT_IPS \RESULTS \head \the file head_steady-state_l1.idf and click the *OK* button.

6 Click the *Properties* button ( ) next the input field it displays:

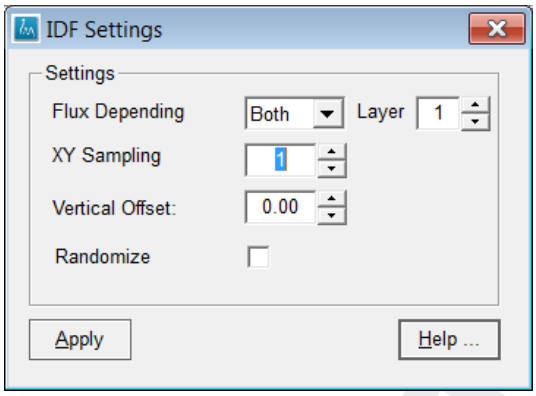


Figure 11.89: *The 'IDF Settings' window allows specifying starting positions of particles using an existing IDF (e.g. calculated groundwater heads) as a reference.*

Flux depending is incorrect here, but means that you can use the positive or negative values on the IDF to start particles. Leave it like this.

7 Enter for the *XY-Sampling* 5 to decrease the number of particles (every 5 cells there will be a starting point).
8 Leave the option Vertical Offset that can be used to position particles a bit higher or lower than the selected values in the IDF file.
9 The option *Randomize* places the particles at random locations, activate that option.
10 Click Apply.

To add this selection of particles to the simulation click the *Plus* button ( ), so:

11 Click the Plus button ( ), you'll notice that 1813 particles are added.
12 Click the Glass button ( ) to display the particles in red.
13 Go to the tab *IDF's* and select the BEDROCK IDF only than you'll see the particles.

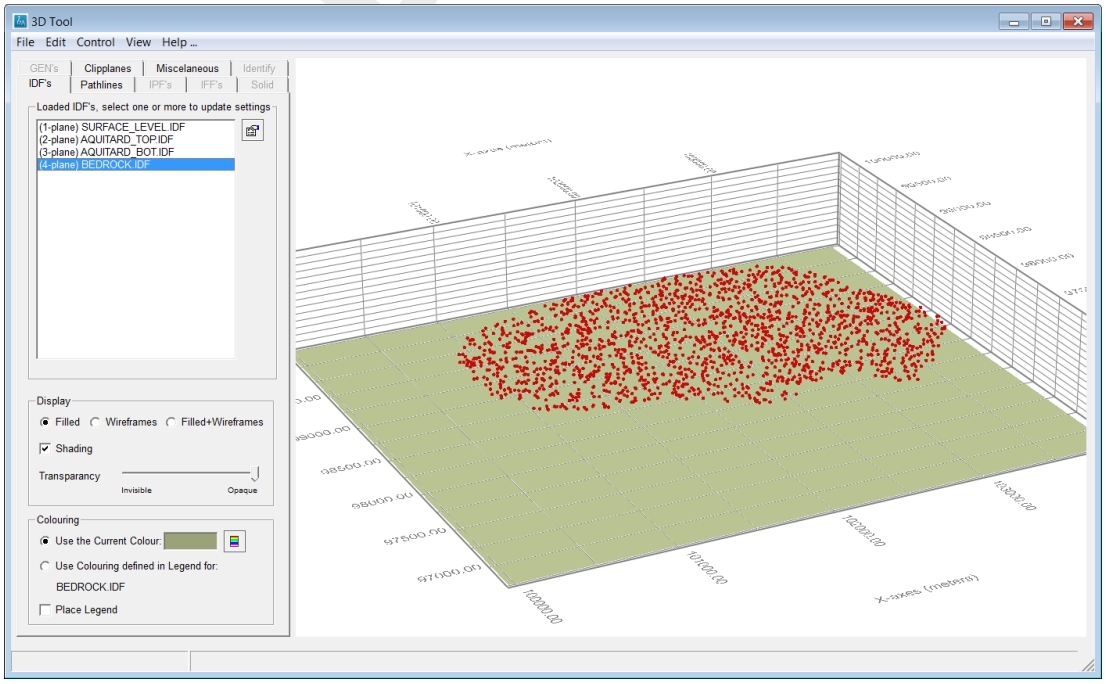


Figure 11.90: *Randomly generated particles (in red).*

Starting the Particle Tracking

To start the particles simulation follow the next steps:

14 Select the tab *Pathlines*.
15 Select the option “Repeat when trapped” , in this way particles are restarted automatically whenever they might be captured by a weak or strong sink.
16 Click the *Start* button.

Pretty nice, isn’t it? You can turn the particle start location on and off by clicking the *Glass* button

17 Click the *Glass* button ( ) on and off.
18 You can rotate, zoom and pan during the simulation by the regular mouse functionalities for the 3-D tool, try it!

You can pause the simulation by pressing the Pause button and restart it by clicking the *Continue* button, so:

19 Click the *Pause* button, notice that you can change all other settings in the 3-D tool as well, just experiment with that.
20 Click the *Continue* button to restart the particle simulation.

Enhance the Appearance

The appearance of the particle is given by lines or point, standard points are selected, these are the most efficient, but lines give a more realistic view, so:

21 Click the *Lines* option.

We can configure the appearance and settings of the current particle set, so:

22 Click the *Configure Particles. . .* button.

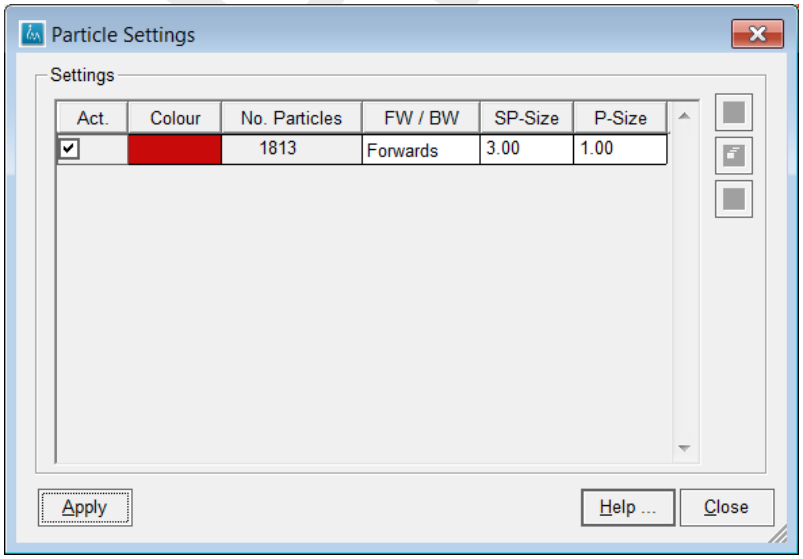


Figure 11.91: *The ’Particle Settings’ window that appears after clicking the ’Configure Particles...’ button in the ’Pathlines’ tab of the 3D Tool.*

You are now in the *Configure Particles* settings window. We alter some settings such as activation, colour, size and direction of the particles. In fact it is possible to have different groups of particles that

go forward and backward.

23 Click the Colour (red) field and change the colour into blue and click OK.
24 Increase the size of the line by specifying a 3.0 in P-Size.
25 Click *Apply*.

The appearance of the particles is updated immediately.

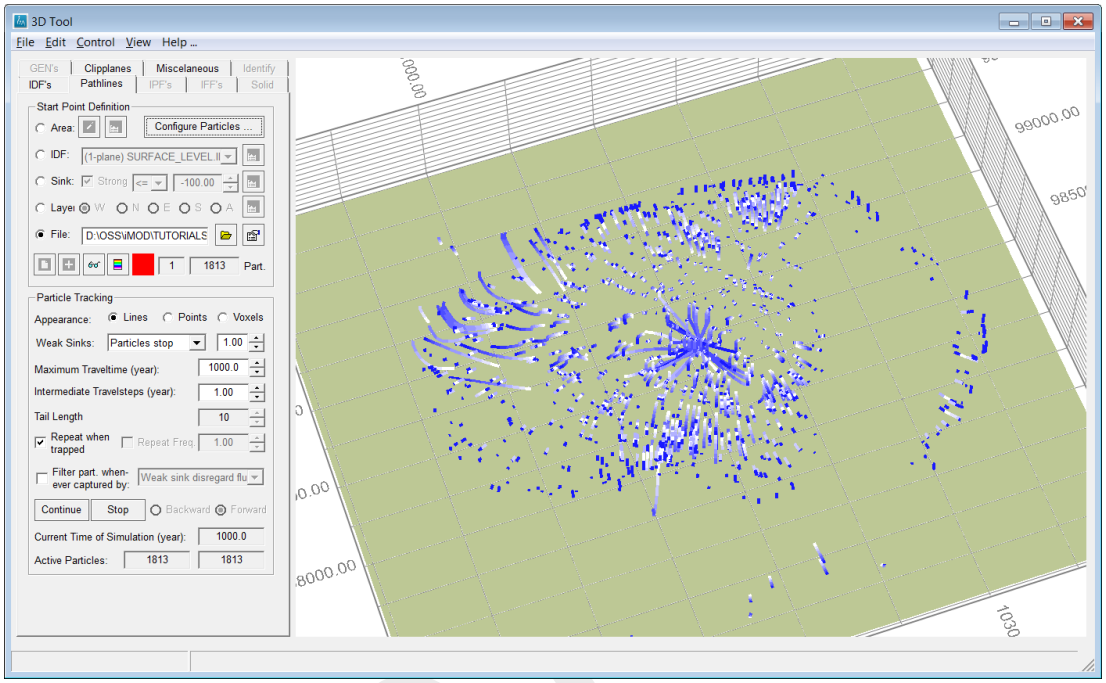


Figure 11.92: *Screen shot of a particle simulation in the 'Pathline' tab of the 3D Tool.*

Let us add another group of particles.

26 Select from the Start Point Definition the option *Sink*.
27 Click the *Properties* button ( ) associated with this *Sink* option.

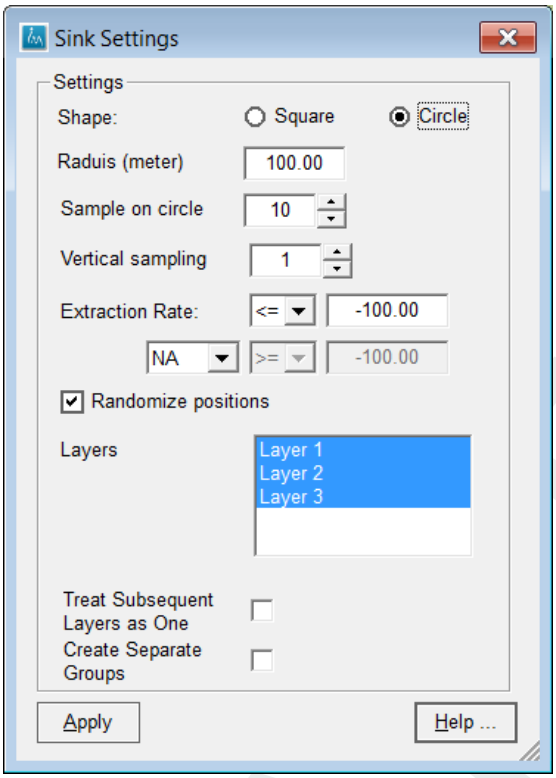


Figure 11.93: *The 'Sink settings' window appears after selecting the 'Sink' option and clicking the 'Properties' button in the 'Start Point Definition' part of the 'Path-lines' tab in the 3D Tool.*

Starting Points around Sinks

Here you can specify how the particles need to be positioned in combination with sinks (a sink is a source that takes water out of the model, can also be a river, drain of well). There is a single well in layer 3 that extract 500 m^3/d, so we would like to put particles around that well, so:

28 Enter a value of 5 for *Radius*, the cell size is 10 meter and we want to have the particles all in the strong sink.
29 Enter 10 for *Vertical Sampling*, so we have 10 intervals in the vertical and 10 per interval on a circle, so in total 10x10=100 particles.
30 Click the *Apply* button.

If the *Plus* button has been greyed out, that means that there is a simulation active that need to be stopped first, so:

31 Click the *Stop* button.
32 Click the *Plus* button and notice that 2 groups of particles are now available and in total we have 1913 particles.
33 Click the *Glass* button to examine the particles.

Change of Particle Tracking Directions

The particle from the well should migrate backwards, so let us change that:

34 Click the *Configure Particles...*
35 Select the option *Backward* for the last group.

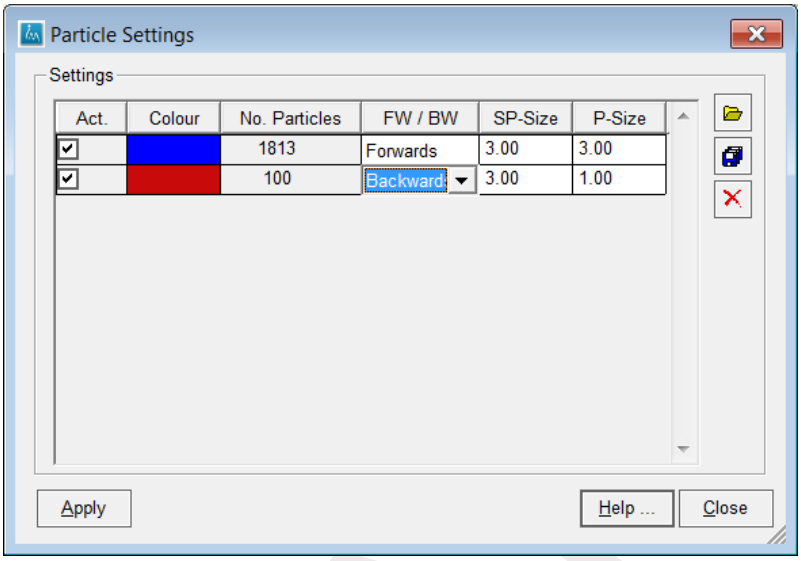


Figure 11.94: *Setting the direction of a group of particles to 'Backward'.*

36 Click the Apply button.
37 Click the Start button.

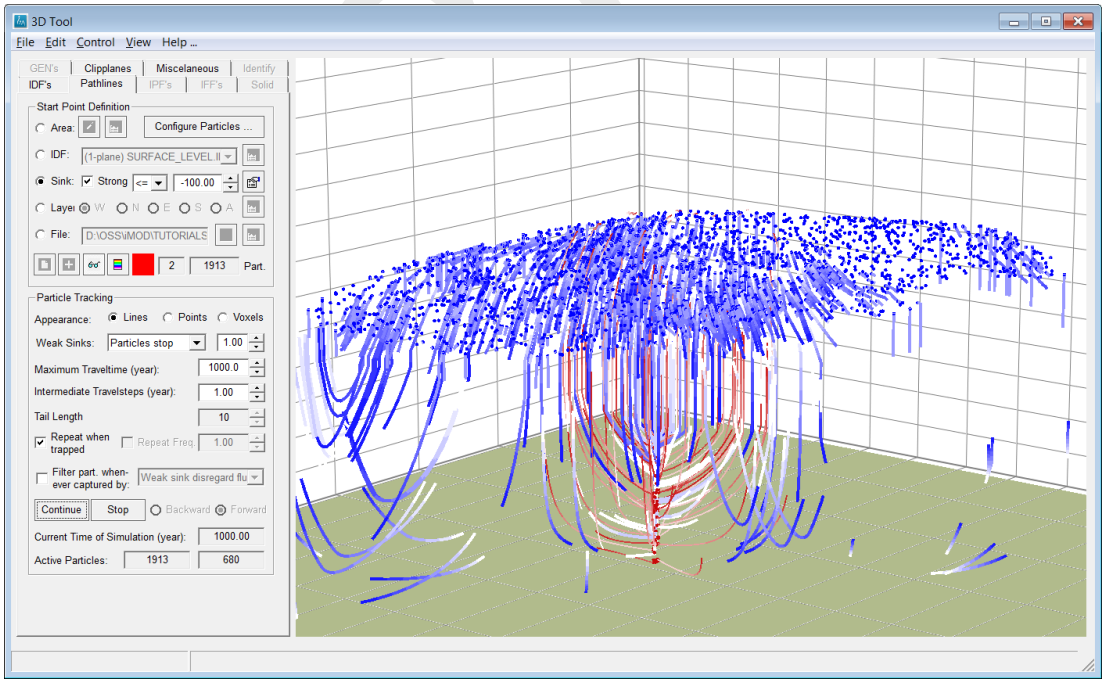


Figure 11.95: *Simultaneous pathlines simulation for two groups of particles, each having its own colour.*

The blue particles can be turned off temporarily:

38 Click the *Configure Particles...* button.

39 Deactivate the first particles group. **Note:** The particles settings can be saved from this window using the *Save As* button, allowing complex particles settings to be re-used, also in a different extent.
40 Click the *Apply* button.
41 Click the *Stop* button to stop the current simulation.
42 Click the *Start* button to start with only the particles from the well.

We can increase the density of the simulation by release more particles after each other, before they actually terminate, so:

43 Click the *Stop* button to stop the current simulation.
44 Deselect the option “Repeat when trapped”
45 Select the option “Repeat Freq.”
46 Click the *Start* button.

You see that after a full length of a particle (in this case 10, namely the “tail length”), another set of particles is released. You can try to see the effect of changing the “Release Freq.”

47 Click the *Spinner* () next to the “Release Freq.” input field to see the effect of this. Bigger values will allow more time in between the next particles, as smaller values will release another particle quicker.

Filtering of Particles

The final thing we do it to filter out particles during a simulation.

48 Click the *Configure Particles...* button.
49 Deselect the second particles group.
50 Select the first set of particles again.
51 Click the *Apply* button.
52 Deselect the option “Release Freq.”.
53 Select the “Repeat when trapped” again.
54 Click the “*Filter part. whenever captured by*” option and select the option “Strong Sink”.
55 Click the *Start* button to start the simulation.

Now, only the particles that are captured by a strong sink will be repeated. In this way a sense of a capture zone if created. Okay. That it for today, so close the IPS by:

56 Selecting the option “File” from the main menu and than “Close 3-D Tool.”.

11.8 Tutorial 8: Surface Flow Routing (SFR) and Flow Head Boundary (FHB) Package

This tutorial gives an introduction to a steady-state, surface water routing package (see section 12.28, and Prudic *et al.* (2004)). See for more detailed references regarding *ISG-Edit* (see section 6.10.3) and the ISG-file format regarding the SFR package (section 9.9). The tutorial also outlines the use of the FHB package (section 12.26) which facilitates a combination of constant head- and constant flux boundaries.

Outline

This is what you will do:

- ◇ Define a simple, single-layer, steady-state model and head- and flux boundaries using the FHB package;
- ◇ Define the outline of the stream network;
- ◇ Set the characteristics of each stream and define the connections within the stream network;
- ◇ Start the SFR simulation and examine the outcome.

Required Data

For this tutorial you need the following iMOD Data Files/folders:

- ◇ The entire folder (and subfolders) in {path of tutorialfolder}\TUT_SFR, containing:
  - .\DBASE\TOP.IDF – the uppermost elevation of the model;
  - .\DBASE\BND.IDF – boundary conditions of the model (to be created in tutorial);
  - .\DBASE\FHB.IDF – constant head and constant flow boundary of the model (to be created in tutorial);
  - .\DBASE\SFR.ISG – ISG with surface flow routing information (to be created in tutorial);
  - .\DBASE\CROSSSECTION.CSV – CSV file with a complex cross-section;
  - .\MODEL.PRJ – model project file;

Getting Started

1. Start iMOD.
2. Select the option *Create a New iMOD Project*.
3. Click the *Start* button.
4. Go to *View* in the menu bar and select the *iMOD Manager* (or use the shortcut *Ctrl+M*), or click the *iMOD Manager* button ( ) from the main window to start the *iMOD Manager* window.
5. Click the *Open IDF* button ( ) from the *Maps* tabs on the *iMOD Manager* and open {path of tutorialfolder}\TUT_SFR\DBASE\TOP.IDF.

This IDF describes the upper most elevation of the model, the top of our single layered aquifer that declines from 512 m+MSL in the west towards 505 m+MSL in the east. We use this IDF to create the boundary IDF.

Creating the Boundary File BND.IDF

6. Click the *Calculator* ( ) on the *iMOD Manager* in the *iMOD Manager* window to start the *Map Operations* window.
7. Enter the output name “{path of installfolder}\IMOD_USER\DBASE\TUT_SFR\BND.IDF” at *Map C*.
8. Make sure the *Formulae* is “C=0*A”.
9. Select the option *Map A* at the section *Select the extent for which the computation applies*.
10. Click the *Compute ...* button.

We will use this BND.IDF file to specify how the boundary conditions need to be. On the west we apply an open boundary condition with constant heads (511 m+MSL, that is 1 meter below the surface level

TOP.IDF). On the east we apply an open boundary condition with a constant outflow flux boundary (-950 m^3/d; a negative number will be used to take water out of the groundwater system, use a positive number to insert water to the groundwater system instead). We will modify the BND.IDF via IDF-Edit, which has been part of Tutorial 2, make sure you have applied this tutorial already.

11 Click the menu option *View*, *Show IDF features* and then *IDF Raster Lines* to display the gridlines of the IDF files.
12 Click the right mouse button and the select from the dropfown menu, the option *IDF Options* and then the option *IDF Edit ...* to start the *IDF-Edit* window.
13 In the 'Selection'-tab click the *Draw ...*-button to start the *IDF Edit Draw* window.
14 Move your mouse in the graphical canvas (note that the cursor symbol changes) and drag, **while holding your left-mouse button**, all model cells in the left column of our model.

You can release the left mouse button to position the mouse on a different location without selecting the underlying cells. To continue selecting cells, you need to press the left mouse button again. If you need to remove some of the selected cells, click the *Remove Cells* from the *IDF Edit Draw* window. Restart selecting cells, click the *Add Cells* again from the *IDF Edit Draw* window. Below is an image of the final selected cells.

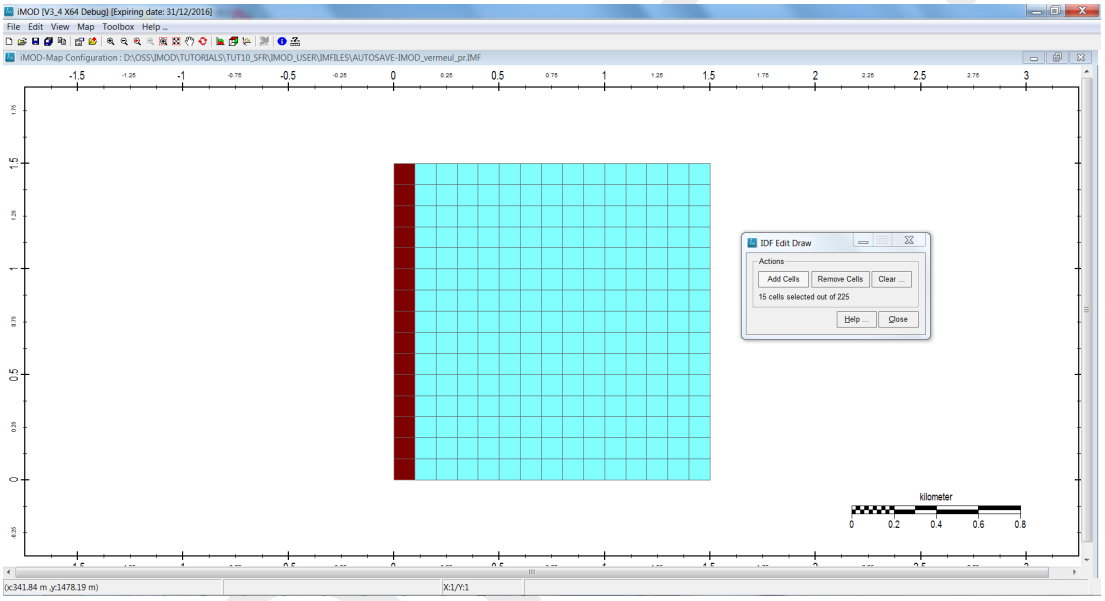


Figure 11.96: *Image after selecting all cells of the most left column of the model.*

Now we are going to change the values for those cells.

15 Click the *Close* button on the *IDF Edit Draw* window to leave the mode to select cells.
16 Click the *Calculate* option from the *IDF Edit* window to start the *IDF Edit Calculation* window.
17 Selection the option *New Value* and enter the value "-2" in the input field next the the "=" sign.
18 Click the *Calculate* button to assign those values to the BND.IDF file.
19 Click the *Close* button.
20 Click *Yes* to leave the editing mode.

A constant head boundary is specified by a negative number, so all values less than zero are appropriate. However, whenever the FHB package is used and a constant head boundary needs to be combined with a constant flux boundary, it is necessary to specify a -2 for constant head cells. For constant flux boundaries we need to specify a +2. Let's do that for the right boundary.

21 Click the *Clear* button and agree to the question whether you are sure to delete the current selection.
22 Click the option *Draw* to start the *IDF Edit Draw* window.
23 Move your mouse in the graphical canvas and drag, **while holding your left-mouse button**, all

model cells in the right column of our model.
24 Click the *Close* button on the *IDF Edit Draw* window to leave the mode to select cells.
25 Click the *Calculate* option from the *IDF Edit* window to start the *IDF Edit Calculation* window.
26 Selection the option *New Value* and enter the value “2” in the input field next the the “=” sign.
27 Click the *Calculate* button to assign those values to the BND.IDF file.
28 Click the *Close* button.
29 Click *Yes* to leave the editing mode.

We’re almost done, we only need to change all the zero values in the BND.IDF to be 1.

30 Click the *Clear* button and agree to the question whether you are sure to delete the current selection.
31 Click the option *Select* to start the *IDF Edit Select* window.
32 Select “BND.IDF” at *IDF-File:*; select “=” at *Logic:* and select “0.0” at *Value:*.
33 Click the *Get Selection* button and notice all rows are selected for column 2 up 14.
34 Click the *Close* button.
35 Click the *Calculate* option from the *IDF Edit* window to start the *IDF Edit Calculation* window.
36 Selection the option *New Value* and enter the value “1.0” in the input field next the the “=” sign.
37 Click the *Calculate* button to assign those values to the BND.IDF file.
38 Click the *Close* button to leave the *IDF Edit Calculation* window..
39 Click *Yes* to leave the editing mode.
40 Click the *Close* button to leave the *IDF Edit* window.

We’re finished !

Specify the characteristic of the boundary

We need two other files that specify the actual constant head values and the constant flux rates. For this we need to copy the BND.IDF into a file called FHB_H.IDF, give the values that are -2 (in BND.IDF that represent a constant head boundary) the value 511 m+MSL.

Note: To estimate the flux over the edge of model, apply Darcy’s Law. In our case we have a head gradient of $\Delta h = 7$ m over distance $d = 1500$ m, a permeability of $k = 60$ m/d and an average thickness of $T = 34$ m, so the horizontal conductance $c = k \times T = 2040$ m^2/d. Filling these in in Darcy’s Law we come up with:

$$\begin{aligned} q &= k \times T \frac{\Delta h}{d} \\ &= 60 \times 34 \times \frac{7}{1500} \\ &= 9.52 \mathrm{m}^2/\mathrm{d} \end{aligned} \tag{11.1}$$

The cell width is 100 m, so the total volume of water is $Q = 952$ m^3/d.

Secondly, we need to copy the BND.IDF to a file called FHB_Q.IDF and give the values that are +2 (in BND.IDF that represent a constant flux boundary) a value of -950 m^3/d (constant flux boundary). It is more-or-less a repetition of the previous steps, but let’s dot that together.

41 Click the *Calculator* () on the *iMOD Manager*in the *iMOD Manager* window to start the *Map Operations* window. Make sure you have selected the BND.IDF in the *iMOD Manager*.
42 Enter the output name “{path of installfolder}\IMOD_USER\DBASE\TUT_SFR\FHB_H.IDF” at *Map C*.
43 Make sure the *Formulae* is “C=A”.
44 Select the option *Map A* at the section *Select the extent for which the computation applies*.
45 Click the *Compute ...* button.

Let’s create the IDF file for the constant flux boundary as well.

46 Click the *Calculator* () on the *iMOD Manager* in the *iMOD Manager* window to start the *Map Operations* window. Make sure you have selected the BND.IDF in the *iMOD Manager*.
47 Enter the output name “{path of installfolder}\IMOD_USER\DBASE\TUT_SFR\FHB_Q.IDF” at *Map C*.
48 Make sure the *Formulae* is “C=A”.
49 Select the option *Map A* at the section *Select the extent for which the computation applies*.
50 Click the *Compute ...* button.

Enter the IDF Edit to change the values.

51 Click the right mouse button and the select from the dropfown menu, the option *IDF Options* and then the option *IDF Edit ...* to start the *IDF-Edit* window.
52 Click the option *Select* to start the *IDF Edit Select* window.
53 Select “BND.IDF” at *IDF-File:*; select “=” at *Logic:* and select “-2.0” at *Value:*.
54 Click the *Get Selection* button and notice all rows are selected for column 1.
55 Click the *Close* button.
56 Click the *Calculate* option from the *IDF Edit* window to start the *IDF Edit Calculation* window.
57 Selection the option *New Value* and enter the value “511.0” in the input field next the the “=” sign.
58 Make sure the FHB_H.IDF is selected in the *Available IDF Files* at the *Assign Value to* section.
59 Click the *Calculate* button to assign those values to the FHB_H.IDF file.
60 Click *Yes* to leave the editing mode.
61 Click the *Close* button to leave the *IDF Edit Calculation* window.

Now for the constant flux boundary on the right.

62 Click the option *Select* to start the *IDF Edit Select* window.
63 Select “BND.IDF” at *IDF-File:*; select “=” at *Logic:* and select “2.0” at *Value:*.
64 Click the *Get Selection* button and notice all rows are selected for column 15.
65 Click the *Close* button.
66 Click the *Calculate* option from the *IDF Edit* window to start the *IDF Edit Calculation* window.
67 Selection the option *New Value* and enter the value “-950.0” in the input field next the the “=” sign.
68 Make sure the FHB_Q.IDF is selected in the *Available IDF Files* at the *Assign Value to* section.
69 Click the *Calculate* button to assign those values to the FHB_Q.IDF file.
70 Click the *Close* button to leave the *IDF Edit Calculation* window.
71 Click *Yes* to leave the editing mode.
72 Click the *Close* button to leave the *IDF Edit* window.

Done regarding the boundary definition. We need to add this to the model via the *iMOD Project Manager*. For the Tutorial we have done already a small amount of work to fill in an iMOD Model Project file (*.PRJ). We need to add the BND.IDF, FHB_H.IDF and FHB_Q.IDF to the current PRJ file. We will do this after we have created the input for the SFR package.

Creating the SFR Package

In our model a stream flows from the west towards the east and splits halfway into two separate streams (see Figure 11.97). We will model this stream in iMOD. First we need to create an ISG file that is capable of generating the SFR input.

73 Select from the main window, the option *Edit*, *Create Feature*, *ISGs* and then *SFR Applicable ....*
74 Enter the ISG file name “{path of installfolder}\IMOD_USER\DBASE\TUT_SFR\SFR.ISG” in the *Save* window.

The entered ISG file will be added to your *iMOD Manager*, but since it will be completely empty, you’ll not see anything appearing on the screen. So, let us create the stream network and the corresponding characteristics.

75 In the *iMOD Manager* select and draw () the file FHB_H.IDF.

76 Select the *Legend* button () from the *iMOD Manager* to start the *Legend* window.
77 Deselect the option () to ignore the colouring of the FHB_H.IDF file.
78 Click the *Apply* button to leave the *Legend* window.
79 Select from the main window the option *View*, *Show IDF Features* and then the option *IDF Indices* to display the cell indices (row-column numbers) of the IDF file FHB_H.IDF. This is handy to produce the ISG file in the next steps.
80 Add the created SFR.ISG file in the *iMOD Manager* to the current selection of files: use the key "Ctrl" and click your left mouse button on the SFR.ISG file to this file to the selection. So both SFR.ISG and FHB_H.IDF should be selected now.
81 Select from the main window, the option *Map*, *ISG options* and then *ISG Edit ...* to start the *ISG Edit* window.

So because we cannot visualize the file SFR.IDF yet (it is still completely empty) we also selected the FHB_H.IDF in the *iMOD Manager* to use it as a guide to draw the stream network in the next steps.

Drawing the Stream Network

We are ready now to start drawing our stream network.

82 Select the *Draw* option () from the *ISG Edit* window.
83 Start drawing the first stream from west to east, we start by clicking the left mouse button at the cell (8-1) (row are numbered from top to bottom); and then make a straight line and click the left mouse button at cell (8-7).
84 Right mouse click to stop drawing.

You'll notice that a stream has been created in the menu field on the *ISG Edit* window called "Segment_1". This was our first stream, let's create another one.

85 Select the *Draw* option () from the *ISG Edit* window.
86 Start drawing the second stream by clicking your left mouse button at cell (8-7), towards the north and click the left mouse button at cell (4-7), further up north-east click at cell (3-10) and finally towards the east, mouse click at cell (3-15).
87 Right mouse click to stop drawing.

and the final one:

88 Select the *Draw* option () from the *ISG Edit* window.
89 Start drawing the second stream from cell and click the left mouse button at cell (8-7), towards the south and click at cell (12-7 , further down south-east click at cell (13-10) and finally towards the east click at cell (13-15).
90 Right mouse click to stop drawing.
91 Select all stream from the menu field, i.e. "Segment_1"; "Segment_2" and "Segment_3".
92 Check the options *Nodes*; *C.Section*; *Seg.Nodes*; *Clc.Pnts.* and *Direction* in the *Show* section at the bottom of the *ISG edit window*.
93 Click the *Update* button.

When you did it right (I'm sure you did), the following stream network should be displayed on your graphical canvas.

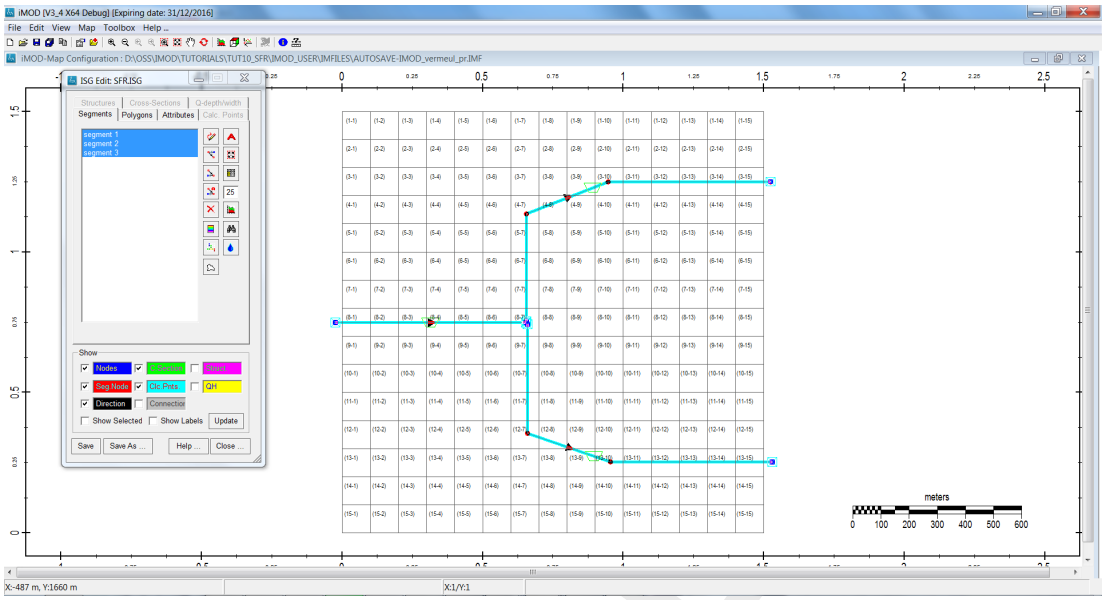


Figure 11.97: *Image of the 3 added ISG segments after turning on the labels Nodes, C.Section, Seg.Nodes, Clc.Pnts. and Direction.*

So, what do we see?

First of all, each stream consists of segment nodes (red dots, *Seg.Nodes*). Each stream contains a single cross-section (green polygons, *C.Section*), and a calculation point at the beginning and end of a stream (blue rectangle with a cross, *Clc.Pnts.* and the start and end of a stream (blue dot, *Nodes*). Furthermore, as we draw the stream, the order in which the coordinates of each stream are entered by clicking the left mouse button, determines the direction of the flow (black arrow, *Direction*). In our case, the order of the coordinates is such that water is flowing from the west to the east and splits at the bifurcation in a north- and south branch. Finally, all segments are selected and therefore those are highlighted in cyan.

Characterizing the Stream Network

The next thing to do is to characterise the stream with appropriate water levels, bottom height, cross-sections and so on.

94 Select "Segment_1" in the 'Segment'-TAB of the 'ISG EDit: SFR.ISG' window.

95 Click the *Attributes* button ( ) to open the *ISG Attributes* window, stretch the window a bit such that the entire table is visible.

We will enter some data in the table. First of all, we will apply this SFR to a steady-state model, so the date and time are irrelevant in this case. We will leave it as it is.

96 Enter "511" for the *Water Level* (column 3).
97 Enter "510" for the *Bottom level* (column 4).
98 Enter "20" for the *Stream width* (column 5).
99 Enter "1" for the *Bed thickness* (column 6).
100 Enter "1" for the *Bed Permeability* (column 7).
101 We will define a rectangular cross-section: select "1" from the drop down menu for *Calc Opt.* (column 10).
102 Enter "13.9" for the *Q flow* (column 12). There is a flow rate entering from the west into the stream of 13.9 m^3/s, that is 1.2 million m^3/day.

We accept the default values for the remaining columns; the table should look similar to the figure below.

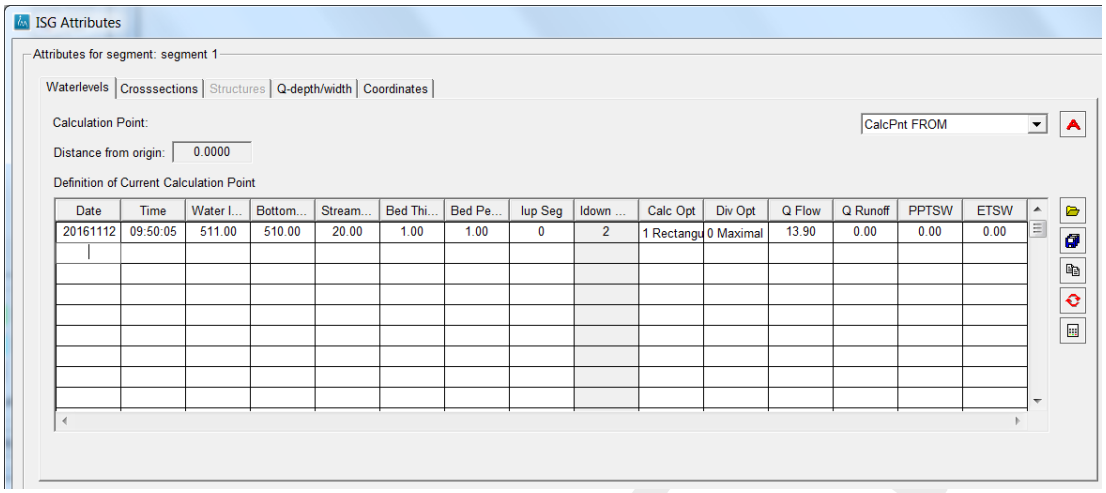


Figure 11.98: *The 'Waterlevels'-tab in the 'ISG Attributes' window for the Calculation point 'FROM' for segment 1.*

The next thing is to enter data for the downstream node.

103 Select the "CalcPnt TO" from the drop down menu *Calculation Point:*.
104 Enter "508" for the *Water Level* (column 3).
105 Enter "507" for the *Bottom level* (column 4).
106 Enter "20" for the *Stream width* (column 5).
107 Enter "1" for the *Bed thickness* (column 6).
108 Enter "1" for the *Bed Permeability* (column 7).

We accept the default values for the remaining columns.

Even though we specified the width and depth of the stream already on the tab *Waterlevels*, we need to specify the Manning's Resistance Coefficient *MRC* for the cross-section. In case you need to use a more sophisticated cross-section, you can specify that in the table on the tab *Crosssections*. We modify the table such that it will align with our entered width ($w = 20$ m) and maximal depth ($d = 2$ m), and assign a Manning's Resistance Coefficient $n = 0.03$.

Note: Manning's Resistance Coefficients n range roughly from n=0.01-0.06; some important values are given in table Table 11.5.

Table 11.5: *Manning's Resistance Coefficients n (source: http://www.engineeringtoolbox.com/mannings-roughness-d_799.html)*

Surface Material	Coefficient n	Surface Material	Coefficient n
Asphalt	0.016	Brick	0.015
Clay tile	0.014	Concrete (Cement)	0.012
Earth - clean	0.022	Earth channel - weedy	0.030
Floodplains - pasture	0.035	Floodplains - heavy brush	0.075
Metal - corrugated	0.022	Natural streams - clean / straight	0.030
Natural streams - major rivers	0.035	Natural channels, poor condition	0.060

109 Select the tab *Crosssections* from the *ISG Attributes* window.
110 Enter "-20" in column=1 and row=1 and 2 (*Distance*).
111 Enter "+20" in column=1 and row=3 and 4 (*Distance*).
112 Enter "2.0" in column=2 and row=1 and 4 (*Z*).
113 Enter "0.0" in column=2 and row=2 and 3 (*Z*).

114 Enter “0.03” in column=3 for all rows (*MRC*).

115 Click the *Redraw* button ( ) to update the display with your modified cross-section.

After you did it correctly, the *ISG Attributes* window should look like the figure below.

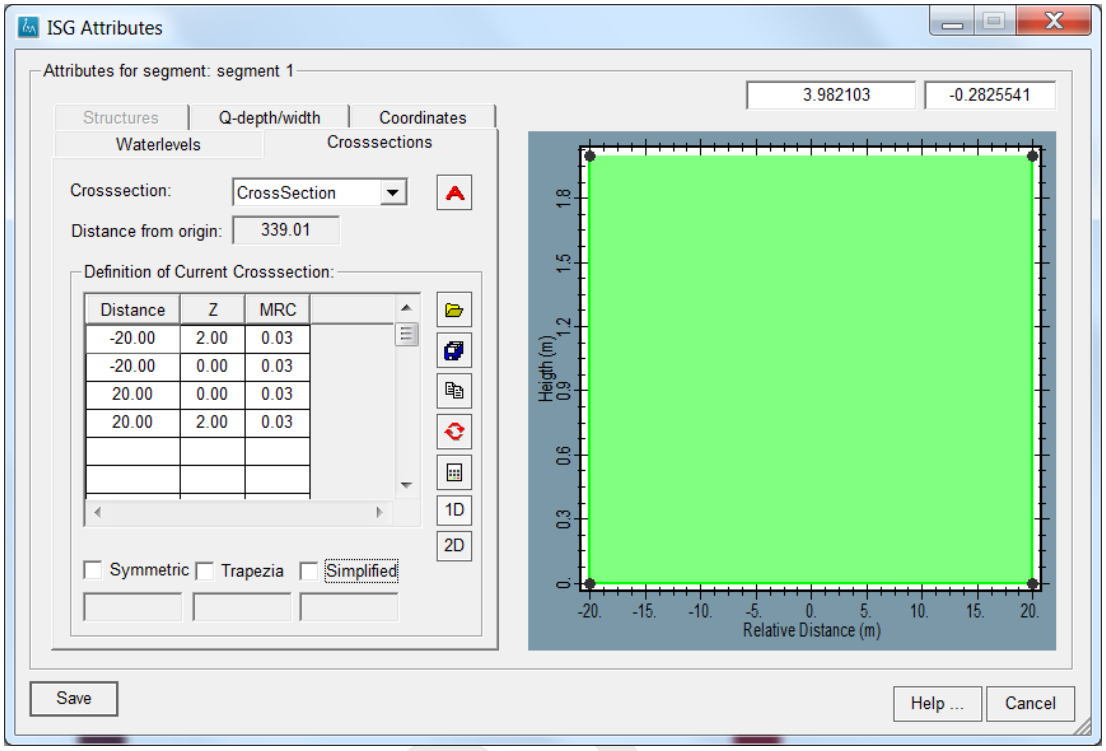


Figure 11.99: *The ’ISG Attributes’ window after entering the Manning’s Resistance Coefficient in the ’Crosssection’-tab for segment 1.*

Bear in mind that iMOD stores all modifications in memory. To actually save your modification on disk, you need to save your data explicitly, let’s do that.

116 Click the *Save* button to store our adjusted stream data and return to the *ISG Edit* window.
117 Click the *Save* button on the *ISG Edit* window to save your adjusted SFR.ISG to disk.
118 Click the *Yes* button to accept overwriting the existing SFR.ISG file.

Okay, one third done! You need to apply the modifications to the other segments by applying the previous steps (76 to 91) using the data from Table 11.6.

Table 11.6: *Parameters per Stream Segment.*

Up stream -	Down stream -	Water Level m+MSL	Bottom Level m+MSL	Stream Width m	Bed Thickness m	Bed Permeability m/d	Q Inflow m^3/s
1		511.0	510.0	20	1.0	1.0	13.9
	1	508.0	507.0	20	1.0	1.0	0.0
2		508.0	507.0	15	1.0	1.0	0.0
	2	504.0	503.0	15	1.0	1.0	0.0
3		508.0	507.0	5	1.0	1.0	0.0
	3	504.0	503.0	5	1.0	1.0	0.0

Note: To compute the total steady-state influx of 13.9 m^3/s, we apply the simplified Manning’s Equation for a rectangular stream. The gradient of Segment_1 is $S = \Delta h = 7$ m over $d = 1500$ m; its width is $w = 20$ m, its depth is $d = 1$ m and its roughness is $n = 0.03$. Filling these in in the simplified

Mannings' Equation:

$$\begin{aligned} Q &= \frac{1.0}{n} \times w \times y^{\frac{5}{3}} \times S^{\frac{1}{2}} \\ &= 45.5 \times 20.0 \times 1.0^{\frac{5}{3}} \times \left(\frac{7.0}{1500.0}\right)^{\frac{1}{2}} \\ &= 1239393 \ \mathrm{m}^3/\mathrm{d} \\ &= 13.9 \ \mathrm{m}^3/\mathrm{s} \end{aligned} \tag{11.2}$$

Note: A nice functionality to check whether you didn't make any typo's entering the data is the profile option; this functionality is also very handy when you need to inspect the result of the simulation (more on that later). Let's do that.

119 Click the *Profile* button ( ) on the *ISG Edit* window (**not** the same icon on the main iMOD window) to start the *ISG Profile* window.
120 Select "Segment_1" and "Segment_2" from the menu field at *Profile Along Selected Segments*.
121 Select "Bottom Level" from the drop down menu *Parameter A:* window.
122 Check the checkbox at *Parameter B:* and select "Water Level" from the drop down menu *Parameter B:* window.

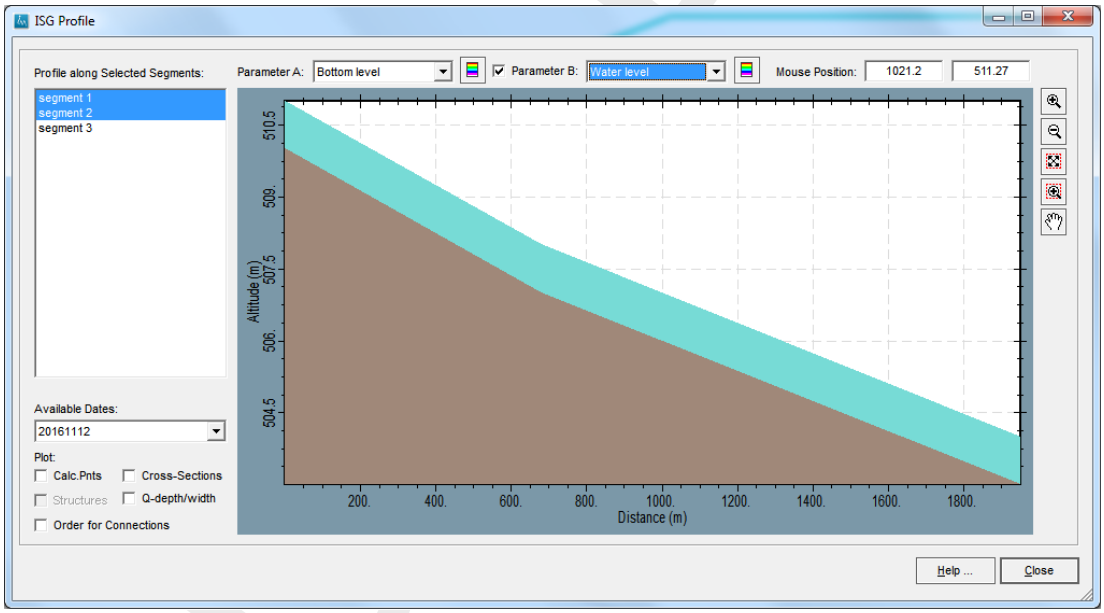


Figure 11.100: *The ISG Profile window facilitates inspecting ISG-variables of selected segments.*

123 Click the *Close* button to return to the *ISG Edit* window.
124 Click the *Save* button in the *ISG Edit* window to save your adjusted stream data to disk.
125 Click the *Yes* button to accept overwriting the existing SFR.ISG file.

Connecting the Stream Network

Now that we have given all streams their appropriate characteristics, the streams need to be connected. This can be done manually or automatically. When many streams are to be connected, this automatic option is very handy, it connects streams within a certain distance automatically. To give an idea of how easy streams can be connected manually, we will practice that right now.

126 Select the stream "Segment_1" on the graphical canvas by clicking your left mouse button near a node or in between two nodes. If the stream is selected it turns into a cyan-coloured line.

127 Click the option *Show Selected* to draw features on the selected stream only, just for reasons of simplification of the image on the graphical canvas.

128 Click the *Connect To* button ( ) and move your mouse toward “Segment_2” until it becomes a red line.

129 Click the left mouse button to indicate that this stream “Segment_1” will be connected to “Segment_2”. If you click next to a line (so no segment is selected), the connection will be removed.

130 Click the right mouse button to stop this selection process and return to the *ISG Edit* window.

131 Click the option *Connections* on the *ISG Edit* window to display the connection as a grey arrow.

132 Click the *Update* button to refresh the graphical canvas.

When you did it right (I’m sure you did), the your display should look similar to the figure below.

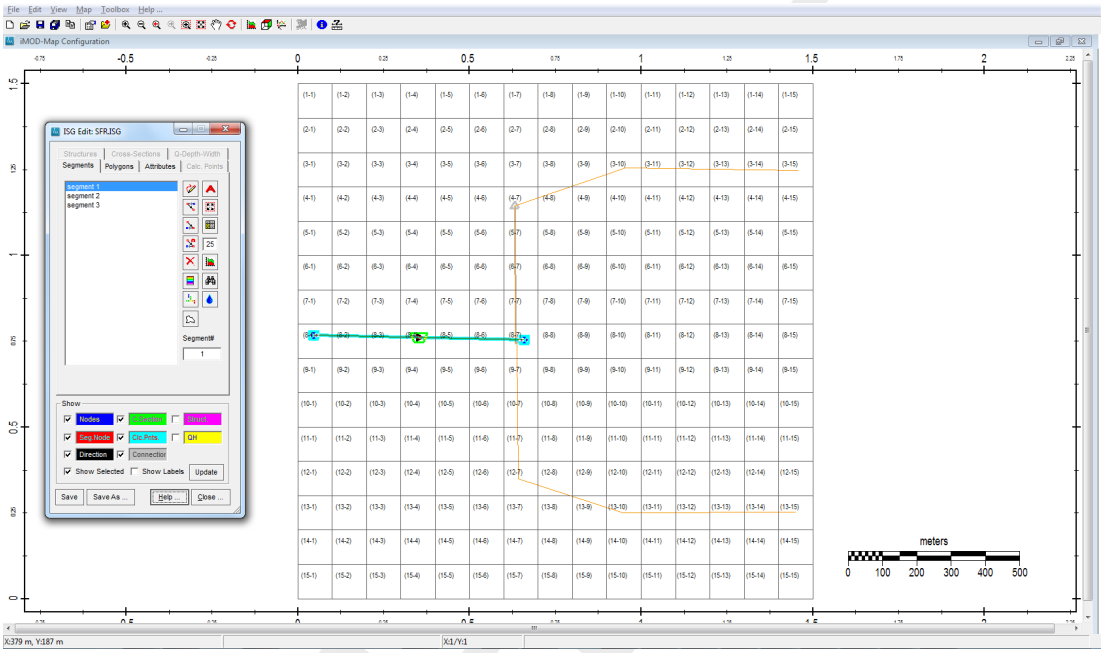


Figure 11.101: *Showing the connection (light grey arrow) to Segment 2 from Segment 1 (cyan line) by selecting the ’Connection’-option in the ’Show’-part of the ’ISG Edit’-window.*

So, “Segment_1” flow into “Segment_2” , but in fact it also flows towards “Segment_3”. We call this a diversion. In order to achieve this, we need to define for “Segment_3” that its inflow is diverted from “Segment_1”. We can do that interactively using the *Connect From* button ( ), or specifying this explicitly in *ISG Attributed* window.

133 Select “Segment_3” from the menu field on the *ISG Edit* window.

134 Click the *Attributes* button ( ) to start the *ISG Attribute* window.

135 Make sure the “ClcPnt FROM” from the menu field *Attributes for:*.

136 Enter the stream number “1” in column *Iup Seg*. This specifies the model to divert from segment Segment_1 (first in the segment list).

137 Select the option “-2” in column *Div. Opt.* This specifies the model to divert from Segment_1 as a fraction of the total inflow.

138 Enter the value “0.30” in column *Q Flow*. This specifies that the fraction of diversion is 0.30 of the outflow of Segment_1.

139 Click the *Save* button to store your modification in memory and return to the *ISG Edit* window.

It is not necessary to specify a diversion for Segment_2 as it automatically receives 100-30=70% of the outflow of Segment_1.

I think we’re done with this ISG, let’s quit the *ISG Edit* window.

140 Click the *Save* button in the *ISG Edit* window to save your adjusted stream data to disk.
141 Click the *Yes* button to accept overwriting the existing SFR.ISG file.

Defining the Model Project

Now that we have created all necessary packages for our model, let's get them together in a Model Project.

142 Select from the main menu the option *View* and then *Project Manager* to start the *Project Manager* window.
143 Click the *Open Projectfile* button ( ) and select the file {path of tutorialfolder} \TUT_SFR\MODEL.PRJ.
144 Click *OK*.

For a detailed exercise on how to create a Project-file from scratch, see Tutorial 4: Create your First Groundwater Flow Model, we will not exercise that here. The opened project file MODEL.PRJ contains all necessary parameter definitions.

The Project Manager will look as follows:

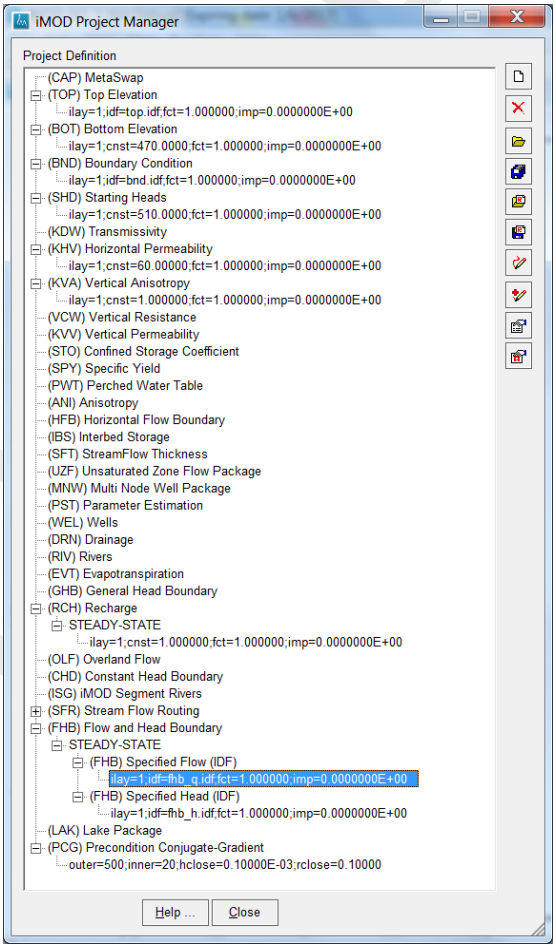


Figure 11.102: *The Project Manager after loading the project file MODEL.PRJ.*

We have a permeability of KHV=60 m/d, a bottom height of our aquifer of BOT=470 m+MSL, a uniform starting head of SHD=510 m+MSL, a uniform recharge of RCH=1 mm/d. Notice that we filled in the FHB- and SFR package for you.

Running the Model

Let's run this model.

145 Click the *Save As Run* button () to start the *Define Simulation Configuration* window.
146 Select the option *MODFLOW2005*.
147 Click the *Simulate* button and enter the following *.NAM file "{path of installfolder}\IMOD_USER\MODELS \TUT_SFR\TUT_SFR.NAM to export the model to Modflow2005 files and start the simulation. Note, that you need to create the folder TUT_SFR yourself first.

Note: There is an option to start a model simulation in the background so that you can continue working with iMOD once the model has been started. As this model is very, very small, we will run the model in the foreground and we have to wait until it has finished before we can continue working with iMOD - probably, reading this sentence was enough time for the model to be finished.

Once iMOD converts your model to MF2005 files, it creates a conventional ISG-file that can be used to transfer the results of the SFR package into iMOD. In this way we can use the existing functionalities in ISG-Edit (such as displaying time series, profiles) for the output of the model. Four items are converted to iMOD after the simulation has finished using the iMOD Batch function SFRTOISG (see section 8.3.8). This iMOD Batch function is part of the run-script (. \TUT_SFR \run.bat) and has been carried out already, so let's see some results.

148 Click the *OK* button once the simulation has been finished.
149 Click the *Close* button on the *iMOD Project Manager* window to close it.
150 Select from the main menu the option *Map*, *Add Map ...* and select the ISG file {path of installfolder} \MOD_USER \MODELS \TUT_SFR \BDGSFR \ISG \SFR.ISG.
151 Select from the main menu the option *Map*, *ISG Options* and than *ISG Edit ...* to start the *ISG Edit* window.

You probably notice that instead of three segments, we have now 35 segments.

152 Check *Seg. Nodes* whenever it has not been checked yet.
153 Click on the *Update* button.

The names of the individual segments still contain the original segment name, so it is easy to select all streams that belong to the same original stream.

154 In the 'ISG Edit' window select all items from the menu field that belong to the original Segment_1 and Segment_2.

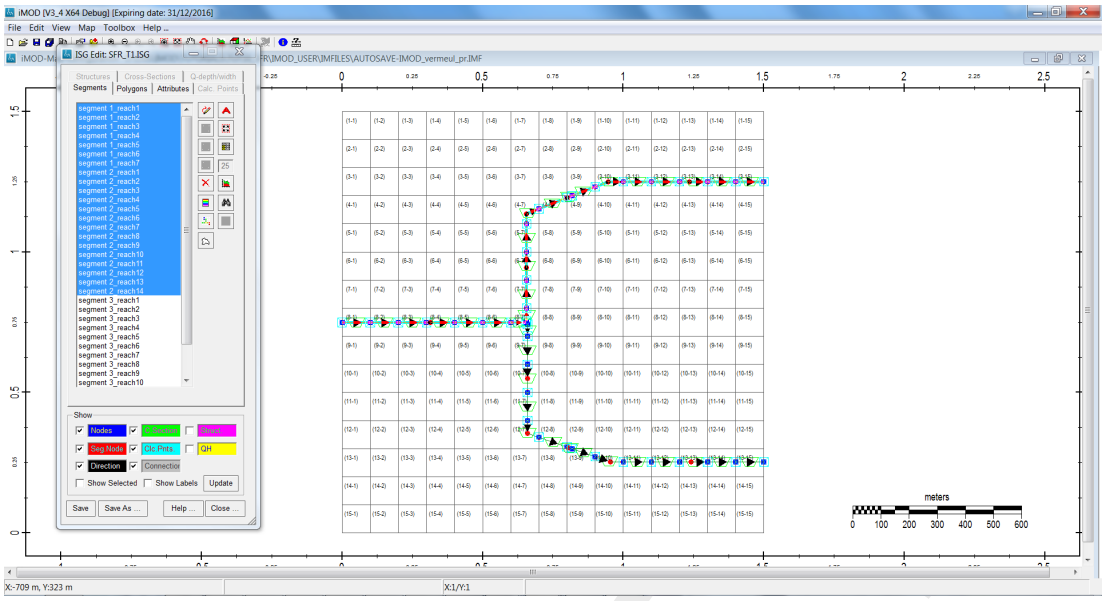


Figure 11.103: *Image after selecting all Segment 1 and 2 streams of SFR.ISG in the ISG Edit window.*

Now we want to see the decline of the water level, or change in discharge per segment.

155 Click the *Profile* button () to start the *Profile* window.

In this picture we observe that the computed surface water level is declining from west to east. We can see how the discharge distribution aligns with our predefined diversion fractions.

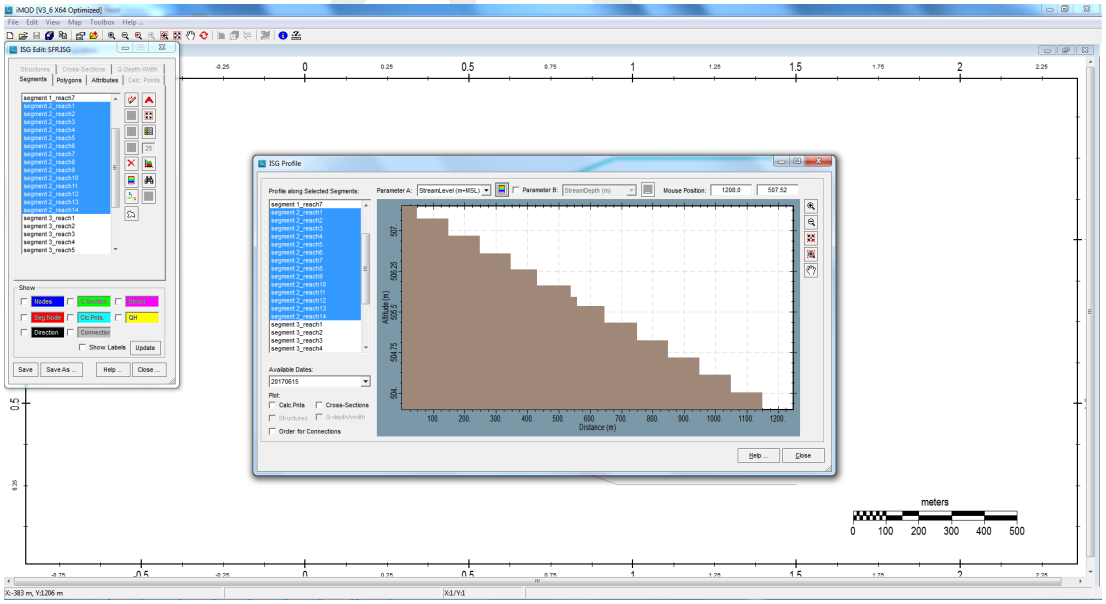


Figure 11.104: *Stream levels in the ISG Profile window.*

156 Select the option “Stream Discharge“ from the drop down menu *Parameter A:*.
157 In the ’ISG Profile’ window (not the ’ISG Edit’ window) select all segments from the menu field.
158 Try the other options from the drop down menu *Parameter A:*.

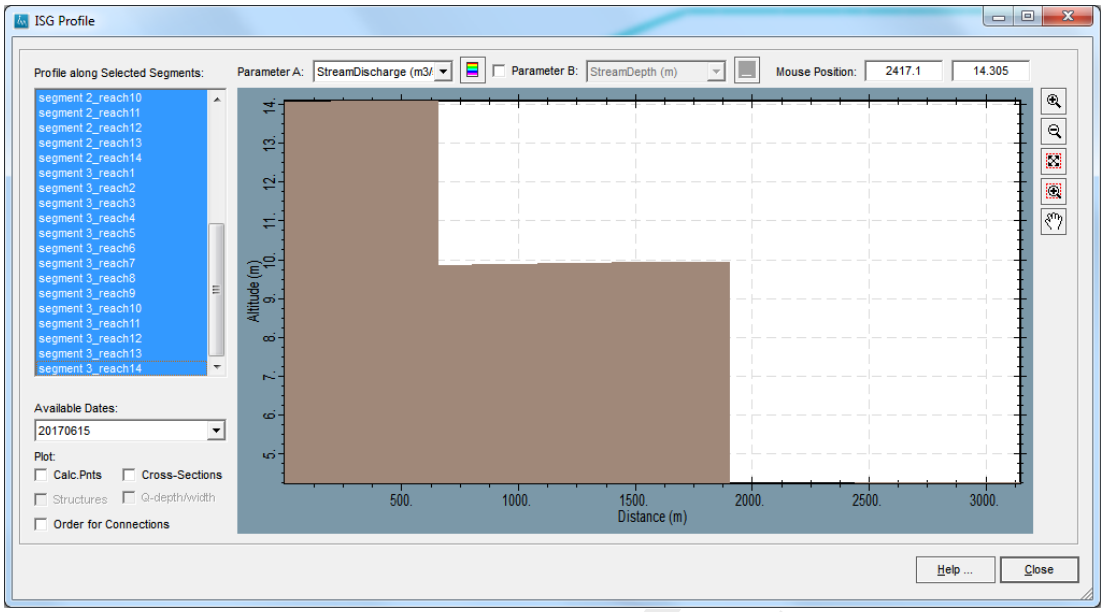


Figure 11.105: *Stream discharges along segments 1 to 3.*

In the graph we see that the inflow volume in “Segment_1” is 14 m^3/s, and the volumes for the “Segment_2” and “Segment_3” are 10 m^3/s ($\approx$ 70%) and 4.25 m^3/s ($\approx$ 30%), respectively.

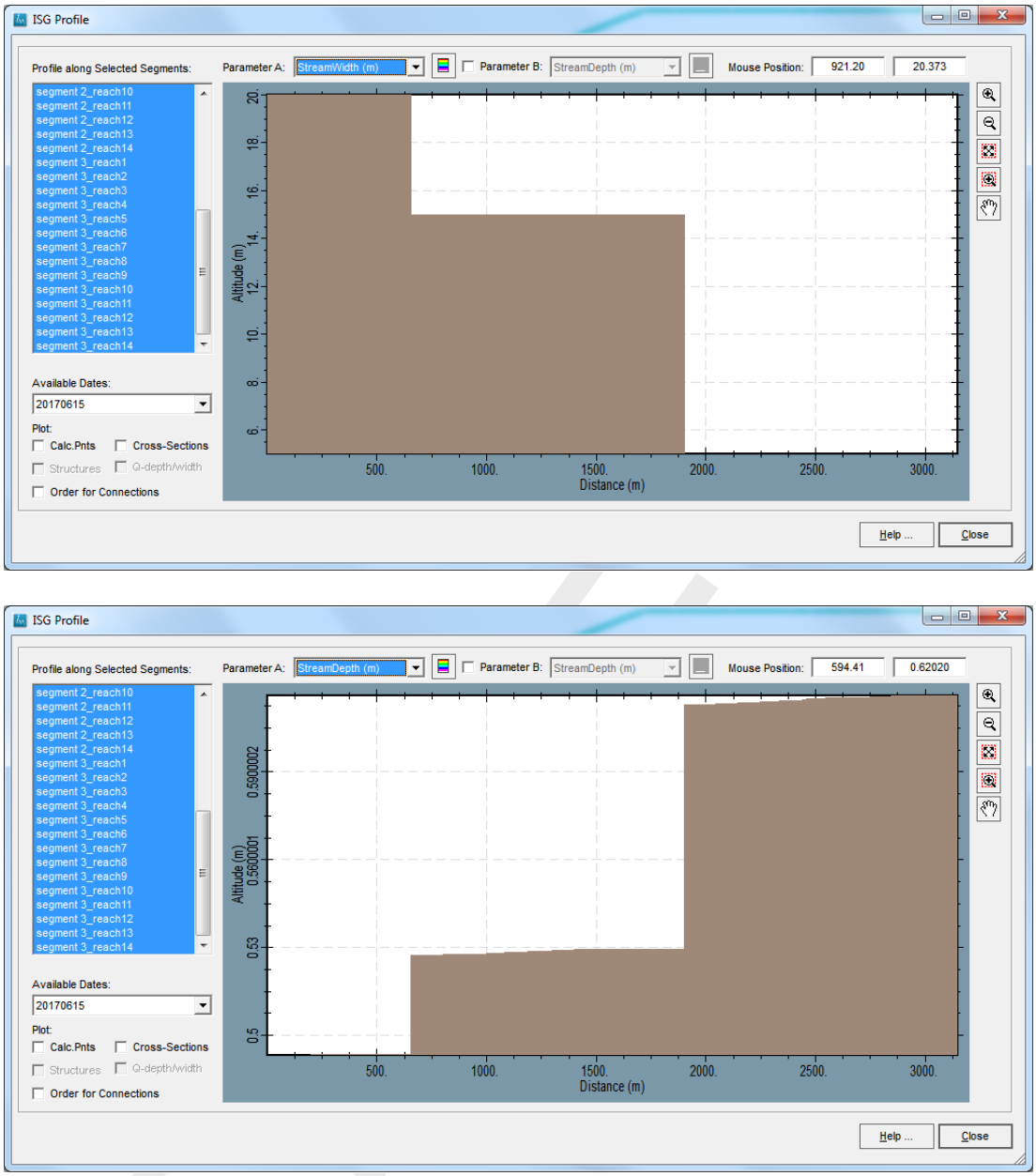


Figure 11.106: *Stream width and stream depth along segments 1 to 3.*

Another (fancy) way to look at your results is to use a legend to colour the lines for a selected output item, such as water levels, discharges.

159 Click the *Close* button to return to the *ISG Edit* window.

160 Select the *Legend* button ( ) to start the *ISG Colouring* window.

161 Select the option Current window to colour all segments within the current graphical window.

162 Increase the line-thickness to 5.

In this visual we observe that the surface water is indeed declining from west to east.

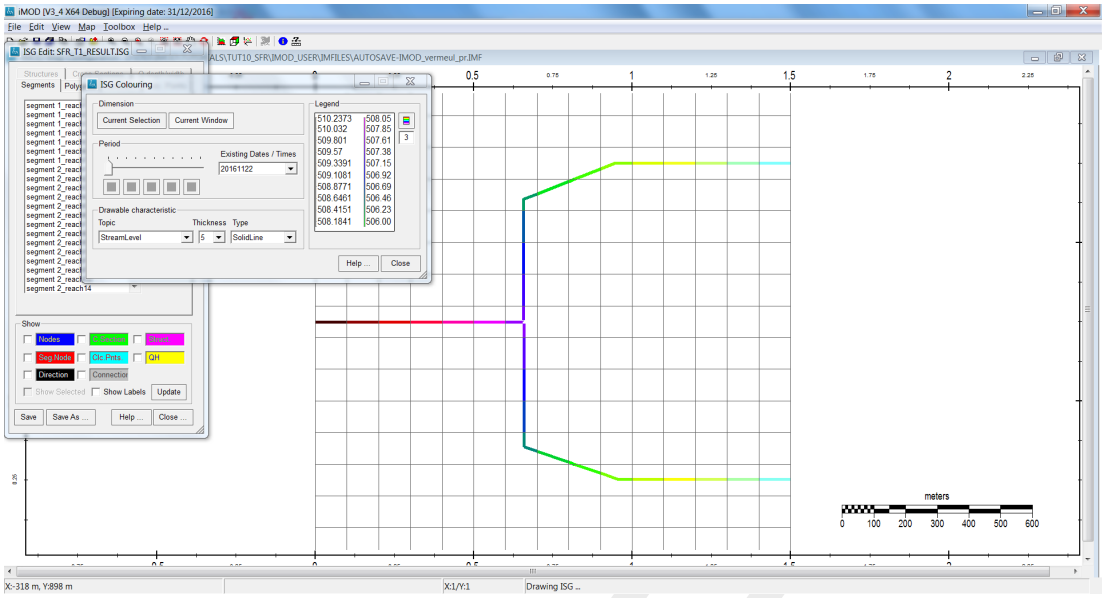


Figure 11.107: *Stream levels visualised when using a colour legend.*

It is easy to visualize the other model outcomes as well:

163 Select “StreamDepth” ,“StreamWidth” and “StreamDischarge” as well.

The legend is computed automatically based on the data of the ISG file. Each stream characteristic in the ISG file has its own legend. This can be modified by using the default *Legend* window that starts whenever you select the option *Legend* ( ) from the *ISG Colouring* window. For any transient simulation you might do in future, you can drag the slider in the ’Period’ part of the of the ’ISG colouring’-window to visualise stream characteristics for different periods.

Now it’s time to visualise the total exchange flux between the surface water and groundwater.

164 Click the *Close* button on the *ISG Edit* window to close the *ISG Edit* window and *ISG Colouring* window, accept the question upon closing.

165 Select from the main menu the option *Map*, *Add Map ...* and select the ISG file {path of installfolder} \IMOD_USER \MODELS \TUT_SFR \BDGSFR \BDGSFR_STEADY-STATE_L1.IDF.

166 Click the *Adjust Legend* button ( ) from the *iMOD Manager* to start the *Legend* window. If the *iMOD Manager* is not visible, display it again by selecting the iMOD Manager button ( ) from the main iMOD window.

167 Use your skills to create the legend as displayed in the next figure. If you find difficulties reproducing this legend, have a look again at Tutorial 1: Map Display.

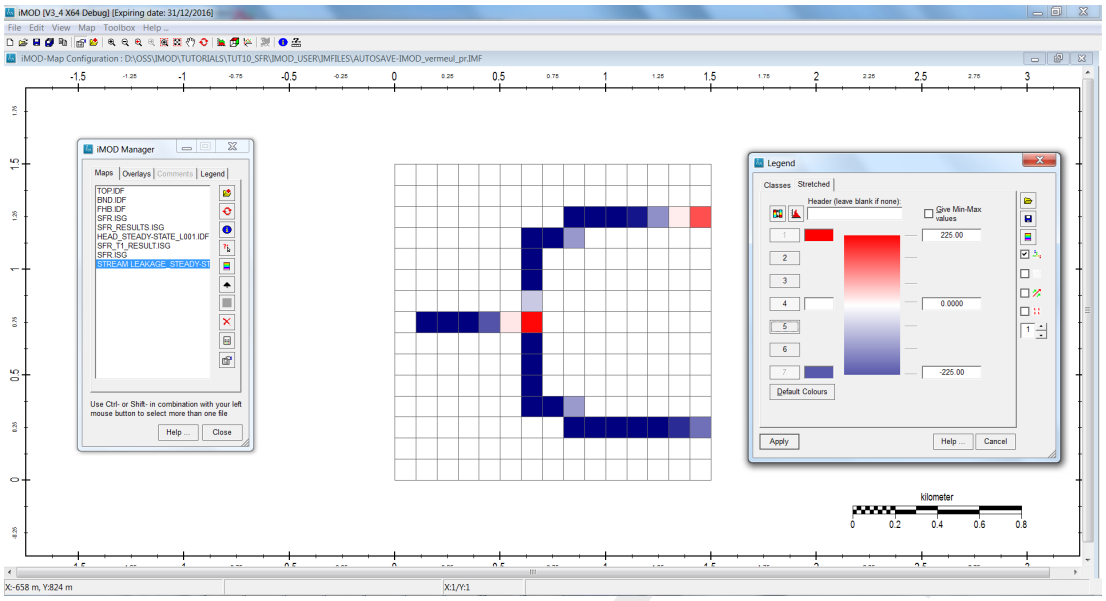


Figure 11.108: *Visualising the computed fluxes between surface water and groundwater.*

Regarding practising the visualisation of ISG's it's OK for now.

168 Click the *Close* button to stop *ISG Edit* window.

It would be nice if you try experimenting with different parameters of the SFR package, such as stream width, Manning's Resistant Coefficients and/or implement an extraction in the model to see whether that effects the surface water level. To estimate the extraction rate, such that the surface water level might change with 0.10 m, use the following equation of the re-organised Manning's Equation:

$$y = \left[\frac{Q \times n}{C \times w \times S^{\frac{1}{2}}} \right]^{\frac{3}{5}} \tag{11.3}$$

If you apply this for stream "Segment_2" , the extraction need to be at least 3.3 m^3/s=285,000 m^3/d.

Enhance the model by an Eight Point Cross-Section

In the coming few steps we will enhance the model a bit more, adding a more complex cross-section and apply a q-width/depth relationship for a segment.

169 Select the option *ISG Options* and then *ISG Edit* to start the *ISG Edit* window.
170 Select "Segment_1" from the menu list of segment names.
171 Click the *Attributes* button ( ) to start the *ISG Attributes* window.
172 Select the tab *Cross-sections* from the *ISG Attributes* window.
173 Select the *Open* button ( ) and select the file {path of tutorialfolder} \TUT_SFR \DBASE \CROSSSECTION.CSV.
174 Click *OK* to read the selected file.

iMOD will open the *Read CSV file* window. Here you can specify what column from the CSV-file you want to use for each of the columns of the cross-section, such as "Distance" , "BottomLevel" and "MRC". We leave the default values as shown below.

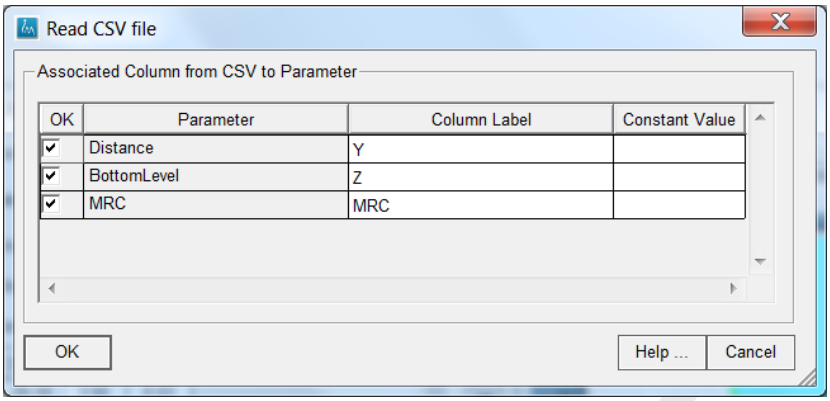


Figure 11.109: *The 'Read CSV file' window.*

175 Click *Ok* to accept the column definitions and return to tab *Cross-sections* on the *ISG Attributes* window.

176 Click the checkbox *Simplified* to observe a simplified cross-section.

The SFR Package has the limitation that only eight-point cross section geometries can be assumed. Eight values each for the horizontal and vertical distances are specified for the segment. Vertical walls are assumed at the end of each cross section. Stream depth, width, and wetted perimeter (hydraulic radius) are computed from the cross section for a given flow using Manning's equation and by dividing the cross section into three parts, one part for the points 1-2-3, a second part for the points 3-4-5-6 and a third part for the points 6-7-8. All those together form the total wetted perimeter and the area. As this can be rather complex, the SFR package uses an iterative procedure to estimate the total discharge (sum of the three parts) until the computed flow is more-or-less equal to the stream flow. This method may not solve for all geometries, especially wide, flat bottom geometries might cause problems, in those case an other option is advised to be used for computing the stream depth.

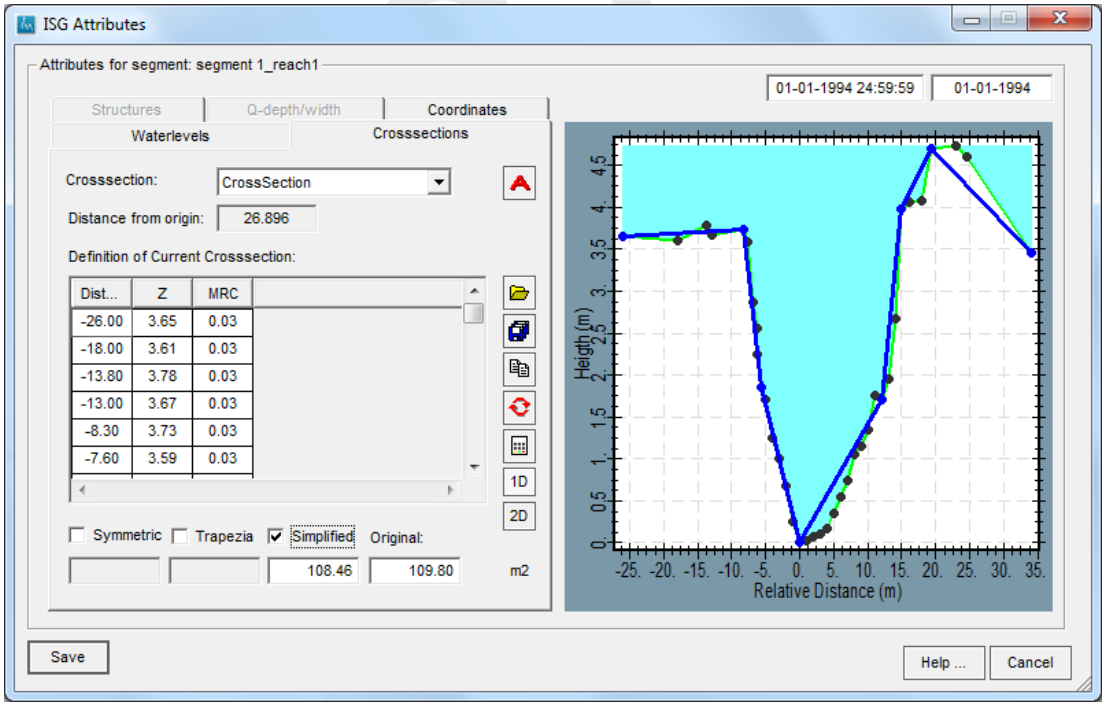


Figure 11.110: *The cross-section as read from the CSV file (black dots) and the 8-points simplified cross-section (blue dots) after selecting 'Simplified' in the 'ISG Attributes'-window, including the corresponding areas of the original and simplified cross-section.*

From the figure above, it seems that the simplified cross-section has a more-or-less similar wetted area (108.46 m^2) compared to the original one (109.80 m^2). Actually, this simplification is done once the model is exported to the SFR package of MODFLOW2005, automatically.

Next thing is to modify the ISG file a little bit more such that it knowns to use this eight-point cross-section.

177 Select the tab *Water Levels* on the *ISG Attributes* window.
178 Select the option “2 Eight Point” from column 10 with label *Calc Opt.*.
179 Click the *Save* button to save the modification in memory and return to the *ISG Edit* window.
180 Click the *Save As* button to save the modification on disk in another name, enter the file name {path of installfolder} \IMOD_USER\DBASE\TUT_SFR\SFR2.ISG.
181 Click the *Close* button to leave to the *ISG Edit* window.
182 Accept the question by clicking the *Yes* button.

11.9 Tutorial 9: Lake Package

This tutorial gives an introduction to a transient implementation of the Lake package (LAK), see section 12.29.

Outline

This is what you will do:

- ◇ Define a simple, five layered, transient model and constant head boundaries along the model;
- ◇ Define the input for the LAK package;
- ◇ Start the model simulation and examine the outcome;
- ◇ Combine the LAK package with the SFR package.

Required Data

For this tutorial you need the following iMOD Data Files/folders:

- ◇ The entire folder (and subfolders) in {path of tutorialfolder} \TUT_LAK \DBASE, containing:
  - ▫ BND.IDF – boundary conditions of the model (to be created);
  - ▫ TOP.IDF – uppermost interface of the model (to be created);
  - ▫ LAK_ID.IDF – lake identification number (to be created);
  - ▫ LAK_BATHYMETRY.IDF – lake bathymetry number (to be created);
- ◇ MODEL.PRJ – initial model project file;
- ◇ MODEL_LAK.PRJ – model project file with the LAK package (to be created);

Getting Started

1. Start iMOD.
2. Select the option *Create a New iMOD Project*.
3. Click the *Start* button.

Create the boundary conditions

We start to create our first IDF file.

4. Select the main menu option *Edit Create Feature*, *IDFs from ...* and then *Scratch ...* to start the *Create IDF* window;
5. Enter the following values:
   - ◇ XLLC / XURC (ft) : “0.00” and “8500.0”;
   - ◇ YLLC / YURC (ft) : “0.00” and “8500.0”;
   - ◇ CellSize (ft) : “250.0”.
6. Activate the *iMOD Manager* (Ctrl+M) (if not active already); the map BND should be selected now.
7. Select the option *Apply ...* and enter the file name {path of installfolder}\IMOD_USER\DBASE\TUT_LAK\BND.IDF.
8. Select the menu option *View*, *Show IDF Features* and then *IDF Raster Lines* to display the rasterlines of the IDF just created.

Now we copy the geometry of this BND file to a to be created TOP file.

9. Select the main menu option *Map*, *IDF Option* and then *IDF Calculate ...* to start the *Map Operations* window;
10. Enter at *Map C* the output file {path of installfolder}\IMOD_USER\DBASE\TUT_LAK\TOP.IDF;
11. Select the option *Map A* to determine the extent for which the computation applies;
12. Click the *Compute ...* button.

We will assign the IBOUND value '-1' (fixed heads) to all cells of columns and rows 1 and 34. There are a number of possibilities to select the appropriate cells; we will now select them by dragging the mouse of the cells, similar to step 14 of Tutorial 8: Surface Flow Routing (SFR) and Flow Head Boundary (FHB) Package:

13 Select the main menu option *Map*, *IDF Option* and then *IDF Edit ...* to start the *IDF Edit* window;
14 Click the option *Draw* to start the *IDF Edit Draw* window.
15 Move your mouse in the graphical canvas and drag, **while holding your left-mouse button**, all model cells in the first row, the right column, the last row and the first column of our model; it should result in 132 selected cells.
16 Click the *Close* button on the *IDF Edit Draw* window to leave the mode to select cells.
17 Click the *Calculate ...* button to start the *IDF Edit Calculation* window;
18 Select the option *New Value* and enter the value "-1" at the utter most right input field;
19 Select the IDF file "BND.IDF" from the dropdown menu at *Available IDF-file*;
20 Click the *Calculate* button to adjust the selected cells in the IDF file.
21 Click the *Close* button to close this window;
22 Click the *Yes* button to confirm the question whether you want to leave this window.

Now we need to define the active area in the boundary conditions.

23 Click the *Select ...* button to start the *IDF Edit Select* window;
24 Select the option "BND.IDF" from the *IDF-file:* dropdown menu;
25 Select the option "NodataValue" from the *Logic* dropdown menu;
26 Click the *Get Selection* button, 1024 cells are selected;
27 Click the *Close* button to leave the *IDF Edit Select* window;
28 Click the *Calculate ...* button to start the *IDF Edit Calculation* window;
29 Select the option *New Value* and enter the value "1" at the utter most right input field;
30 Select the IDF file "BND.IDF" from the dropdown menu at *Available IDF-file*;
31 Click the *Calculate* button to adjust the selected cells in the IDF file.
32 Click the *Close* button to close this window and return to the *IDF Edit* window;
33 Click the *Yes* button to confirm the question whether you want to leave this window. .

Now, we will create the surface level. The surface level declines gradually from west to east, starting at 160 m and ending at 140 m.

34 Select the *Draw Polygon* () and draw a polygon around all cells of the first column. iMOD will select IDF cells that are in a polygon with their midpoint;
35 Click the *Select ...* button to start the *IDF Edit Select* window;
36 Select the IDF file "TOP.IDF" from the *IDF-file* dropdown menu;
37 Select the option "All" from the *Logic* dropdown menu;
38 Click the *Get Selection* button and 34 cells are selected;

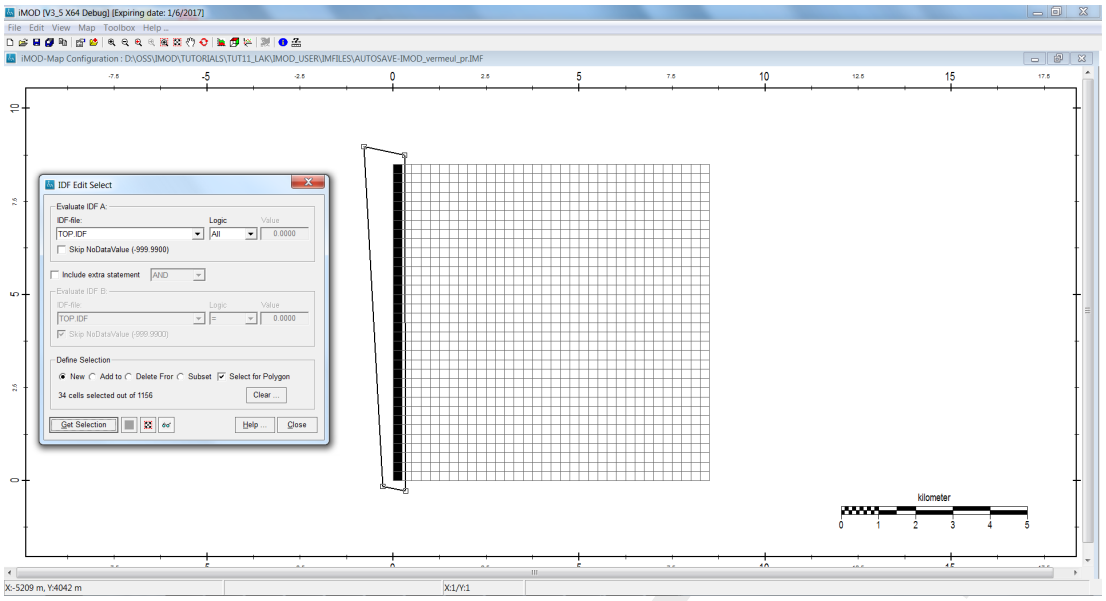


Figure 11.111: *34 cells selected after clicking the 'Get selection' button.*

39 Click the *Close* button to leave the *IDF Edit Select* window;
40 Click the *Calculate* button to start the *IDF Edit Calculation* window;
41 Enter a value of "160" in the input field right of the option *New Value*;
42 Click the *Calculate* button to adjust the selected cells in the IDF file.
43 Click the *Close* button to close this window and return to the *IDF Edit* window;
44 Click the *Yes* button to confirm the question whether you want to leave this window. .

Now we compute the right in a similar manner and give the east side of the model the value 140.0.

Note: It is easy to just move the drawn polygon to the right and than follow the steps 25-30 again.

Let's interpolate the surface level.

45 Click the *Select ...* button to start the *IDF Edit Select* window;
46 Select the IDF file "TOP.IDF" from the *IDF-file* dropdown menu;
47 Select the option "NodataValue" from the *Logic* dropdown menu;
48 Uncheck the option *Select for Polygon*;
49 Click the *Get Selection* button and 1088 cells are selected in between column 1 and column 34;
50 Click the *Calculate* button to start the *IDF Edit Calculation* window;
51 Select the option *Interpolate*;
52 Select the option *PCG* to use a linear interpolation;
53 Click the *Calculate* button to adjust the selected cells in the IDF file.
iMOD will start the *Solver Settings* window in which you can specify the accuracy of the interpolation. We will accept all the default settings.

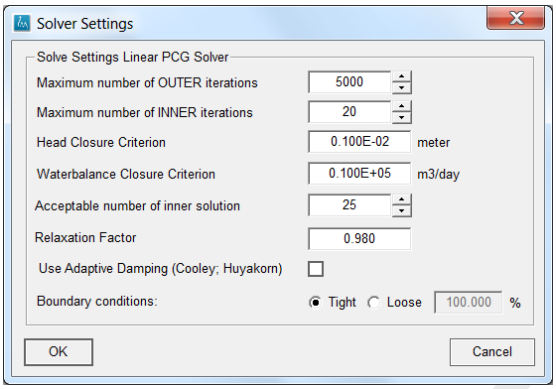


Figure 11.112: *The Solver Settings window.*

54 Click the *OK* button to close the *Solver Settings* window and start the interpolation;
55 Click the *Close* button to close the *IDF Edit Calculation* window and return to the *IDF Edit* window;
56 Click the *Yes* button to confirm the question whether you want to leave this window.
57 Click the *Close* button to close the *IDF Edit* window.

Let's see how the interpolation looks like.

58 Right mouse click on the graphical canvas and select the option *Current Zoom Level* and then *Linear* to display the top of our system than gradually declines from west to east.

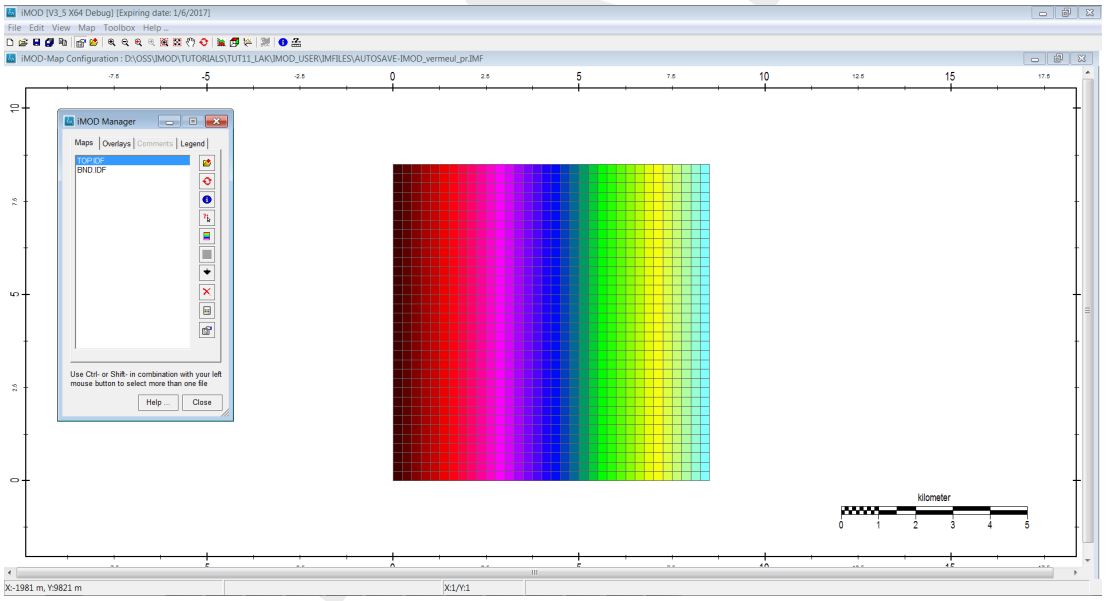


Figure 11.113: *The interpolated surface level.*

Create the Lake

We would like to introduce a lake in the middle of our model. We need an IDF file that describes the maximal extent of the lake, and another IDF file that describes the bathymetry of the lake.

59 Create the following two IDF files; apply the *Formulae* "C=0.0*A"in Map Operations when copying the geometry of the map BND:

- ◇ {path of installfolder}\IMOD_USER\DBASE\TUT_LAK\LAK_ID.IDF - this file will store the location of the Lake;
- ◇ {path of installfolder}\IMOD_USER\DBASE\TUT_LAK\LAK_BATHYMETRY.IDF - this file will

store the depth (bathymetry) of the lake.

I'm sure you know by now how *IDF Edit* works, so:

60 Start *IDF Edit* and modify the files as follows:

- ◇ fill in the LAK_ID.IDF file such that it contains a value of "1" for the rectangle given by the cell indices (row=12,column=15) and (row=18,column=21);
- ◇ fill in the LAK_BATHYMETRY.IDF file such that it contains a value of "107" for the rectangle given by the cell indices (row=12,column=15) and (row=18,column=21), and a value of "97" for the rectangle given by the cell indices (row=14,column=17) and (row=16,column=19).

61 Turn on the cell-indices via *View*, *Show IDF Features* and than *Cell Indices*. We have done this before in step 79 in section 11.8.

You can configure your IDF files to display the actual IDF values as follows:

62 Select the file LAK_BATHYMETRY.IDF in the *iMOD Manager* (if not visible select from the main window, the menu option *View* and than *iMOD Manager* or click Ctrl+M alternatively);

63 Select the *Legend* button ( ) from the *iMOD Manager* window to start the *Adjust Legend* window;

64 Select the option *Data Numbers* ( ) to display the IDF values;

65 Increase the textsize by entering a "3" underneath the *Data Numbers* option;

66 Click the *Apply* button to close the *Adjust Legend* window and redraw the graphical canvas.

67 Select the option *Map*, *Entire Extent* and than *Unique Values* to display the content of the IDF files with a colouring legend for unique values only.

You should have the following IDF files created:

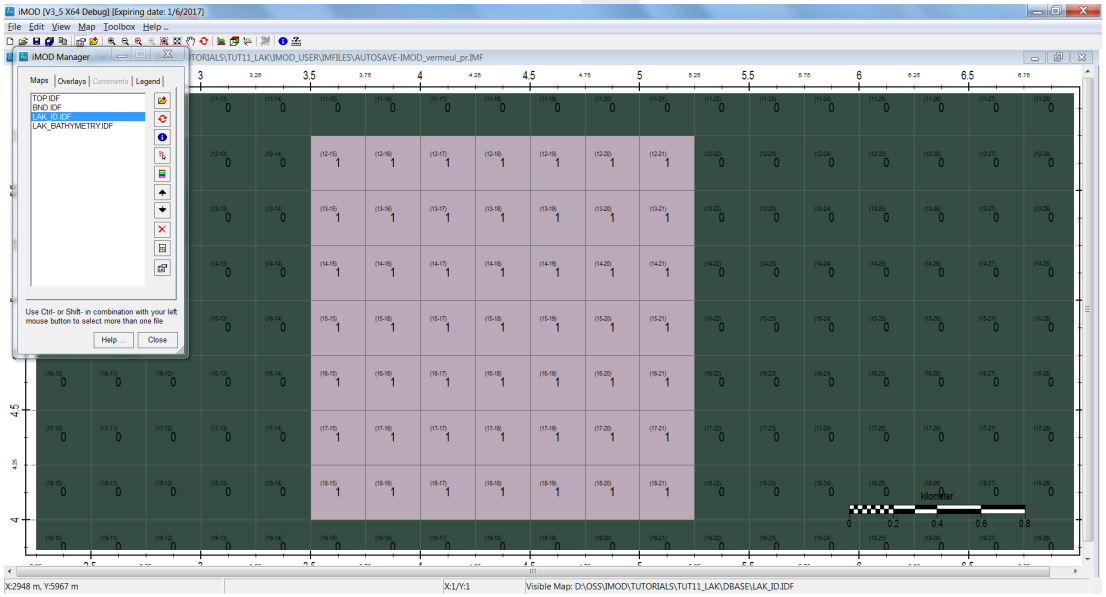


Figure 11.114: *Lake Identification.*

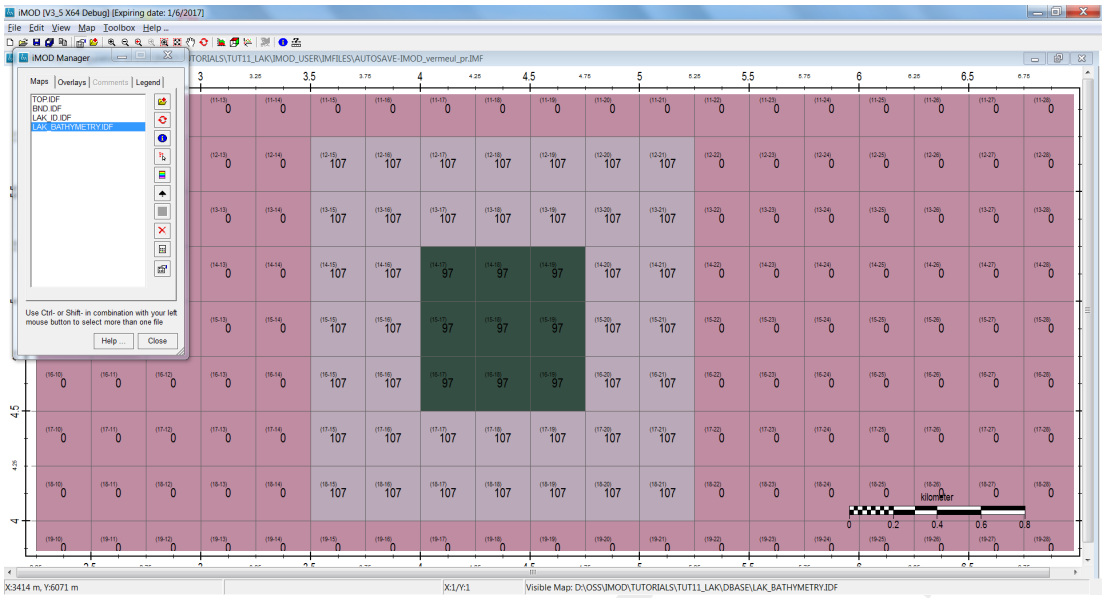


Figure 11.115: *Lake Bathymetry.*

Add the Lake to the Modelling Project

Now, we will add the parameters for the Lake Package in our modelling project.

68 Select the option *View* and then select *Project Manager* to start the *iMOD Project Manager* window;

69 Click the *Open Projectfile* button ( ) and select the file {path of tutorialfolder}\TUT_LAK\MODEL.PRJ;
70 Click the *Open* button.

The entire model has been filled in already. Notice that this MODEL.PRJ file refers to the prepared IDF files given with the iMOD install in the folder {path of tutorialfolder}\TUT_LAK\DBASE instead of the files you created yourself and saved in the folder {path of installfolder}\IMOD_USER\DBASE\TUT_LAK. With the *Define Characteristics* button on the *Project Manager* window you can change the file reference if you like (see also section 5.5.1).

You may inspect the model for a while and you will notice that it is a model with 5 model layers. The BND.IDF and TOP.IDF are filled in as well. The BND.IDF is used to define the boundary types for all model layers and the TOP.IDF is used at the module (TOP) for the first model layer. That file is also used for the Surface Elevation used by the (EVT) module. We will now enter the parameters for the (LAK) module.

71 Select the option (LAK) in the tree view *Project Definition*;

72 Click *Properties* button ( ) to start the *Define Characteristics for* window.
73 Select the option *Transient, start from* and enter the date “1 December 2016” in the date entry fields. All the other packages (RCH and EVT) start at that period as well;

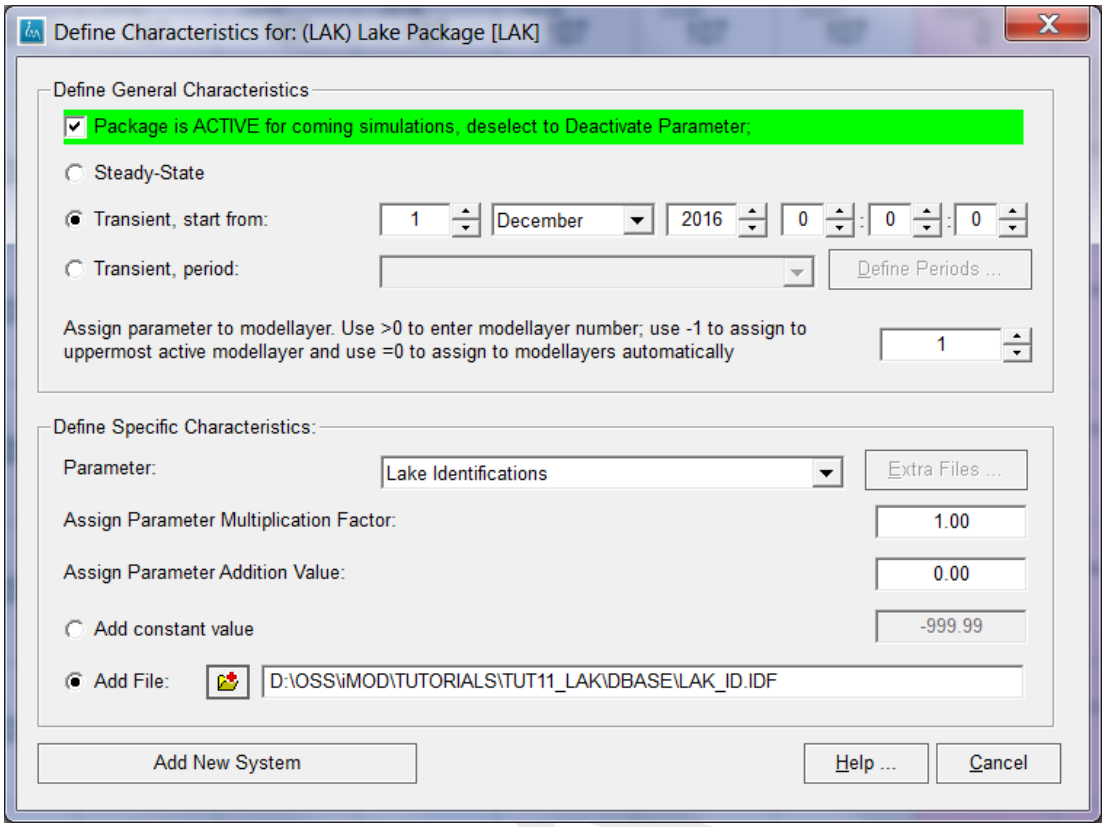


Figure 11.116: *Example of the 'Define Characteristisc for: (LAK) Lake Package' window; the part 'Define Specific Characteristics' contains a pull-down Parameter list which should be parameterized according to the values given in Table Table 11.7.*

74 We will enter the following values for the different input parameters by selecting the appropriate parameter from the *Parameter* dropdown list sequentially, see Table 11.7.

75 Click the *Add New System* button to add the parameter to the modeling project.

Table 11.7: *Modeling Parameters for the Lake Package.*

-	**Parameter**	**Entry**	**Units**
1	Lake Identifications	LAK_ID.IDF	-
2	Lake Bathymetry	LAK_BATHYMETRY.IDF	m+MSL
3	Initial Lake Levels	110.0	m+MSl
4	Minimal Lake Levels	97.0	m+MSl
5	Maximal Lake Levels	145.0	m+MSL
6	Lakebed Resistance	10.0	days
7	Precipitation at surface Lake	0.0116	m/d
8	Evaporation at surface Lake	0.0103	m/d
9	Total Overland runoff	0.0	m^3/d
10	Total Lake Withdrawall	0.0	m^3/d

So, let's first save our configuration in a new project file.

76 Close the *Define Characteristics for* window by selecting the *Add System* button.

77 Click the *Save As* button ( ) and save a new modeling project file at {path of installfolder} \IMOD_USER \RUNFILE \MODEL_LAK.PRJ.

Start the model simulation

With this project file we can generate a Runfile and/or a standard MODFLOW2005 model. As we use the LAK-package and this is not supported by a Runfile we need to create standard MODFLOW2005 files, let's do that.

78 Click the *Save Model* button ( ) to start the *Define Simulation Configuration* window.

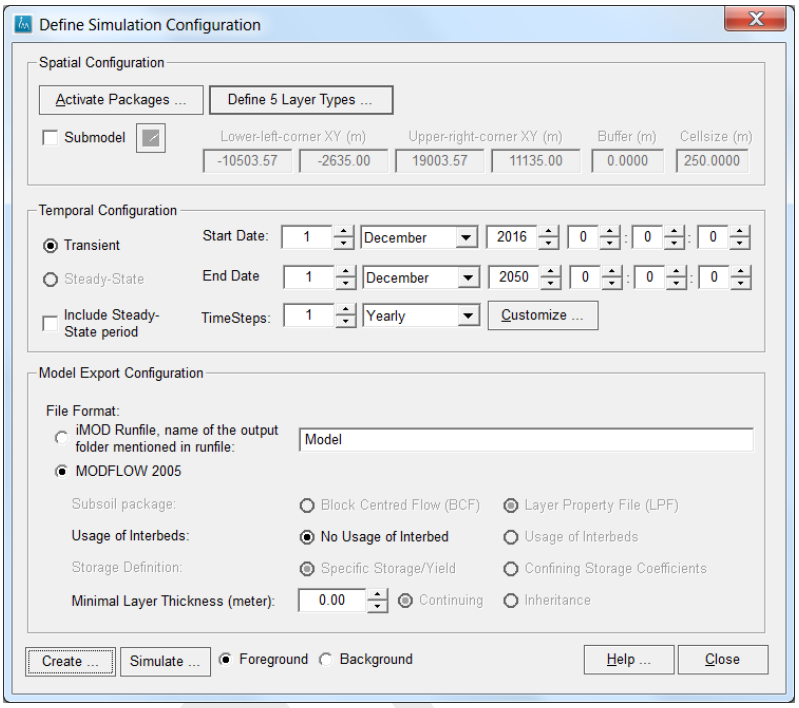


Figure 11.117: *Example of the iMOD Define Simulation Configuration window.*

79 Enter "2050" at the *End Date* input field.
80 Select the option "Yearly" from the *TimeSteps:* dropdown menu field.
81 Select the option "MODFLOW2005" at the *File Format:* radio button.

So, we will create a model that is transient, starts at the 1st of December 2016 00:00:00 and ends at the 1st of December 2050 00:00:00. The model will generate output after each year.

82 Click the *Simulate* button and enter the following *.NAM file {path of installfolder} \IMOD_USER \MODELS \TUT_LAK \TUT_LAK.NAM to export the model to MODFLOW2005 files and start the simulation. Note, that you need to create the folder TUT_LAK yourself first, use the option *New Folder* in your current *Save* window.
83 Click the *Save* button.

iMOD will now first create the necessary MODFLOW2005 file; as the model is tiny, this will be finished rapidly, then the simulation will start immediately. You'll see that the model start in a separate DOS-command window and it will echo the simulation progress. As it is a transient simulation with 34 stress periods, it will consume probably 30 seconds to accomplish.

Inspect the result of lake simulation

The lake levels will be part of the saved hydraulic heads, so we only have to open, e.g. the hydraulic of the first stress-period to generate time series. The lake exchange with groundwater will be saved in a separate budget file, we will open that as well.

84 Select the option *Map* and then the option *Quick Open* to start the *Quick Open* window, see section 6.2. With this window it is easy to open and view results from a model simulation.
85 Select the option “HEAD” from the *Topic* dropdown menu.
86 Select the option “20171201” from the *Time:* dropdown menu.
87 Select the options “1 2 3 4 5” from the *Layer* dropdown menu. Tip: drag your mouse to select all layers.
88 Click the *Open* button.
89 Repeat the above mentioned steps to open the results for BDGLAK as well, do this for model layers 1,2 and 3.

iMOD will load all selected results files into the *iMOD Manager* and displays the result on the graphical canvas. Use your experience learned from the previous Tutorials to display the computed heads as time series as shown in the following figure.

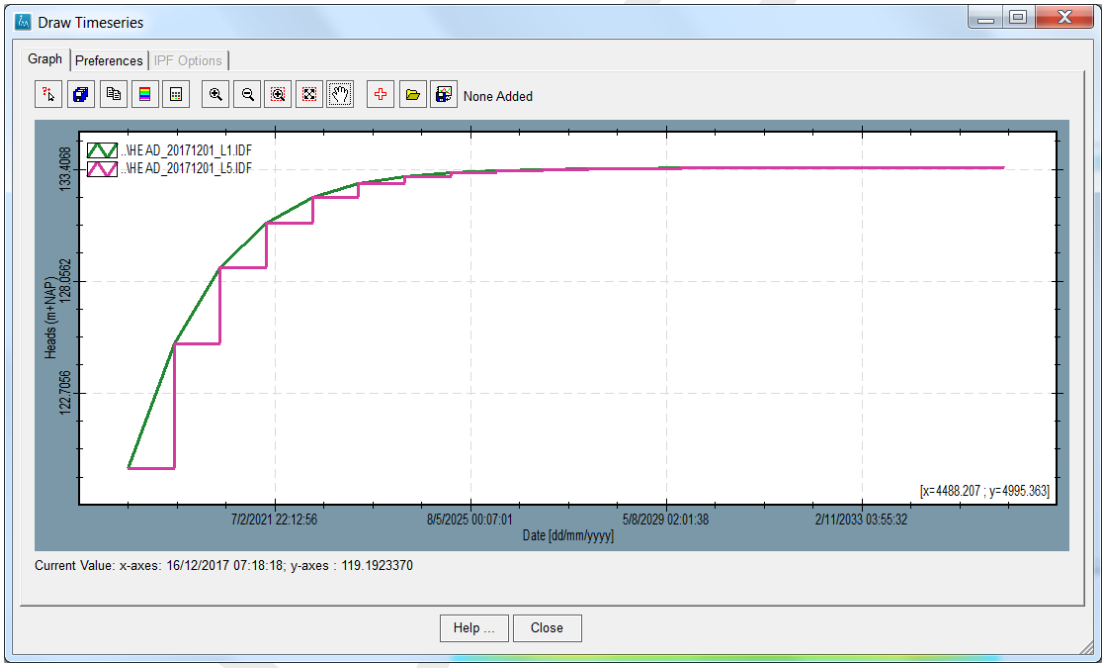


Figure 11.118: *Time Series of lake levels.*

The lake package simulates the exchange of groundwater and surface water such that the water balance of the lake equals (more-or-less). So, in the end it finds a water level of 133.47 m+MSL. At this lake level there is an equilibrium between the nett recharge of the lake (precipitation minus evaporation) and the drainage to the lake.

Note: The total volumes can be found, per stress-period, in the list file after the simulation. You can find the file here: {path of installfolder} \IMOD_USER \MODELS \TUT_LAK \TUT_LAK.LIST. To get this water balance, search for a part of the string “HYDROLOGIC BUDGET SUMMARIES FOR SIMULATED LAKES”. For the first year, the total inflow to the lake is 2.7264E+07 m^3/year (we have time step lengths of one year). This is equal to 74695 m^3/d, that is the sum of all fluxes from the BDGLAK-files. You can find the total fluxes per BDGLAK-file via the *Map Info* option ( ) and then select the *Statistic* button ( ) you can read the *Sum* of the individual flux file.

Let's look at the lake spatial exchange volumes.

90 Select the file "BDGLAK_20171201_L1.IDF" from the *iMOD Manager* and redraw the canvas by clicking the *Redraw* button ( ).
91 Add to the selection of files in the *iMOD Manager*, the files "LAK_ID.IDF", "BDGLAK_20171201_L2.IDF" and "BDGLAK_20171201_L3.IDF". Use your Ctrl-Left mouse button to select multiply files.
92 Click the *Map Value* button ( ) to start the *Map Value* window.
93 Mouse your mouse over the image and observe the values of the selected maps.

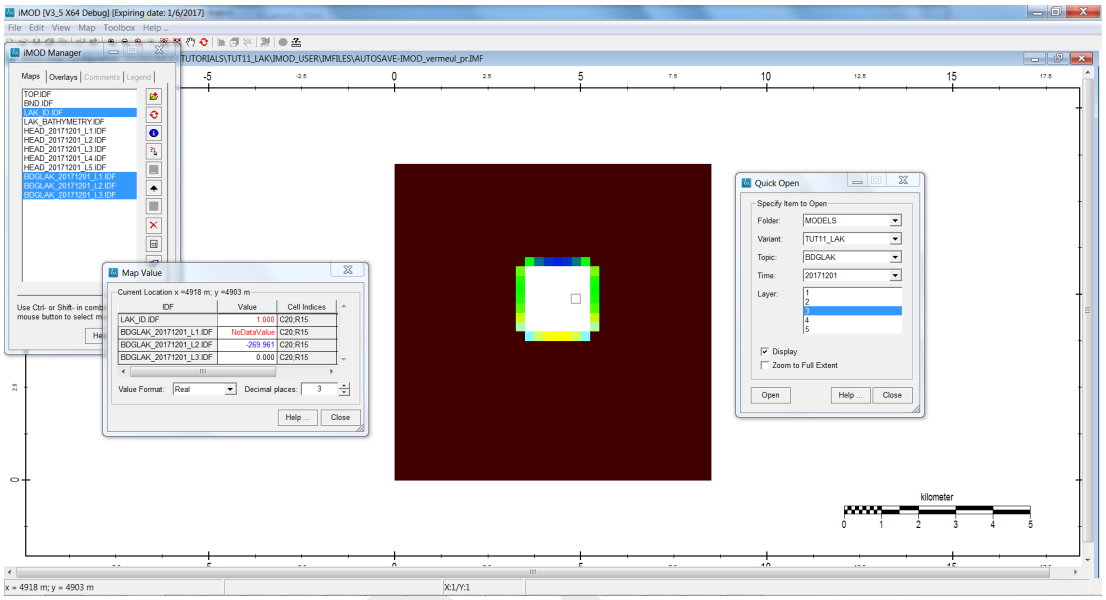


Figure 11.119: *Computed spatial Lake fluxes.*

The fluxes to- and from the lake are given by the BDGLAK-files. These fluxes are stored within the first model cell next to the lake. So whenever you hoover your mouse, you'll notice that the fluxes are all zero at the location of the lake for the first model layer. As the lake connects for only a part to the second model layer, you'll notice that lake fluxes appear in the second layer only underneath the lake and even for model layer 3, the bottom of the lake. Try to understand the pattern of the fluxes. Having a look at the head differences between layer 1 and 5 in a cross-section (utilizing the Cross-Section Tool) may also shed some extra light on how the flux pattern looks like.

Connect the Lake with the SFR package

In this final step, we will connect the lake with the surface water model as described by the SFR package. The implementation of this package is explained in section 11.8. We have created the SFR layout and stored the file in {path of tutorialfolder} \TUT_LAK \DBASE \SFR.ISG. Let us open the file.

94 Select the *Map Open* button ( ) and select the file {path of tutorialfolder} \TUT_LAK \DBASE \SFR.ISG.
95 Select the file "LAK_ID.IDF" from the *iMOD Manager*.
96 Click the redraw button ( ) to refresh the graphical canvas.
97 Select from the main menu the option *Map*, *ISG Options* and then *ISG Edit ...* to start the *ISG Edit* window.
98 Select "Segment 1" from the list of segments.

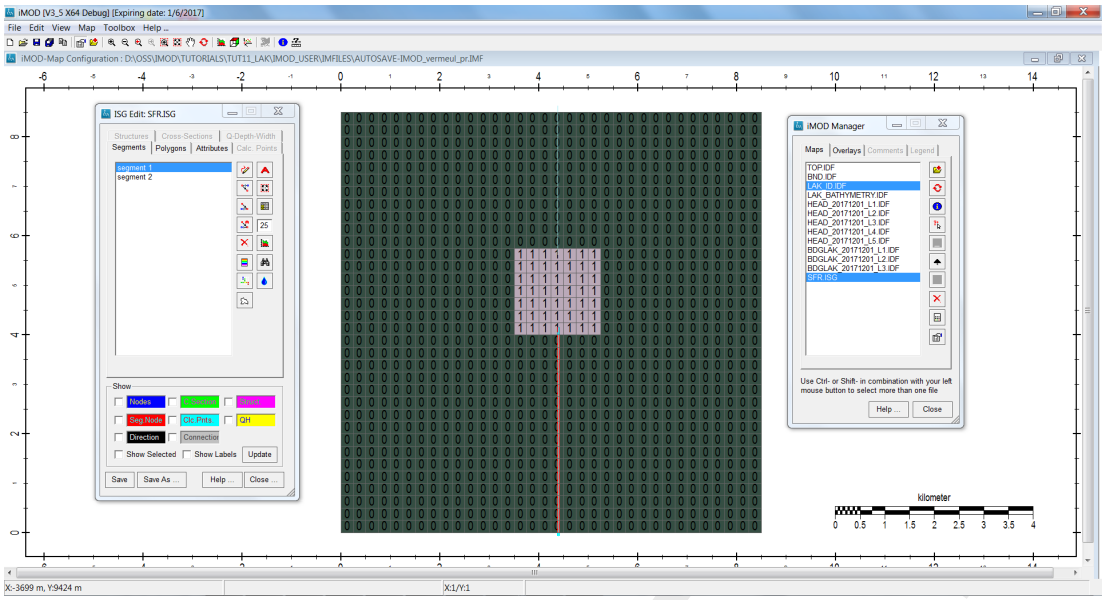


Figure 11.120: *Current layout of the SFR and LAK maps.*

In the figure you can see that the streams are connected to the lake on the south and north sides. From the south, the stream feeds the lake, from the north, it drains the lake.

To connect the lake (LAK) to the stream (SFR), we need to specify a negative lake number in the SFR.ISG, let's do that.

99 Click the *Attributes* button ( ) to open the *ISG Attributes* window for 'Segment 2' ('Segment 2' should be selected in the previous step).
100 Select the "CalPnt TO" from the *Calculation Point:* dropdown menu.
101 Enter the Lake number at "-1" *Idown Seg.* (column 9). Use a minus to indicate that the connection is a lake number and not a stream number.
102 Click the *Save* button to store the modification in memory.

Now we need to connect the upstream segment 2, such that it can receive water from the lake.

103 Select "Segment 2" from the list of segments.
104 Click the *Attributes* button ( ) to open the *ISG Attributes* window.
105 Enter the Lake number at "-1" *Iup Seg.* (column 8). Use a minus-sign to indicate that the connection is a lake number and not a stream number.
106 Click the *Save* button to store the modification in memory.

That's all, we need to add this ISG file to our project.

107 Click the *Save* button to save the modification in your model database as file {path of installfolder} \IMOD_USER \DBASE \TUT_LAK\SFR.ISG.
108 Select the option *View* and then select *Project Manager* to pop-up the *iMOD Project Manager* window;
109 Select the option (SFR) in the tree view *Project Definition*.
110 Click *Properties* button ( ) to start the *Define Characteristics for* window.
111 Select the option *Transient, start from* and enter the date "1 December 2016" in the date entry fields.
112 Click the *Open* button ( ) and select the file you just created {path of installfolder} \IMOD_USER \DBASE \TUT_LAK \SFR.ISG.
113 Click the *Add New System* button to add the parameter to the modeling project and close the *Define Characteristics for* window.

114 Click the *Save As* button ( ) and save a new modeling project file at {path of installfolder} \IMOD_USER \RUNFILE \MODEL_LAK_SFR.PRJ.

Simulate the enhanced model

As we use the LAK-package in combination with the SFR we need to create standard MODFLOW2005 files, let's do that.

115 Click the *Save Model* button ( ) to start the *Define Simulation Configuration* window.
116 Enter "2036" at the *End Date* input field.
117 Select the option "Yearly" from the *TimeSteps:* dropdown menu field.
118 Select the option "MODFLOW2005" at the *File Format:* radio button.
119 Click the *Simulate* button and enter the following *.NAM file {path of installfolder} \IMOD_USER \MODELS \TUT_LAK_SFR \TUT_LAK.NAM to export the model to MODFLOW2005 files and start the simulation. Note, that you need to create the folder TUT_LAK_SFR yourself first, use the option *New Folder* in your current *Save* window.
120 Click the *Save* button.

Inspect the result of the enhanced model simulation

Use the steps from 84 to open the result files from the the folder {path of installfolder} \IMOD_USER \MODELS \TUT_LAK_SFR. Use yor skills to observe that the steady-state lake level becomes 112.10 m+MSL and the stream stage up- and downstream of the lake are 118.63 and 111.57, respectively.

Note: Use *ISG Edit* to explore the results of the results of the SFR package, see section 11.8, step 148 onwards.

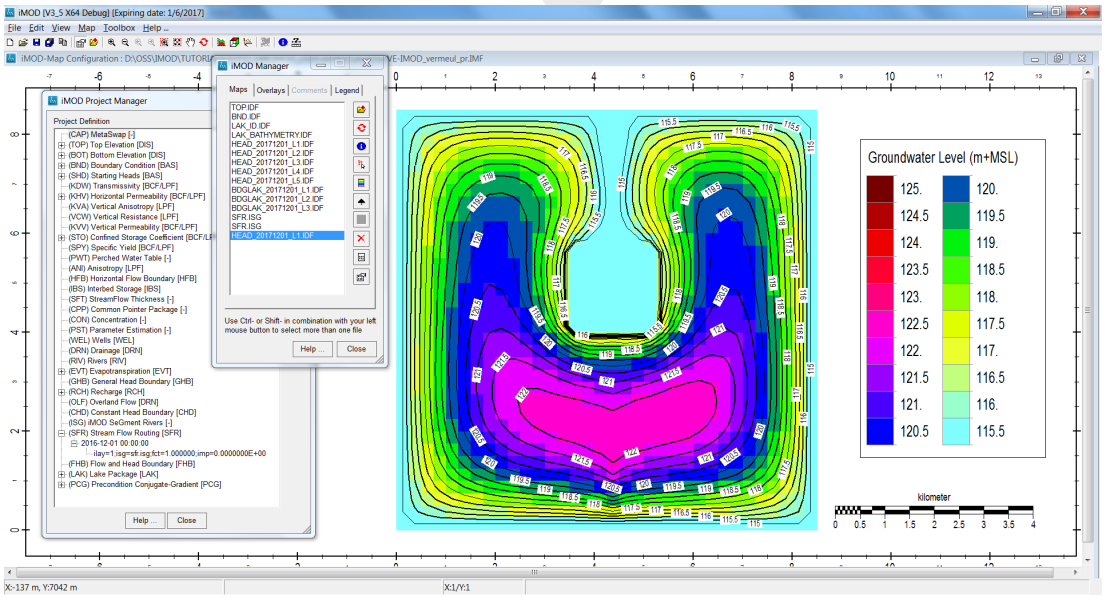


Figure 11.121: *Current result of the groundwater levels for 31st of December 2037.*

Use the {path of installfolder} \IMOD_USER \MODELS \TUT_LAK \TUT_LAK.LIST to explore the water balance for the lake at the last stress period.

It follows that:

Table 11.8: *Summary of Lake Water balance.*

Parameter	**Value**	**Unit**
Lake Stage	112.09	m+MSL
Lake Volume	2.121050E+07	m^3
Precipitation	1.300215E+07	m^3/year
Evaporation	1.154501E+07	m^3/year
Groundwater Inflow	5.7897E+07	m^3/year
Groundwater Outflow	0.0000E+00	m^3/year
Surface Water Inflow	2.5738E+08	m^3/year
Surface Water Outflow	3.1528E+08	m^3/year

Okay, please feel free to experiment more with several parameters for the LAK package.

11.10 Tutorial 10: Multi-Node Well- and HFB Package

This tutorial gives an introduction to the Multi-Node Well Package (MNW, see section 12.30) by using it in an unconfined quasi 3-D transient model. It also compares the MNW package with the conventional WEL package. We also add the HFB package to block the horizontal flow from a particular direction.

Outline

This is what you will do:

- Load an existing model project and display the model in 3-D;
- Construct a quick and simple model project with the WEL package;
- Define the model as an unconfined model and simulate the model;
- Modify the model project with the MNW package and simulate the model;
- Inspect both results;
- Change some parameters in the MNW package to simulate well losses;
- Include the horizontal barrier flow package (HFB) and simulate the results for that configuration.

Required Data

For this tutorial you need the following iMOD Data Files/folders:

- The entire folder (and subfolders) in {path of tutorialfolder} \TUT_MNW \DBASE, containing:
  - BND \BND.IDF – boundary conditions of the model;
  - TOP \TOP.IDF – top elevation of each model layer;
  - BOT \BOT.IDF – bottom elevation of each model layer;
  - HFB \SHEET_PILE.GEN – location of the sheet piling (to be created);
  - WEL \WEL.IPF – location of the extraction well;
  - WEL \WEL.TXT – time series of the extraction rate of the well;
  - MNW \WEL_THIEM.IPF – location of the extraction well configured for a well loss based on Thiem equation;
  - MNW \WEL.TXT – time series of the extraction rate of the well;
- MODEL_WEL.PRJ – initial model project file;
- MODEL_MNW.PRJ – model project file with the MNW package (to be created);
- MODEL_HFB.PRJ – model project file with the HFB package (to be created);

Getting Started

1. Start iMOD.
2. Select the option *Create a New iMOD Project*.
3. Click the *Start* button;
4. Activate the *iMOD Manager* (short-cut is *Ctrl+M*)

Load the Modelling Project in 3-D

We will load the modelling project and generate a 3-D image of our model.

5. Select the option *View* and then select *Project Manager* to start the *iMOD Project Manager* window;
6. Click the *Open Projectfile* button () and select the file {path of tutorialfolder} \TUT_MNW \MODEL_WEL.PRJ;
7. Click the *Open* button;
8. Select the option (WEL) in the tree view *Project Definition*;
9. Click the *Draw* button () to add the well file to the *iMOD Manager*;
10. Click the *Zoom All* button () to view the location of the well.

We want to display the well in 3-D together with the modellayers, so we need to configure the Z-settings of the well via IPF Configure, let's do that.

11 Right click your mouse button and select the option *IPF Options* and than *IPF Configure* to start the *IPF Configure* window;
12 Select "Z1" at the dropdown menu at *Z-coordinate*. (top elevation of the screen);
13 Check the option *Sec.Z-Crd* to define the secondary Z-coordinate. (botom elevation of the screen);
14 Select "Z2" at the dropdown menu at *Sec.Z-Crd*;
15 Click the button *Pick Colour*;
16 Select a red colour from the *Colour* window;
17 Click the *Ok* button to leave the *Colour* window;
18 Click the *Close* button to leave the *IPF Configure* window.

We will now load the upper- and lower elevations per model layer, we use the *Special Open* option from the *Project Manager*. This option allows you to quickly read a selection of IDF files in a particular order from the current model project to the *iMOD Manager*. In that way, it is easy to port the files in the right order to the *Profile Tool* and/or *3-D Tool*.

19 In the Project Manager window click the *Special Open* button ( ) to start the *Special Open* window.

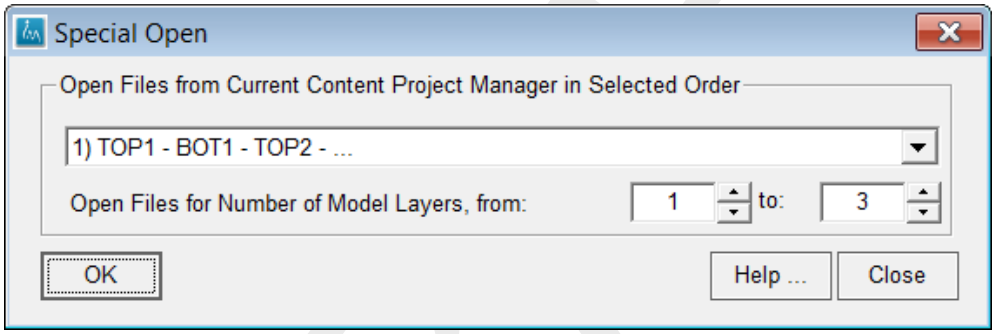


Figure 11.122: *Example of the Special Open window.*

20 Click the *OK* button to load the selected files to the Project Manger and leave the *Special Open* window;
21 Click the *Zoom All* button ( ) to set the graphical display to the extent of the IDF files loaded in the *iMOD Manager*;
22 Select in the *iMOD Manager* all IDF-files together with the WEL.IPF;
23 Click the *3-D* button ( ) from the iMOD Main window;

You'll notice that prior to the 3D tool the *3D IDF Settings* dialog appears. In this dialog the appearance of the IDF-files can be configured. For example, an IDF can be represented by planes (quads between mids of gridcells giving a smooth surface) and/or cubes (representing the grid cells as flat surfaces, like Lego-blocks). To visualize aquitards as solids we will combine each bottom of an aquifer with the top of the aquifer lying underneath it.

24 Select the option "Quasi 3D Model (aquitard)" from the *Configuration* dropdown menu;
25 Click the *Apply* button.

To show the well we need to instruct iMOD to ignore the associated txt file temporarily and use the *Z* and *Sec.Z-Crd* as set previously. Therefore do the following:

26 Select the tab *IPF's* from the *3-D Tool* window;
27 Select the option *Deact. Associated Files*.

Now we see our well.

28 Select the tab *IDFs* from the *3-D Tool* window;
29 Select the option *Transparancy* to create translucent blocks in order to see the well clearly;
30 Rotate the image with your left mouse button.

The image might look like this:

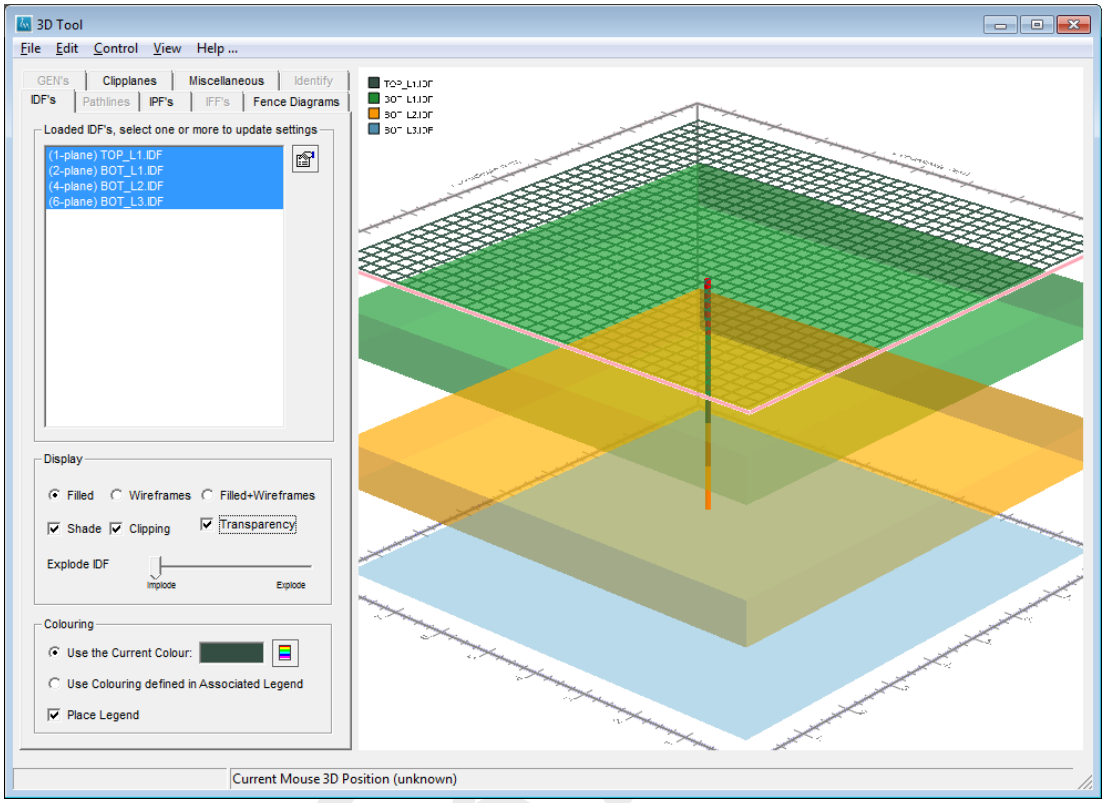


Figure 11.123: *3-D image of our model.*

The 3D-image allows you to observe that the well is penetrating all model layers; in fact the well screen is for a length of 1.0 meter in model layer 1, model layer 2 is fully penetrated and layer 3 contains 5 meters (half of the thickness of that aquifer) of the well screen. The well extracts from all three model layers, proportional to the respective length of the well screen in each layer; this will be computed by iMOD when the model definition is translated to the MF2005 WEL package. The total strength of the well is 10,000.0 m^3/d from December 1^{st} 2016 up to December 1^{st} 2040. Starting from December 1^{st} 2040 the well is turned off (0.0 m^3/d). This is specified in the WEL.TXT file associated to the WEL.IPF. Below is the content of that file.

```
2
2
DATE , -9999.0
Q , -9999.0
20161201,-10000.0
20401201, 0.0
```

31 Quit the *3-D Tool* window by clicking the option *File* and then *Quit 3-D Tool*;

Run the Model

Because we want to be able to simulate layers falling dry we apply a model with unconfined model layers. In that way the areas that fall dry are no part of the simulation until these model cells are re-wetted again. When unconfined model layers are applied in iMOD, iMOD includes the wetting option of MF2005 automatically.

32 Whenever the *Project Manager* may have disappeared, pop-it-up by selecting the menu option *View* and than *Project Manager*;

33 Click the *Save Model* button ( ) to start the *Define Simulation Configuration* window;

34 Click on the button *Define 3 Layer Types* to start the *Layer Types* window.

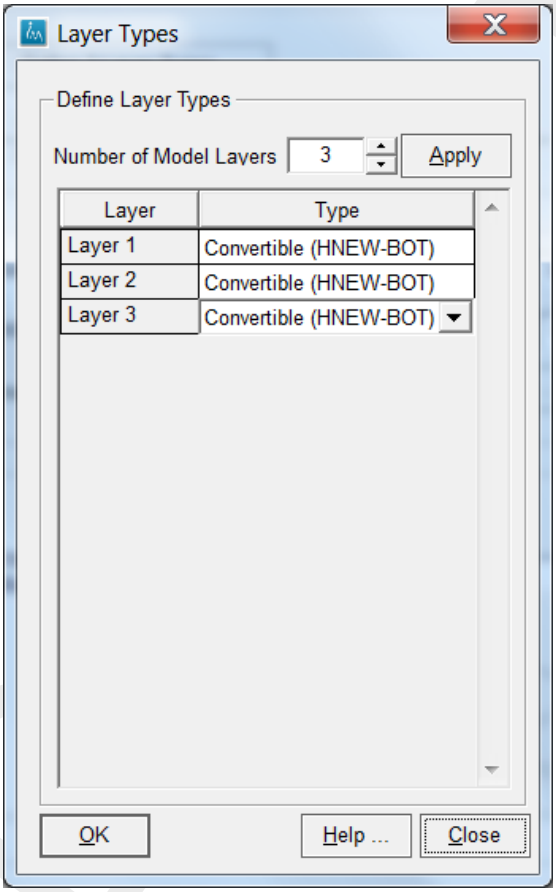


Figure 11.124: *Example of the Layer Types window: assigning layer type 'Convertible (HNEW-BOT)' to all layers.*

35 Select the option Convertible (HNEW-BOT) for model layer 1, 2 and 3. In this way all model layers will be unconfined and the transmissivity is a function of the computed head (HNEW) minus the bottom of each model layer (BOT).

It is important to know that MF2005 includes the option to simulate model cells becoming dry when the hydraulic head of that cell drops below the bottom of that model cell. To ensure that dry cells can become part of the simulation again, iMOD includes the wetdry-option in the LPF-package automatically: it is not needed to specify extra input for this option. iMOD defines the wetdry-option to all active model cells that are part of an unconfined model layer. Whenever the head underneath the dry cell (h_n) is higher than 0.1 meter above the bottom of the model layer (BOT), it becomes wet again. Using this option, is more stable than using all four adjacent model cells as well. In the iteration, the head at

that cell is than initially set to by following equation:

$$h = \mathrm{BOT} + \mathrm{WETFCT}\,(h_n - \mathrm{BOT})\,, \tag{11.4}$$

whereby WETFCT=0.1. These are programmed internally in iMOD as they give the most robust approach. However, whenever it is still needed to modify this, the (advanced) model user can modify the exported MF2005-files outside iMOD themselves.

36 Click the *OK* button;

From the PRJ-file iMOD has read the transient characteristics of your model; it starts at 1$^{\mathrm{st}}$ of December 2017 00:00:00 and ends at the same date. More input was not yet given to the model, but we can extent the simulation period of the model by simply defining another end date, let's do that.

37 Enter the year "2050" at the *Enddate*;
38 Enter "Monthly" at the *TimeSteps*;
39 Select the option "MODFLOW 2005" at the *File Format:* radio button.

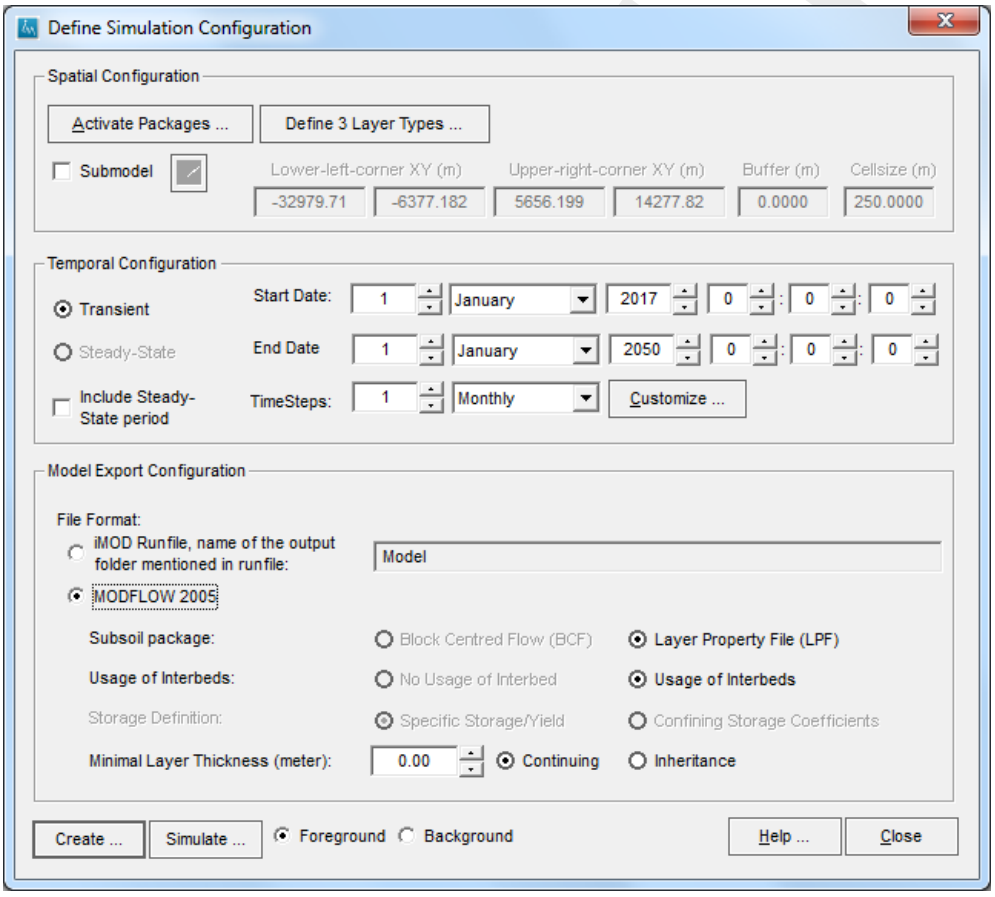


Figure 11.125: *Example of the iMOD Define Simulation Configuration window.*

After step 39 the Define Simulation Configuration window should look like in Figure 11.125.

The model will generate results on a monthly time step which is based on the definition of the input data, we can inspect this.

40 Click the *Simulate* button and enter the following *.NAM file {path of installfolder} \IMOD_USER \MODELS \TUT_WEL \TUT_WEL.NAM to export the model to MODFLOW2005 files and start the simulation immediately. Note, that you need to create the folder TUT_WEL yourself first, use the option *New Folder* in your current *Save* window.

iMOD will now first create the necessary MODFLOW2005 files; as the model is tiny this will be will finished quickly. Immediately thereafter the simulation starts. You'll see that the model starts in a separate DOS-command window and it will echo the simulation progress. It is a transient simulation of 408 stress periods, it probably will take something like 20 seconds of runtime (e.g. on a computer with a 2.6 GHz processor).

Inspect the result of simulation

Let's inspect the hydraulic head of the first model layer and the well rates and generate time series.

41 Select the option *Map* and then the option *Quick Open* to start the *Quick Open* window, see section 6.2. With this window it is easy to open and view results from a model simulation.
42 Select the option "HEAD" from the *Topic* dropdown menu.
43 Select the option "20171201" from the *Time:* dropdown menu.
44 Select the options "1", "2" and "3" from the *Layer* dropdown menu. Tip: drag your mouse to select multiply entries of the menu field.
45 Click the *Open* button; the specified head maps are added to the *iMOD Manager*;
46 Repeat the above mentioned steps to open the results for BDGWEL as well;
47 Click the *Close* button to leave the *Quick Open* window.

iMOD will load all selected result files into the *iMOD Manager* and displays the result on the graphical canvas. Use your experience learned from the previous Tutorials to display the computed heads (HEAD) and the extraction rate (BDGWEL) as time series as shown in the following figure. Be aware that it can take a few seconds, as iMOD needs to open 1224 files.

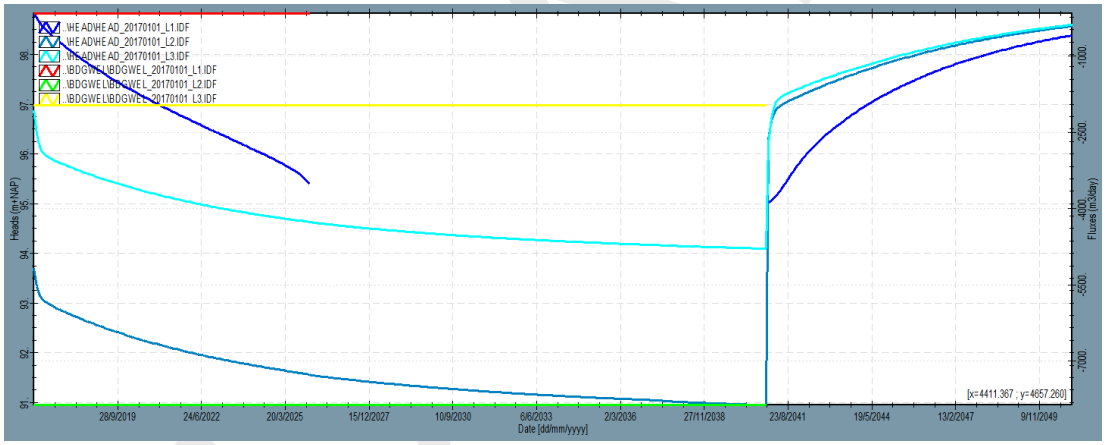


Figure 11.126: *Time Series of computed hydraulic heads and abstraction rates at the location of the well using the WEL package: heads in layer 1 (blue line), layer 2 (turquoise line) and layer 3 (cyan line), abstraction rates [m3/day] in layer 1 (red line), layer 2 (green line) and layer 3 (yellow line).*

Note: The extraction in model layer 1 is inactive as soon as the layer becomes dry. Also observe that the layer is re-wetted and the extraction of layer 1 is re-activated again as a result of deactivation of the extractions in the other model layers.

The total extraction for the entire duration of the model is less than was assigned to the model. So, instead of taking out 87.8E^6 m^3/d, the amount of extracted water was 86.8E^6 m^3/d. One of the advantages of the MNW package is that the total extracted amount remains intact once a model layer falls dry. Another improvement is that the extraction rate declines gradually as a model layers tends to dry, instead of abrupt as with the WEL package. The other layers will get an increased extraction for those case. Let's observe that in the coming part of this tutorial.

Creating the Multi-Node Well (MNW) input

48 Select the option *View* and then select *Project Manager* to start the *iMOD Project Manager* window;

49 Click the *Open Projectfile* button ( ) and select the file {path of tutorialfolder} \TUT_MNW \IMOD_USER \RUNFILE \MODEL_MNW.PRJ;

50 Click the *Open* button;

This will clean the entire *Project Manager* first before loading in the selected PRJ file.

51 Select the option (MNW) in the tree view *Project Definition*;

52 Click the *Draw* button ( ) to add the well file to the *iMOD Manager*;

53 Select the option *Map*, *IPF Options* and than *IPF Analyse ...* to start the *IPF Analyse* window.

54 Click your right mouse button on the graphical canvas and select the option *Select the Entire Domain*.

Now we have selected our MNW-well and the values for the different attributes are presented in the table. We can see that that screens of the well starts at 96.0 m+MSL and ends at 70.0 m+MSL. This is similar to our previous well modelled by the conventional WEL package. You can also see that the methodology of computing well loss is given by the keyword THIEM and the appropriate parameter relevant to that is the RADUIS ($r_w = 0.25$ m), see section section 12.30 for more detailed information about the MNW package.

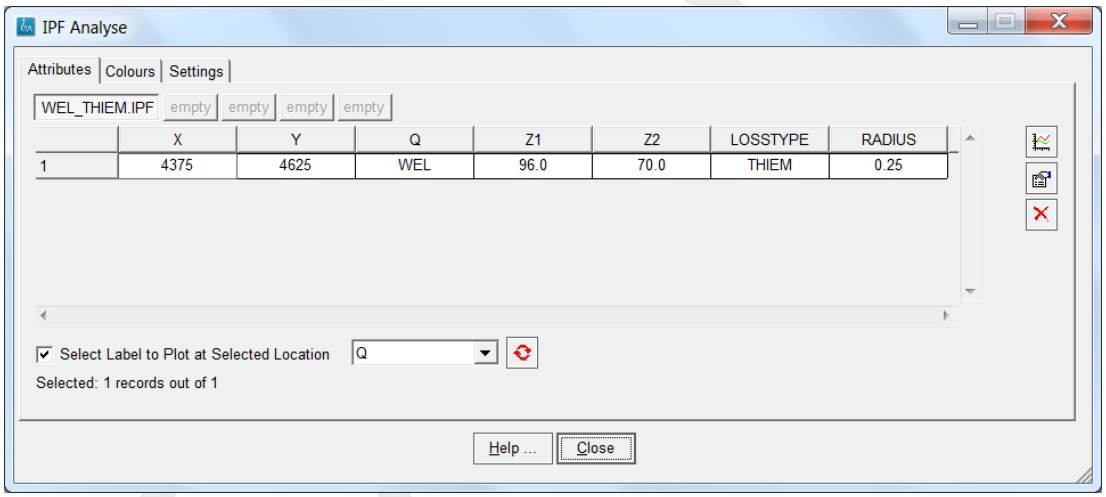


Figure 11.127: *Attribute values for the MNW-well.*

MNW computes a hydraulic head in the cell h_n such that it equals the computed hydraulic head at the well minus a head loss term (e.g. the Thiem equation, see Konikow *et al.* (2009)). Here we neglect head loss due to skin and local turbulence effects for that particular cell, so:

$$h_{\mathrm{WELL}} - h_n = \frac{Q_n}{2\pi T} \ln \frac{\mathrm{r_0}}{\mathrm{r_w}}, \tag{11.5}$$

where Q_n is the well rate (m^3/d), T is transmissivity of the aquifer (m^2/d) at the well, r_0 is the effective radius of a finite-difference cell (m), this is assumed for isotropic conditions as $r_0 = 0.14\sqrt{\Delta x^2 + \Delta y^2}$; and r_w is the actual radius of the well.

Note: Because r_0 is typically much larger than r_w, the head in a pumping well will be lower than the model-computed head. The head in the pumping well is not equal to the hydraulic head saved by the model.

Okay, let's run the model with the MNW package.

55 Apply steps 32 and further;
56 Click the *Simulate* button and enter the following *.NAM file {path of installfolder} \IMOD_USER \MODELS \TUT_MNW \TUT_MNW.NAM to export the model to MODFLOW2005 files and start the simulation. Note, that you need to create the folder TUT_MNW yourself first, use the option *New Folder* in your current *Save* window.

Again, iMOD will first create the necessary MODFLOW2005 files and start the simulation immediately. Similar to the model using the WEL package, this model including the MNW package will also probably take no more than something like 20 seconds to run.

Compare the result of WEL and MNW simulation

Let's inspect the hydraulic head of the first model layer and the computed distribution of extraction rates and generate time series.

57 Select the option *Map* and then the option *Quick Open* to start the *Quick Open* window, see section 6.2. With this window it is easy to open and view results from a model simulation.
58 Select the option "TUT_MNW" from the *Variant* dropdown menu.
59 Select the option "HEAD" from the *Topic* dropdown menu.
60 Select the option "20171201" from the *Time:* dropdown menu.
61 Select the options "1", "2" and "3" from the *Layer* dropdown menu. Tip: drag your mouse to select multiply entries of the menu field.
62 Click the *Open* button;
63 Repeat the above mentioned steps to open the results for BDGMNW as well;
64 Click the *Close* button to leave the *Quick Open* window.

iMOD will load all selected result files into the *iMOD Manager* and displays the result on the graphical canvas.

65 Use your experience learned from the previous Tutorials to display the extraction rate (BDGMNW) as time series as shown in the following figure. Be aware that it can take a few seconds, as iMOD needs to open 1224 files.

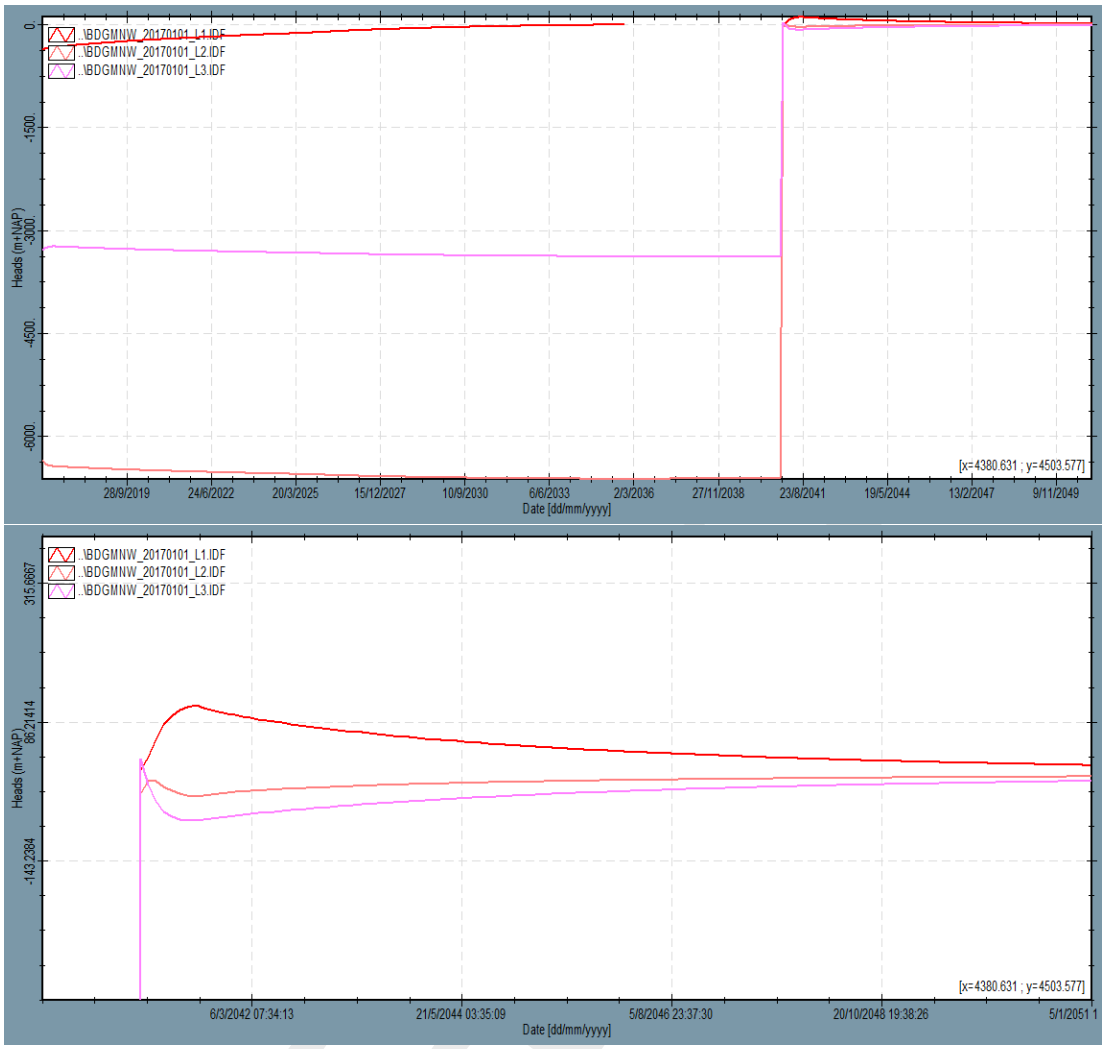


Figure 11.128: *Time Series of computed extraction rates using the MNW package in layer 1 (red), layer 2 (orange) and layer 3 (violet); total time series (above) and zoomed in from 2040 onwards (below).*

As expected, you might observe that the total extraction rate varies and that the extraction rate for the first model layer slowly decrease to zero. At the same time the extraction of the deeper aquifers, increases to sum up to 10,000 m^3/d. If we look at the zoomed in image (bottom) for the period after 2040, where we turned off the well, we observe that the well rates vary although there is no external rate specified.

So, what is happening?

Well, one of the features of the MNW package is the capability of simulating intra borehole flow, actually water can move from one aquifer - through the borehole - to another aquifer. Due to the stopping of the pumping, the deeper aquifers recover quicker from the computed draw down than the unconfined aquifer, mainly due to the low storage coefficient. This causes an overpressure from the deep aquifers to the shallow one and generates a groundwater flow that migrates directly through the borehole into the first aquifer.

66 Use your experience learned from the previous Tutorials to display the hydraulic head (HEAD) for our model with the WEL- and MNW package as time series as shown in the following figure. Be aware that it can take a few seconds, as iMOD need to open 1632 files.

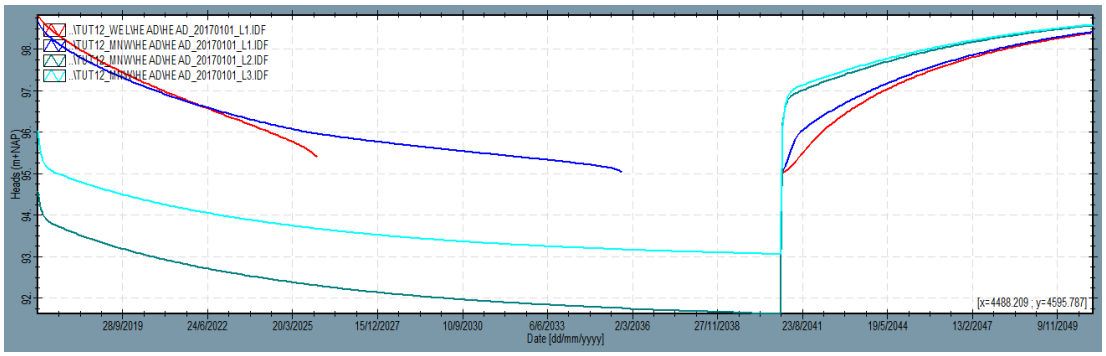


Figure 11.129: *Time Series of computed hydraulic heads at the location of the abstraction well: in layer 1 using the WEL package (red line), and heads in layers 1 to 3 using the MNW package (blue, turquoise and cyan lines respectively).*

The figure of timeseries of computer hydraulic heads at the location of the well clearly shows that there is an overpressure that causes this intra borehole flow. Moreover, when comparing the hydraulic heads in layer 1 cells at the well location, in the model with the MNW package the cell remains wet for a longer period of time compared to the model with the WEL package. This is caused by the MNW package decreasing the extraction amount gradually and therefore decreases the draw down rate of the ground water head. Due to the early mentioned intra borehole flow, the heads in the model with an MNW package recover more quickly than in the model with the WEL package.

So the MNW package can really add some extra features concerning the behaviour of a well in your model. Speaking of more detail, it seems that there is a horizontal barrier (sheet pile wall) blocking the flow to our well. Let's see how to incorporate this with iMOD into our model.

Enhancing the model with a Horizontal Flow Barrier (HFB) input

Let's create our sheet pile wall.

67 Select from the main menu the option *Edit*, *Create Features* and then *GENs ...* to start the *Create GENs* window.
68 Click the *Draw* button ( ), this will start the *Select* window;
69 Select the option *Line* from the *Shape types*;
70 Click the *Ok* button.
71 Start drawing a line at the west side of the well. Click your left mouse button to position the first point of the line. Each left mouse click will insert another point.
72 Click your right mouse button to finish the drawing.

The following line could be on your screen.

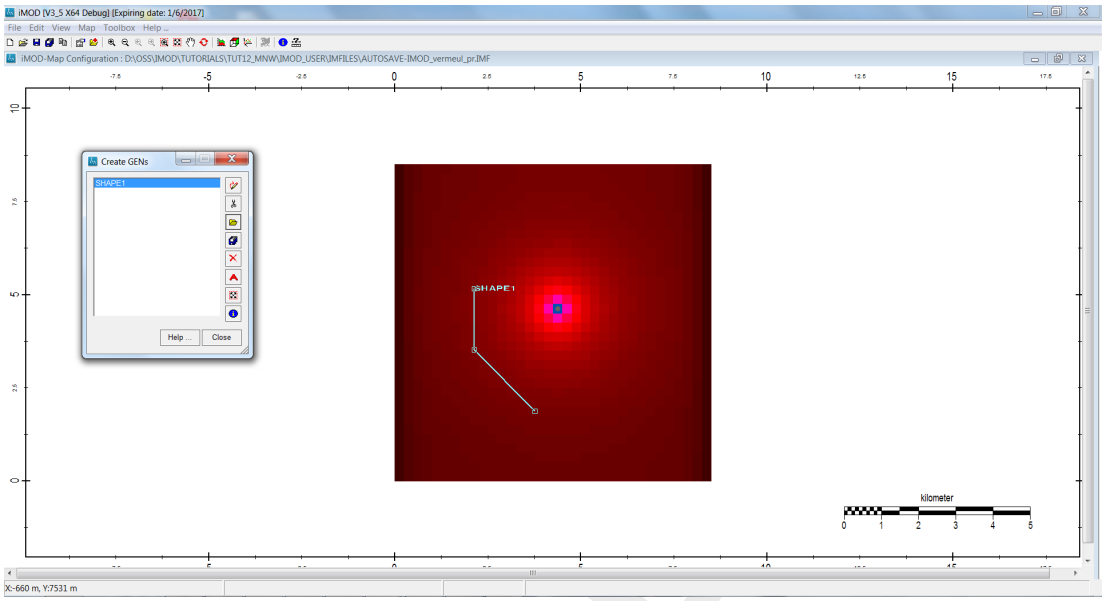


Figure 11.130: *Outline of our sheet pile.*

Let's save the sheet pile wall.

73 Click the *Save As* button ( ) and save the sheet pile at {path of installfolder} \IMOD_USER \DBASE \TUT_MNW \SHEET_PILE.GEN.
74 Click the *Save* button.
75 Click the *Close* button to close the *Create GENs* window.

Now we have to add this sheet pile to our modelling project.

76 Select the option *View* and then select *Project Manager* to start the *iMOD Project Manager* window;
77 Select the option (HFB) in the tree view *Project Definition*;
78 Click *Properties* button ( ) to start the *Define Characteristics for* window.
79 Enter a value of "2"at the *Assign Parameter to model layer ...*, our pile sheet wall will act as barrier for the second model layer only;
80 Enter a value of "1000.0"at the *Assign Parameter Addition Value*, our pile sheet wall will have a resistance of 1000.0 days;
81 Click the *Add File* button ( ) and select the file we just created: {path of installfolder} \IMOD_USER \DBASE \TUT_MNW \SHEET_PILE.GEN.

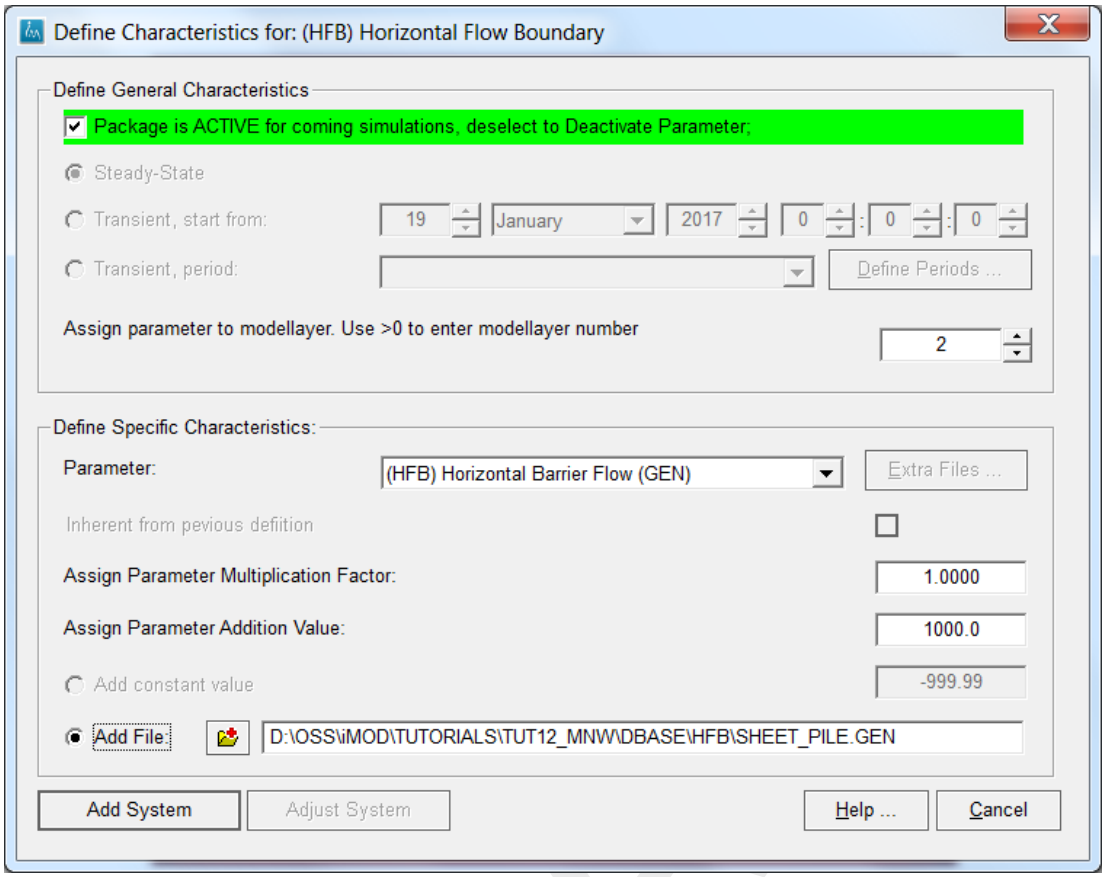


Figure 11.131: *Example of the iMOD Project Manager window.*

82 Click the *Add New System* button to add the parameter to the modeling project; the *Define Characteristics for* window will be closed.

83 Click the *Draw* button ( ) to open the sheet pile to the *iMOD Manager*;

84 Click the *Save As* button ( ) and save a new modeling project file at {path of installfolder} \IMOD_USER \RUNFILES \MODEL_HFB.PRJ.

So, we're ready to run this model.

85 Apply steps 32 and further;

86 Click the *Simulate* button and enter the following *.NAM file {path of installfolder} \IMOD_USER \MODELS \TUT_HFB\TUT_HFB.NAM to export the model to MODFLOW2005 files and start the simulation. Note, that you need to create the folder TUT_HFB yourself first, use the option *New Folder* in your current *Save* window.

Also here again iMOD will first create the necessary MODFLOW2005 files and starts the simulation. Similar to the previous 2 models, this model including the MNW + HFB package will also probably take no more than something like 20 seconds to run.

Compare the results of the MNW and HFB simulation

Let's inspect the hydraulic head of the first model layer and generate time series.

87 Click the *Close* button to leave the *Define Simulation Configuration* window;

88 Select the option *Map* and then the option *Quick Open* to start the *Quick Open* window, see section 6.2. With this window it is easy to open and view results from a model simulation.

89 Select the option "TUT_HFB" from the *Variant* dropdown menu.

90 Select the option “HEAD” from the *Topic* dropdown menu.
91 Select the option “20171201” from the *Time:* dropdown menu.
92 Select the options “2” from the *Layer* dropdown menu.
93 Click the *Open* button;
94 Click the *Close* button to leave the *Quick Open* window.

iMOD will load all selected result files into the *iMOD Manager* and displays the result on the graphical canvas.
During the simulation iMOD translates the manually drawn sheet pile wall - which we saved earlier as SHEET_PILE.GEN - to a continuous (kinked) line coinciding exactly with the lateral cell faces it intersects; when utilizing the HFB package the specified resistance is assigned to these cell faces. It is always a good idea to examine the result of such a translation, e.g. to check whether the discretization has resulted in a sheet pile wall that is fully continuous and thus behaving like a true barrier. Let’s open that file.

95 Click the *Add File* button () from the main toolbar and select the file {path of installfolder} \IMOD_USER \MODELS \TUT_HFB\MODELINPPUT \TUT_HFB_L2.GEN.
96 Select the file SHEET_PILE.GEN and HEAD_20171201_L2.IDF from the *iMOD Manager* window.
97 Click the *Redraw* button ().
98 Use your experience to change the colour of the lines.

You should see, more-or-less, the following image. In white is the actual position of the sheet pile in the model. Due to the chosen grid size, it is a little bit shifted and crenelated due to the rectangular simulation network.

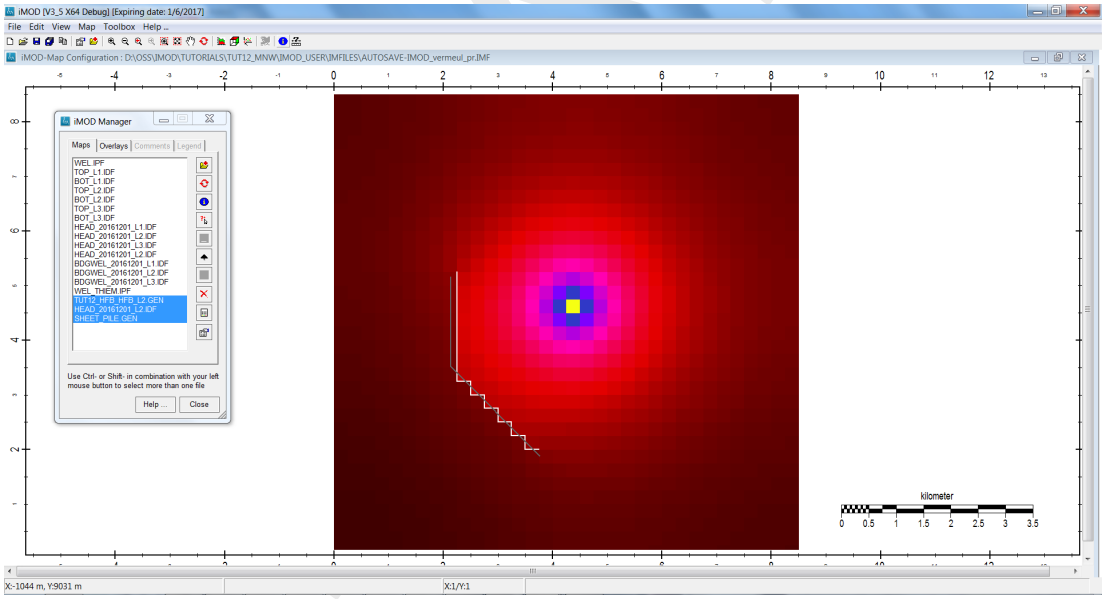


Figure 11.132: *Display of the possible outcome of our HFB model.*

You could try to experiment with more complex shapes for the HFB and/or modify the resistance of the sheet pile.

11.11 Tutorial 11: Unsaturated Zone Package

This tutorial gives an introduction to a transient implementation of the Unsaturated Zone package (UZF), see section 12.31.

Outline

This is what you will do:

- ◇ Create a transient PRJ file with TOP, BOT, KHV, RCH and EVT package;
- ◇ Simulate the RCH and EVT package for an unconfined model and examine the results;
- ◇ Modify the PRJ file with the UZF package;
- ◇ Simulate the UZF package and examine the results and compare it with the conventional RCH and EVT model;
- ◇ Modify the parameters of the UZF package to see the impact of parameters;

Required Data

For this tutorial you need the following iMOD Data Files/folders:

- ◇ The entire folder (and subfolders) in {path of tutorialfolder} \TUT_UZF \DBASE, containing:
  - ▫ . \TOP \TOP_L*.IDF – IDF files with top of model layers (3);
  - ▫ . \BOT \BOT_l*.IDF – IDF files with bottom of model layers (3);
  - ▫ . \PREC \PREC_*.IDF – IDF files with precipitation on a daily base;
- ◇ MODEL_RCH_EVT.PRJ – model project file with RCH and EVT package (to be created);
- ◇ MODEL_UZF.PRJ – model project file with UZF package (to be created);

Getting Started

1. Start iMOD.
2. Select the option *Create a New iMOD Project*.
3. Click the *Start* button;

Create a PRJ file

We will quickly generate a PRJ file using the auto fill option of the *Project Manager*.

4. Select the option *View* and then select *Project Manager* to start the *iMOD Project Manager* window;
5. Select the option (TOP) in the tree view *Project Definition*;
6. Click *Define Characteristics Automatically* button () to start the *Define Characteristics for* window.
7. Add the following string to the second column in the table “{path of tutorialfolder} \TUT_UZF \DBASE \TOP \TOP_L*.IDF”;
8. Select the option *Select files within Given Layer Range* and enter “1” and“3” in the input fields to the right.

Your window should look as follows:

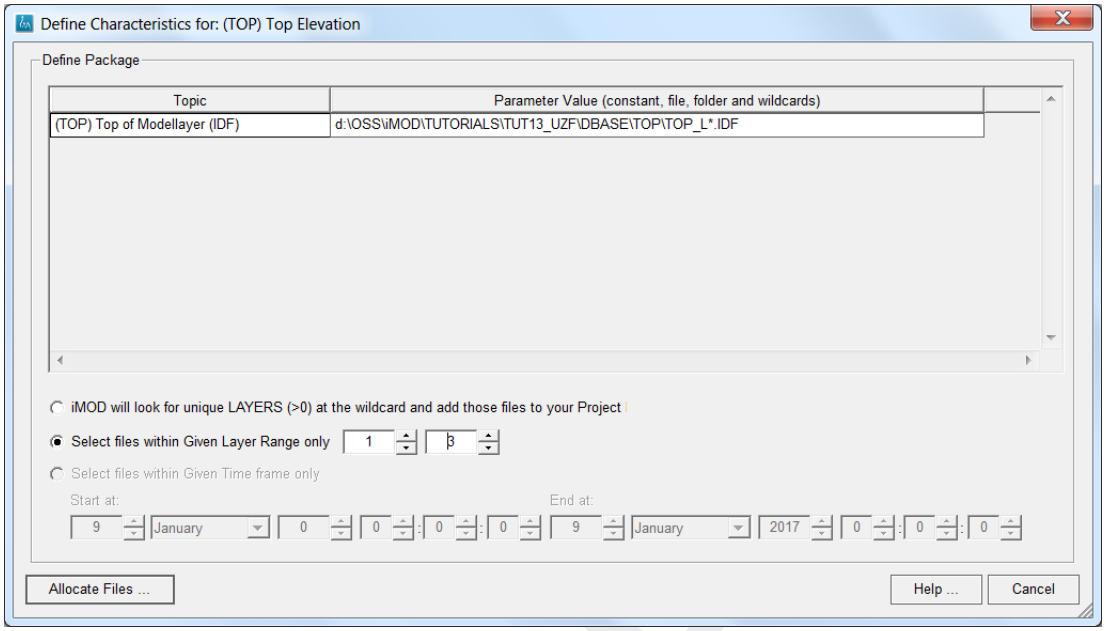


Figure 11.133: *Example of the Define Characteristics Automatically window.*

In this manner we tell iMOD to add the given files for model layer 1,2 and 3 automatically. Let's see how that works.

9 Click the *Allocate Files ...* button.

iMOD will pop-up a window with the files found in the *Automatic Package Allocation* window.

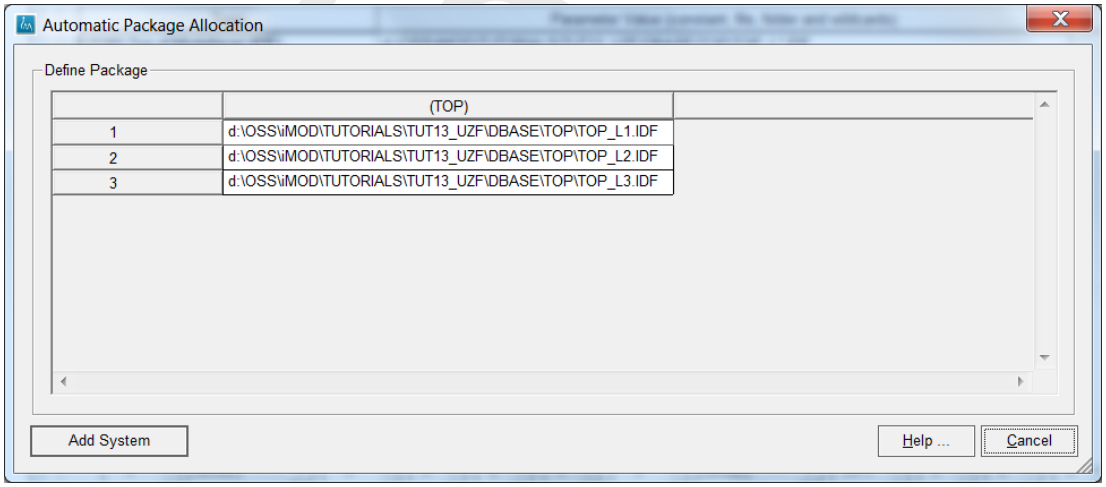


Figure 11.134: *Example of the Automatic Package Allocation window.*

Here we can inspect the results, or even modify this. We leave it like this as it is correct and continue.

10 Click the *Add System* button, this will add the files to our modelling project and closes the *Automatic Package Allocation* window.
11 Repeat the steps 5 up to 10 for the bottom of the model layers (keyword is BOT), these are stored in "{path of tutorialfolder} \TUT_UZF \DBASE \BOT \BOT_L*.IDF".

Also constant values for a package or modules can be inserted in this manner, as the boundary conditions of our three-layered model are all equal one, we insert this information as follows:

12 Select the option (BND) in the tree view *Project Definition*;
13 Click *Define Characteristics Automatically* button () to start the *Define Characteristics for* window.
14 Add the following string to the second column in the table “1”;
15 Select the option *Select files within Given Layer Range* and enter “1” and“3” in the input fields to the right.
16 Click the *Allocate Files ...* button.
17 Click the *Add System* button, this will add the files to our modelling project and closes the *Automatic Package Allocation* window.

That’s convenient, right? Let’s add the constant information for the other packages repeating the steps 12 up to 17.

18 Add a constant starting head of “95.0” m+MSL for the SHD package (starting heads) for the model layers 1,2 and 3;
19 Add a constant starting head of “10.0” m/d for the KHV package (horizontal permeability) for the model layers 1,2 and 3;
20 Add a constant starting head of “1.0” for the KVA package (vertical anisotropy) for the model layers 1,2 and 3;
21 Add a constant starting head of “0.3” for the SPY package (specific yield) for the model layers 1,2 and 3.

Next thing is to add the confined storage coefficients (STO).

22 Select the option (STO) in the tree view *Project Definition*;
23 Click *Define Characteristics Automatically* button () to start the *Define Characteristics for* window.
24 Add the following string to the second column in the table “0.2E-03”;
25 Select the option *Select files within Given Layer Range* and enter “1” and“3” in the input fields to the right.
26 Click the *Allocate Files ...* button.
27 Modify the value “0.2E-03” for model layer 1 into “1.0”;
28 Click the *Add System* button, this will add the files to our modelling project and closes the *Automatic Package Allocation* window.

Almost done, we will add the Evaporation package (EVT) on the conventional way as it will be defined for our first stress period only.

29 Select the option (EVT) in the tree view *Project Definition*;
30 Click *Properties* button () to start the *Define Characteristics for* window.
31 Select the option *Transient, start from* and enter the date “6 October 2013” in the date entry fields;
32 Select the option *Add constant value*;
33 We will enter the following values for the different input parameters by selecting the appropriate parameter from the *Parameter* dropdown list sequentially:
   - ◇ **(EVA) Evapotranspiration Rate (IDF)** = “10.0” mm/day;
   - ◇ **(SUR) Surface Level (IDF)** = “100.0” m+MSL;
   - ◇ **(EXD) Extinction Depth (IDF)** = “8.0” m.
34 Click the *Add New System* button to add the parameter to the modelling project and close the *Define Characteristics for* window.

Note: By defining a surface level (SUR) of 100.0 m+MSL and an extinction depth (EXD) of 8.0, there will be a linear reduction of evaporation with depth. At a depth of 8 m+MSL (2 meter above the bottom of model layer 1), the evaporation is 0.0.

Precipitation is the input parameter that varies per stress period. We will add that parameter with the automatic package allocation.

35 Select the option (RCH) in the tree view *Project Definition*;
36 Click *Define Characteristics Automatically* button () to start the *Define Characteristics for* window.
37 Add the following string to the second column in the table “{path of tutorialfolder} \TUT_UZF \DBASE \PREC \PREC_*.IDF”.;
38 Select the option *iMOD will look for unique TIME STEPS ...*;
39 Click the *Allocate Files ...* button.
40 Click the *Add System* button, this will add the files to our modelling project and closes the *Automatic Package Allocation* window.

We have add now 459 definitions for precipitation in a single mouse click, that’s awesome isn’t it. Final thing to do is to add the configuration for the PCG solver.

41 Select the option (PCG) in the tree view *Project Definition*;
42 Click *Properties* button () to start the *PCG settings* window.
43 Click the *Apply* button to add the parameter to the modelling project and close the *PCG settings* window.

Okay, we’re done, let’s save the modelling project.

44 Click the *Save As* button () and save a new modeling project file at {path of installfolder} \IMOD_USER \RUNFILE \MODEL_RCH_EVT.PRJ.

Simulate the model

With this project file we generate a standard MODFLOW2005 model, let’s do that.

45 Click the *Save Model* button () to start the *Define Simulation Configuration* window;
46 Click on the button *Define 3 Layer Types* to start the *Layer Types* window.

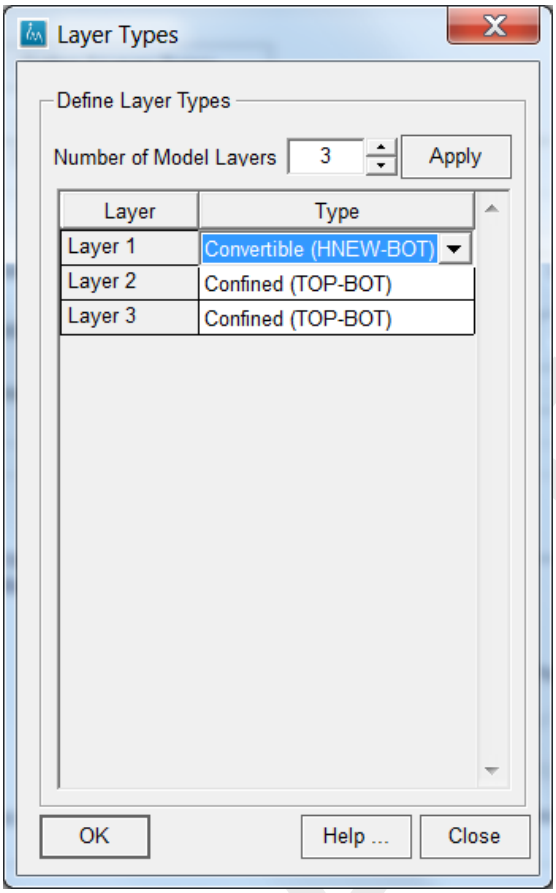


Figure 11.135: *Example of the Layer Types window: assigning layer type 'Convertible (HNEW-BOT)' to layer 1.*

47 Select the option Convertible (HNEW-BOT) for model layer 1 by clicking on the cell of the second column. In this way the first model layer will be unconfined and the transmissivity is a function of the computed head (HNEW) minus the bottom of the model layer (BOT);
48 Click the *OK* button;
49 Select the option "MODFLOW 2005" at the *File Format:* radio button.

iMOD has found out what the transient content of your model is, it starts at 6st of October 2013 00:00:00 and ends at the 7st of January 2015 00:00:00. based on the entered precipitation (RCH) and evaporation (EVT) data.

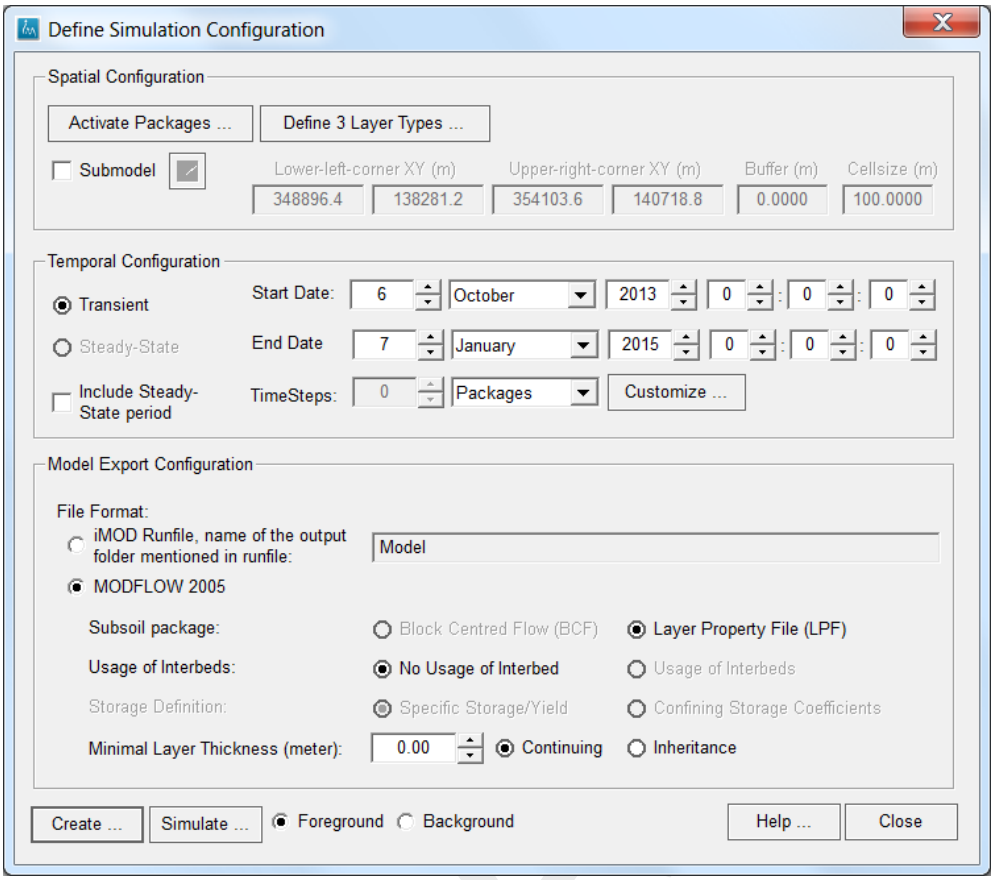


Figure 11.136: *Example of the iMOD Define Simulation Configuration window.*

The model will generate results on a daily time step which is based on the occurrence of the input data, we can inspect this.

50 Click the option *Customize ...* (on the *Define Simulation Configuration* window at the *Temporal Configuration* settings) to start the *Time Discretization Manager for Simulation* window.

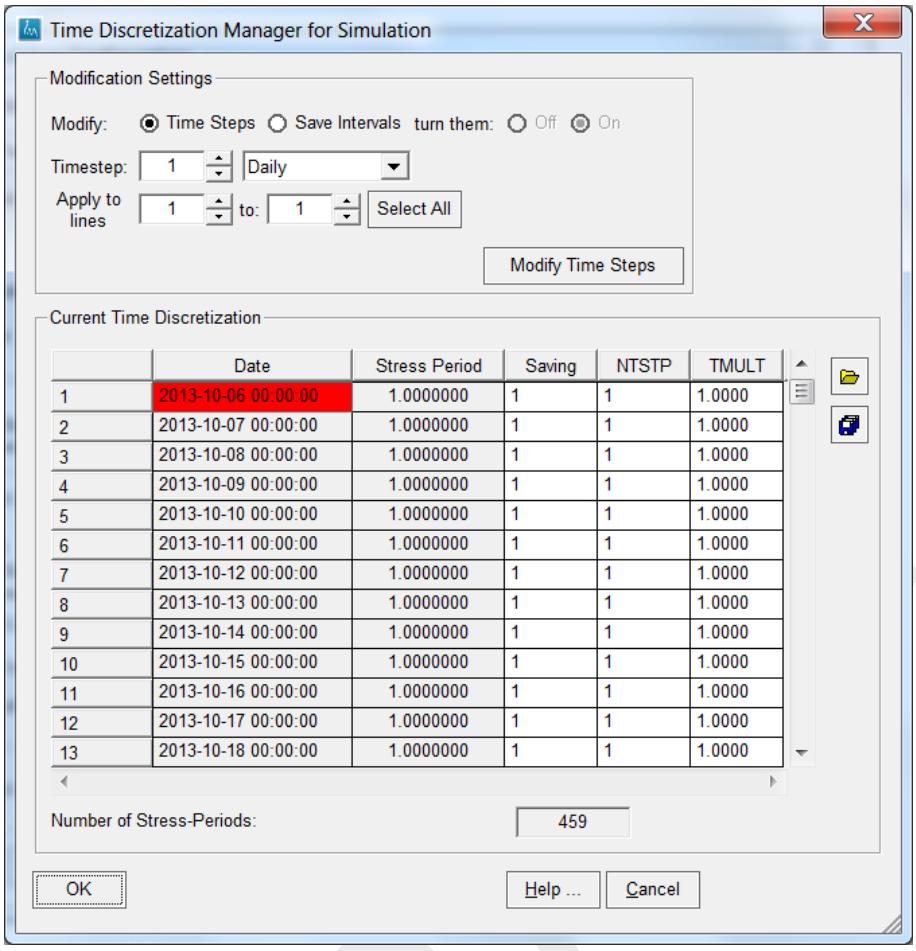


Figure 11.137: *Example of the iMOD Time Discretization Manager for Simulation window.*

We will leave the definition of the stress-periods for now.

51 Click the *OK* button to close the *Time Discretization Manager for Simulation* window.
52 Click the *Simulate* button and enter the following *.NAM file {path of installfolder} \IMOD_USER \MODELS \TUT_RCH_EVT \TUT_RCH_EVT.NAM to export the model to MODFLOW2005 files and start the simulation. Note, that you need to create the folder TUT_RCH_EVT yourself first, use the option *New Folder* in your current *Save* window.
53 Click the *Save* button.

iMOD will now create the necessary MODFLOW2005 file and runs the model, as the model is tiny, this will be finished rapidly. It will start the simulation directly thereafter. You'll see that the model start in a separate DOS-command window and it will echo the simulation progress. As it is a transient simulation with 458 stress periods, it will consume probably 10 seconds to accomplish.

Inspect the result of simulation

Let's inspect the hydraulic head of the first model layer and the computed recharge (equal to the input in fact) and generate time series.

54 Select the option *Map* and then the option *Quick Open* to start the *Quick Open* window, see section 6.2. With this window it is easy to open and view results from a model simulation.
55 Select the option "HEAD" from the *Topic* dropdown menu.
56 Select the option "20131006" from the *Time:* dropdown menu.

57 Select the option “1” from the *Layer* menu.
58 Click the *Open* button.
59 Repeat the above mentioned steps to open the results for BDGRCH as well.

iMOD will load all selected results files into the *iMOD Manager* and displays the result on the graphical canvas. Use your experience learned from the previous Tutorials to display the computed heads as time series as shown in the following figure.

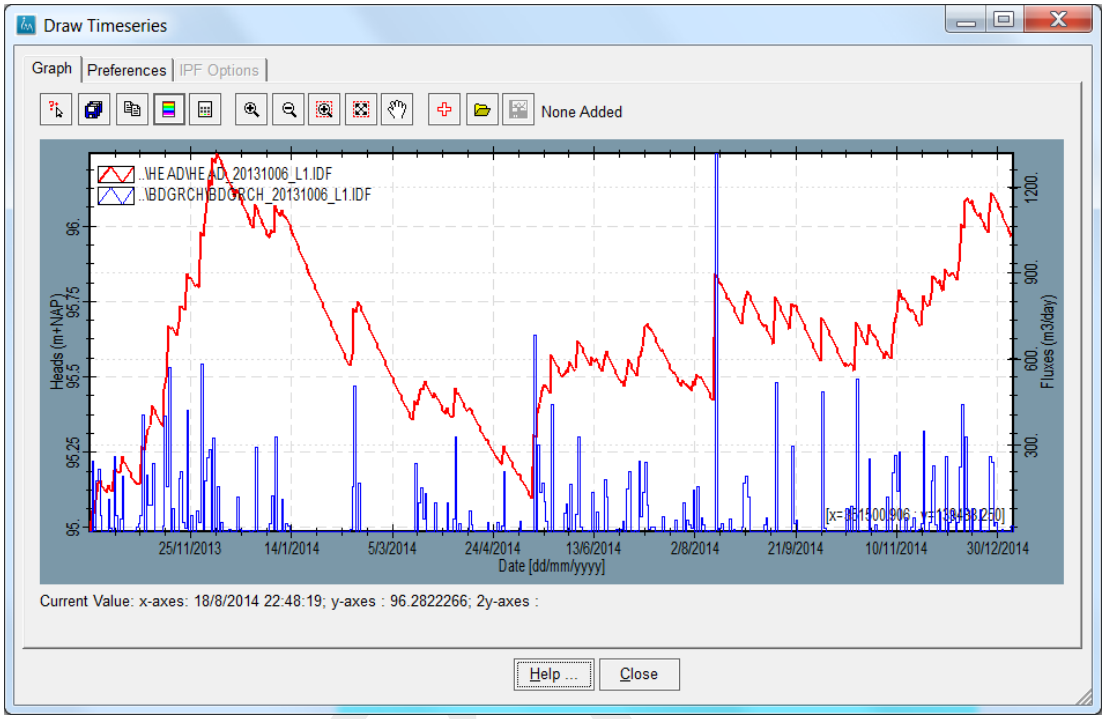


Figure 11.138: *Time Series of computed groundwater levels and precipitation.*

Note: Modify the settings of the time series to assign the recharge from the BDGRCH on the second y-axes and use “BlockLines” as *LineStyle*.

As you may have noticed, the groundwater levels respond directly to the net-recharge. This is often, especially by these deep groundwater levels (> 2 meter depth) unrealistic. A delay of recharge through the unsaturated zone is a process that is taken care of by the UZF package.

Creating the UZF input

So, this tutorial is called *UZF Package*, but up to now we didn’t do anything with that. Well, most of the preparation we have done and running the model with the RCH and EVT packages, gives us a nice comparison of two different commonly used concepts. First we will delete the RCH and EVT packages.

60 Select the option (RCH) in the tree view *Project Definition*;
61 Click the *Delete* option and confirm that you really want to remove the package content;
62 Select the option (EVT) in the tree view *Project Definition*;
63 Click the *Delete* option and confirm that you really want to remove the package content;

Now we will add the input for the UZF package which is more-or-less a combination of the RCH and EVT package input.

64 Select the option (UZF) in the tree view *Project Definition*;

65 Click *Define Characteristics Automatically* button () to start the *Define Characteristics for* window.
66 Add the following strings to the second column in the table:

- **(AEA) Areal Extent of Active Model (IDF)** = “1”;
- **(BCE) Brooks-Corey Epsilon (IDF)** = “4”;
- **(SWC) Saturated Water Content of Unsat. Zone (IDF)** = “0.3”;
- **(IWC) Initial Water Content (IDF)** = “0.05”;
- **(INF) Infiltration Rates at Land Surface (IDF)** = “{path of tutorialfolder} \TUT_UZF \DBASE \PREC \PREC_*.IDF”;
- **(EVA) Evaporation Demands (IDF)** = “10.0”;
- **(EXD) Extinction Depth (IDF)** = “8”
- **(EWC) Extinction Water Content (IDF)** = “0.01”;

Below is an example of your current window.

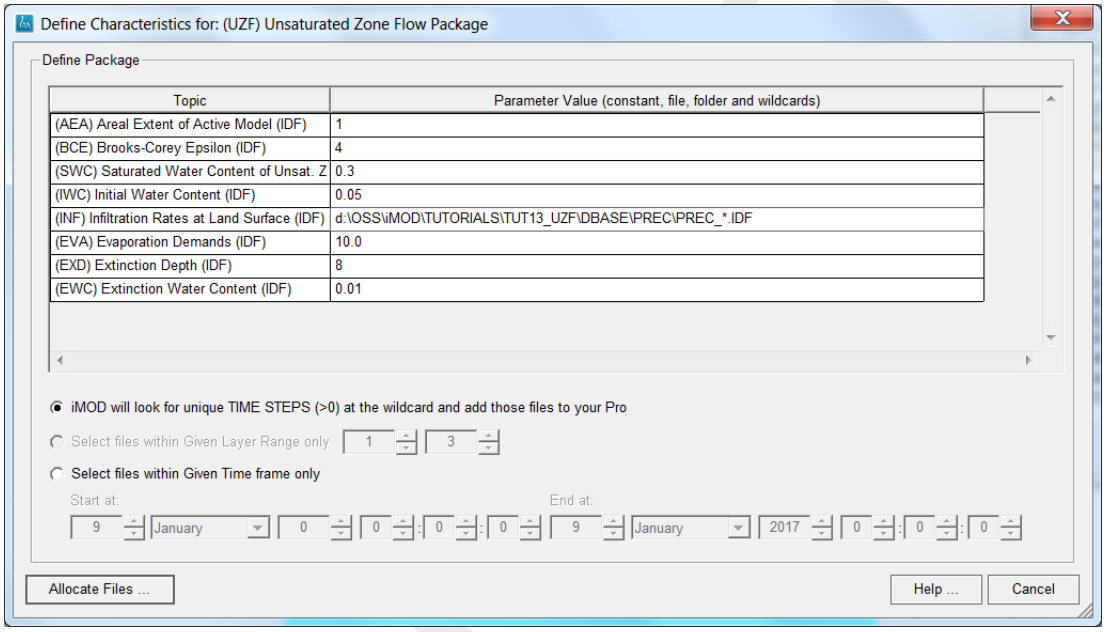


Figure 11.139: *Example of the Define Characteristics Automatically window.*

Let’s gather the appropriate files.

67 Select the option *iMOD will look for unique TIME STEPS ...*;
68 Click the *Allocate Files ...* button.

iMOD will combine the constant given values with the wild cards. At those location where the “inherent” is mentioned, iMOD will use the previous mentioned value/file.

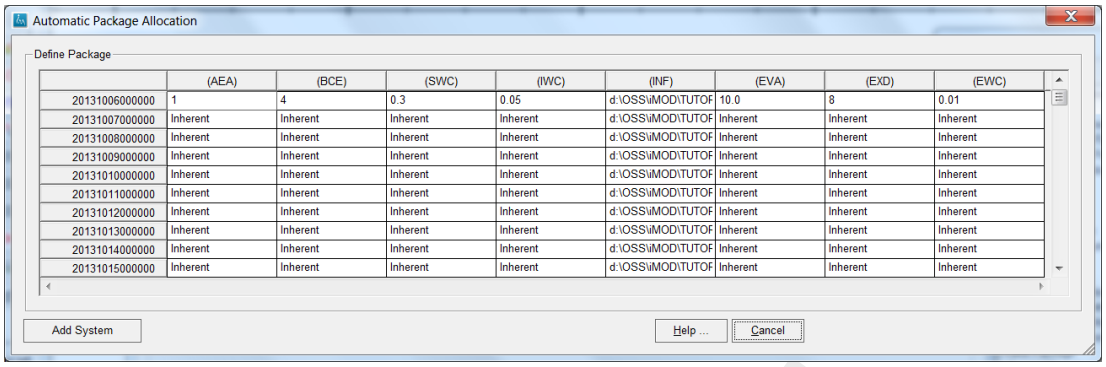


Figure 11.140: *Example of the Define Characteristics Automatically window.*

We will leave it like it is.

69 Click the *Add System* button, this will add the files to our modelling project and closes the *Automatic Package Allocation* window.

It can take a little bit longer to refill the *Project Manager*. But once it is finished, we're ready to start the model.

Model Simulation

70 Click the *Save Model* button ( ) to start the *Define Simulation Configuration* window;
71 Select the option "MODFLOW 2005" at the *File Format:* radio button.
72 Click the *Simulate* button and enter the following *.NAM file {path of installfolder} \IMOD_USER \MODELS \TUT_UZF \TUT_UZF.NAM to export the model to MODFLOW2005 files and start the simulation. Note, that you need to create the folder TUT_UZF yourself first, use the option *New Folder* in your current *Save* window.
73 Click the *Save* button.

iMOD will now create the necessary MODFLOW2005 file and runs the model, it will consume probably a little bit longer than our previous model to accomplish.

Inspect the result of simulation

Let's inspect the hydraulic head of the first model layer, the computed groundwater recharge, the precipitation and evaporation and generate time series.

74 Select the option *Map* and then the option *Quick Open* to start the *Quick Open* window, see section 6.2. With this window it is easy to open and view results from a model simulation.
75 Select the option "TUT_UZF" from the *Varient* dropdown menu.
76 Select the option "HEAD" from the *Topic* dropdown menu.
77 Select the option "20131006" from the *Time:* dropdown menu.
78 Select the options "1" from the *Layer* menu.
79 Click the *Open* button.
80 Repeat the above mentioned steps up to to open the results for:

- ◇ **BDGGET** = the amount of computed evapotranspiration (m^3/d);
- ◇ **BDGGRC** = the amount of computed groundwater recharge (m^3/d);
- ◇ **UZFET** = the amount of computed evapotranspiration in the unsaturated zone (m^3/d);
- ◇ **UZFINF** = the amount of precipitation (infiltration) in the unsaturated zone (m^3/d).

iMOD will load all selected results files into the *iMOD Manager* and displays the result on the graphical

canvas.

81 Select the files HEAD_20131006_L1.IDF for both the model TUT_RCH_EVT and TUT_UZF from the *iMOD Manager* window;
82 Add to the current selection of files the files BDGGET, BDGGRC, UZFINF and UZFET;
83 Use your experience learned from the previous Tutorials to display the selected files as time series as shown in the following figure.

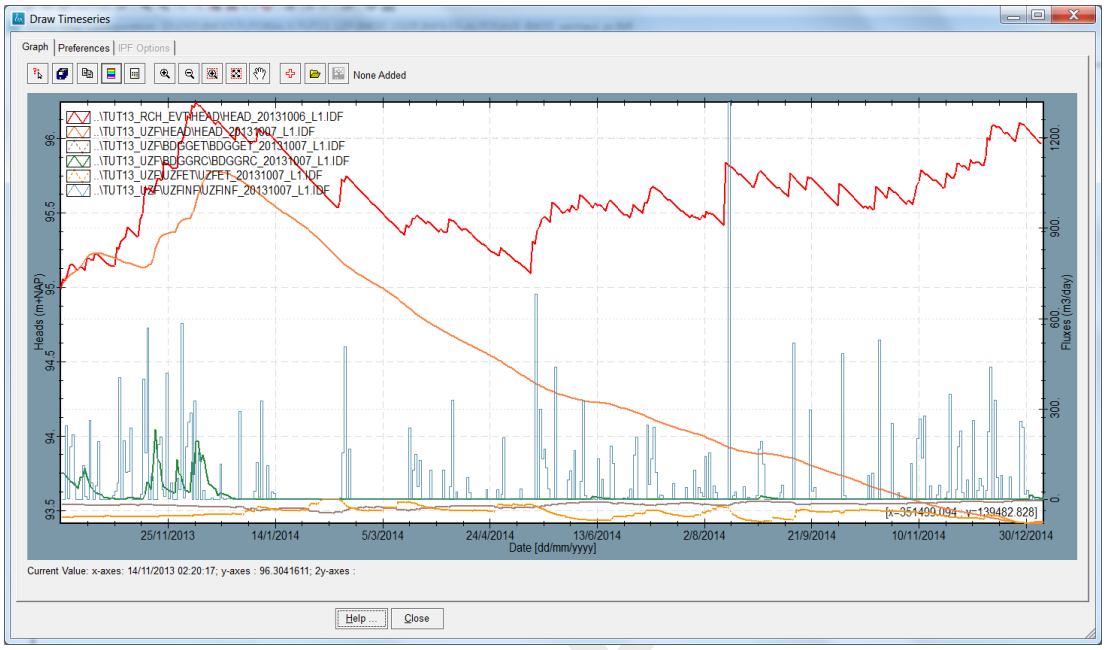


Figure 11.141: *Time Series of computed groundwater levels with the RCH and EVT and the UZF package.*

Modify the parameters of the UZF package

As you may have noticed, the UZF package generates some ground water recharge whenever multiply rainfall events appear on a short notice (dark green line). Short, heavy rainfall events, may not even feed the groundwater due to the strong evaporation or instant surface runoff. The evaporation has been modelled with an extinction depth of 8 meter, this may be a bit unrealistic and deplete the groundwater more than it should; let us change some parameters to see the effect of this.

First we should know what the Brooks-Corey Exponent (ϵ) represents. The Brooks-Corey Exponent is used to compute the unsaturated hydraulic conductivity $K(\theta)$ as a function of the moisture content θ.

$$K\left(\theta\right) = K_s \left[\frac{\theta - \theta_r}{\theta_s - \theta_r}\right]^{\epsilon}, \tag{11.6}$$

whereby K_s is the saturated hydraulic conductivity; θ_r is the residual water content; θ_s is the saturated water content; and ϵ is the Brooks-Corey exponent. This function describes how the hydraulic conductivity approaches the saturated conductivity as the water content θ increases. This can be a linear relation ship ($\epsilon = 1.0$), or a non-linear whereby the hydraulic conductivity increases more slow than the water content increases ($\epsilon > 1.0$) or faster than the water content increases ($\epsilon < 1.0$). The next figure shows the relation ship for different values of ϵ and the effect on the hydraulic conductivity $K(\epsilon)$.

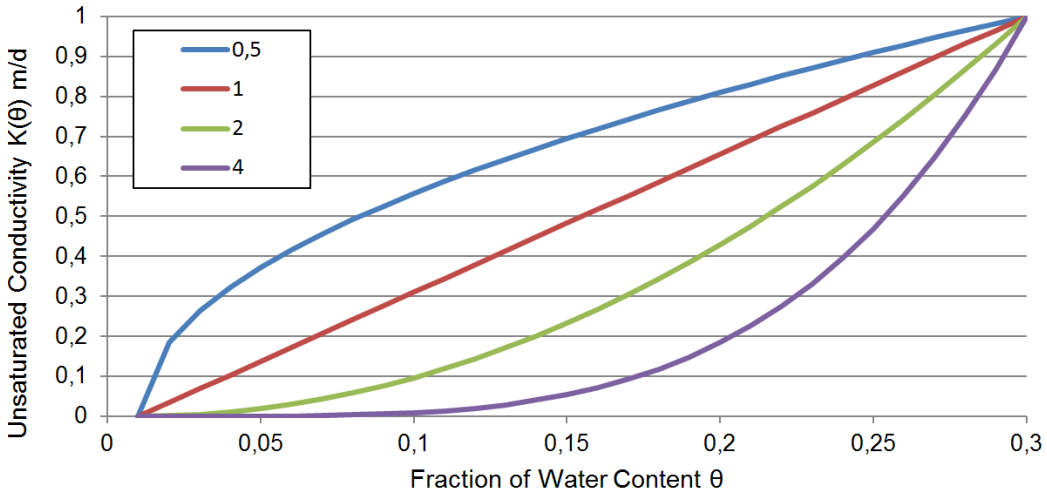


Figure 11.142: *Empirical relation between water content (θ) and hydraulic conductivity $K(\theta)$ for different values for the Brooks-Corey Exponent (ϵ).*

In our case, it seems that our model has too less amount of recharge. In order to increase the recharge we need to decrease the Brooks-Corey Exponent (ϵ). The hydraulic conductivity will increase more rapid for a slight increase of the water content, allowing precipitation seeps through the subsoil more quicker.

The easiest way to modify the parameters for the UZF package is to re-read the input, let's do that:

84 Repeat the steps 60 up to 66, but use the value 2.0 for the Brooks-Corey Exponent;
85 Repeat the steps 70 up to 73 to run the adjusted model, save the results in the folder {path of installfolder}\IMOD_USER \MODELS \TUT_UZF2;
86 Repeat the steps starting from 74 to load the results into iMOD and generate time series;

If you have successfully carried out the above mentioned steps, the time series should like the figure below.

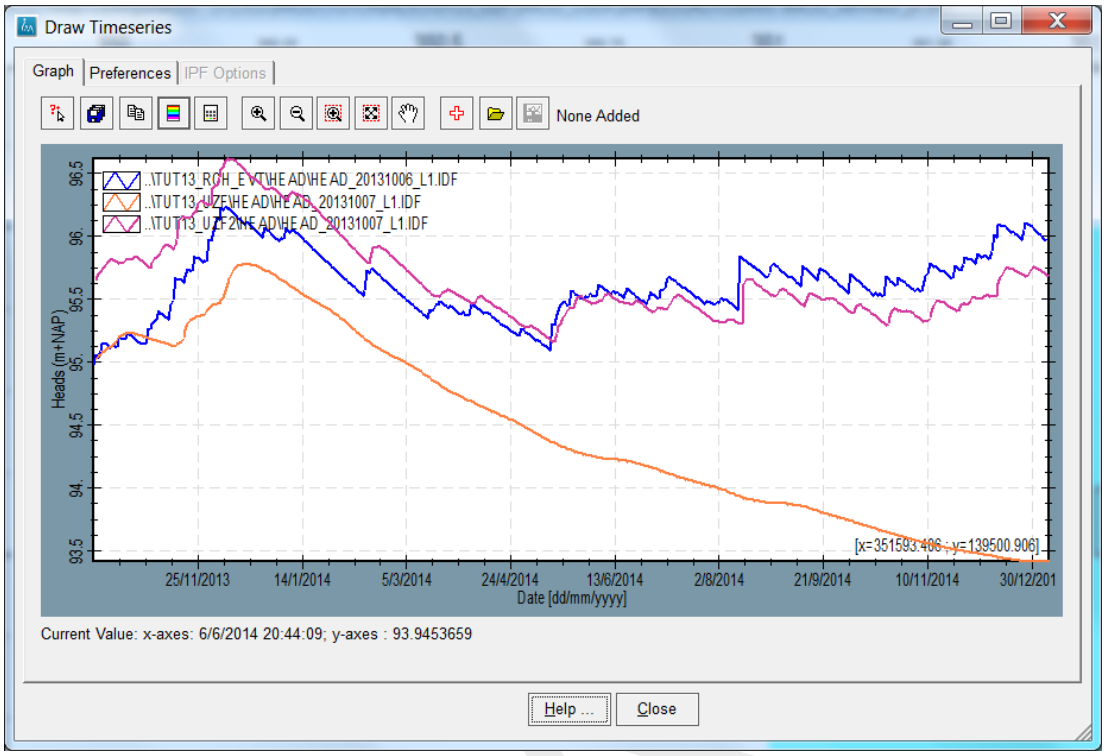


Figure 11.143: *Time Series of computed groundwater levels for the combination RCH-EVT and the two variants with the UZF package.*

With a Brooks-Corey (ϵ=2.0) the computed hydraulic head is significantly different and almost align (though more smooth) with the RCH-EVT combination. If you examine the water balance in the output files (*.LST-files), the following tables can be derived to clearly show the influence of the UZF package on the net recharge of the aquifer.

Table 11.9: *Summary of water balance for the different model configurations for the unsaturated zone (uz) and saturated zone (sz).*

Parameter	**RCH-EVT**	**UZF(ϵ=4)**	**UZF(ϵ=2)**
Precipitation$_{uz}$	-	2365470	2365470
Evaporation$_{uz}$	-	-2053611	-285329
Seepage$_{uz}$	-	-403764	-2311417
Net Storage$_{uz}$	-	-86867	201259
Total$_{uz}$	-	-5038	-954
Recharge$_{sz}$	2365470	403764	2311417
Evaporation$_{sz}$	2075648	-878899	-2107888
Net Storage$_{sz}$	290137	475197	203549
Total$_{sz}$	-315	62	-20

The UZF package is able to reduce the net recharge significantly, the Brooks-Corey Exponent (ϵ) is very sensitive in the amount of ground water recharge that seeps through the unsaturated zone and influences the behaviour of the groundwater level. Feel free to experiment more with the UZF parameters and observe how the influence the model outcomes. If you decrease the (SWC) Saturated Water Content of Unsat. Zone (IDF), the yielding hydraulic heads will be almost similar to the RCH-EVT combination. It is also interesting to start with a very wet subsoil (IWC) Initial Water Content (IDF), and see how the model responds on that.

Note: It is possible to examine per cell the moisture content in more detail, especially during the

model building phase this could be desirable. This functionality is not steered via iMOD, but can be easily added to the MF2005 file directly, please visit the website of the USGS for this adjustments.

Note: It is possible to examine per cell the moisture content in more detail, especially during the model building phase this could be desirable. This functionality is not steered via iMOD, but can be easily added to the MF2005 file directly, please visit the website of the USGS for this adjustments.

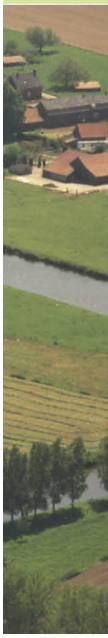
12 Theoretical background

In this chapter the iMOD-packages are described in detail. In the table below these iMOD-packages are listed by their acronyms including their MODFLOW-equivalent (if available). Also any differences between an iMOD-package and its MODFLOW-equivalent is described.

iMOD Key	Req.	Description	section nr	Equivalent MODFLOW package
CAP		Unsaturated zone package	section 12.1	n.a.
BND	x	Boundary conditions (compulsory)	section 12.2	BAS
SHD	x	Starting heads (compulsory)	section 12.3	BAS
KDW		Transmissivity	section 12.4	LPF/BCF
VCW		Vertical resistances	section 12.5	LPF/BCF
KHV		Horizontal permeabilities	section 12.6	LPF/BCF
KVA		Vertical anisotropy for aquifers	section 12.7	LPF/BCF
KVV		Vertical permeabilities	section 12.8	LPF/BCF
STO		Storage coefficients	section 12.9	LPF/BCF
SSC		Specific storage coefficients	section 12.10	LPF/BCF
TOP		Top of aquifers	section 12.11	DIS
BOT		Bottom of aquifers	section 12.12	DIS
PWT		Perched-water table package	section 12.13	n.a.
ANI		Horizontal anisotropy package	section 12.14	LPF
HFB		Horizontal flow barrier package	section 12.15	HFB
IBS		Interbed Storage package	section 12.16	IBS
SFT		Streamflow thickness package	section 12.17	n.a.
WEL		Well package	section 12.18	WEL
DRN		Drainage package	section 12.19	DRN
RIV		River package	section 12.20	RIV
EVT		Evapotranspiration package	section 12.21	EVT
GHB		General-head-boundary package	section 12.22	GHB
RCH		Recharge package	section 12.23	RCH
OLF		Overland flow package	section 12.24	n.a.
CHD		Constant-head package	section 12.25	CHD/BAS
FHB		Flow Head Boundary package	section 12.26	FHB
ISG		Segment package	section 12.27	RIV/DRN
SFR		Surface water Flow Routing package	section 12.28	SFR
LAK		Lake package	section 12.29	LAK
PST		Parameter estimation package	section 12.33	n.a.
MNW		Multi-Node Well package	section 12.30	MNW
UZF		Unsaturated Zone package	section 12.31	UZF
PKS		Parallel Krylov Solver	section 12.32	n.a.

12.1 CAP MetaSWAP Unsaturated zone module

The process of groundwater recharge and discharge through the unsaturated zone is simulated in iMODFLOW with the MetaSWAP concept (see Annex 1). MetaSWAP is developed by Alterra, Wageningen as part of the SIMGRO model code (Van Walsum, 2017b), (Van Walsum, 2017a). The SIMGRO framework is intended for regions with an undulating topography and unconsolidated sediments in the (shallow) subsoil. Both shallow and deep groundwater levels can be modelled by MetaSWAP. This model is based on a simplification of 'straight Richards', meaning that no special processes like hysteresis, preferential flow and bypass flow are modelled. Snow is not modelled, and neither the influence of frost on the soil water conductivity. A perched watertable can be present in the SVAT column model, but interflow is not modelled. There are plans for including the mentioned special processes in MetaSWAP Inundation water can be modelled as belonging to both groundwater and surface water at the same time. Processes that are typical for steep slopes are not included. The code contains several parameterized water management schemes, including irrigation and water level management.
The input data required for MetaSWAP are (Van Walsum *et al.*, 2016):



- ◇ **BND**: Boundary setting, used to specify active MetaSWAP elements
- ◇ **LGN**: Landuse code, should be referred to by the file luse_mswp.inp
- ◇ **RTZ**: Rootzone thickness in cm (min. value is 10 centimeter).
- ◇ **SFU**: Soil Fysical Unit should be referred to by fact_mswp.inp.
- ◇ **MET**: Meteo Station number, should be referred to by mete_mswp.inp.
- ◇ **SEV**: Surface Elevation (m+MSL).
- ◇ **ART**: Artificial Recharge Type, 0=no occurrence, ART>0 means present at current location whereby ART=1: from groundwater, ART=2: from surface water extraction
- ◇ **ARL**: Artificial Recharge Location, number of modellayer from which water is extracted.
- ◇ **ARC**: Artificial Recharge Capacity (mm/d).
- ◇ **WA**: Wetted Area (m^2) specifies the total area occupied by surface water elements. Value will be truncated by maximum cellsize.
- ◇ **UA**: Urban Area (m^2) specifies the total area occupied by urban area. Value will be truncated by maximum cellsize.
- ◇ **PD**: Ponding Depth (m)
- ◇ **PWT**: Depth of the perched water table level (m-SL)

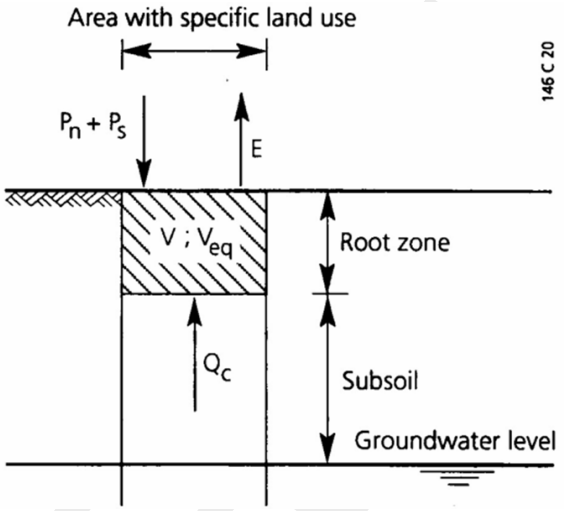


Figure 12.1: *Unsaturated zone with P_n = nett precipitation, P_s = irrigation, E = evapotranspiration, V = soil moisture, V_{eq} = soil moistureat equilibrium and Q_c = rising flux.*

12.2 BND Boundary conditions

The boundary conditions (-) consist of one IDF (or a constant value) for each modellayer specifying for each cell whether

- ◇ **Boundary value** < 0
  Those values denote areas that fixated head. The model will not change these values and they act as a fix boundary condition;
- ◇ **Boundary value** $= 0$
  Those values denote areas that are excluded for the simulation. No groundwater flow will go through those areas;
- ◇ **Boundary value** > 0
  Those values denote areas that take part of the simulation, groundwater flow goes through them and the head are computed. An important constraint to those locations is that need to be connected to at least a single fixed boundary condition, e.g. a boundary condition < 0 or one of the other packages that are head-dependent, such as the RIV, GHB, DRN package. The latter could be risky since that boundary condition might be removed whenever the head is below the drainage base.

The cell values correspond with the IBOUND values specified in the MODFLOW BAS package.

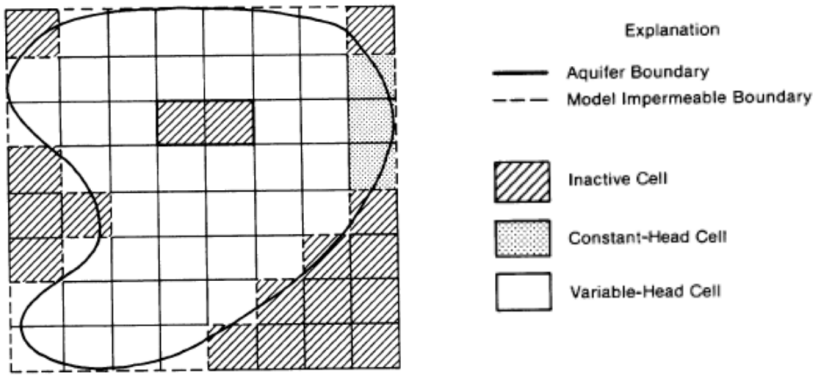


Figure 12.2: *Example of the boundary conditions for a single layer (source McDonald and Harbaugh, 1988)*

12.2.1 Scaling

For different grid resolutions the boundary is scaled internally via iMOD or iMODFLOW. For downscaling (e.g. from an input of 100 x 100 meter to a finer resolution of 10 x 10 meter) all finer grid cells obtain the identical value of the original cell. No interpolation of boundary conditions is performed. That means that the image looks identical to the original dataset but then at a finer resolution. For upscaling on the other hand, a different approach is used, which can be described best by the following rules that lists the steps to come from local values (the original values of the fine grid) to the global value (the upscaled courser grid):

1 If the local value is equal to its *NodataValue* it is set to zero, it becomes inactivated;
2 If the (corrected) local value is less than 0.0 (fix boundary condition) the global value becomes equal to the local value;
3 If the global value is equal to zero and the local value is greater than zero, the global value becomes equal to the local value.

To summarize the above, fix boundary conditions goes before variable locations, which go before inactive locations. As a consequence, an up-scaled model is always equal or larger in size than its finer representation, as well as the total size of fix boundary conditions. To compare an up-scaled model with its finer representation might yield differences in fluxes from boundary conditions as well as differences in RCH, EVT, UZF packages.

12.3 SHD Starting Heads

The starting head (L) consists of one IDF (or a constant value) for each modellayer specifying for each cell the initial head to start the model simulation. The starting heads correspond with the initial heads specified in the MODFLOW BAS package.

12.3.1 Scaling

For different grid resolutions the starting head is scaled internally via iMOD or iMODFLOW. For downscaling (e.g. from an input of 100 x 100 meter to a finer resolution of 10 x 10 meter) all finer grid cells obtain an interpolated value of the original cell. A four-point polynomial interpolation is used for this, which gives a smooth interpolation based upon the direct neighbouring grid cell centres that are not equal to the *NodataValue*. If none is found, the up-scaled value remains equal to the *NodataValue*. It might be advisable to store starting head conditions on a coarse scale, as they will be smoothed anyhow, prior to the simulation. For up-scaling, a simple approach is applied by computing the average value of all finer grid cells inside the coarse cell. Values that are equal to the *NodataValue* are excluded and if they all appear to be equal to the *NodataValue*, the up-scaled value becomes *NodataValue* as well. Bear in mind that iMOD does not take into account whether grid cells are partially overlapping cells. It simply uses grid cells that are inside a coarse grid cell based upon their grid centres.

12.4 KDW Transmissivity

The transmissivity (L^2/T) of each modellayer is defined by one IDF (or a constant value). Alternatively the transmissivity of a modellayer may be defined by the product of the horizontal permeability defined in the KHV package and the layer thickness derived from the TOP and BOT package (see Figure 12.3). The KDW transmissivity corresponds with the TRAN variable specified in the MODFLOW BCF package.

12.5 VCW Vertical resistances

The vertical resistance (T) of each modellayer is defined by one IDF (or a constant value). Alternatively the vertical resistance of a model layer may be defined indirectly by the layer thicknesses derived from the TOP and BOT package, the vertical permeability defined in the KVV package, the horizontal permeability defined in the KHV package and the vertical anisotropy defined in the KVA package (see Figure 12.3). The VCW vertical resistance corresponds with the reciprocal of the VCONT variable specified in the MODFLOW BCF package, so $VCW = \frac{1}{VCONT}$.

12.6 KHV Horizontal permeabilities

The horizontal permeability (L/T) of each modellayer is defined by one IDF (or a constant value). The horizontal permeability is used in combination with the layer thickness to calculate the transmissivity of a modellayer (see Figure 12.3). The KHV horizontal permeability corresponds with the HY variable specified in the MODFLOW BCF package and the HK variable specified in the MODFLOW LPF package.

12.7 KVA Vertical anisotropy for aquifers

The vertical anisotropy (-) of each modellayer is defined by one IDF (or a constant value). The vertical anisotropy is multiplied with the horizontal permeability to calculate the vertical permeability in the permeable part of a modellayer (see Figure 12.3). The KVA vertical anisotropy corresponds with the VKA variable specified in the MODFLOW LPF package.

12.8 KVV Vertical permeabilities

The vertical permeability (L/T) of the resistance layer between two modellayers is defined by one IDF (or a constant value). The vertical permeability is used in combination with the thickness of the resistance layer to calculate the vertical resistance between two modellayers (see Figure 12.3). The KVV vertical permeability corresponds with the HY variable specified in the MODFLOW BCF package and the VKCB variable specified in the MODFLOW LPF package.

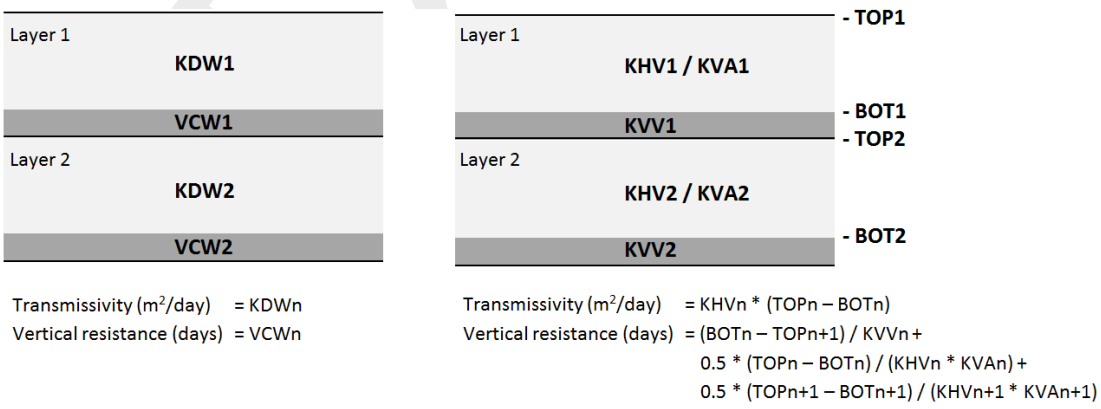


Figure 12.3: *Hydraulic layer parameters used in iMODFLOW*

12.9 STO Storage coefficients

The storage coefficient (for confined conditions) or specific yield (for unconfined conditions) of each modellayer is defined by one IDF (or a constant value). The value depends on the lithology of the modellayer. The storage coefficient in confined aquifers varies between 1x10^{-5} to 1x10^{-3}. The specific yield ranges between 0.02 for clay to 0.25 for gravel. The STO storage coefficient corresponds with the SF1 variable specified in the MODFLOW BCF package and the SS and SY variable specified in the MODFLOW LPF package.

12.10 SSC Specific storage coefficients

The storage coefficient for modellayers which can change from confined to unconfined conditions is defined by two IDFs (or constant values) for each modellayer. The SSC storage coefficient corresponds with the SF2 variable specified in the MODFLOW BCF package and the SS and SY variable specified in the MODFLOW LPF package.

porosity is sum of void spaces as specific yield is drainable volume (always les then porosity as some pores are undrainable due to capillary suction forces)

12.11 TOP Top of aquifers

The top level of the permeable part of each modellayer (see Figure 12.3) is defined by one IDF (or a constant value). The TOP level corresponds with the TOP variable specified in the MODFLOW DIS package.

12.12 BOT Bottom of aquifers

The bottom level of the permeable part of each modellayer (see Figure 12.3) is defined by one IDF (or a constant value). The BOT level corresponds with the BOTM variable specified in the MODFLOW DIS package.

12.13 PWT Perched water table package

A perched water table (or perched aquifer) is a (temporary) water table that occurs above the regional groundwater table in the unsaturated zone. This occurs when there is a (relatively) impermeable layer above the regional groundwater table in the unsaturated zone. With the PWT-package a perched water table can be schematized in iMOD, the perched water table concept is given in Figure 12.4.

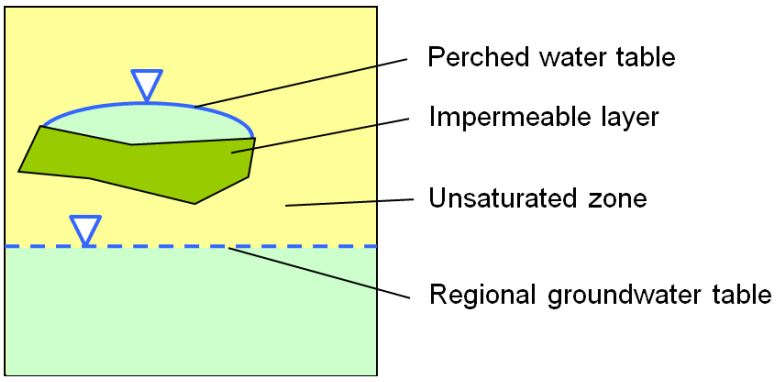


Figure 12.4: *Conceptual schematization of a perched water table.*

In the following pages the concept of the perched water table package is described and illustrated by several hydrologic situations. Hereby, the following figure (Figure 12.5) is used which that represents the perched water table in terms of model parameters. Important to understand is that there can be only a single perched water table in each vertical column. Once a perched water table exists, both the

horizontal and vertical flow component will be reduced up to zero when the pressure head above the perched water table drops below the top of the aquitard that creates the perched water table.

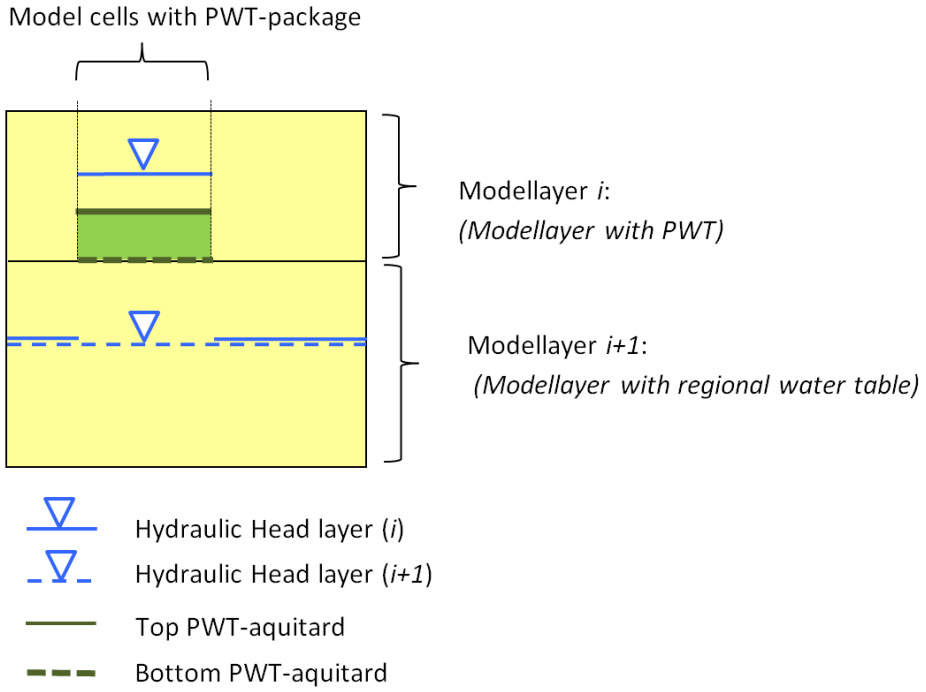


Figure 12.5: *Conceptual schematization of a perched water table in a groundwater model.*

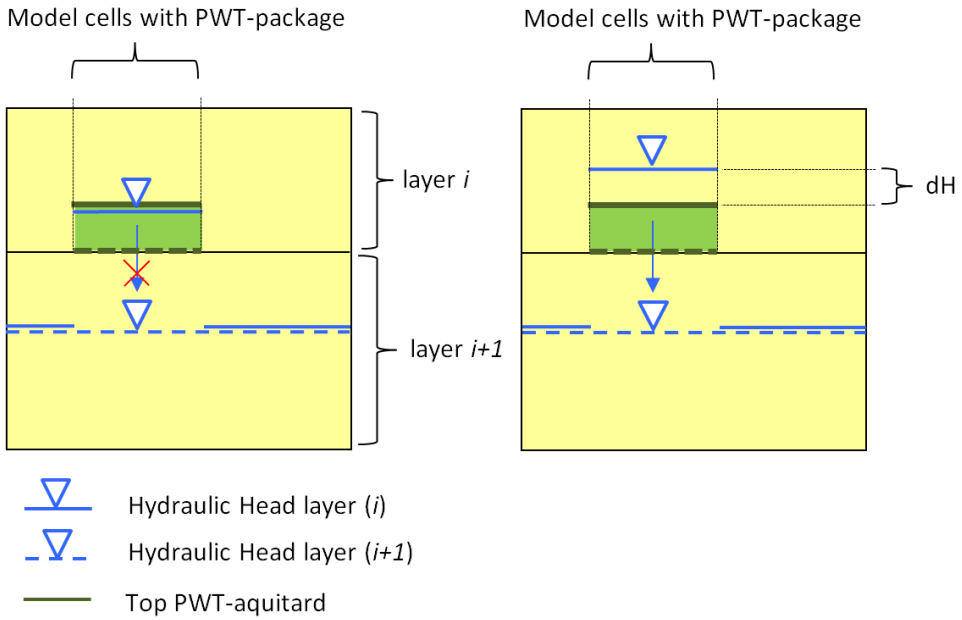
The PWT package is applied using the following assumptions, these are described in the following table in more detail.

I. There is no storage in the PWT aquitard and the driving force for vertical flow equals the pressure head of layer x minus the top of the PWT-aquitard.

There are two situations to distinguish

- **No Perched Water table**
  This situation is depicted on the left figure, the perched water table is below the top of the aquitard yielding a zero flux through the aquitard
- **Thickness of a perched water table**
  This situation is show on the right figure, in this particular case the vertical flux through the aquitard is computed as:

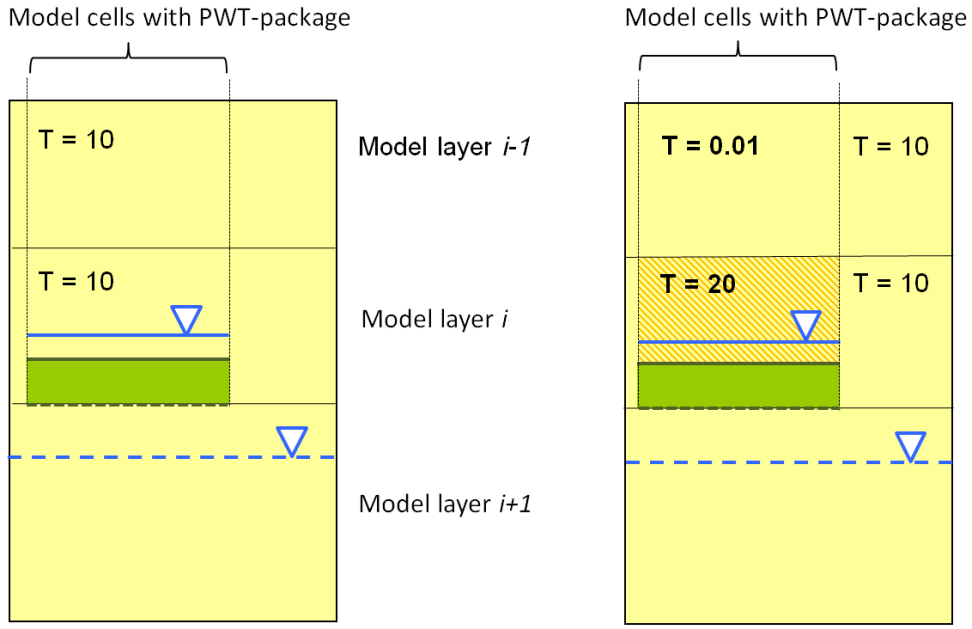
$$\mathrm{d}H/(H_i - H_{i+1}) \text{ whereby}$$
$$\mathrm{d}H = H_i - T \text{ : thickness of the perched water table}$$
$$T \text{ : top of the aquitard} \qquad (12.1)$$
$$H_i \text{ : pressure head of modellayer } i$$



Schematization of vertical fluxes using the PWT-Package

II. The model cells with the PWT-package are considered to be the top most layer with saturated groundwater.

If the PWT cells are not within the first model layer the transmissivity above the PWT cells is recalculated. On the figure left, the transmissivities are 10 and 10 for the first two modellayer above the PWT. Since the PWT package will compute the transmissivity only for the first modellayer above the PWT layer, iMODFLOW will redistribute the transmissivities such that they all are lumped in the first modellayer above the PWT layer. This is shown in the figure right. Now, the transmissivitiy of the first modellayer is 0.01 (actually this is equal to the parameter MINKD in the runfile) and the first modellayer above the PWT layer has 20.



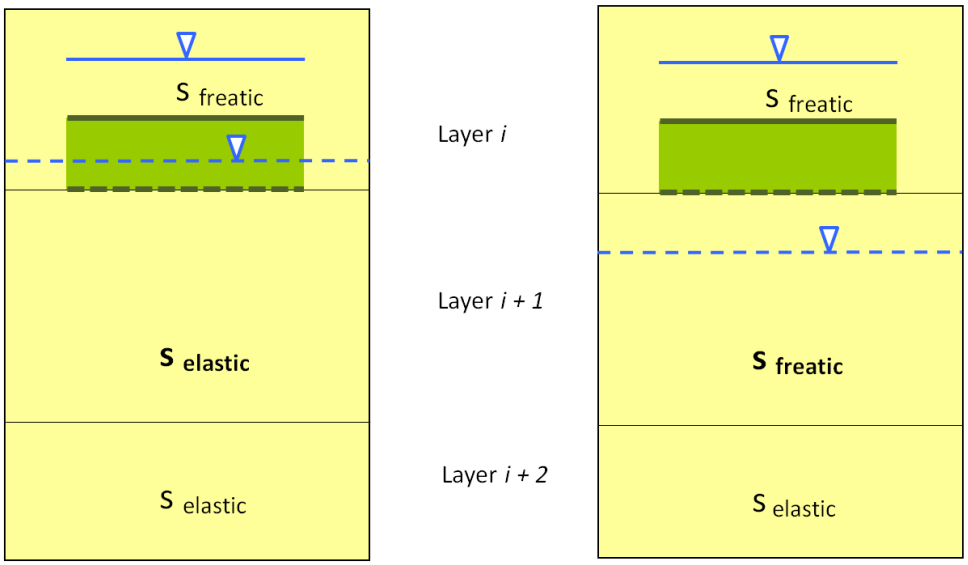
Schematization of transmissivity when the PWT-Package is used in second model layer

III. The model cells with the PWT-package are considered to be unconfined and thus also have a phreatic storage coefficient.

In order to compute the effective transmissivity T_{e}, the permeability is computed initially by $k = T/(TOP_{\text{aquifer}} - TOP_{\text{aquitard}})$. This permeability is used to compute the transmissivity T_{e} as function of the pressure head as $T_{\text{e}}(h) = k(h - TOP_{\text{aquitard}})$.

IV. The model cells in layer $i+1$ have a phreatic storage coefficient, unless the pressure Head of layer $i+1$ is greater than the bottom of the PWT aquitard. In this case an elastic storage coefficient is used.

In case the underlying aquifer becomes unconfined, and therefore the elastic storage coefficient is used, this is illustrated in the figures below.



Schematization of transmissivity when the PWT-Package is used in second model layer

The numerical implementation is such that the horizontal conductances and the vertical resistances are calculated on the heads of timestep $t-1$. This is in order to avoid numeric instability.

12.14 ANI Horizontal anisotropy module

12.14.1 Introduction

Anisotropy is a phenomenon for which the permeability *k* is not equal along the x- and y Cartesian axis, k_{xx} and k_{yy}, respectively. It can be notated that for isotropic conditions $k_{xx} = k_{yy}$ (see Figure 12.6a), and for anisotropic conditions $k_{xx} \neq k_{yy}$ (see Figure 12.6b).

(a) *Isotropic conditions, flow [q] perpendicular to piezometric head [h]*

(b) *Anisotropic conditions, flow [q] non perpendicular to piezometric head [h]*

Figure 12.6: *Example of groundwater flow [q] for (a) isotropic and (b) anisotropic flow conditions.*

To express the amount of flow along the x- and y-axes of a Cartesian coordinate system, the following equations are valid to compute the flow along these direction; q_x and q_y, respectively (Strack ODL

(1989), *Groundwater Mechanics*, Princeton Hall, Inc., Englewood Cliffs, New-Jersey):

$$\begin{bmatrix} q_x \\ q_y \end{bmatrix} = \begin{bmatrix} -k_{xx} & -k_{xy} \\ -k_{yx} & -k_{yy} \end{bmatrix} \begin{bmatrix} \frac{\partial h_x}{\partial x} \\ \frac{\partial h_y}{\partial y} \end{bmatrix} \tag{12.2}$$

From equation (12.2), it can be seen that in anisotropic conditions ($k_{xx} \neq k_{yy}$), flow along the x-direction is not influenced solely by the hydraulic gradient along this x-axis, but also by a hydraulic gradient along the y-axis. The permeability's k_{xy} and k_{xy} are equal to each other and depend on the angle φ of the principal axis to the x-axis:

$$\begin{aligned} k_{xx} &= f \times T \times \cos(\varphi)^2 + T \times \sin(\varphi)^2 \\ k_{xy} &= k_{yx} = ((f \times T) - T) \times \cos(\varphi) \times \sin(\varphi) \\ k_{yy} &= f \times T \times \sin(\varphi)^2 + T \times \cos(\varphi)^2 \end{aligned} \tag{12.3}$$

For values φ=0.0; φ=90.0; φ=180.0; φ=270.0, k_{xy} and k_{xy}become 0.0.

12.14.2 Parameterisation

Anisotropy is expressed by an angle φ and anisotropic factor *f*. The angle φ denotes the angle along the main principal axis (highest permeability *k*) measured in degrees from north (0°), east (90°), south (180°) and west (270°). The anisotropic factor *f* is **perpendicular** to the main principal axis. The factor is between 0.0 (full anisotropic) and 1.0 (full isotropic), see Figure 12.7.

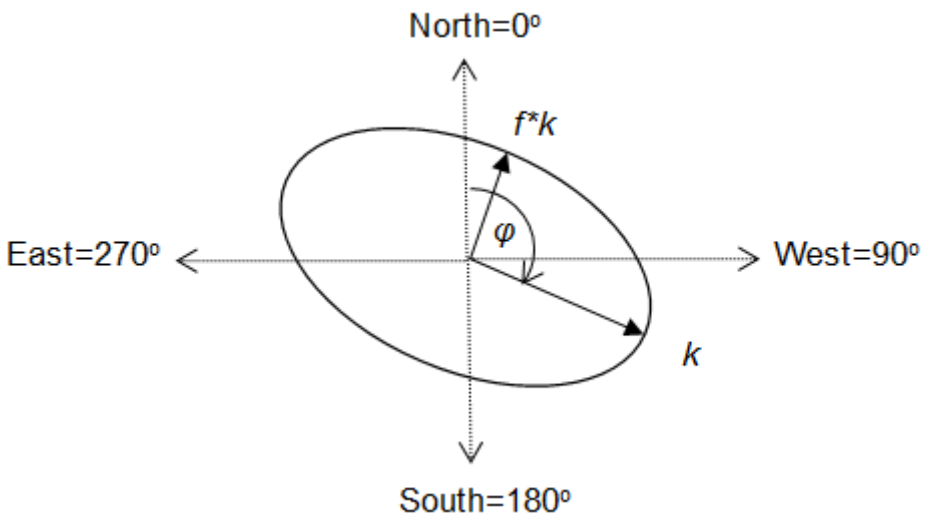
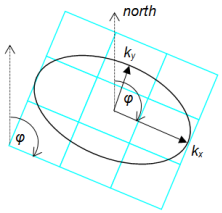
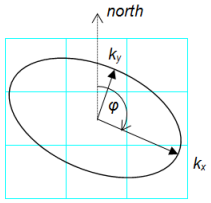


Figure 12.7: *Anisotropy expressed by angle φ and anisotropic factor f*

Most optimally, the model discretisation should follow the configuration of the anisotropy, see Figure 12.8a. However, anisotropy could be folded in many different directions (principal directions), which probably yield for anisotropy in many angles throughout the modeling domain. With the chosen mathematical method (finite-differences) in iMODFLOW, it is impossible to fold the model network according to the anisotropy, see Figure 12.8b.



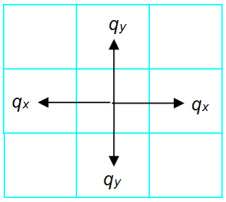
(a) $K_x < K_y; \varphi = 120.0°$



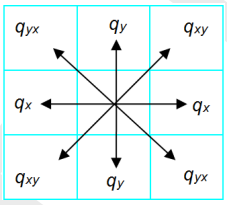
(b) $K_x < K_y; \varphi = 120.0°$

Figure 12.8: *Example of (a) anisotropy aligned to the model network and (b) anisotropy non-aligned to the model network.*

Since the principal direction of the permeability is not aligned to the axes of the modeling network, it is necessary to add extra flow terms to the finite difference equation to take into account the diagonal flow, caused by the anisotropy, see Figure 12.9.



(a) *Isotropic conditions*



(b) *Anisotropic conditions*

Figure 12.9: *Example of (a) flow terms in isotropic flow conditions and (b) flow terms in anisotropic flow conditions.*

For more detailed explanation on the computation of these extra flow terms, see Vermeulen PTM (2006) *et al. Limitation to Upscaling of Groundwater Flow Models dominated by Surface Water Interaction,* Water Resources Research 42, W10406, doi:10.1029/2005WR004620.

For each cell in the model network, anisotropic angles φ and factors *f* can be specified. For those situations where a single model cell contains more than one of these anisotropic parameters, they will be up-scaled to the model cell. For the anisotropic angle, the most frequent occurrence will be used, as for the anisotropic factor, a mean value will be computed. This seems to be the most robust and fair trade-off between a coarsened model network and loss in detail.

The ANI horizontal anisotropy corresponds with the TRPY variable specified in the MODFLOW BCF package and the HANI variable specified in the MODFLOW LPF package.

12.15 HFB Horizontal flow barrier module

Horizontal barriers obstructing flow such as semi- or impermeable fault zone or a sheet pile wall are defined for each model layer by a *.GEN line file. The behaviour of this is twofold:

◇ **Factor** f
This is used automatically whenever the packages TOP and BOT are omitted in the runfile. By lines that obstruct groundwater with a particular reduction factor f for the hydraulic conductance or permeability, see Figure 12.9a, resulting in variable resistances along the line. The factor f is applied to the computed harmonic conductances in between cells i (icol index) and j (irow index).

$$\begin{aligned} CR_{i,j} &= f\,\frac{2T_2T_1DY_j}{(T_1DX_i+T_2DX_{i-1})} \\ CC_{i,j} &= f\,\frac{2T_2T_1DX_i}{(T_1DY_j+T_2DX_{j-1})} \end{aligned} \tag{12.4}$$

◇ **Resistance** r This is used automatically whenever the packages TOP and BOT are included in the runfile. By lines that obstruct groundwater flow with a variable resistance r in days for that line, see Figure 12.9b, resulting in variable resistances $C^{'}$ along that line. This combined resistance $C^{'}$ (as a sum of the original resistance and the additional fault resistance) is computed internally as:

$$\begin{aligned} CR_{i,j} &= \frac{2T_2T_1DY_j}{(T_1DX_i+T_2DX_{i-1})} \\ DZ_{i,j} &= \tfrac{1}{2}(TOP_{i,j}-BOT_{i,j})+\tfrac{1}{2}(TOP_{i+1,j}-BOT_{i+1,j}) \\ C_{i,j} &= \tfrac{1}{2}DX_iDX_{i+1}\frac{CR_{i,j}}{DZ_{i,j}} \\ C^{'}_{i,j} &= r+C_{i,j} \\ CR_{i,j} &= C^{'}_{i,j}\frac{2T_2T_1DY_j}{(T_1DX_i+T_2DX_{i-1})} \end{aligned} \tag{12.5}$$

Whenever r is negative, the resulting resistance $C^{'}$ is equal to abs(r). In that way the resistance between cells can become less than the resistance that is based on the permeability of the geological material.

Note: A **Factor** f or **Resistance** r of 0.0, means that the fault is completely impermeable.

In iMOD faults can be simulated by entering GEN files in the runfile directly. iMOD will define automatically at which cell faces the permeabilities need to be adjusted based on the specifications of the fault.

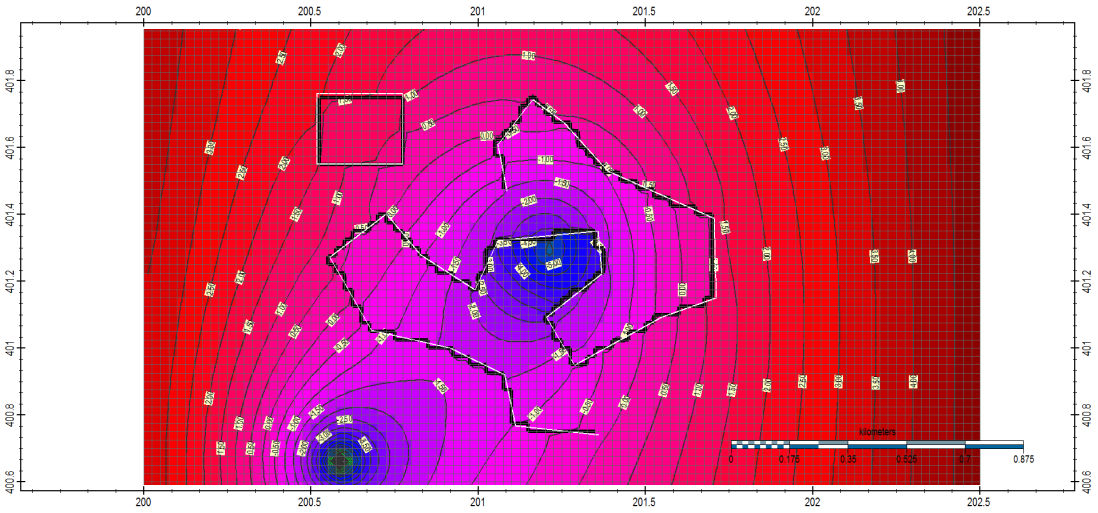


Figure 12.10: *Example of a horizontal flow barrier parameterization in case of a uniform model network consisting of model cells of 25 x 25 m. Based on the location of an irregular shaped fault line (white line) the cell faces (thick black lines) are identified where the conductance between the cells is adjusted using the parameter values of the fault line. The computed hydraulic heads (thin black contour lines) illustrate the local effects of the barriers on groundwater flow.*

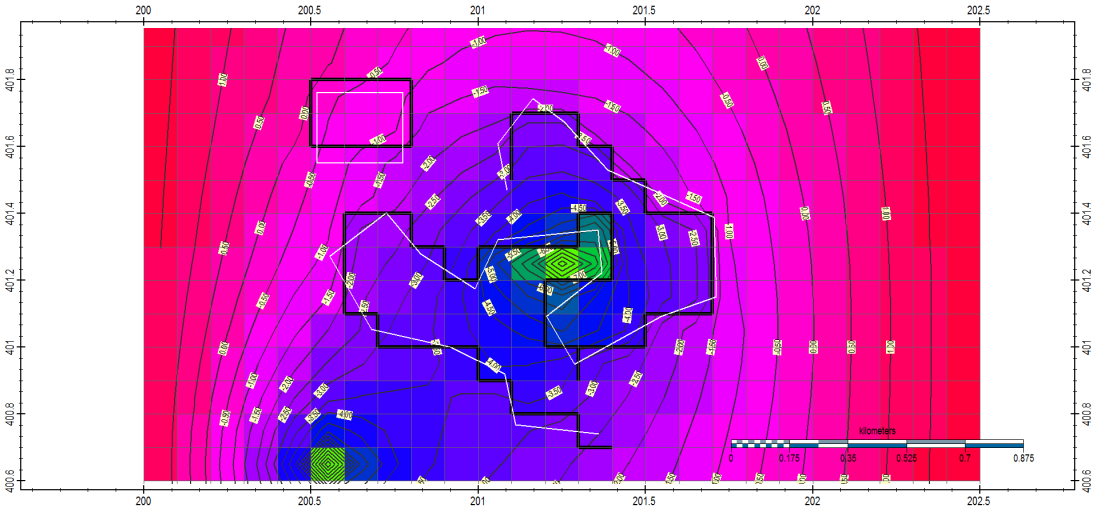


Figure 12.11: *The same example as above, but now for a uniform model network consisting of model cells of 100 x 100 m.*

The line *.GEN file defines the location of the barrier. The multiplication factor is used to create the obstruction by reducing the conductance between model cells. The HFB module corresponds with the MODFLOW HFB package.

Whenever GEN files are assigned to layer number 0, iMOD will assign the fault to the appropriate model layers automatically. In that case the GEN file needs to be a 3D GEN (see section 9.10). Based on the elevation in the 3D GEN file and the TOP and BOT elevations of each model layer, the fault and the nett resistance will be assigned according the following method:

- ◇ Whenever the thickness of the model layer is < 0.5 meter:
  - ▫ Compute the nett resistance r or factor f of the fault as a weighted arithmetic mean for the thickness of each individual fault along that particular cell.
- ◇ Whenever the thickness of the model layer is > 0.5 meter:
  - ▫ Each grid cell interface will be filled in until no space is available that can be occupied by a fault line;
  - ▫ Each contribution of an individual fault is harmonic (c^{-1});
  - ▫ The system number as used in the model, is equal to the fault that contributed mostly to the nett resistance;
- ◇ The final nett resistance is the harmonic mean between:
  1. the summed resistance weighted to the level of occupation index ($i = 0.0 - 1.0$), and
  2. a resistance of 1 day for the remaining part of the model layer ($1.0 - i$).

  Be aware that whenever the occupation index is 90 %, the resistance will be already significantly less. An example is demonstrating what the nett resistance are for different settings.

Total thickness m	Thickness m	Resistance d	Weighted harmonic md^{-1}	Nett Resistance c	Confined Resistance c
	2.5	100	0.025		
	2.5	1,000	0.0025		
5.0	5.0		0.0275	181.82	181.82
	2.0	100	0.02		
	2.0	1,000	0.002		
5.0	4.0		0.022	227.27	4.89

(12.6)

12.16 IBS Interbed Storage package

The compaction of modellayers by a reduction in water pressure is calculated using four IDFs (or constant values): preconsolidation head or preconsolidation stress in terms of head in the aquifer (L); dimensionless elastic storage factor for interbeds present in the modellayer; dimensionless inelastic storage factor for interbeds present in the modellayer; starting compaction in each layer with interbed storage (m). The IBS package is comparable to the IBS package of MODFLOW.

12.17 SFT Streamflow thickness package

The streamflow thickness is defined by two IDFs (or constant values): the streamflow thickness (L) and the permeability (L/T).

12.18 WEL Well package

The well package defines the groundwater abstractions for each modellayer from wells by IPF-files. The IPF-files contain the coordinates of the well locations and may contain an average abstraction rate (L^3/T) or a link to a text-file with abstraction time series (L^3/T). The screen depth may be added to assign automatically the modellayer from which the abstraction takes place. The WEL package is comparable to the WEL package of MODFLOW.

12.19 DRN Drainage package

The drainage package defines the location, the elevation (L) and the conductance (L^2/T) of the drainage system by two IDFs. The drainage system represents drainage pipes and drainage ditches by which water is removed from the model when the calculated head in a modellayer exceeds the elevation of the drainage system. The drainage package is usually connected to the first modellayer only. The DRN package is comparable to the DRN package of MODFLOW.

Drainage simulated in cells with surface water becomes inactive when the drainage elevation is below the water level in the same cell as defined in the RIV package. Herefor it is necessary to set the parameter ICONCHK=1 in the runfile.

12.20 RIV River package

The river package defines the location, the water level (L), the bottom level (L), the conductance (L^2/T) and the infiltration factor (-) by four IDFs. The river package represents the presence of permanent water from which water may infiltrate or to which water may discharge. The source of water in the river package is unlimited which means that rivers never dry out. The RIV package is comparable to the RIV package of MODFLOW, except for the infiltration factor which is added in iMODFLOW.

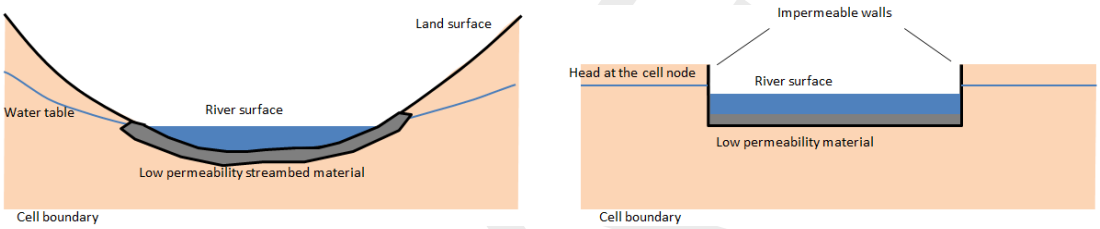


Figure 12.12: *Principle of the RIV package (adapted from Harbaugh, 2005)*

The RIV package may be replaced by the ISG package which defines the surface water in segments.

12.21 EVT Evapotranspiration package

The evapotranspiration package defines the evapotranspiration by plant transpiration or directly from the saturated groundwater by three IDFs: evapotranspiration rate (0.001L/T), top elevation (L) for maximal evapotranspiration rate and thickness (L) over which the evapotranspiration rate is reduced to zero. The EVT package is comparable to the EVT package of MODFLOW. Remember that the units for the evapotranspiration rate are mm/d for iMODFLOW.

The EVT package may be replaced by the CAP module which makes a more sophisticated simulation of the processes in the unsaturated zone.

12.22 GHB General-head-boundary package

The general head boundary package simulates flow to or from a model cell from an external source by two IDFs: the elevation (L) and the conductance (L^2/T) of the general head boundary. The GHB package assumes an unlimited source of water and is often used to model large water bodies which border the area of interest.

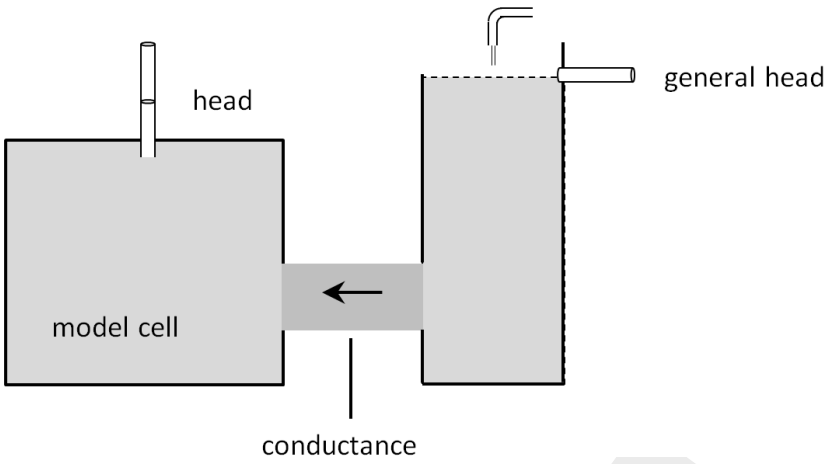


Figure 12.13: *Principle of the General Head Boundary package (Harbaugh, 2005)*

12.23 RCH Recharge package

The recharge package defines the quantity of water (0.001L/T) from precipitation that percolates to the groundwater by one IDF (or a constant value). The RCH package is comparable to the RCH package of MODFLOW. Remember that the units are mm/d for iMODFLOW.

The RCH package may be replaced by the CAP module which makes a more sophisticated simulation of the recharge process through the unsaturated zone.

12.24 OLF Overland flow package

The overland flow package defines the elevation (L) above which outflow of groundwater will occur when exceeded by the groundwater head. The package simulates the effect of outflow of water across the land surface. The water is discharged out of the model and does not return to the groundwater. The OLF elevation may be determined at a few centimetres above ground elevation to represent shallow ponding caused by small obstructions against outflow. The flow rate of the OLF package is calculated assuming a fixed resistance against outflow of 1 day. The OLF package is not available in MODFLOW, however, the OLF-concept can also be applied by using the conventional DRN package, see section 12.19. The modeller might prefer to use the OLF package because 1) the assigned resistance value is cell size independent and 2) in the model output the resulting volumetric budget are kept seperate from e.g. the DRN volumetric budgets.

12.25 CHD Constant-head package

The constant head package defines the elevation (L) of groundwater heads at cells where the BND value < 0 by one IDF (or a constant value). The CHD package is comparable to the definition of the CHD in the BAS package of MODFLOW.

12.26 FHB Flow and Head Boundary package

The existing Flow and Head Boundary package of MODFLOW2005 (Leake (1997)) was implemented in iMOD. With this package it is possible to combine constant head- and constant flux boundaries for all model layers independently. For each model cell it is possible to define whether it becomes a constant head or constant flux boundary condition.

12.27 ISG iMOD Segment package

The iMOD Segment package defines the surface water system with an ISG-file which contains all relevant information used by surface water elements which are in direct relation with groundwater. The ISG-file stores:

- ◇ the actual outline of the surface water element;
- ◇ (time-dependent) stages, bottom heights, infiltration factors and the resistance of the riverbed;
- ◇ the cross-sections 2D and/or 3D;
- ◇ (time-dependent) up- and downstream stages at weirs;
- ◇ discharge/width/depth relation ships.

To store all these different types of information the ISG-file format consists of associated files that are connected by the ISG-file. The ISG package as is, is not available in MODFLOW, but it generates the input for the conventional RIV package. For the SFR package (see section section 12.28), this type of an ISG file is expanded with more data, see section section 9.9 for a detailed description of both types of ISG files.

The ISG package file format is based on vectors and time series and therefore has a much more efficient disk use than the RIV package. iMOD and iMODFLOW both, can handle those ISG files to generate model input. Within ISG Edit (see section section 6.10.3) it is possible to compute IDF files from the ISG file for the different model parameters such as conductance, stage, bottom heights and infiltration resistances. Or, more efficient, it is possible to use the ISG directly in the RUN- and/or PRJ file and let iMOD/iMODFLOW grid the ISG file internally to a conventional/modified RIV-file, see section section 10.11.

The way iMOD/iMODFLOW grids the vector based ISG file onto the simulation raster is as follows.

- ◇ Each segment, containing at least two nodes, is treated separately from the other segments and intersected with the model network;
- ◇ All intersected model cells are given the linear interpolated values for stage s, bottom heights b, infiltration factor f and resistances c in between all existing calculation nodes along the segment;
- ◇ For each location along the stream the appropriate cross-section is assigned. If a single cross-section is specified, that cross-section is valid for the entire stream. If more cross-sections are specified, the application of each cross-section is the stretch along the stream between the location of that cross-section up to the next specified cross-section - along the direction of the FROM- and TO-node. As a exception, the first specified cross-section is applied as well for the stretch between the FROM-node up to the first specified cross-section;
- ◇ The conductance is a function of the interpolated stage s and bottom height b. The water depth d (difference between the stage and bottom height; $d = s - b$) is used to compute the wetted perimeter wp at each location along the segment with the appropriate cross-section. The conductance (m^2/d) is the product of the wetted perimeter wp times the length of the intersection in the particular model cell l, divided by the resistance, so $cond = \frac{wp \times l}{c}$. The infiltration factor f is used as a package entry and corrects the conductance iteratively whenever the stage is higher than the computed groundwater level h, in that case the conductance becomes $cond = \frac{wp \times l}{f \times c}$. The conductance has a lower values for short intersections than for longer ones, this is clearly seen in the following figure.

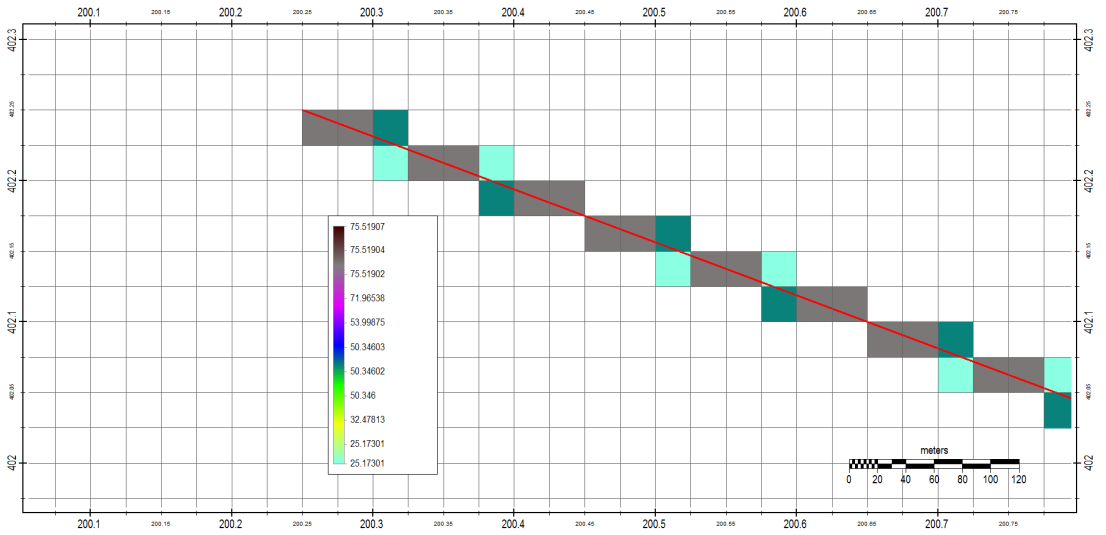


Figure 12.14: *Example of the conductance (m^2/d) of a segment (red line) in an ISG file gridded on a model network.*

The cross-sections in a ISG can yield different appearances of the conductance and therefore the outline of the segment in the model network. The following configurations might occur:

- ◇ The cross-section is a 2D cross-section perpendicular to the stream and describes the bathymetry as a function of distance x and height z. The distance x is at all times less than the width of current location in the model network;

- ◇ The cross-section is a 2D cross-section perpendicular to the stream and describes the bathymetry as a function of distance x and height z. The distance x can be more than 1.5 times the width of current location in the model network. In this particular case, the influence of the segment will be distributed over more than the single intersected model cell. A ”brush“ (circle with a radius equal to the half the width at the current location along the segment) is used to compute the fraction in which each model cell is influenced by the segment. The fraction is computed as the number of points in a grid cell that occupied by a circle, a grid cell which is completely covered by a circle is given a fraction of 1.0, other location with less covering receive a fraction < 1.0, see the following figure.

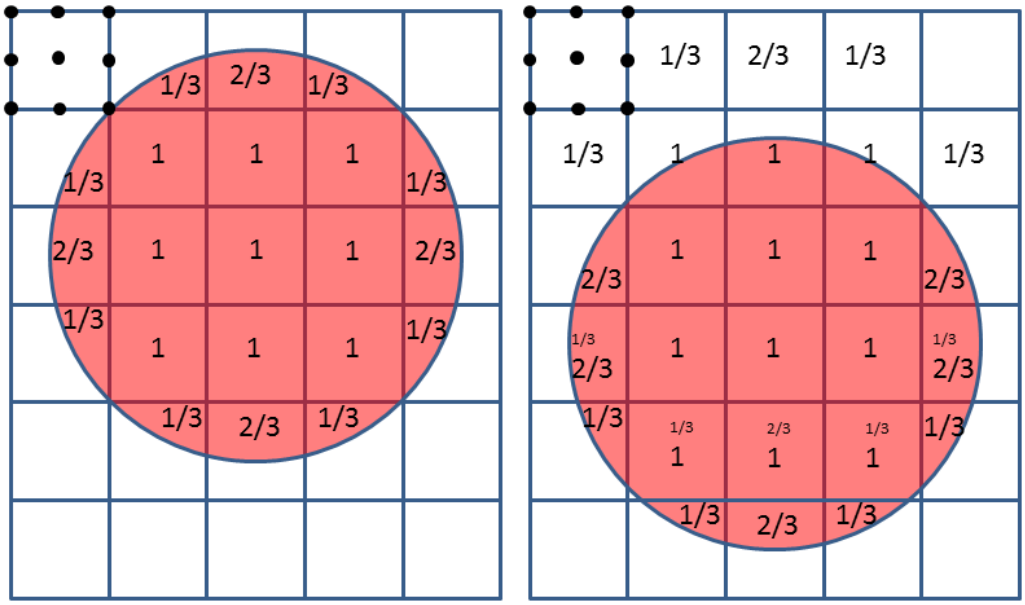


Figure 12.15: *Example of the brush method; (left) showing the fractions for the first location of the brush; (right) showing the updated and new fractions when the brush is moved one row down.*

◇ The cross-section is a 3D cross-section which describes the bathymetry on a local and regular x, y, z raster. The local regular raster is projected in the (ir)regular model network. The conductance is the sum of the overlying areas of the local regular bathymetry raster divided by the entered resistance. Whenever an indicator i is present in the ISG file - to denote inundation which depend on a reference height h_{ref} - the final resistance is multiplied with the value of the indicator i. This is only done whenever the stage s is higher than the river bathymetry b at that particular location.

Each of the above mentioned configuration yield a different assignment of conductances to the underlying model network, as shown in the following figure.

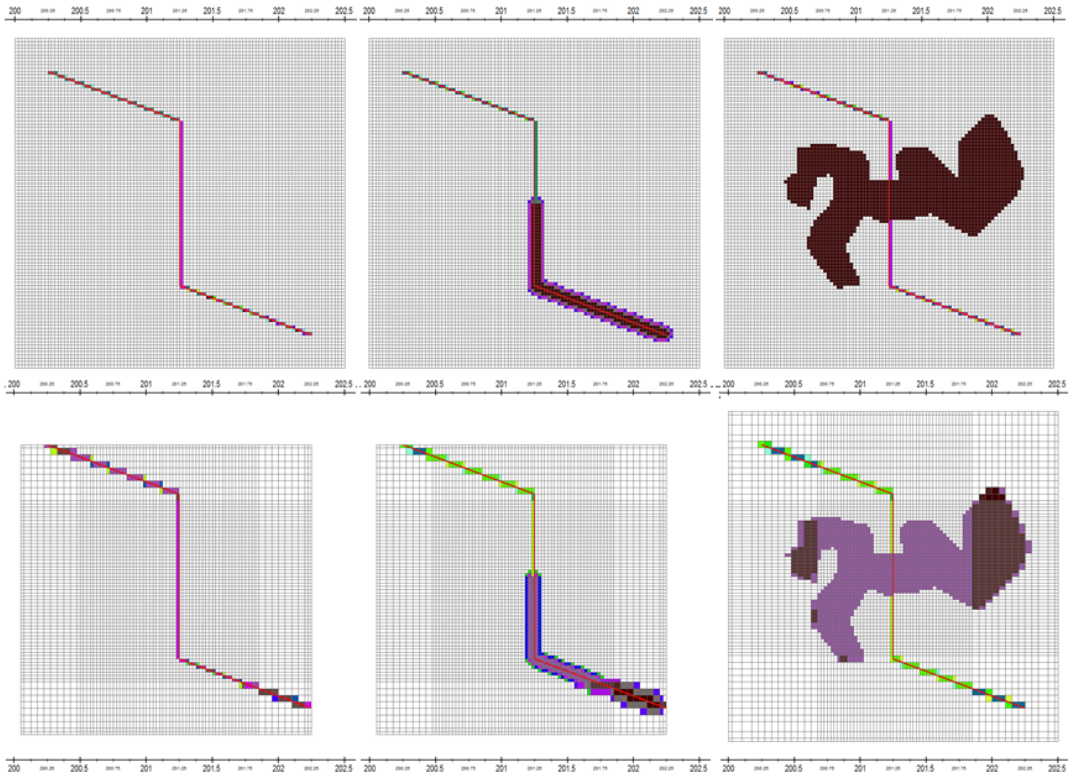


Figure 12.16: *Example of different conductances for a segment in an ISG file gridded on different model network with and without local sub grid refinements and for different type of cross-sections.*

The ISG file format also makes it more easy to convert a surface water model data from SOBEK into iMOD using the SOBEK import tool.

12.28 SFR Surface water Flow Routing Package

The existing Surface Flow Routing package of MODFLOW2005 (Prudic *et al.* (2004)) has been implemented in iMOD. With this package it is possible to route surface water through a network of segments. These segments are stored in an additional type of ISG file that contains more records than the conventional ISG. These additional records are specific for the SFR package and describe the connection and diversion between individual segments and how the water depth in the segments needs to be computed. The water depth is a function of precipitation, evaporation, external discharges and the computed hydraulic head. The cross-section of each segment can be specified by a rectangle, an eight-point cross-section and/or a relationship between discharge, width and depth. The SFR can discharge surface water in and from a lake as described by the LAK Package. The SFR is therefore a significant improvement in the interaction between surface- and groundwater.

There are a few limitations to the SFR package due to the iMOD implementation:

- ◇ the input units for discharge are m^3/s;
- ◇ it is not allowed to define the water depth by an empirical equation;
- ◇ a connection to a lake (up- and/or downstream) needs to be specified by the negative lake identification, it is not necessary that the stream segment truly enters the lake, in fact any connection can be made to any lake, even if the lake isn't in the neighbourhood of the segment;
- ◇ by default the output of each segment is written in an output file that is converted to an ISG file after the simulation is finished. For large networks, this file may become very large. The interaction between the SFR and the groundwater is saved in the BDGSFR file and is a per cell lumped quantity;
- ◇ inspect the *.LST-file for more detailed information on total water balances per lake and in- and/or

outflow to streams that are connected;
- ◇ the SFR may be subject to convergence problems; an increase of the riverbed resistance may avoid these problems.
- ◇ the value used to set the tolerance level of stream depth used in computing leakage between each stream reach and active model cell is set to 0.0001;
- ◇ no solute transport is supported;
- ◇ a unsaturated zone underneath the segments is not supported.

12.29 LAK Lake Package

The existing Lake package of MODFLOW2005 (Merritt and Konikow (2000)) has been implemented in iMOD. With this package it is possible to compute lake stages in relation to the groundwater head. Given a bathymetry of the lake, iMOD will assign the lake to the appropriate model layers and assign the correct conductances to the lake. The SFR can be connected to locations where a lake is defined to route water in- and from a lake.

The implementation of the LAK package in iMOD is straightforward as several IDF files (and or constant values) need to be specified. iMOD computes the Lake Identifications and Lake conductances underneath and along the sides of the Lakes. In that same process, iMOD can adjust some of the geohydrologic parameters, such as top- and bottom elevation of model layers as well as permeability values. It is assumed that the position and spatial extent of the lake volume is defined by the specification of a volume of inactive cells within a three dimensional model grid. Because the model grid is used to define the lake volume, the lateral and vertical grid dimensions must be appropriately chosen so that the spatial extent and bathymetry of the lake are defined with the necessary accuracy. In some cases this may require a finer horizontal discretization in the vicinity of the lake and a finer vertical discretization than would be necessary to simulate heads in the aquifer. iMOD will modify the vertical discretization internally, this is explained the following figure.

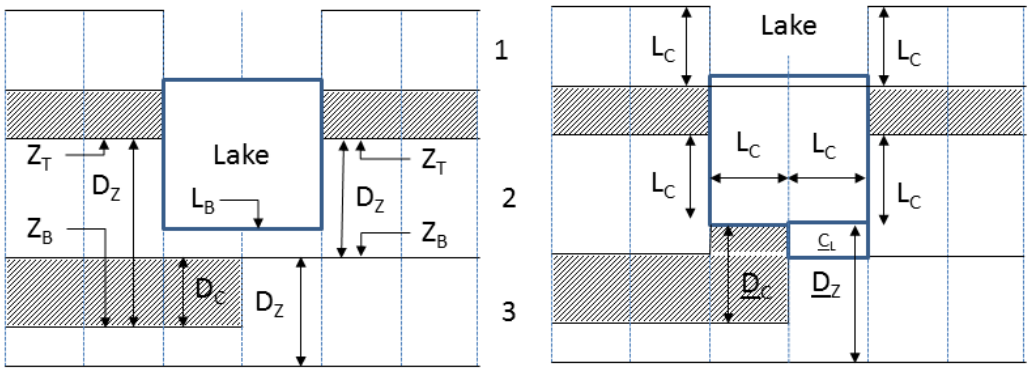


Figure 12.17: *Scheme of the implementation of the LAK package in iMOD.*

Suppose a Lake is considered that intersects two model layers, the user enters the bathymetry of the lake (L_B). Whenever the lake bathymetry is lower than the top of a model layer (Z_T) and higher than the bottom of that model layer (Z_B) the bottom of the lake is supposed to be in that particular model layer. The bottom of a model layer contains an existing interbed with thickness D_C as well. For the Lake package it is essential that the elevation of the model layers describe the bathymetry of the Lake as these are used to compute the table with depth, area, and volume relations. iMOD will adjust the bottom of a model layer to reflect the bathymetry and corrects for the removed aquifer and/or increased thickness of the interbed. In case there is an interbed underneath the Lake, iMOD will increase the permeability of that interbed such that the vertical resistance of that interbed remains identical though its thickness has been increased ($\underline{D_C} > D_C$). In case the lake bathymetry intersects the interbed itself, nothing is changed.

In case there is no interbed, the permeability that existed underneath the Lake bathymetry and in

the same model layer, will be added to the layer below such that the transmissivity for ($\underline{D}_Z$)remains identical to the sum of the original two model layers. The permeability of the model layer underneath will be increased or decreased which depends on the permeability fraction between the two model layers. Furthermore, the vertical resistance of the removed aquifer ($\underline{C}_L$), is added to the Lake resistance.

The side conductances (L_C) are the sum of resistances of a) the lake itself and b) the resistance through the subsoil.

There are some build-in limitations and assumptions:

- ◇ The RCH and EVT packages need to be assigned to model layer 0, which means they will be assigned to the upper active model layer. This is necessary in case a lake or part of lake dries up and blocks any recharge to the underlying aquifer;
- ◇ iMOD does not support the usage of sub lakes;
- ◇ the numbering of the lakes needs to be unique but it is not necessary to number them continuously. The lake number can have the format of a real number (e.g. 1.2 or 1.43) instead of an integer. Usage of integers is preferred as any connection to the SFR needs to be specified in the SFR package by an integer. iMOD will reassign a unique lake number internally, but the original lake number will be displayed in the LAK-package (*.lak);
- ◇ theta is the time weighting factor for computing lake stages during transient time steps. A theta of 0.5 represents the average lake stage during a time step. In iMOD the lake package is configured with a theta of -1.0 that represents the lake stage at the end of the time step; Moreover, a negative THETA applies for a SURFDEPTH that decreases the lakebed conductance for vertical flow across a horizontal lakebed caused both by a groundwater head that is between the lakebed and the lakebed plus SURFDEPTH and a lake stage that is also between the lakebed and the lakebed plus SURFDEPTH. This method provides a smooth transition from a condition of no groundwater discharge to a lake, when groundwater head is below the lakebed, to a condition of increasing groundwater discharge to a lake as groundwater head becomes greater than the elevation of the dry lakebed. The method also allows for the transition of seepage from a lake to groundwater when the lake stage decreases to the lakebed elevation. Values of SURFDEPTH ranging from 0.01 to 0.5 have been used successfully in test simulations. SURFDEP is configured by default to a value of 0.25.
- ◇ the interaction between the lake and groundwater is saved in the BDGLAK and is a per cell lumped volume.
- ◇ inspect the *.LST-file for more detailed information on total water balances per lake and in- and/or outflow to streams that are connected;
- ◇ the LAK package is very sensitive to non-convergence, a quasi steady-state approach and/or a transient implementation with small time steps is recommended.
- ◇ no solute transport is supported.

12.30 MNW MultiNode Well Package

The existing Multi Node Well package of MODFLOW2005 (Halford and Hanson (2002), Konikow *et al.* (2009)) was implemented in iMOD. The Multi-Node Well package is used to simulate “long” wells that are connected to more than one model layer; the abstraction rate is vertically distributed proportional to the transmissivity adjacent to the well screen; e.g. when a hydraulic head gradually drops below the top of a well screen the yield of this shallow part of the well will also gradually drop.

MODFLOW computes the head at a block-centered node of a finite-difference grid on the basis of a fluid mass balance for fluxes into and out of the volume of the cell of interest, including flow in or out of a well located within the surface area (and volume) of that cell. However, because of differences between the volume of a cell and the volume of a well-bore, as well as differences between the average hydraulic properties of a cell and those immediately adjacent to a well, it is not expected that the computed head for the node of a finite-difference cell will accurately reproduce or predict the actual head or water level in a well at that location. Furthermore, if the length of the open interval or screen of a vertical well is greater than the thickness of the cell, then the head in the well would be related to the head in the ground-water system at multiple levels (and at multiple locations for a non-vertical well). Thus, if the user needs to estimate the head or water level in a well, rather than just the head at the nearest node, then additional calculations are needed to correct for the several factors contributing to the difference between the two.

As denoted by the name of the package, the advantage of the MNW package benefits mostly whenever wells are considered that discharge from a multi-aquifer system. Since the MNW package can deal with intra borehole flow and computes a realistic head loss at the well, this makes the package mostly applicable for multi-layered unconfined systems. In the case that the well falls dry, this is more realistic simulated with the MNW package, more-over, the total strength of a multi-layered extraction system remains intact as-much-as possible during a simulation. This example also demonstrates that the well discharge is not simply proportional to the transmissivities of the multiple aquifers screened by the well. For single penetrating, confined system the MNW is similar to the WEL package.

MNW computes a hydraulic head in the cell h_n such that it equals the computed hydraulic head at the well minus a head loss term (e.g. the Thiem equation, we neglect in this tutorial head loss due to skin and local turbulence effects) for that particular cell, so:

$$\begin{aligned} h_{\mathrm{WELL}} - h_n &= \tfrac{Q_n}{2\pi T}\ln\tfrac{\mathrm{r_0}}{\mathrm{r_w}} \\ \mathrm{CWC}_n &= \tfrac{\ln\frac{\mathrm{r_0}}{\mathrm{r_w}}}{2\pi T} \\ Q_n &= (h_{\mathrm{WELL}} - h_n)\,\mathrm{CWC}_n \\ h_{\mathrm{WELL}} &= \tfrac{\mathrm{CWC}_n h_n + Q_n}{\mathrm{CWC}_n} \end{aligned} \tag{12.7}$$

where CWC_n is the cell-to-well conductance m^2/d; Q_n is the well rate (m^3/d), T is transmissivity of the aquifer (m^2/d) at the well, r_0 is the effective radius of a finite-difference cell (m), this is assumed for isotropic conditions as $r_0 = 0.14\sqrt{\Delta x^2 + \Delta y^2}$; r_w is the actual radius of the well. Because r_0 is typically much greater than r_w, the head in a pumping well will be lower than the model-computed head.

The problem is solved by MODFLOW via estimations of h_{WELL} and Q_n that lag an iteration behind estimated of h_n because the above mentioned equation are solved explicitly. This causes slow convergence of the solver if the MNW cells are incorporated in MODFLOW as a general-head boundary (subtract CWC$_n$ from HCOF and subtract CWC$_n \times h_{\mathrm{WELL}}$ from RHS). Convergence is accelerated by alternately incorporating the MNW cells as specified rates in odd iterations (subtract Q_n from RHS) and as general-head boundaries in even iterations.

Note: Multi-node wells with cell-to-well conductances that are “too great” tend to make MODFLOW numerically unstable. Cell-to-well conductances increase as cell size is decreased, which also decreases the effective external radius (r_0). Cell-to-well conductances become greater as r_0 approaches rw and are undefined if r_0 is less than or equal to r_w. For these small cells, a pumped well should be simulated as a high-conductivity zone as cell area approaches the cross-sectional area of a well.

12.31 UZF Unsaturated Zone Package

The existing Unsaturated Zone Flow package of MODFLOW2005 (Niswonger *et al.* (2006)) has been implemented in iMOD. Percolation of precipitation through unsaturated zones is important for recharge of ground water. Rain and snow melt at land surface are partitioned into different pathways including runoff, infiltration, evapotranspiration, unsaturated-zone storage, and recharge. The package was developed to simulate water flow and storage in the unsaturated zone and to partition flow into evapotranspiration and recharge. A kinematic wave approximation to Richards' equation is solved by the method of characteristics to simulate vertical unsaturated flow. The approach assumes that unsaturated flow occurs in response to gravity potential gradients only and ignores negative potential gradients (upward capillary flow); the approach further assumes uniform hydraulic properties in the unsaturated zone for each vertical column of model cells.

The iMOD implementation of the UZF package has the following limitation and/or assumptions:

- ◇ the UZF can be added to the model in a spatially distributed way and therefore for user-specified areas. The pointer value for this needs to be negative to ignore any UZF at that particular location, the absolute value in this IDF file is the model layer for which the upper elevation will be used to specify the surface level;

- ◇ the RCH- and or EVT packages can be used with the UZF package simultaneously, it is a user responsibility not to overlap these concepts with the UZF;
- ◇ whenever the UZF package is active, iMOD will include the WETDRY option automatically for each model layer;
- ◇ the computed net recharge is saved in the BDGUZF file.
- ◇ the package also accounts for land surface runoff to streams and lakes, however this is not supported by iMOD.
- ◇ by default iMOD uses the permeability as specified for model layer 1, to be equal to the saturated permeability used by the UZF package.

12.32 PKS Parallel Krylov Solver Package

12.32.1 Introduction

The Parallel Krylov Solver (PKS) for speeding-up computations is a new package that is being developed in cooperation with the USGS, Utrecht University, Alterra and Delft University of Technology (Verkaik *et al.* (2016)). PKS is basically a linear solver that is largely based on the PCGU-solver in MODFLOW-USG for unstructured grids (Panday *et al.* (2003)) and is used in iMODFLOW for structured grids. PKS is suitable for coarse-grained problems, where the computational time for each subdomain is much larger than the actual communication overhead.

12.32.2 Mathematical model

Subject for PKS is solving the linear system of equations, resulting from the finite-volume discretization of the groundwater flow equation (Harbaugh (2005)), to be denoted by:

$$A\mathbf{h} = \mathbf{b}, \tag{12.8}$$

where $\mathbf{A}$ is a square, symmetric, positive-definite coefficient matrix [L^2/T] that includes cell-by-cell conductances, calculated using cell dimensions and hydraulic conductivities at cell interfaces, and unknown components of sink/source and storage terms; $\mathbf{h}$ is the vector of unknown heads at time t; and $\mathbf{b}$ is a known vector of source/sink and storage terms.

It can be shown mathematically that convergence of iterative methods is strongly related to the spectral properties (eigenvalues) of the coefficient matrix. Instead of solving the system (12.8), it is more efficient to solve the (left-)preconditioned equivalent

$$M^{-1}A\mathbf{h} = M^{-1}\mathbf{b}, \tag{12.9}$$

where M is called the preconditioner. Choosing M is usually done such that M is a good approximation of A, the eigenvalues of $M^{-1}A$ are clustered around 1, and the system involving M is much easier to solve than the original system.

The PKS solver is a so-called Schwarz domain decomposition solver (Dolean *et al.* (2015)) and uses a parallel Additive Schwarz (AS) preconditioner[1]:

$$M_{jac}^{-1} = \sum_i A_i^{-1}, \tag{12.10}$$

where A_i corresponds to the (local) interior coefficients of the (global) coefficient matrix A for subdomain i. Basically, this corresponds to the block-Jacobi (block-diagonal) preconditioner, after numbering the unknowns over the sub-domains.

PKS solves the preconditioned sytem (12.9) with the AS-preconditioner (12.10) using the Conjugate Gradient (CG) Krylov subspace method, resulting in the so-called CG-Schwarz method. Within CG, application of M_{jac}^{-1} can be fully done in parallel. The corresponding sub-domain solutions are obtained inaccurately by ILU(0) (zero fill-in incomplete LU-factorization) preconditioning of A_i only. PKS only supports Dirichlet interface transmission conditions and does not (yet) support coarse-space correction like deflation.

[1] To be more precise, an overlapping Restricted Additive Schwarz preconditioner is implemented, but for iMOD, only a fixed overlap of 1 is being used, corresponding to the non-overlapping case.

12.32.3 Implementation and some practical considerations

Inter-subdomain communication

PKS follows a so-called distributed memory parallellization approach, where each subdomain is uniquely assigned to one computational core (process), using local memory (RAM) only. In this way PKS is scalable regarding problem size and hardware. Exchanging data between subdomains is done by the Message Passing Interface, and typically involves communication for each CG inner (Schwarz) iteration, ensuring a tight coupling: local (point-to-point) communication for the vector update and global (all-reduce) communications for computing interior products and evaluating stopping criteria. Since this is done for each inner iteration the expected speed-up in computational time with PKS largely depends on MPI (latency and bandwidth).

Typically, PKS is suitable through MPI on fast networks, like Infiniband/Myrinet on Linux clusters, or MPI through memory such as multi-core CPUs on desktop/laptop machines or shared-memory machines like SGI-clusters. For the current generation of multi-core CPUs, e.g. Intel Haswell E5-2698v3 16-core CPU, one should keep in mind that for large, memory-driven, groundwater models the actual bandwidth between processor and memory limits the parallel performance. In practice, this means it may be sometimes more beneficial to use less than the maximum number of cores.

Note: On desktops/laptops with multi-core CPUs it may be beneficial not to use all cores.

Load balance

Besides MPI communication, load balance is also very important for parallel performance. This means that the actual work/load should be distributed as equally as possible over the multiple computational cores. PKS now supports two methods: uniform in x,y-direction, and the Recursive Coordinate Bisection (RCB) method. The RCB method, recursively, weight the user-defined load, by alternating in horizontal and vertical direction. Figure 12.18 shows an example of both methods for the Netherlands Hydrological Model (De Lange *et al.* (2014)) and 128 subdomains.

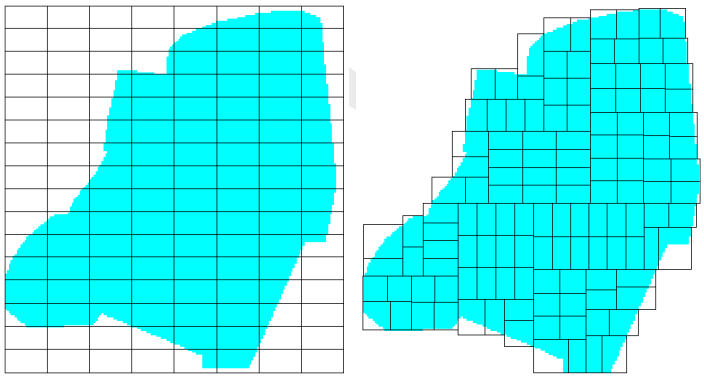


Figure 12.18: *Two partitioning methods for the Netherlands Hydrological Model based on weights as specified by the boundary grid. Left: uniform partitioning; right: recursive coordinate bisection partitioning.*

For this example the uniform partitioning method (left) results in some subdomains containing zero cells, since in this case the model boundary (IBOUND) is irregular, while the RCB method (right) results in a more optimal partitioning. Although RCB results in optimal load, it does not result in an optimal communication scheme regarding MPI. One should notice that even with the RCB and irregular boundaries, finding the optimal weight can be hard and subject for trial-and-error. For each MODFLOW cell, the weight can varying due to for example boundary conditions (stresses) and coupling concepts.

Input/output

Another parallel performance aspect that is important is input/output (I/O). PKS supports parallel binary (clipping) reading of model IDF input files and parallel writing of IDF output files, that can be merged by the master MPI process afterwards. However, the actual I/O performance depends on the given hardware (e.g. type of hard drive) and file system configuration. When too many MPI processes simultaneously try to access one hard drive it can slow down I/O. This is typically the case for common off-the-shelf desktops or laptops. On the other hand, super computers use parallel file-systems - e.g. Lustre file systems - to tackle this problem by accessing multiple hard drives simultaneously.

Note: In general, it is always recommended to try to minimize model output.

12.33 PST Parameter estimation

12.33.1 Introduction

In mathematics and computing, the Levenberg-Marquardt algorithm (LMA) provides a numerical solution to the problem of minimizing a function, generally nonlinear, over a space of parameters of the function. These minimization problems arise especially in least squares curve fitting and nonlinear programming. The LMA interpolates between the Gauss–Newton algorithm (GNA) and the method of gradient descent. The LMA is more robust than the GNA, which means that in many cases it finds a solution even if it starts very far off the final minimum. For well-behaved functions and reasonable starting parameters, the LMA tends to be a bit slower than the GNA. LMA can also be viewed as Gauss–Newton using a trust region approach. The LMA is a very popular curve-fitting algorithm used in many software applications for solving generic curve-fitting problems. However, the LMA finds only a local minimum, not a global minimum.

The used algorithm is known as the Levenberg-Marquardt algorithm (LMA) that goes back to 1943 as Kenneth Levenberg presented his work to the Mathematical Society at their annual meeting (Levenberg, K. *The Quarterly of Applied Mathematics 2*, 164 (1944)). Marquardt on the other hand popularized the method by distributing his FORTRAN code for free. In the following section a brief overview is given of the Levenberg-Marquardt algorithm and its implementation in iMODFLOW.

Most of the following LMA implementation has been based on the paper of Olsthoorn (1995), *Effective Parameter Optimization for Ground-Water Model Calibration*, Groundwater, Vol. 33, no.1) and the paper of Knorr BM (2011), *The Levenberg-Marquardt Algorithm*, Seminar in Modern Physics, Summer 2011 and the PEST Manual of Doherty (2010).

12.33.2 Methodology

The core of parameter estimation is the minimization of some error criterion, cost or objective function $\Phi_{\mathrm{m}}(\mathbf{p})$, that depends on a parameter vector **p** with elements $p_i \rightarrow i = 1, N_p$ where N_p denotes the number of unknowns to be optimized (i.e. the amount of parameters). In general the objective function $\Phi_{\mathrm{m}}(\mathbf{p})$ is the sum of squares sum of the individual errors notated as:

$$\Phi_{\mathrm{m}}(\mathbf{p}) = (\mathbf{y} - \phi(\mathbf{p}))^{\mathrm{T}} \mathbf{Q}_1 (\mathbf{y} - \phi(\mathbf{p}))$$

where **y** are the measurements with elements $y_i \rightarrow i = 1, N_h$; where N_h denotes the number of observations; $\phi(\mathbf{p})$ are the computed head for the parameters defined in **p** and **Q** is the weight matrix assigned to the observations defined as:

$$Q_{i,i} = \frac{1}{\sqrt{\sigma_i^2}}$$

where $Q_{i,i}$ is the weight for the i^{th} observation. The variance σ_i^2 is the squared standard deviation σ_i that measures the amount of variation from the average. A low σ_i indicates that an particular observation y_i should be able to the meet the corresponding computed head ϕ_i more closely that observations with higher values of variations σ. It is possible to specify weight values $Q_{i,i}$ or variances σ_i^2 in iPEST.

The Levenberg-Marquardt algorithm is applied to minimize the objective function value by adjusting the individual values for the parameter vector with $\alpha_i \mathbf{p}_i$ where α_i is the optimal multiplication factor for the i^{th} parameter that yields a minimal objective function value. In order to arrive at a valid minimal objective function value, it is advisable to let N_h substantially larger than the number of parameters N_p (Yeh and Yoon (1981), *Aquifer parameter identification with optimum dimension in parameterization*, Wat.Res.Res, v17, no. 3, pp. 664-672) or apply regularisation as done by the Pilot Point concept (Doherty, 2003), see subsection 12.33.4.

The Gradient Descent Method (the simplest method) approaches the minimum of the objective function $\Phi_m(\mathbf{p})$ by adjusting each parameter according to:

$$\mathbf{p}_{i+1} = \mathbf{p}_i - \zeta_i \nabla \Phi_{\mathrm{m}}(\mathbf{p}_i),$$

where the subscript *i* denotes the sequential parameter iteration (parameter update cycle) and ζ_i is a weighting factor for the i^{th} cycle. It says that for each cycle i, the individual parameter sensitivities ($\nabla \Phi_{\mathrm{m}}(\mathbf{p})$) to the residual surface (i.e. the multi dimensional representation of the objective function), multiplied with a weighting factor ζ will contribute to a reduction on the objective function. In this way steps are taken towards the minimum according to the gradient ∇ of the residual surface. The problem is that the Gradient Descent Method takes large steps in those areas of the residual surface that have small gradients and it takes small steps for those areas with large gradients. It normally leads to zigzagging in long narrow valleys on the $\Phi_{\mathrm{m}}(\mathbf{p})$ residual surface. The Gauss-Newton method replaces the scaling factor ζ by the inverse of the curvature (second derivative$\nabla^2 \Phi_{\mathrm{m}}(\mathbf{p})$ often called the Hessian) of the $\Phi_{\mathrm{m}}(\mathbf{p})$ surface and interchanges the undesired behaviour of the Gradient Descent Method. It therefore converges faster, so:

$$\mathbf{p}_{i+1} = \mathbf{p}_i - \left(\nabla^2 \Phi_{\mathrm{m}}(\mathbf{p}_i)\right)^{-1} \nabla \Phi_{\mathrm{m}}(\mathbf{p}_i).$$

The gradient $\nabla \Phi_{\mathrm{m}}(\mathbf{p})$ is denoted as the Jacobian **J** and each column in that matrix **J** is defined by:

$$\mathbf{J} = \frac{\partial \phi}{\partial \mathbf{p}_i} = \frac{\phi(\mathbf{p}_i) - \phi(\mathbf{p}_0)}{\Delta \alpha_i},$$

where **J** is a Jacobian matrix $N_h \times N_p$ (number of observations row wise and number of parameters column wise) and represents the sensitivity of the residual for each observation point according to a small perturbation $\Delta \alpha_i$ in the i^{th} parameter compared to the original parameter value $\mathbf{p}_0$. Since the algorithm assumes linearity over the interval $\Delta \alpha_i$, the second derivative $\nabla^2 \Phi_{\mathrm{m}}(\mathbf{p})$ is approximated by $\mathbf{J}^{\mathrm{T}}\mathbf{QJ}$ in the neighbourhood of $\mathbf{p}_0$. This yields the following Gauss-Newton parameter update formula:

$$\mathbf{p}_{i+1} = \mathbf{p}_i + \Delta \mathbf{p}_i$$

$$\Delta \mathbf{p}_i = -2 \left(\mathbf{J}^{\mathrm{T}}\mathbf{QJ}\right)^{-1} \mathbf{J}^{\mathrm{T}}\mathbf{Q} \left(\mathbf{y} - \phi(\mathbf{p})\right)$$

where **Q** is the diagonal weight matrix with individual weight values q_i in the diagonal, where the product $-2\mathbf{J}^{\mathrm{T}}\mathbf{Q}\left(\mathbf{y} - \phi(\mathbf{p})\right)$ represents the steepest gradient on the residual surface. If parameters are far from their optimum, which they are probably initially, this $\mathbf{J}^{\mathrm{T}}\mathbf{QJ}$ is only a crude approximation of the true Hessian matrix. As a result the parameter update vector $\Delta \mathbf{p}$ might be quite wrong which can result in a failure to converge. The great insight of Levenberg was to simply combine the Gradient Descent and the Gauss-Newton Methods to include a damping factor λ which determines how much of the Gradient Descent or Gauss-Newton Method to include, so:

$$\Delta \mathbf{p} = -2 \left(\mathbf{J}^{\mathrm{T}}\mathbf{QJ} - \psi \mathbf{I}\right)^{-1} \mathbf{J}^{\mathrm{T}}\mathbf{Q} \left(\mathbf{y} - \phi(\mathbf{p})\right)$$

where **I** is the identity matrix. Whenever ψ is large, the parameter update will be determined more by the Gradient Descent Methods and whenever ψ is small, the Gauss-Newton Method will be included significantly. The great insight of Marquardt is to replace the identity matrix **I** by the diagonal of the matrix $\mathbf{J}^{\mathrm{T}}\mathbf{QJ}$, so:

$$\Delta \mathbf{p} = -2 \left(\mathbf{J}^{\mathrm{T}}\mathbf{QJ} - \psi \mathrm{diag}(\mathbf{J}^{\mathrm{T}}\mathbf{QJ})\right)^{-1} \mathbf{J}^{\mathrm{T}}\mathbf{Q} \left(\mathbf{y} - \phi(\mathbf{p})\right)$$

Olsthoorn (1994) suggested to adjust ψ such that the yielding parameter update vector $\Delta \mathbf{p}$ is within a certain trust hyper sphere (with a radius around the current parameter vector which is determines

by their minimal en maximal values and a maximal $\Delta\mathbf{p}$ compared to the previous iteration). So, by starting at a small value for ψ (full confidence in the contribution of Gauss-Newton), it will increase until all parameter vectors are within their trust hyper sphere. It should be noticed that parameters that exceed their upper and or lower bounds, within their trust hyper sphere, will influence the parameter update for the other parameters and coming iterations. This is circumvented by temporarily remove the Jacobian vector $\mathbf{J}_i$ for that particular parameter i and hold the parameter at their upper- or lower bounds (i.e. a *frozen* parameter). On later iterations, the parameter will be included whenever the parameters vector will be calculated that moves parameters from their bounds back into the allowed parameter domain.

In the figure below it is demonstrated how the trusted hyper sphere affects the behaviour of convergence to the minimum of the $\Phi_{\mathrm{m}}(\mathbf{p})$ surface. In practice, small trust hyper spheres will yield more iterations than large trust hyper spheres; however, the latter can zigzag from one side of the valley onto the other side. In table the following behavior has to be expected for the different settings.

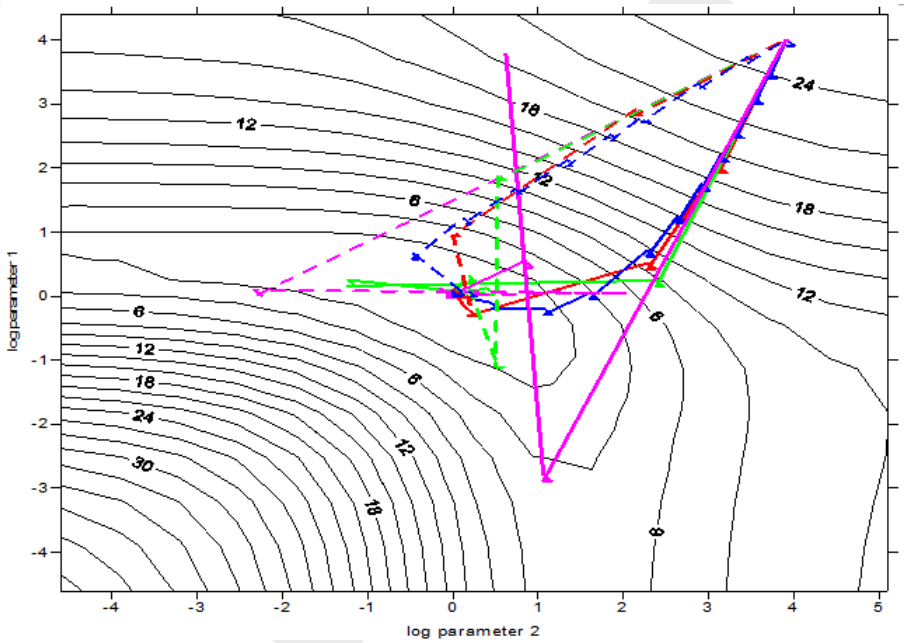


Figure 12.19: *Example of the different behaviours in a common $\Phi_{\mathrm{m}}(\boldsymbol{p})$ surface for different trust hyper spheres, purple=1000, green=100, red=10 and blue=2. Solid lines are Levenberg and dashed lines are Marquardt.*

12.33.3 Eigenvalue Decomposition

Identifiability of the parameters to be optimized is a prerequisite of model calibration. This means that a unique set of parameters values **p** yields a unique head ϕ field. In fact, all information for identifiability of the parameters is contained in the Jacobian matrix of the model at the optimum values of the parameters.

$$\mathbf{J} = \left[\frac{\partial\phi}{\partial p_i}, \frac{\partial\phi}{\partial p_{i+1}}, ..., \frac{\partial\phi}{\partial p_{n_p}}\right]$$

The Jacobian reveal the observability of the parameters by virtue of its rows and columns. Each row expresses the sensitivity of a single observation with respect the set of parameters. Each column expresses the sensitivity of all observations with respect to a single parameter. Some fundamental studies have been carried out which shed light of the inverse problem with respect to the identifiability parameters (Dietrich, 1990, Speed and Ahlfeld, 1996). These are based on the singular value decomposition of the sensitivity matrix, they even define the dimensions of the inverse problem, given the model and the data. Their conclusions should be valid if they are based on the optimum parameters and if the model is no too non-linear near its optimum.

$$\left(\mathbf{J}^{\mathrm{T}}\mathbf{Q}\mathbf{J}\right)\mathbf{v} = \Lambda\mathbf{v}$$

where Λ contains the ordered eigenvalues (singular values are $\Lambda^{\frac{1}{2}}$) and **v** are the eigenvectors. The eigenvectors are representing the axis of the residual surface, the eigenvalues $\sqrt{\lambda_i}$ represent the length of axis i. The ratio between the first and last eigenvalues is the condition number $\kappa = \lambda_1/\lambda_n$. As a general rule of thumb, if the condition number $\kappa(A) = 10^k \implies k = \log(\kappa)$, then you may lose up to k digits of accuracy on top of what would be lost to the numerical method due to loss of precision from arithmetic methods. However, the condition number does not give the exact value of the maximum inaccuracy that may occur in the algorithm. It generally just bounds it with an estimate (whose computed value depends on the choice of the norm to measure the inaccuracy). An informal rule of thumb is that if the condition number $\kappa = 15$, multicollinearity is a concern (multicollinearity or collinearity is a statistical phenomenon in which two or more variables in a multiple regression model are highly correlated); if it is greater than $\kappa > 30$ multicollinearity is a very serious concern. But again, these are just informal rules of thumb. The size of the condition number determines the shape of the minimum in the residual surface. Whenever the condition number is small, the parameters are identifiable, they all contribute significantly and unique to the residual surface. Whenever the eigenvalues decomposition leads to eigenvalues of zero, it means that the matrix $\mathbf{J}^{\mathrm{T}}\mathbf{QJ}$ is singular; in that case the determinant is zero. Whenever this occurs, it means that the current *rank* of the Jacobian matrix is less than the actual dimensions of the Jacobian matrix, in other words, there is a redundancy in the data, as a consequent it is impossible to generate a unique solution. There is a true linear dependency in the data. More often, the smallest eigenvalue might be very small that blow up small errors in the observations and cause large error in (some of) the estimated parameters. It is important to compute those eigenvalues from the covariance matrix stored in double precision.

12.33.4 Pilot Points and Regularisation

12.33.4.1 Kriging

Kriging is a stochastic interpolation method used to estimate the value of an attribute at an unknown location. The beauty of Kriging is that coincides that the unknown values, that need to be interpolated, meet the statistics of the measurement points. Hence the predicted location obtains the lowest statistical probability of not yielding the desired result. Kriging uses prior knowledge about the spatial distribution of an event such as permeability's: this prior knowledge encapsulates how permeability varies over a given region. Then, given a series of measurements of permeability's, Kriging can predict permeability's at unobserved points x^*. The predicted value $\hat{z}$ at location x^* is defined by:

$$\hat{z}(x^*) = \begin{bmatrix} w_1, & w_2, & \cdots & w_n \end{bmatrix} \cdot \begin{bmatrix} z_1 \\ z_2 \\ \vdots \\ z_n \end{bmatrix}$$

where w_i are the weighting variables for each of the measurements z_i. It is different than Inverse Distance Weighting (IDW) which also ignores the spatial location of the sample points and assign weight variables based on the distances between measuring points and unknown points. In Kriging, however, statistics is used to assign values to the weighting variable, such that all locations are related but nearby locations are more related than more distant locations. Moreover, for locations that are strongly correlated, those with less correlation with the estimation location, are effectively ignored. The covariances (interrelationship) and thus the Kriging weights, are determined entirely by the data. The measure that determines the interrelationship between all locations is given by the *semivariance*. It describes the variance of variables γ when the corresponding pair of points x_i and x_j are at some distance ($d = |x_i - x_j|$) apart, such that:

$$\gamma(d) = \frac{1}{2N(d)} \sum_{i=1}^{N(d)} \left(z(x_i) - z(x_i + d)\right)^2$$

where z_i is the value of the measurement at location i, z_j at location j within the distance d and $N(d)$ represents the number of data points that belong to d, normally d represents a certain tolerance of distance. The correlation between the variables solely depends on the spatial distance that separates them and is independent of its true location (i.e. stationarity of the second moment). The key here is to take as many samples within the study region as possible to insure the statistical accuracy of the *semivariance* at each distinct distance d and be able to calculate an average *semivariance* (from all

sample points). Those *semivariances* combined, defines a *semivariogram*. Intersecting the *semivariogram* in three places provides the *nugget* (the height of the jump of the *semivariogram* at the origin), the *sill* (highest level of *semivariance*) and the *range* (where the line of best fit for the *semivariance* has slope of zero). This means that beyond the *range* no relationship exists between corresponding pair of points. The nugget effect can be attributed to measurement errors or spatial sources of variation at distances smaller than the sampling interval or both. Measurement error occurs because of the error inherent in measuring devices. Natural phenomena can vary spatially over a range of scales. Variation at microscales smaller than the sampling distances will appear as part of the nugget effect. Before collecting data, it is important to gain some understanding of the scales of spatial variation. It often happens that the *semivariogram* is a good match for spherical functions, however, linear and/or exponential functions (acts as a kind of IDW) can be applicable as well. The following expressions are used for those relationships:

$$\gamma(d) = c_0 + c_1 * \frac{d}{\text{range}} \qquad \text{linear}$$

$$\gamma(d) = c_0 + c_1 * 1.5\frac{d}{\text{range}} - 0.5(\frac{d}{\text{range}})^3 \qquad \text{spherical}$$

$$\gamma(d) = c_0 + c_1 * 1.0 - \exp(-3.0\frac{d}{\text{range}}) \qquad \text{exponential}$$

$$\gamma(d) = c_0 + c_1 * 1.0 - \exp(-3.0\frac{d^2}{\text{range}^2}) \qquad \text{gaussian}$$

$$\gamma(d) = c_0 + c_1 * d^{0.5} \qquad \text{power}$$

where d is the true distance between two locations. With those *semivariances* the weights are computed to be used in estimating values of the attribute value in unknown locations. Using the *semivariances* predicted from the *semivariogram* the results from the linear equations are the weights producing an interpolation minimizing the error in the predicted values. In these, two approaches are described and implemented in iPEST.

Simple Kriging

Simple Kriging assumes stationarity of the first moment (all variables have the same mean $\bar{z}$) over the entire domain. Eventually the following system is solved for each location x^* to obtain the weights **W**:

$$\mathbf{W} = \mathbf{K}_{\mathrm{S}}^{-1}\mathbf{k}_{\mathrm{S}} = \begin{bmatrix} \gamma(x_1,x_1) & \cdots & \gamma(x_1,x_n) \\ \vdots & \ddots & \vdots \\ \gamma(x_n,x_1) & \cdots & \gamma(x_n,x_n) \end{bmatrix}^{-1} \begin{bmatrix} \gamma(x_1,x^*) \\ \vdots \\ \gamma(x_n,x^*) \end{bmatrix}$$

The assumption that you will know the exact mean $\bar{z}$ is often unrealistic. However, sometimes it makes sense to assume that a physically based model gives a known trend. Then you can take the difference between that model and the observations, called residuals, and use Simple Kriging on the residuals, assuming the trend in the residuals is known to be zero. This is done in Simple Kriging.

Ordinary Kriging

In Ordinary Kriging a constant unknown mean $\bar{z}$ is assumed only over the search neighborhood of x^*. So, instead of subtracting the global mean for each random variable, the mean is computed for the values in the search neighborhood of each individual location x^*.

$$\begin{bmatrix} \mathbf{W} \\ \mu \end{bmatrix} = \mathbf{K}_{\mathrm{O}}^{-1}\mathbf{k}_{\mathrm{O}} = \begin{bmatrix} \gamma(x_1,x_1) & \cdots & \gamma(x_1,x_n) & 1.0 \\ \vdots & \ddots & \vdots & \vdots \\ \gamma(x_n,x_1) & \cdots & \gamma(x_n,x_n) & 1.0 \\ 1.0 & \cdots & 1.0 & 0.0 \end{bmatrix}^{-1} \begin{bmatrix} \gamma(x_1,x^*) \\ \vdots \\ \gamma(x_n,x^*) \\ 1.0 \end{bmatrix}$$

where μ is the Lagrange multiplier used in the minimization of the Kriging error $\sigma_k^2(x)$ to honor the unbiasedness condition. In other words, this μ forces the sum of the Kriging weights to become 1.0. Ordinary Kriging can be used for data that seems to have a trend.

Kriging Variance

By Kriging it is possible to compute the variance σ^2 of the estimated points as well. Therefore, the computed Kriging weights **w** need to be multiplied with the matrix **k**, so:

$$\sigma_{\mathrm{O}}^2 = \mathbf{w}^{\mathrm{T}}\mathbf{k}_{\mathrm{O}} = \begin{bmatrix} w_1, & \cdots & w_n, & \mu \end{bmatrix} \begin{bmatrix} \gamma(x_1, x^*) \\ \vdots \\ \gamma(x_n, x^*) \\ 1.0 \end{bmatrix}$$

The standard deviation of the estimate becomes $\sqrt{\sigma_{\mathrm{O}}^2}$. The variance estimate depends entirely on the data configuration and the covariance function, not on the data values themselves.

12.33.5 First-Order Second Moment Method (FOSM)

Uncertainty in groundwater may be divided into two classes; intrinsic uncertainty and information uncertainty. The first class derives from the variability of certain natural properties or processes and is an irreducible uncertainty inherent to the system. The second class is the result of ”noisy“ or incomplete information about the system and may be reduced by various strategies, notable further measurements and analyses (Dettinger and Wilson, 1981). The spatial and temporal variation of parameters, such as the recharge rate, and spatial variability of properties such as hydraulic conductivity are extremely complicated. The first idea for the First-Order Second Moment Method (FOSM) was applied to groundwater by Dettinger and Wilson (1981). The first moment of the heads is assumed to first-order accuracy by the mean heads obtained as the solution of the model using the optimized values for the model parameters. The variance of the head is denoted as the Second-Order moment. The FOSM method is a approximation method and one of the most widely applied in engineering design. One of the advantages of the method is that it allows the estimation of uncertainty in the output variables without knowing the shapes of pdfs of input variables in detail.

How the uncertainty of the parameters propagates into the head uncertainty is given by the parameter covariance matrix $\mathbf{C}^{\mathrm{p}}$. Here for we can compute the parameter covariance matrix as the total objective function value Φ_{m} divided by the degrees of freedom, so:

$$\mathbf{C}^{\mathrm{p}} = \frac{\Phi_{\mathrm{m}}}{N_h - N_p} \left(\mathbf{J}^{\mathrm{T}}\mathbf{Q}\mathbf{J}\right)^{-1}$$

$$\mathbf{e}_i^{\mathrm{p}} = \sqrt{C_{i,i}^{\mathrm{p}}}$$

the entries of the diagonal elements of this parameter covariance matrix are the squared standard deviations of the uncertainties of the *zonal* values; the confident limits (standard error $\mathbf{e}_i^{\mathrm{p}}$) of the parameters. So, the true parameter value p_i might be in between:

$$p_i - e_i^{\mathrm{p}} \leq p_i \geq p_i + e_i^{\mathrm{p}}$$

This standard parameter error $\mathbf{e}^{\mathrm{p}}$ is a measure of how unexplained variability in the data propagates to variability in the parameters, and is essentially an error measure for the parameters. The variance indicates the range over which a parameter value could extend without affecting model fit too adversely.

Moreover, from this parameter covariance matrix the correlation coefficients can be computed as:

$$R_{i,j}^{\mathrm{p}} = \frac{C_{i,j}^{\mathrm{p}}}{\sqrt{C_{i,i}^{\mathrm{p}} \cdot C_{j,j}^{\mathrm{p}}}}$$

The correlation matrix shows those parameters that are highly correlated whenever they have correlation factors of > 0.90 or < -0.90. This means that whenever it appears that parameter A would be

larger in reality, this also will be the case for parameter B. In other words, one can be linearly predicted from the others with a non-trivial degree of accuracy.

The Jacobian expresses the sensitivity in ϕ to variations or uncertainties in the parameters. In this case the Jacobian is computed for all nodes in the model, and not only for the location where measurements are available, as done by the Levenberg-Marquardt Algorithm (LMA), so in this case $\mathbf{J}_{\mathrm{fosm}}$ is a $N_M \times N_P$ matrix as the Jacobian **J** for the LMA is $N_H \times N_P$, where N_M is the total number of model nodes.

$$\mathrm{var}(\phi) = \mathrm{diag}(\mathbf{J}_{\mathrm{fosm}}\mathbf{C}^{\mathrm{p}}\mathbf{J}_{\mathrm{fosm}}^{\mathrm{T}})$$

Finally the standard head error σ^{ϕ} due to the standard parameter error σ^{p} becomes $\sqrt{\mathrm{var}(\phi)}$. Owing to its simplicity the First-Order Second Moment Method (FOSM) is one of the most widely used technique. Its problem, though, is it linearisation of the model-output function about the mean values of the input variables assuming to represent the statistical properties of the model output over the complete range of input values.

12.33.6 Scaling

For several problems, especially those involving different types of observations and parameters whose magnitudes may differ greatly, the elements of J can be vastly different in magnitude. This can lead to roundoff errors. Fortunately, this can be circumvented to some extent through the use of a scaling matrix **S**. Let **S** be a square, $n \times n$ matrix with diagonal elements only, the i^{th} diagonal element of S being given by:

12.33.7 Sensitivity

The possibility that a parameter estimation problem runs smoothly decreases with the number of parameters. In highly parameterized problems some parameters are likely to be more sensitive in comparison with others. As a result, the Levenberg-Marquardt algorithm may decide that large adjustments are required for their values if they are to make any significant contribution to reducing the objective function *F*(**p**). However, limits are set on parameter changes, such that the magnitude (but not the direction) of the parameter update vector is reduced. If a parameter is particularly insensitive compared to others, it may denominate the parameter update vector, yielding a large update vector. This need to be trimmed to fit the limits of the parameter update and as a result the update for other parameters might not change much at all, with a result that the objective function might be not reduced significantly at all. The result is that the convergence takes place intolerable slowly (or not at all), with a huge wastage of model runs. The relative sensitivity for a parameter is computed by:

$$s_i = \frac{m^{-1}\sum_{j=1}^{m} w_j j_{ij}}{\sum s_i} \cdot 100\%,$$

where s_i is the sensitivity of the i^{th} parameter and is the product of the observational weigh times the Jacobian value for that particular observation j in relation to the parameter *I*, divided by the total observation m. In the figure below an example is given of the relative sensitivity of different parameter during parameter estimation.

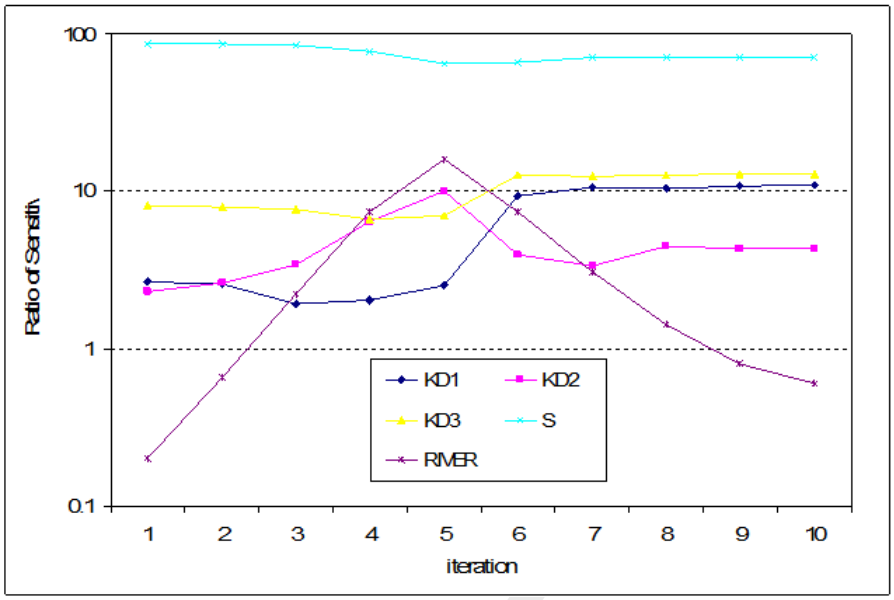


Figure 12.20: *Sensitivity ratio of different parameters during the parameter estimation process.*

The most sensitive parameter is the storage (S) in Figure 12.20, however, the parameter adjustment is adjusting the storage the least since the least sensitive parameter, the RIVER, determines the final parameter update vector. As can see in the decrease of the objective function, it is not the best thing to do. Whenever the sensitivity of the Rivers increase, the storage becomes more important in the gradient and the objection function declines more significantly.

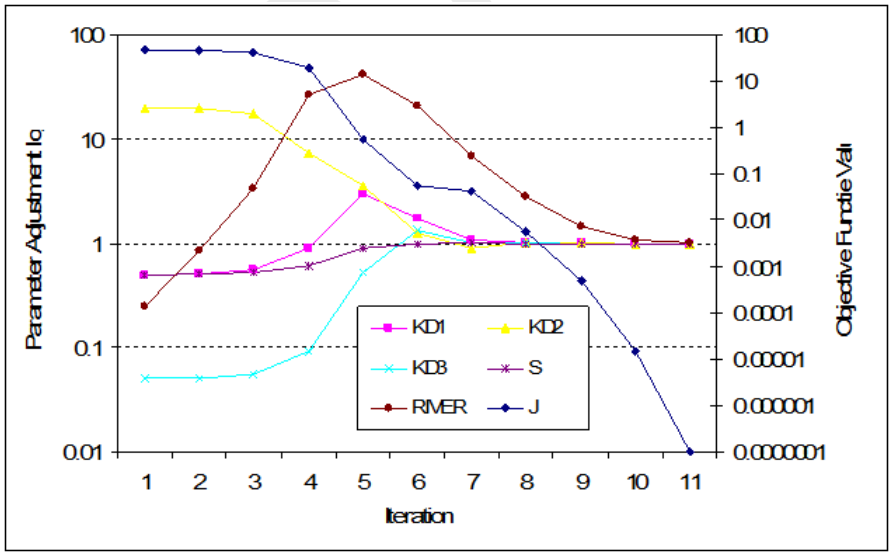


Figure 12.21: *Parameter adjustments in relation to the reduction of the objective function value.*

In order to avoid the disturbance, and therefore slower convergence, due to insensitive parameters, iMODFLOW temporary holds those parameter(s). Whether a parameter is insensitive or not is determined by the ratio of their s_i value compared to the total sensitive value, see Figure 4.8.

12.33.8 Example

In this section a simple example is given for a single optimalisation step using the Levenberg-Marquardt algorithm as outlines in the previous sections. Suppose two parameters $p_1 = p_2 = 1.0$ need to be optimized that describe two different zones for permeability in a single modellayer. Two measurements are available, one measurement in each zone. The first simulation is to run the model with $p_1 = p_2 = 1.0$, thereafter with $p_1 = \Delta p$ and $p_2 = 1.0$ and finally with $p_1 = 1.0$ and $p_2 = \Delta p$ with $\Delta p = 1.1$. For each measurement location the measurements z_j is subtracted from the computed head ϕ_j^i for the i^{th} simulation and stored in the matrix **D** [m], thus:

$$\mathbf{D} = \begin{bmatrix} \phi_1^0 - z_1 & \phi_1^1 - z_1 & \phi_1^2 - z_1 \\ \phi_2^0 - z_2 & \phi_2^1 - z_2 & \phi_2^2 - z_2 \end{bmatrix} \Rightarrow \mathbf{D} = \begin{bmatrix} -1.484 & -1.391 & -1.406 \\ -1.686 & -1.645 & -1.513 \end{bmatrix}$$

From **D** it is obvious that both parameters decline the objective function by a multiplication of Δp. An adjustment in p_1 does reduce the measurement z_1 mostly and p_2 reduces the second measurement z_2. However, both parameters influence both measurements and therefore both parameters cannot be estimated separately as they are correlated to eachother. This is more clear in the Jacobian **J** [m/Δp], defined as:

$$\mathbf{J} = \begin{bmatrix} (d_1^2 - d_1^1)/\Delta p_1 & (d_1^3 - d_1^1)/\Delta p_1 \\ (d_2^2 - d_2^1)/\Delta p_2 & (d_1^3 - d_1^1)/\Delta p_2 \end{bmatrix} \Rightarrow \mathbf{J} = \begin{bmatrix} 0.976 & 0.818 \\ 0.430 & 1.815 \end{bmatrix}$$

where d_j^i represent the i^{th} column and j^{th} row from the **D** matrix. As it is common to apply a $\log$ transformation whenever permeabilities are calibrated, the value for $\Delta p = \log(1.1) = 0.0953$. Based on the Jacobian, the Hessian **H** [$\Delta p^2/m^2$] is computed as:

$$\mathbf{H} = (\mathbf{J}^{\mathrm{T}}\mathbf{Q}\mathbf{J} + \psi\mathbf{I})^{-1} = \begin{bmatrix} 1.137 & 1.585 \\ 1.585 & 3.977 \end{bmatrix}^{-1} = \begin{bmatrix} 1.979 & -0.789 \\ -0.789 & 0.566 \end{bmatrix}$$

where **Q** is an identity matrix with diagonal values of 1.0, so every measurement has equal weights. The Hessian is proportional to the curvature of the plane of the objective function Φ, it implies a large step in the direction with low curvature (i.e., an almost flat terrain) and a small step in the direction with high curvature (i.e, a steep incline). From the Hessian the parameter covariance $\mathbf{C}^{\mathrm{p}}$ [Δp^2] is estimated as:

$$\mathbf{C}^{\mathrm{p}} = \frac{\Phi_m}{\max(1.0, N_h - N_p)}\mathbf{H} = \frac{4.266}{1}\begin{bmatrix} 1.979 & -0.789 \\ -0.789 & 0.566 \end{bmatrix} = \begin{bmatrix} 8.442 & -3.365 \\ -3.365 & 2.414 \end{bmatrix}$$

where the total objective function $\Phi = 4.266$ m^2 and the number of measurements $N_h = 2$ and the number of parameter $N_p = 2$. From these parameter covariance matrix the parameter standard error $\mathbf{e}^{\mathrm{p}}$ [Δp] can be computed as:

$$\mathbf{e}^{\mathrm{p}} = \sqrt{\mathrm{diag}(\mathbf{C}^{\mathrm{p}})} = \begin{bmatrix} 2.905 \\ 1.554 \end{bmatrix}$$

In other words, the twice the parameter standard error represents 95% of the confident limits of the parameter. In this case the current parameter value is highly uncertain as it $\Delta p_1 \pm 2.905$ and $\Delta p_2 \pm 1.554$. Since the parameter is log transformed, the confident limits of the parameters are currently:

$$\exp(\Delta p_1 = 0.0 \pm 2.905) \longrightarrow 0.0033 > \Delta p_1 < 297.32$$
$$\exp(\Delta p_2 = 0.0 \pm 1.554) \longrightarrow 0.0476 > \Delta p_2 < 21.02$$

As the optimization of the parameters continues, this parameter standard error should decline as the objective function value Φ declines and the Jacobian will be become more flattened in the minimum of the plane of Φ. As a result the confident limits decreases as well. Another useful parameter is the correlation matrix $\mathbf{R}^{\mathrm{p}}$ that can be computed from the covariance matrix $\mathbf{C}^{\mathrm{p}}$ easily.

$$\mathbf{R}^{\mathrm{p}} = \begin{bmatrix} 1.0 & -0.745 \\ -0.745 & 1.0 \end{bmatrix}$$

From this correlation matrix is can be observed that both parameters are not correlated significantly and can be determined in combination. Another method of determining the uniqueness of the covariance matrix is the computation of its eigenvalues and eigenvectors, so:

$$(\mathbf{J}^{\mathrm{T}}\mathbf{QJ})\mathbf{v} = \Lambda\mathbf{v} = \begin{bmatrix}1.137 & 1.585\\ 1.585 & 3.977\end{bmatrix}\begin{bmatrix}0.408 & 0.913\\ 0.913 & -0.408\end{bmatrix} = \begin{bmatrix}91.611\\ 8.389\end{bmatrix}\begin{bmatrix}1.137 & 1.585\\ 1.585 & 3.977\end{bmatrix}$$

where Λ contains the ordered eigenvalues (singular values are $\Lambda^{\frac{1}{2}}$) and **v** are the eigenvectors. The eigenvectors are representing the axis of the residual surface, the eigenvalues $\sqrt{\lambda_i}$ represent the length of axis i. The ratio between the first and last eigenvalues is the condition number κ, thus $\kappa = 91.611/8.389 = 10.920$. This condition number κ is relatively small and therefore the problem is well-conditioned; the parameters are identifiable, they all contribute significantly and unique to the residual surface.

The Gradient Descent gradient $\Delta\mathbf{p}^*$ of the parameter adjustment becomes:

$$\Delta\mathbf{p}^* = -\mathbf{J}^{\mathrm{T}}\mathbf{Qd}_1 = -\begin{bmatrix}0.976 & 0.818\\ 0.430 & 1.815\end{bmatrix}\begin{bmatrix}1.0 & 1.0\\ 0.0 & 0.0\end{bmatrix}\begin{bmatrix}-1.484\\ -1.686\end{bmatrix} = \begin{bmatrix}2.177\\ 4.280\end{bmatrix}$$

The parameter update becomes the $\exp()$ of the computed Gradient Descent gradient, so $\Delta p_1 = 8.82$ and $\Delta p_2 = 72.24$, which is rather huge that can be reduced within the limits of a *trust* hypersphere. Whenever the Hessian **H** is included (at this stage the ψ is assumed to be 0.0 meaning that the full Gauss-Newton algorithm is applied, this ψ normally increases during the optimisation such that in the final stage the parameter update will be more equal to the Gradient Descent method) the final parameter update vector $\Delta\mathbf{p}$ becomes:

$$\Delta\mathbf{p} = -\mathbf{HJ}^{\mathrm{T}}\mathbf{Qd}_1 = -\begin{bmatrix}1.979 & -0.789\\ -0.789 & -0.566\end{bmatrix}\begin{bmatrix}-2.177\\ -4.280\end{bmatrix} = \begin{bmatrix}0.931\\ 0.705\end{bmatrix}$$

The elements for the final parameter update $\Delta\mathbf{p}$ becomes $\Delta p_1 = \exp(0.0 + 0.931) = 2.537$ and $\Delta p_2 = \exp(0.0 + 0.705) = 2.024$. After this the sequence starts again.

12.33.9 Remarks

Notes:

- ◇ Do not use parameter adjustments that are too large, whenever many parameters are concerned use a step size of 2, whenever you have less parameters you can increase this to a maximum of 10;
- ◇ Experiment with different starting values for a parameter to see whether you end up with the same optimal values;

12.34 Serial runtimes

The time that a simulation will consume depends on many things, e.g. the type of machine that you're using (hardware), and the configuration of your model. So, the consumption of the ANI package is more than whenever the HFB package is used to simulate any type of horizontal anisotropy.

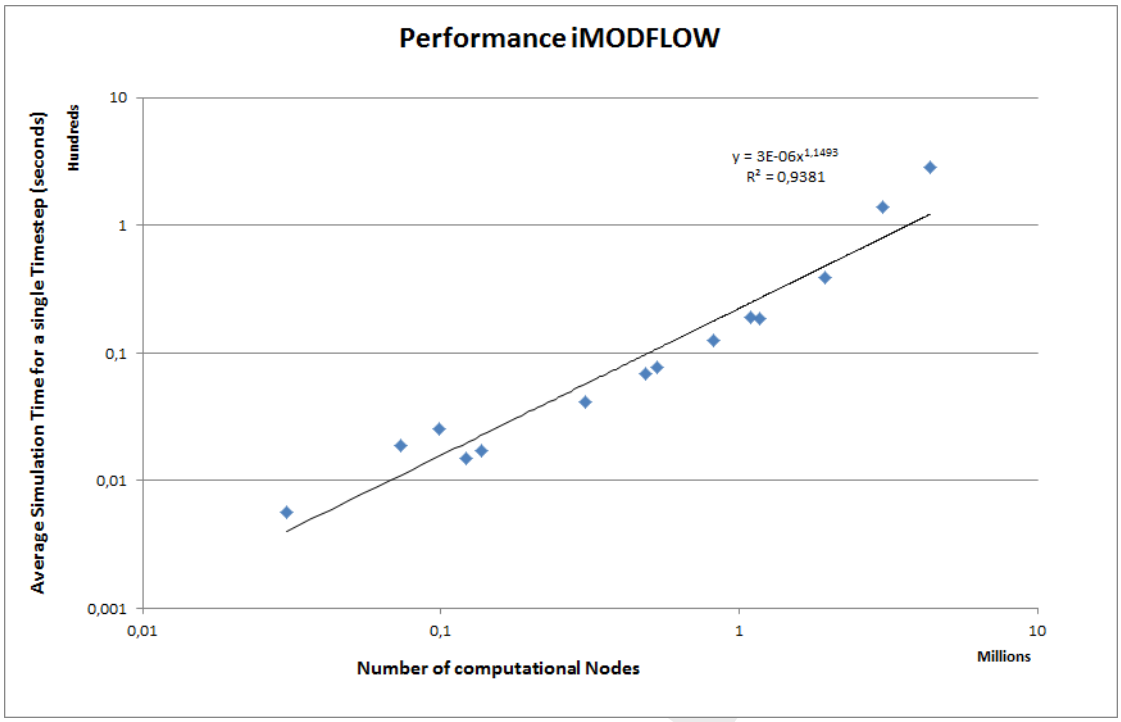


Figure 12.22: *Computed run times for a single time step, for several different amount of nodes. The results are based on the simulation of the IBRAHYM model for 5843 time steps, and cell sizes varying in between 25m^2 and 1000m^2.*

On average is seems that the simulation time is related to the number of nodes as follows:

$$\textbf{Time}\,(seconds) = 3.0^{-6} \times \textbf{Nodes}^{1.15} \times \textbf{Number of Time steps}$$

$$\textbf{Nodes} = \left(\frac{\textbf{Time}\,(seconds)}{3.0^{-6} \times \textbf{Number of Time steps}} \right)^{\frac{1}{1.15}}$$

12.35 Timestep

Just a small nodal spacing is desirable, one would like to use small time steps to obtain an accurate solution as well. A good order-of-magnitude is to estimate the critical time step Δt_c with a formulae given by de Marsily (1986). For a homogeneous and isotropic aquifer Δt_c can be estimated by:

$$\Delta t_c = S \frac{(\Delta x \times \Delta y)^2}{4T}$$

whereby Δx and Δy are the cell sizes of the model (m), S (-) is the porosity values and T is the transmissivity value (m^2/day). The result of the equation is given in the following figure:

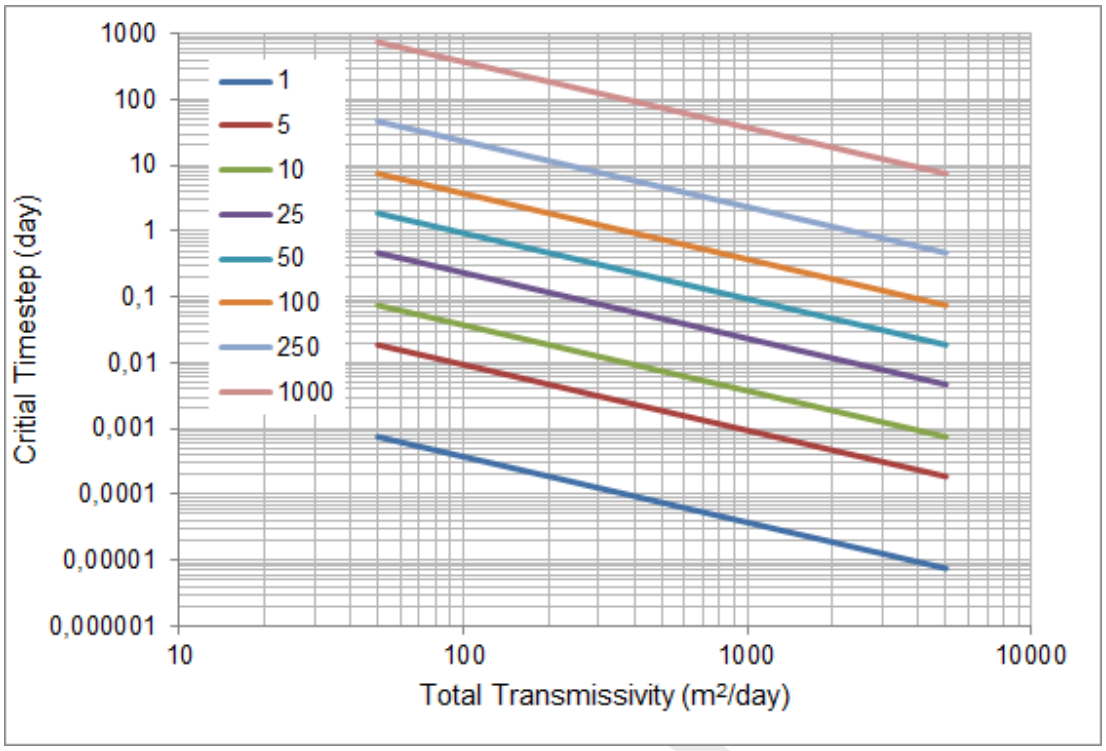


Figure 12.23: *Estimated critical time step (y-axis) for a porosity of $S = 0.15$ and different values for transmissivity (x-axis) and cell sizes (coloured lines)*

References

Ang, A.-S. and W. Tang, 1975. *First-Order-Second-Moment (FOSM) analyse, Probability Concepts in Engineering Planning and Design,*, vol. I Basic Principles. John Wiley & Sons, Inc., New York, USA.

Berendrecht and Others, 2007. "A Methodology for Interactive Planning for Water Management." In *Oxley, L. and Kulasiri, D. (eds) MODSIM 2007 International Congress on Modelling and Simulation Modelling and Simulation Society of Australia and New Zealand*, pages 74-80.

De Lange, W. J., G. F. Prinsen, J. C. Hoogewoud, A. A. Veldhuizen, J. Verkaik, G. H. P. Oude-Essink, P. E. V. van Walsum, J. R. D. J. C. Hunink, H. T. L. Massop and T. Kroon, 2014. "An operational, multi-scale, multi-model system for consensus-based, integrated water management and policy analysis: The Netherlands Hydrological Instrument." *Environmental Modelling & Software* 59: 98-108.

Dettinger, M. and J. Wilson, 1981. "First Order Analysis of Uncertainty in Numerical Models of Groundwater Flow." *Water Resources Research* 17 (1): 149-161. Part 1. Mathematical Development.

Doherty, J. E., 2003. "Ground water model calibration using pilot points and regularization." *Groundwater* 41 (2): 170-177.

Doherty, J. E., 2010. *Approaches to highly parameterized inversion - A guide to using PEST for groundwater-model calibration*. Tech. Rep. 2010-5169, U.S. Geological Survey Scientific Investigations, Reston, Va. U.S. Dept. of the Interior, U.S. Geological Survey.

Dolean, V., P. Jolivet and F. Nataf, 2015. *An introduction to domain decomposition methods*. Society for Industrial and Applied Mathematics (SIAM), Philadelphia, PA. Algorithms, theory, and parallel implementation, PDF: https://www.ljll.math.upmc.fr/nataf/OT144DoleanJolivetNataf_full.pdf.

Glasgow, H., M. Fortney, J. Lee, A. Graettinger and H. Reeves, 2003. "MODFLOW 2000 head uncertainty, a first-order second moment method." *Groundwater* 41 (3): 342-50.

Halford, K. and R. Hanson, 2002. *User Guide for the Drawdown-Limited, Multi-Node Well (MNW) Package for the U.S. Geological Survey's Modular Three-Dimensional Finite-Difference Ground-Water Flow Model, Versions MODFLOW-96 and MODFLOW-2000*. Tech. Rep. Open-File Report 02-293, 33 p., U.S. Geological Survey.

Harbaugh, A., 2005. *MODFLOW-2005, The U.S. Geological Survey Modular Ground-Water Model—the Ground-Water Flow Process*. Tech. rep., USGS.

Konikow, L. F., G. Z. Hornberger, K. J. Halford, R. T. Hanson and A. W. Harbaugh, 2009. *Revised Multi-Node Well (MNW2) Package for MODFLOW Ground-Water Flow Model*. Tech. rep., USGS.

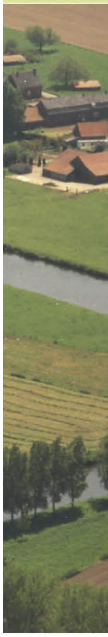
Kunstmann H., K. W. and S. T., 2002. "Conditional first-order second-moment method and its application to the quantification of uncertainty in groundwater modelling." *Water Resources Research* 38 (4).

Leake, D., S.A. Prudic, 1991. *Documentation of a computer program to simulate aquifer-system compaction using the modular finite-difference ground-water flow model.* Tech. rep., U.S. Geological Survey, Techniques of Water-Resources Investigations, Book 6, Chapter A2.

Leake, M. R., Stanley A. Lilly, 1997. *Documentation of computer program (FHB1) for assignment of transient specified-flow and specified-head boundaries in applications of the modular finite-diference ground-water flow model (MODFLOW)*. Tech. rep., U.S. Geological Survey. Open-File Report 97-571.

Maskey, S. and G. V., 2012. "Improved first-order second moment method for uncertainty estimation of flood forecasting." *Hydrological Sciences Journal* .

McDonald, M. and A. Harbaugh, 1988. *A modular three-dimensional finite-difference ground-water flow model: Techniques of Water-Resources Investigations of the United States Geological Survey*. Tech. rep., USGS.



Merritt, M. L. and L. F. Konikow, 2000. *Documentation of a Computer Program to Simulate Lake-Aquifer Interaction Using the MODFLOW Ground-Water Flow Model and the MOC3D Solute-Transport Model*. Tech. Rep. Water-Resources Investigations Report 00-4167, U.S. Geological Survey.

Minnema, B. e. a., 2013. "Utilization of Interactive MODeling (iMOD) to Facilitate Stakeholder Engagement in Model Development Using a Sustainable Approach with Fast, Flexible and Consistent Sub-Domain Modeling Techniques." In *MODFLOW AND MORE 2013: TRANSLATING SCIENCE INTO PRACTICE.* Colorado, The United States of America.

Niswonger, R., D. Prudic and R. Regan, 2006. *Documentation of the Unsaturated-Zone Flow (UZF1) Package for modeling unsaturated flow between the land surface and the water table with MODFLOW-2005*. Tech. Rep. Techniques and Methods Book 6, Chapter A19, 62 p., U.S. Geological Survey.

Panday, S., C. D. Langevin, R. G. Niswonger, M. Ibaraki and J. D. Hughes, 2003. *An unstructured grid version of MODFLOW for simulating groundwater flow and tightly coupled processes using a control volume finite-difference formulation*. Tech. Rep. book 6, chap. A45, 66 p., U.S. Geological Survey. Open-File Report.

Pollock, D., 1994. *User's Guide for MODPATH/MODPATH-PLOT, Version 3: A particle tracking post-processing package for MODFLOW, the U.S. Geological Survey finite-difference ground-water flow model*. Tech. Rep. 94-464 6 ch., U.S. Geological Survey. Open-File Report.

Prudic, D. E., L. F. Konikow and E. R. Banta, 2004. *A New Streamflow-Routing (SFR1) Package to Simulate Stream-Aquifer Interaction with MODFLOW-2000.* Tech. rep., U.S. GEOLOGICAL SURVEY.

Van Walsum, P. E. V., 2017a. *SIMGRO V7.3.3.2, Input and Output reference manual*. Tech. Rep. Alterra-Report 913.3, Alterra, Wageningen. 98 pp.

Van Walsum, P. E. V., 2017b. *SIMGRO V7.3.3.2, Users Guide*. Tech. Rep. Alterra-Report 913.2, Alterra, Wageningen. 111 pp.

Van Walsum, P. E. V. and P. Groenendijk, 2008. "Quasi Steady-State Simulation on of the Unsaturated Zone in Groundwater Modeling of Lowland Regions." *Vadose Zone Journal* 7: 769-778.

Van Walsum, P. E. V., A. A. Veldhuizen and P. Groenendijk, 2016. *SIMGRO V7.2.27, Theory and model implementation*. Tech. Rep. Alterra-Report 913.1, Alterra, Wageningen. 93 pp 491.

Verkaik, J., J. D. Hughes, E. H. Sutanudjaja and P. E. V. van Walsum, 2016. "First Applications of the New Parallel Krylov Solver for MODFLOW on a National and Global Scale." In *2016 AGU Fall Meeting*. San Francisco, California, USA.

Vermeulen, P., 2006. *Model-Reduced Inverse Modeling.* Ph.D. thesis, Delft, University of Technology. With summary in Dutch. ISBN-10: 90- 9020536-5. ISBN-13: 978-90-9020536-6.

Vermeulen, P., B. Becker and T. Heinz, 2014. *Coupling iMOD-SOBEK. Coupling of a surface water- and groundwater flow model to compute bank storage effects in wetlands along the Elbe River on different grid resolutions*. Tech. rep., Bundesanstalt für Gewässerkunde. Koblenz, Germany.

Vermeulen, P. and Others, 2013. "Groundwater modeling for the Mekong delta using iMOD." In *20th International Congress on Modelling and Simulation (MODSIM2013)*.

Release Notes iMOD-GUI

Version	**3.01.00**	**iMOD_V3_01_00_X32R.exe (for 32-bit systems)** **iMOD_V3_01_00_X64R.exe (for 64-bit systems)**
Date	18-9-2015	
Based On	3.00.00	
Changed Functionality	SVN 32	- Displays Bitmaps in the SOLID TOOL, in cross-sections and 3D displays.
	SVN 33	- MODFLOW2000 does not have the capability as MODFLOW2005 does, to use LENUNI and ITMUNI; when importing a MODFLOW2000 model LENUNI and ITMUNI are set separately.
	SVN 46	- Keywords MONTHLY and YEARLY added to the functionality of the iMODBATCH function XYZ2IDF. In combination with a transient IPF (including a TXT file), it is possible to grid the IPF for mean values for selected years or months.
	SVN 46	- Changed ACCURACY from EPSILON(1.0) to 0.0 in the IMODPATH. This influences the minimal velocity that determines whether a particles does not move anymore, by changing it into 0.0 m/day, particles will continue until they truly stop. The value EPSILON(1.0) yielded the value 1.1920929E-07 m/day.
	SVN 48	- Changed the method to write the borehole information in TXT file for IPF files created by the iMODBATCH function DINO2IPF, in situation whereby no values are read, the value becomes "None".
	SVN 70	- The iMODBatch function IMPORTMODFLOW has been modified such that it can read external files with a MODFLOW-88 format. - The iMODBatch function ISGGRID has been extended to export the gridded data to a MODFLOW river file (SCD format).
	SVN 79	- The keywords for the SOLIDTOOL are changed from TOP and BOT into INT to make it possible to construct subsoils with an uneven number of interfaces. - The Darcian upscaling method reviewed and updated.
	SVN 91	- The iMODBATCH function ISGGRID extended to support the export to svatswnr_drng.inp used by MetaSWAP. - Problems with rendering on a Remote Desktop Server(s) related to Winteracter 10. Included an iMOD version based on Winteracter 8 that does not have these problems. In this Winteracter 8 - Remote Desktop Server-version some (minor) functionalities of iMOD are not supported on the RDP-server(s).
	SVN 165	- SOLIDTOOL corrects layers that crosses the lowest layer.
	SVN 166	- IMODBATCH the function IPFSAMPLE includes the parameter IACOL to specify the column to start inserting the sampled data.
	SVN 299	- Enlarged fields (20/50) to (52) in *.dlf files.
New Functionality	SVN 39	- Added the iMODBATCH functionality UTM2LATLONG to transform IDF UTM coordinates to a Lat-Long IPF with data, e.g. to be gridded by the IMODBATCH functionality XYZ2IDF.
	SVN 43	- Added functionality to the WATERBALANCE TOOL to use hours, minutes and seconds as time scales, so IDF files with date and time identifications become processed, e.g. HEAD_20141231063000 as the head on the 31^st of December 2014 at 6hours, 30 minutes and 0 seconds. - Added functionality to the TIMESERIES TOOL to plot time series using hours, minutes and seconds as time scale.
	SVN 48	- Increase the size for the grid fields automatically in IPFANALYSE whenever borelogs/time series are identified.
	SVN 51	- Reading IPF files with associated TXT files with long dates (yyyymmddhhmmss).
	SVN 70	- Added the iMOD Batch functionality ISGADDSTAGE to add and/or modify existing waterlevels in an ISG file from a given IPF file with timeseries.

	SVN 71	- Added the functionalities *Go Back to Previous Extent* and *Go to Next Extent* on the main icon bar and the *Cross-Section* window.
	SVN 163	- Export possible from the SOLID tool to the GEO format as used by GeoSoftware of Deltares. - The SOLID tool supports the dynamic use of different cell size for each interface. - In IMPORTMODFLOW function the Modflow scheme 1996 is supported. - Size of the profile tool increased and gave it a red colour.
	SVN 236	- Reading of *.MAP files from PCRaster.
	SVN 269	- Context-sensitive HELP-functionality: adding section-bookmarks to the iMOD User Manual and synchronize the list of bookmarks in iMOD.
	SVN 283	- The IDF-function for exportation of IDF-files to ascii-files is extended with the "Export given extent"-functionality.
	SVN 290	Reading of GEF files, as addition to iMODBATch function GEF2IPF.
Fixed Bugs	SVN 34	- Bug in IPFSAMPLING in combination with CSV-file format.
	SVN 46	- SAVE button didn't work for steady-state configuration, also the selection of a different model layer didn't responded accordingly.
	SVN 47	- Bug in iMODPATH using NCON=0 should be NCON=1.
	SVN 51	- Bug in IDFCALC whenever the function MIN,MAX,MEAN or SUM is selected; the variable LEX was not initiated. - Bug in X64 versions only: in displaying the license agreement, the variable IU was not initiated.
	SVN 60	- Bug in WATERBALANCE as a result of implementation of SVN 43, dates with 8 digits didn't work anymore.
	SVN 70	- Bug in default legends that could not be saved temporarily whenever a relative pathname was specified by the USER keyword in the preference file.
	SVN 72	- Bug on the *Add Topography* window as the coordinates could not be manipulated appropriately.
	SVN 76	- Bug in memory allocation for the Quick-Response Tool. - Bug in reading IPF files as CSV using the double quotes.
	SVN 77	- Bug in display of IFF lines in the 3D Tool that are vertically.
	SVN 88	- Export format for the output files for iMODPATH (IFF and IPF) synchronized. - Identical algorithm used in the postprocessing of pathlines in the iMODBATCH function iMODPATH, to determine appropriate cell indices for points as used in iMODFLOW. This means that points that are exactly on the boundary of model cell will be assigned to the i+1 cell instead of i.
	SVN 163	- Bug in creation of legend where the difference exceeds the range of a single precision real, namely >10.0+e37. For those cases the legend will be inaccurate but iMOD will not hang. - The display of the *NodataValue* of an IDF is displayed correctly in MapInfo and IDF Edit displays.
	SVN 213	- Bug in *Compute Mean Values...*, after measuring the mean the specific dialog window cannot be closed neither it is possible to proceed with the iMOD session.
	SVN 216	- Bug in positioning of labels in 3D-tool. Labels did disappeared when turning the 3D-schematisation under certain angles.
	SVN 218	- Update of keywords vor iMOD Batch in code.
	SVN 226	- Fix coordinates in CreateIDF whenever changes are made in the dialog.
	SVN 240	- Correct reading of run-files in the ModelTool without bounding coordinates in submodels.
	SVN 254	- iMOD Batch reading of keuword with an extra space after the "="-signs raised a problem. This has been fixed, as it was noticed by the GxG-function in iMOD Batch using the keyword IPERIOD=.

	SVN 267	- Changed the keyword CROSS-SECTION_IN to CROSSSECTION_IN
	SVN 287	- Fixed bug in reading *prf having a last empty line.
	SVN 298	- No capitalizing input from *.ini file.
	SVN 299	- Colouring of the correct field using DLF legends.

Version	**3.2.00**	**iMOD_V3_2_X32R.exe (for 32-bit systems)** **iMOD_V3_2_X64R.exe (for 64-bit systems)**
Date	11-11-2015	
Based On	3.01.00	
Changed Functionality	SVN 305	- Reading/assignment of DLF files (maximal 10) for usage within Profile Tool, 3D Tool and IPF Analyse.
	SVN 309	- Usage of the DLF field colourwidth to display boreholes with variable widths. - include the keyword STOPERROR in BAS file for convergence issues in Modflow2005.
	SVN 312	- Labeling of IPF files in the 3D tool can be specific selected for each IPF separately.
	SVN 320	- ISGGRID create nodatavalues (-9999.00) for cells not intersected by lines.
	SVN 325	- IMPORTSOBEK stopped whenever actual length weren't equal to the lengths based on the nodes of the segment. The import now is not stopped but a warning is issued to the file importsobek.log and the process continues. - Enlarged fields (20/50) to (52) in *.dlf files.
New Functionality	SVN 305	- Add screen number for IPF and IFF in the profile tool. - Use different legends for IPF files (*DLF).
	SVN 309	- Save DLF legends in IMF-files. - Manually activate display of IPF/IFF files during moving/drawing the cross-section. - Mousemove coupled to location in identification window in IPF Analyse via Profile Tool.
	SVN 320	- Saving of solid files during editing without leaving the cross-section tool. - Extended the IMODBatch functionality ISGEXPORT with keyword IEXPORT to denote export of calculation points or cross-sections.
	SVN 326	- Remove and/or modify more nodes in SOLIDTOOL simultaneously
	SVN 341	- Added Inf and NaN in IDF Edit to search on those values in the IDF files.
	SVN 343	- Added an active/deactive code per line in the SolidTool. Now per line it can be specified whether or not it need to be included in the solid.
	SVN 351	- Added functionality in the 3D Tool to zoom to predefined zoom scales. - Display the lines in the cross-sections as true splines or straight lines.
	SVN 364	- Change timesteps in the projectmanager. - Save cross-sections and 3D Tool configurations in a iMOD Demo-mode.
	SVN 370	- Include the option sign() as a function in IDF Calc, subtract only whenever the sign of the two are equal and use pointer values to note the type of difference. - the iMODFLOW-executable present in the *{installfolder}* will be invoked instead of the iMODFLOW-executable copied to {installfolder}\MODELS\{Result Folder}.
	SVN 375	- Usage of preference colours for the default legend. - Apply a value in IDFCalc to "trim" outcome of calculation whenever the outcome is less than a specified absolute value

	SVN 401	- Export to Modflow2005, give explicitly if the model is 3D or Quasi 3D.
Fixed Bugs	SVN 306	- iMODBatch ISGSIMPLIFY removal of first and last calculation point in case stage were completely flat. - iMODBatch CREATESOF correct usage of given OUTLET points to stop tracing the drainage level.
	SVN 309	- Screen number were outgreyed in Profile Tool. - Export of BND to Modflow2005 created constant head along submodel as it was filled with nodata from IDF.
	SVN 312	- Display of bitmaps in 3D Tool in combination with bitmaps attached to solid cross-sections.
	SVN 343	- Total length of line in SolidTool didn't match true length, only visible in cross-sections with many points.
	SVN 351	- Delete of spf will not shift attached bitmaps appropriately.
	SVN 376	- Use of small-caps for FUNC in IDFCALC gave errors.
	SVN 378	- Bug in smoothing the IDF files whenever the file to be used for the smoothing exceeds the size of the IDF to be smoothed upon.

Version	**3.2.1**	**iMOD_V3_2_1_X32R.exe (for 32-bit systems)** **iMOD_V3_2_1_X64R.exe (for 64-bit systems)**
Date	24-11-2015	
Based On	3.2	
New Functionality	SVN 440	iMOD-GUI can now invoke iMODFLOW using foldernames containing spaces.
Changed Functionality		
Fixed Bugs		

Version	**3.3**	**iMOD_V3_3_X32R.exe (for 32-bit systems)** **iMOD_V3_3_X64R.exe (for 64-bit systems)**
Date	25-03-2016	
Based On	3.2.1	
New Functionality	SVN 439	- Specify the option to reduce sizes of boreholes if they do not fit next to each other and thus may overlap.
	SVN 466	- Added iMODBatch function RUNFILE, to create *.PRF from *.RUN files and/or create *.RUN files out of *.PRJ files.
	SVN 471	- Project Manager supports now the creation of RUNFILES. - PlugIn Tool is added as an additional tool to support external programs or scripts to be invoked by iMOD and exchange input and output. - Added an extra MetaSWAP output component to the waterbalance tool (BDGPSSW).
	SVN 471	- The Interactive Pathline Simulator tool for animating groundwater flow.
	SVN 487	- Added units to waterbalance items.
	SVN 502	- Added an option to change the transparancy of individual IDF files in the 3D Tool.
	SVN 512	- The IPS functionality can now be started from the Pathline Tool.
	SVN 520	- The usage of breaklines in the SOLID Tool is made available in the GUI. - Coordinates in the Profile Tool can be decreased in number by specifying a minimal distance between coordinates. - Compute differences in IDFTIMESERIES can handle IDF files with hours, minutes and seconds.

	SVN 528	- Automatic spinner in 3-D. - Entry of scale ratio in the Profile Tool. - Display label and size on the cross-section for the SOLID Tool on 2D
	SVN 541	- Kriging settings can be defined per interface.
	SVN 544	- Automatic rendering of the image in a circular movement in 3D by pressing the spacebar.
	SVN 546	- Bitmap in the background of cross-sections in the SOLID Tool can be temporarily hidden and fixed so that it cannot be moved while adjusting the lines of the cross-section.
Changed Functionality	SVN 422	- Solid Tool; Compute Interfaces window. Export to *.geo is with the "version" name attached to the keyword VERSION.
	SVN 439	- Spline mode in Solid Tool is "off" by default, for export to IPF or GEO.
	SVN 489	- The IPS module can create a temporary submodel for particle tracking purposes. - The reading module or the imodpath entries has been made similar to other scaling modules.
	SVN 504	- PNG, PCX and JPG file can now by used as background images, and can be resized and flipped horizontally and vertically. - iMODBatch function CREATEIDF will NOT ask to overwrite existing IDF-files while importing ASC-files, GUI still does. - Read in GEN file in Profile Tool are corrected for duplicated points.
	SVN 507	- Previous foldernames are saved to be re-used in next search in folders.
	SVN 512	- Pathline Tool creates its runfile in the RUNFILE folder instead of the TMP folder, it also creates a stamp of the chosen model result folder in the filename. - Adjustment of the NODATA value in the IDF file, causes that the content of the IDF file itself will be changed accordingly. So values that are equal to the previous NODATA value, will become adapted to the new NODATA value. - Bitmaps that can be positioned behind the cross section in the solid tool, can be stretched and moved interactively whenever the corresponding bitmap is selected from the Add Background Image dialog. - Background Images may be BMP, PNG, PCX and JPG files.
	SVN 516	- IFF attributes can be plotted all, also whenever the number of columns are enlarged above 7.
	SVN 581	- In the waterbalance tool: extra comment line that explains the possible causes for disclosure of the balance, e.g. differences in used units (m3/day or mm/day) for specific fluxes.
Fixed Bugs	SVN 418	- Translate x,y coordinates from shape files into GEN files using the double-precision format. - Non-existing background files are turned off and properly reset.
	SVN 427	- Bug in reading the CLR files and display them in the preference tab.
	SVN 429	- Bug in collecting correct dates in the waterbalance tool.
	SVN 432	- When defining a new function using iMODBatch, now the correct function is presented (instead of the next function from the list of available functions).
	SVN 443	- Whenever IPF files are not active in profile tool setting the snapping option was not available.
	SVN 466	- Bug in ISG grid in combination with 2d and 1d cross-sections.
	SVN 467	- iMODBATCH function MKWELLIPF ignored ICLAY variable.
	SVN 471	- Bugfix in Compute GXG. Selecting correct surface level.
	SVN 472	- Bigfux in iMODBatch PLOT function - usage of IPF files for timeseries plotting.

	SVN 482	- Usage of particles pass through all weak sinks gave problems whenever the weak sinks approaches a strong cell as an internal value 0f FRAC=0.99 was used to denote a strong cell, FRAC is now 1.0 and maximized to be 1.0 numerically. - Applying a fraction for iMODPATH (ISNK=3) didn't work from iMODBatch.
	SVN 486	- Computation of SUMC in IDFSCALE option 14, went wrong. - Applying lower-left coordinates in IDFSCALE went wrong.
	SVN 489	- Default colouring of IDF files was maximized to 50, following files got colour number 1, now the colour numbering continues.
	SVN 502	- Bug reading GEN files for overlays. - Intersect for non-equidistantial cell went wrong, created a killing bug for the profile-tool. - 3D tool with non-equidistantial IDF gave bug. - In profiletool, whenever a knickpoint was positioned outside the selected IDF, the profile length didn't take into account the extra space of the cross-section outside the IDf file.
	SVN 509	- Import of Modflow2005 the LENUNI variabel didn't applied correctly to EVT package and the elevation in the DIS whenever LAYCON=0.
	SVN 515	- Bug in timeseries export.
	SVN 528	- Load SHP file in CreateIDF from Polygons/lines.
	SVN 544	- Rasterizing ISG didn't take into account stages with nodata values for transient mean values.
	SVN 616	- Bug in Solid Tool by using the "pan" function in Zoom-in modus.

Version	**3.4**	**iMOD_V3_4_X32R.exe (for 32-bit systems)** **iMOD_V3_4_X64R.exe (for 64-bit systems)**
Date	30-06-2016	
Based On	3.3	
Changed Functionality	SVN 647	- 3D TOOL: 3D Tool is organized differently whereby the dialog is attached to the graphical screen and the tool operates independently of the existing 2-D screen.
	SVN 666	- IMODBATCH: EXPORTASC Write results per row instead of free-formatted
	SVN 660	- IMODBATCH CUS: Usage of IEXPZONE to include an additional buffer around each isolated formation to ensure a more logical connection within parts of the formation laterally; - Usage of ICLIP to include a clipping IDF for each formation to be internally blanked out to ensure usage of overlapping maps for formations
	SVN 635	- XYZTOIDF: Interpolation of interfaces from IPF files, median values are not supported anymore
	SVN 707	- WATERBALANCE: Always using files with/without *sys* in their given names
	SVN 709	- STARTPOINTS: Tried to read from the non-existing SDF-file the first time an SDF-file is created
	SVN 713	- GENERAL: The numeric format of plotting IDF values is now depending on the accuracy of the IDF values
	SVN 738	- CONTOURING: Improved contouring algorithm, especially to include delineation of flat areas
	SVN 756	- CONTOURING: Legend label on contours uses appropriate number of decimals - LEGEND: Legend plotting now plots a grey rectangle around classes - 3DTOOL: The menu option 'Select' is now part of the main dialog and removed from the main menu

	SVN 762	- LEGEND: Enhanced legend plotting for 255 and 50 classes categories
	SVN 781	- IMODBATCH: Testbank does not subtract results if one-of-the-two is equal to its Nodatavalue
	SVN 787	- IMODPATH: Restored functionality to save deepest model layer during particle tracking (MAXLAY)
New Functionality	SVN 635	- SOLID TOOL: Usage of separate IPF files to include (additional) interpolation points, or use those points solely for the interpolation, with- and without associated txt files that describe the elevations of the individual interfaces. Without associated txt files, each IPF describes the z-elevation at the third column
	SVN 642	- GENERAL: Relative path can be read from IMF-files, those are relative to the name of the current - IMF-file and are converted to global paths
	SVN 734	- LEGEND: Legend can be adjusted with chosen intervals
	SVN 778	- GENERAL: Added debug reporting level
	SVN 784	- GENERAL: Reading of Point Shapefiles will be converted to IPF-files, also Shapefiles can be read from the MAP-tab on the iMOD Manager
	SVN 849	- GEOCONNECT TOOL: Geoconnect tool is usable to determine the geologic origin of 3D models (what geologic formation is within what model layer) and (re)create the parameterisations of a 3D model (compute the correct k-values for each model layer based on the fractions of geologic formations in that model layer)
Fixed Bugs	SVN 625	- TIMESERIES TOOL: Series can be plotted while new files are added to the selected folder in the background as a model is still running
	SVN 630	- IMPORT MODFLOW: Time units to adjust PERLEN to days, usage of TSMULT included to generate additional stress-periods - ISS flag does not to be read from the BCF pcakage in 2000 and 2005 configurations - Conversion of time units to days was wrong for the WEL, DRN, RIV packages - GENERAL: Out greying of the iMOD Info button on the menu-bar didn't synchronize with the rest
	SVN 634	- PROFILE TOOL: Cross-section did not work whenever DX is not equal to DY
	SVN 636	- ISG EDIT TOOL: Gridding of transient data gave error, in combination of computing the mean and entered start and end date
	SVN 640	- GEOCONNECT: Factors in preprocessing were not used; iMODBatch in preprocessing array already allocated; Added coordinates on tab 1; include the identify button on tab 3 to inspect the current existing formations on the current window. - DEMO-MODE: 3D DEMO functionality was not working properly
	SVN 695	- MF2005 EXPORT: HFB was not assigned to the utmost row and column, e.g. irow=1, irow=nrow, icol=1 and icol=ncol - MODELTOOL: Applied quotes for file names
	SVN 698	- PATHLINES: Writing of column/row numbers at start- and end location in IPF files was switched
	SVN 706	- WATERBALANCE: Could not find the *sys* files from iMODFLOW V3.0 and younger
	SVN 709	- GENERAL: Usage of MAXSHAPES is supported from the preference menu - IPS: Usage of constant values from a iMODPATH runfile for TOP and BOT parameters
	SVN 727	- IMODBATCH: Whenever an argument is missing after the "=" of an optional argument an error message appeared

SVN 732	- MODELTOOL: Including the PST parameter while converting to a imodflow.run file - GENERAL: Didn't position a contour line in between a class of 0.0
SVN 744	- PROJECTMANAGER: Saving of number of timeseries in runfile on right position PST was not mentioned in the header of packages - ISGEDIT: Gridding of ISG file did not overrule grid-dimensions entered on the last window
SVN 750	- KRIGING: Improved algorithm, NUGGET effect was not correctly processed
SVN 762	- LEGEND: Bug in class-legend gave a crash whenever a negative value from the legend was selected in the table
SVN 776	- PROFILETOOL: Loading of BMP causes the 21st filename (IDF, IPF of IFF) to be closed by the Winteracter routine IGRFILEINFO.
SVN 781	- PROJECTMANAGER: Removal of PST didn't work properly - GEOCONNECTTOOL: Small issues resolved, crash by repeatedly starting the post-processing

Version	**3.6**	**iMOD_V3_6_X32R.exe (for 32-bit systems)** **iMOD_V3_6_X64R.exe (for 64-bit systems)**
Date	10-05-2016	
Based On	3.4	

Starting from iMOD 3.6 we summarize all new, changed, extended and fixed functionalities on the iMOD-website: http://oss.deltares.nl/web/imod/release-notes. Per release these release notes are also distributed per email to all iMOD-community members,

Release Notes iMODFLOW

Version	**3.00.01**	**iMODFLOW_V3_00_01_METASWAP_SVN1004_X32R.exe** (for X32-bit systems) **iMODFLOW_V3_00_01_METASWAP_SVN1004_X64R.exe** (for X64-bit systems)
Date	15-10-2014	
Based On	3.00.00	
New Functionality		
Changed Functionality		linked with MetaSWAP SubVersion number 1004 from repository https://repos.deltares.nl/repos/GWSobek/trunk/src/modmsw/
Fixed Bugs	SVN 49	When a GEN-file coincides exactly with cell face no HFB-cell face was assigned resulting in a barrier with a hole. This bug has partially been fixed; with the real world test model NHI the bug fix works, however, the standard USGS HFB-test still fails because the test contains a barrier partly at a cell face. Version 3.00.01 was released because the bug manifests only in exceptional cases; a subsequent bugfix is planned to also fix these exceptional cases.

Version	**3.00.02**	**iMODFLOW_V3_00_02_METASWAP_SVN1004_X32R.exe** (for X32-bit systems) **iMODFLOW_V3_00_02_METASWAP_SVN1004_X64R.exe** (for X63-bit systems)
Date	20-11-2014	
Based On	3.00.01	
New Functionality		
Changed Functionality		
Fixed Bugs	SVN 80	IMOD-319: default value added for KVA-module (1.0).
	SVN 81	IMOD-327: bug fixed upscaling anisotropy factor (most frequent occurrence).
	SVN 82	Bug fixed applying factor for recharge.

Version	**3.01.00**	**iMODFLOW_V3_01_00_METASWAP_SVN1031_X64R.exe (for 64-systems)** **iMODFLOW_V3_01_00_X32R.exe (for 32-bit systems)** **iMODFLOW_V3_01_00_X64R.exe (for 64-bit systems)**
Date	17-07-2015	
Based On	3.00.02	
New Functionality	SVN 194	In Perched Water Table PWT-package: new conceptual enhancements implemented for how groundwater flows at the edges of a Perched Water Tables to avoid some numerical instabilities.
Changed Functionality		linked with MetaSWAP SubVersion number 1032 from repository https://repos.deltares.nl/repos/GWSobek/trunk/src/modmsw/
	SVN 188	Update for VS2008 (EXTERNAL statements removed) - Update for interface MODFLOW-TRANSOL.
	SVN 194	HFB-package update (based on an earlier implementation in iMODFLOW 2.6.) improving the discretization of curved lines to the model grid.
	SVN 202	PEST package update based on iMODFLOW 2.6.
	SVN 221	Update for TRANSOL interface.
	SVN 242	Update interface with TRANSOL to previous version.
	SVN 259	Some minor iPEST-messages and I/O adjustments.
	SVN 260	Removed 'modflow' subdirectory from output results directory.

	SVN 214, 215, 228-230, 234, 246	Update of iMOD license text.
Fixed Bugs	SVN 105	In SGWF2BCF7C: adding uninitialized variable CR to variables list.
	SVN 191	Initialization of constant CNSTNT added for u1drel and u2dint.
	SVN 212	In HFB package: minor error-correction for case that lines are outside model domain.
	SVN 238	In Grid2MetaSWAP: reading ascii files standard with xllcorner and yllcorner which also could be xllcenter and yllcenter.
	SVN 257	In coupling Modflow-MetaSWAP with Mozart: change of general missing parameter value.
	SVN 258	In HFB package: minor error-correction for case that lines are outside model domain.
	SVN 261	In ISG package: a small change in the ISG calculation routine was made.
	SVN 261	Automated scaling factor: the computation of the simulation window was sometimes incorrect in case the extent of the entered model was not exactly divisible by the cell size of the model.

Version	**3.2**	**iMODFLOW_V3_2_METASWAP_SVN1044_X64R.exe** (for 64-systems) **iMODFLOW_V3_2_X32R.exe** (for 32-bit systems)
Date	10-09-2015	
Based On	3.01.00	
New Functionality		
Changed Functionality		linked with MetaSWAP SubVersion nr. 1044 from repository https://repos.deltares.nl/repos/GWSobek/trunk/src/modmsw/
Fixed Bugs	SVN 317	Bug fix uninitialized arrays when using the ANI package. This problem may result in a floating point error at some machines.
	SVN 318	Bug fix automatic assigning ISG to layer: a floating point error (division by zero) might occur are specific machines.
	SVN 344	Bug fixes CHD-package: 1. input start/end head was swapped; 2. iMOD usage of factors (e.g. h = 0.5h1 + 0.5h2) was incorrect.
	SVN 352	Bug fix ISG 2-D cross sections.

Version	**3.2.1**	**iMODFLOW_V3_2_1_METASWAP_SVN1044_X64R.exe** (for 64-systems) **iMODFLOW_V3_2_1_X32R.exe** (for 32-bit systems)
Date	20-11-2015	
Based On	3.2	
New Functionality	SVN 438	iMODFLOW can now work with foldernames containing spaces.
Changed Functionality		
Fixed Bugs		

Version	**3.3**	**iMODFLOW_V3_3_METASWAP_SVN1047_X64R.exe** (for 64-systems) **iMODFLOW_V3_3_X32R.exe** (for 32-bit systems)
Date	25-03-2016	
Based On	3.2.1	
New Functionality		
Changed Functionality		linked with MetaSWAP SubVersion nr. 1047 from repository https://repos.deltares.nl/repos/GWSobek/trunk/src/modmsw/

Fixed Bugs	SVN 561	Restoring the gridding functionality of flow-width in ISG.
	SVN 618	Bug fix Metadata package (MET) timestep management in MODFLOW, causing a delayed read of transient well data. This bug was relevant when defining stress period lengths that were larger than the available time discretisation present in the ipf- and txt-files used as the source of the abstraction data.
	SVN 621	Adjusted scaling of well extraction from median value to mean values weighted to time - usage of nodata values in txt files.

Version	**3.4**	**iMODFLOW_V3_4_METASWAP_SVN1047_X64R.exe** (for 64-systems) **iMODFLOW_V3_4_X32R.exe** (for 32-bit systems)
Date	20-06-2016	
Based On	3.3	
New Functionality		
Changed Functionality		
Fixed Bugs	SVN 660	DRN: Memory reallocation of the drain package will be performed whenever the number of drains exceeds the previous allocated memory.
	SVN 662	TIMESERIES: A limited number of unit numbers were available (10-99) for time series, now it is set to (10-999). COMMON: An inactive package in the runfile was seen as "reuse" package instead of a new package definition without any entries.
	SVN 680	PCG: The cleaning of the matrix coefficients was not done correctly.
	SVN 671	IPF: Reading screen depths only needed whenever ilay.eq.0.
	SVN 719	ANI: The incorrect nodata values were assigned to the inactive corner cells at the computational area.
	SVN 720	HFB: The issue with the out of array boundary horizontal flow barrier package is solved.
	SVN 732	COMMON: The initialization problem is solved for iMOD license agreement file.
	SVN 786	COMMON: A non-converging steady-state MODFLOW simulation was not finalized correctly.
	SVN 792	BALANCE: Computing the waterbalans, the ANI-terms were not included in the constant-head boundary flux, also they were not applied as a correction on the BDFFFF and BDGFRF fluxes for the LPF-configuration.
	SVN 799	PWT: The floating point exception for inactive cells should be skipped in the perched water table package.
	SVN 806	CHD: For zero-thickess cells at a subdomain boundary a constant head cell could be activated incorrectly resulting in an error message "no-flow cells cannot be converted to constant head cells".

Version	**3.6**	**iMODFLOW_V3_6_METASWAP_SVN1196_X64R.exe** (for 64-systems) **iMODFLOW_V3_6_X32R.exe** (for 32-bit systems)
Date	30-05-2017	
Based On	3.5	

Starting from iMOD 3.6 we summarize all new, changed, extended and fixed functionalities on the iMOD-website: http://oss.deltares.nl/web/imod/release-notes. Per release these release notes are also distributed per email to all iMOD-community members,

A About SIMGRO and MetaSWAP

A.1 What are the models intended for?

Most regional model codes cover only part of the processes within a region. For coming to grips with many issues of integrated water management it is necessary to have a model that covers the whole (regional) system, including plant-atmosphere interactions, soil water, groundwater and surface water. SIMGRO (a dated acronym of SIMulation of GROundwater) was developed for that purpose.

The name 'SIMGRO' was formerly used for referring to an integrated model code, including submodels for the compartments and processes as shown in Fig. 1. Now it is used in the meaning of a modelling framework. This framework has been connected to a number of 'inhouse' components, but also has possibilities for coupling to other codes. The in-house components are:

1 an SVAT-model that is commonly referred to as 'MetaSWAP', covering the plantatmosphere interactions and soil water;
2 a simplified surface water metamodel;
3 a drainage package, for simulating groundwater drainage with fast feedback from surface water.

The current possibilities for coupling to other codes are:

1 MODFLOW for groundwater (operational);
2 SOBEK-CF for surface water (under development).

The used schematisation assumes that the separate SVAT columns only interact with each other via their connections to groundwater and surface water.

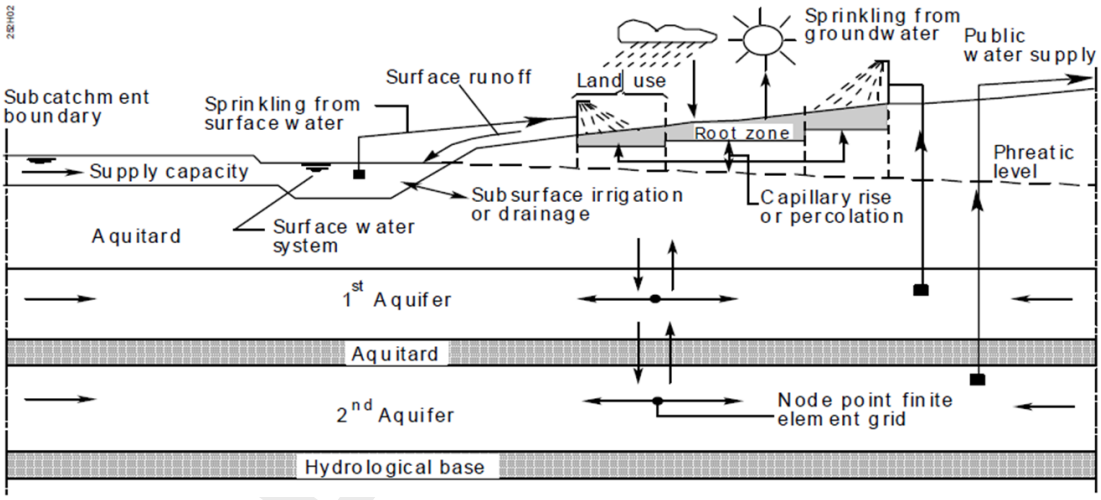


Figure A.1: *Overview of the processes modelled in SIMGRO. MetaSWAP (Van Walsum and Groenendijk, 2008) is used for the SVAT (**S**oil **V**egetation **A**tmosphere **T**ransfer) processes that are modelled within vertical columns. These column models are integrated with the groundwater model (MODFLOW) and a surface water model; for the latter there are several options, including a simplified metamodel that can be linked to form a basin network.*

A.1.1 What is the scope of the model application?

The SIMGRO framework is intended for regions with an undulating topography and unconsolidated sediments in the (shallow) subsoil. Both shallow and deep groundwater levels can be modelled by MetaSWAP. This model is based on a simplification of 'straight Richards', meaning that no special processes like hysteresis, preferential flow and bypass flow are modelled. Snow is not modelled, and neither the influence of frost on the soil water conductivity. A perched watertable can be present in the SVAT column model, but interflow is not modelled. There are plans for including the mentioned special processes in MetaSWAP Inundation water can be modelled as belonging to both groundwater and surface water at the same time. Processes that are typical for steep slopes are not included.

The code contains several parameterized water management schemes, including irrigation and water level management.

A.1.2 What are the used spatial and temporal scales of the model?

The spatial scale of the model is typically for a unit of 1 km^2 and less. Model applications have involved up to 500 000 units (National Hydrological Instrument for the Netherlands). A prototype model of a small basin involved 800 000 cells of 5x5 m. A resolution finer than 5x5 m is considered to be beyond the scope of the model, because then the one-dimensional flow schematization is not adequate.

It is possible to couple several MetaSWAP columns to a single ground water cell. In that way it is possible to use 'tiles' for representing fractions of the soil surface, for e.g. the vegetated part and the built-up part.

The model uses two nested time scales:

1 a fast cycle for the plant-atmosphere interactions and the interactions with surface water;
2 a slow cycle for the unsaturated zone and the coupling to the groundwater model.

Typically an interval of 1 hour is used for the fast processes, and 0.5 or 1 day for the slow processes. The time step of the slow processes and that of the groundwater model should be equal.

A.1.3 What are the necessary input data?

The required input data are described in Alterra Report 913.2. The main categories of input data are:

1 temporal scales;
2 schematization and coupling to other models;
3 soil elevation and soil physical data;
4 land and water use parameters, including irrigation demand;
5 irrigation water supply capacities;
6 meteorological data, including the option of grid files;
7 output control parameters

A.1.4 What output data can the model produce

The main categories of output data are:

1 databases involving up to 129 (optional) data simulated items per SVAT unit, at the time scale the groundwater model, and also for user-defined output periods;
2 files in csv-format that are accessible while the model is still running ('monitoring files')

A.1.5 How does the model communicate with the user, in what language?

The model is run from a DOS-prompt and communicates with the user via files, both binary and ASCII ones. By using the facility of the *csv*-files, the user can follow a simulation as it progresses.

A.1.6 On what platform does the model operate?

The model available for Windows platforms. The used language (Intel Fortran) makes it possible to migrate to a Linux platform, however, this is not supported by Alterra. The model has been coded with 'dynamic' memory allocation, meaning that the amount of used RAM memory is exactly tuned to the needs of the application. Large models will require a 64-bit environment due to RAM memory requirements.

A.1.7 What does the model cost?

The model is freely available, including the source code.

A.1.8 How are the model and its documentation made available?

The model and documentation is available at the SIMGRO ftp-site `ftp://ftp.wur.nl/simgro/` and contains the following folders:

- **doc:** documentation;
- **unsa**: available soil physical databases and automated procedure;
- **tests:** test sets used for the Status A certification;
- **exe:** executable of MainSIMGRO.

The **doc** folder contains (among others) the following documents:

- SIMGRO Release notes
- SIMGRO Theory and model implementation
- SIMGRO User's guide
- SIMGRO Input and output reference manual

A.1.9 Who are the contact persons?

More information about the model can be obtained from:

- Paul van Walsum, paul.vanwalsum@wur.nl.
- Ab Veldhuizen, ab.veldhuizen@wur.nl.

Deltares

PO Box 85467
3508 AL Utrecht
Princetonlaan 6-8
3584 CB Utrecht
The Netherlands

+31 (0)88 335 81 00
imod.support@deltares.nl
www.deltares.nl