TELEMAC
MODELLING SYSTEM

2D HYDRODYNAMICS

TELEMAC-2D SOFTWARE

VERSION 6.1

VALIDATION DOCUMENT

DECEMBER 2011

This manual has been updated for version 6.1 by SOGREAH

EVOLUTIONS OF THE DOCUMENT

DATE	AUTHOR	EVOLUTION
10/2010	SOGREAH	General update for release 6.0
12/2011	SOGREAH	Update for release 6.1
05/2012	Riadh Ata (LNHE)	Update for release 6.1

OUTLINE

INTRODUCTION .... J

CONVENTIONS USED IN THIS MANUAL .... K

ABSTRACT .... K

PREFACE 12

1. INTRODUCTION 14
1.1. MODEL OVERVIEW 14
1.1.1. PURPOSE 14
1.1.2. PRE- AND POST-PROCESSING AND OTHER SOFTWARE FEATURES 14
1.1.3. VERSION INFORMATION 15
1.2. VALIDATION PRIORITIES AND APPROACHES 15
1.3. RELATED DOCUMENTS 16

2. MODEL VALIDITY 16
2.1. PHYSICAL SYSTEM 16
2.2. MODEL FUNCTIONALITY 17
2.2.1. APPLICATIONS 17
2.2.2. PROCESSES 19
2.3. CONCEPTUAL MODEL 22
2.3.1. ASSUMPTIONS AND APPROXIMATIONS 22
2.3.2. CLAIMS AND SUBSTANTIATIONS 24
2.4. ALGORITHMIC IMPLEMENTATION 25
2.4.1. ASSUMPTIONS AND APPROXIMATIONS 25
2.4.2. CLAIMS AND SUBSTANTIATIONS 26
2.5. SOFTWARE IMPLEMENTATION 29
2.5.1. IMPLEMENTATION TECHNIQUES 29
2.5.2. CLAIMS AND SUBSTANTIATIONS 30

3. VALIDATION STUDIES 32
3.1. GAUSSIAN WATER SURFACE CENTRED IN A SQUARE DOMAIN - SOLID BOUNDARIES 32
3.1.1. PURPOSE 32
3.1.2. LINKED CLAIMS 32
3.1.3. APPROACH 32
3.1.4. RESULTS 33
3.1.5. CONCLUSIONS 33

3.1.6. STEERING FILE *34*
3.1.7. FIGURES *36*
3.2. GAUSSIAN WATER SURFACE CENTRED IN A SQUARE DOMAIN - OPEN BOUNDARIES **41**
3.2.1. PURPOSE *41*
3.2.2. LINKED CLAIMS *41*
3.2.3. APPROACH *41*
3.2.4. RESULTS *42*
3.2.5. CONCLUSIONS *42*
3.2.6. STEERING FILE *43*
3.2.7. FIGURES *45*
3.3. BUMP - SUBCRITICAL CONDITION **49**
3.3.1. PURPOSE *49*
3.3.2. LINKED CLAIMS *49*
3.3.3. APPROACH *49*
3.3.4. RESULTS *50*
3.3.5. CONCLUSIONS *51*
3.3.6. STEERING FILE *51*
3.3.7. FIGURES *52*
3.4. BUMP - TRANSCRITICAL CONDITION **56**
3.4.1. PURPOSE *56*
3.4.2. LINKED CLAIMS *56*
3.4.3. APPROACH *56*
3.4.4. RESULTS *58*
3.4.5. CONCLUSIONS *58*
3.4.6. STEERING FILE *58*
3.4.7. FIGURES *60*
3.5. BUMP - TRANSCRITICAL CONDITION WITH HYDRAULIC JUMP **63**
3.5.1. PURPOSE *63*
3.5.2. LINKED CLAIMS *63*
3.5.3. APPROACH *63*
3.5.4. RESULTS *64*
3.5.5. CONCLUSIONS *65*
3.5.6. STEERING FILE *65*
3.5.7. FIGURES *67*
3.6. UNCOVERING OF A BEACH **70**
3.6.1. PURPOSE *70*
3.6.2. LINKED CLAIMS *70*
3.6.3. APPROACH *70*
3.6.4. RESULTS *71*
3.6.5. CONCLUSIONS *71*
3.6.6. STEERING FILE *71*
3.6.7. FIGURES *73*
3.7. PROPAGATION OF A SURFACE WAVE **76**

3.7.1. PURPOSE 76
3.7.2. LINKED CLAIMS....76
3.7.3. APPROACH....76
3.7.4. RESULTS....77
3.7.5. CONCLUSIONS....78
3.7.6. STEERING FILE 78
3.7.7. FIGURES 80
3.8. FLOW AROUND BRIDGE PIERS....82
3.8.1. PURPOSE 82
3.8.2. LINKED CLAIMS....82
3.8.3. APPROACH....82
3.8.4. RESULTS....83
3.8.5. CONCLUSIONS....83
3.8.6. STEERING FILE 83
3.8.7. FIGURES....85
3.9. SILLS IN A CHANNEL TREATED AS SINGULARITIES 87
3.9.1. PURPOSE 87
3.9.2. LINKED CLAIMS....87
3.9.3. APPROACH....87
3.9.4. RESULTS....88
3.9.5. CONCLUSIONS....89
3.9.6. STEERING FILE 89
3.9.7. FILE OF DATA CONCERNING THE SILLS....91
3.9.8. FIGURES....92
3.10. CULVERTS 95
3.10.1. PURPOSE 95
3.10.2. LINKED CLAIMS....95
3.10.3. APPROACH....95
3.10.4. RESULTS....97
3.10.5. CONCLUSIONS....97
3.10.6. STEERING FILE 97
3.10.7. FILE OF DATA CONCERNING THE CULVERT 98
3.10.8. FIGURES....99
3.11. THEORETICAL DAM BREAK WAVE - INITIALLY DRY BED 102
3.11.1. PURPOSE 102
3.11.2. LINKED CLAIMS....102
3.11.3. APPROACH....102
3.11.4. RESULTS....103
3.11.5. CONCLUSIONS....105
3.11.6. STEERING FILE 105
3.11.7. FIGURES....107
3.12. THEORETICAL DAM BREAK WAVE - INITIALLY WET BED 111
3.12.1. PURPOSE 111

3.12.2. LINKED CLAIMS 111
3.12.3. APPROACH 111
3.12.4. RESULTS 112
3.12.5. CONCLUSIONS 113
3.12.6. STEERING FILE 113
3.12.7. FIGURES 115
3.13. WIND SET-UP 118
3.13.1. PURPOSE 118
3.13.2. LINKED CLAIMS 118
3.13.3. APPROACH 118
3.13.4. RESULTS 120
3.13.5. CONCLUSIONS 120
3.13.6. STEERING FILE 120
3.13.7. FIGURES 122
3.14. RIVER CONFLUENCE 123
3.14.1. PURPOSE 123
3.14.2. LINKED CLAIMS 123
3.14.3. APPROACH 123
3.14.4. RESULTS 124
3.14.5. CONCLUSIONS 125
3.14.6. STEERING FILE 125
3.14.7. FIGURES 127
3.15. CAVITY 130
3.15.1. PURPOSE 130
3.15.2. LINKED CLAIMS 130
3.15.3. APPROACH 130
3.15.4. RESULTS 131
3.15.5. CONCLUSIONS 132
3.15.6. STEERING FILE 132
3.15.7. FIGURES 134
3.16. BREAKWATER 138
3.16.1. PURPOSE 138
3.16.2. LINKED CLAIMS 138
3.16.3. APPROACH 138
3.16.4. RESULTS 139
3.16.5. CONCLUSIONS 139
3.16.6. STEERING FILE 140
3.16.7. FIGURES 142
3.17. MALPASSET DAM BREAK WAVE 144
3.17.1. PURPOSE 144
3.17.2. LINKED CLAIMS 144
3.17.3. APPROACH 144
3.17.4. RESULTS 145

3.17.5. CONCLUSIONS 146
3.17.6. STEERING FILE 146
3.17.7. FIGURES 148
3.18. RIVER CULM 151
3.18.1. PURPOSE 151
3.18.2. LINKED CLAIMS 151
3.18.3. APPROACH 152
3.18.4. RESULTS 153
3.18.5. CONCLUSIONS 153
3.18.6. STEERING FILE 153
3.18.7. FIGURES 155
3.19. MERSEY ESTUARY 157
3.19.1. PURPOSE 157
3.19.2. LINKED CLAIMS 157
3.19.3. APPROACH 158
3.19.4. RESULTS 159
3.19.5. CONCLUSIONS 159
3.19.6. STEERING FILE 159
3.19.7. FIGURES 161
3.20. WESTERN EUROPEAN CONTINENTAL SHELF 164
3.20.1. PURPOSE 164
3.20.2. LINKED CLAIMS 164
3.20.3. APPROACH 165
3.20.4. RESULTS 166
3.20.5. CONCLUSIONS 166
3.20.6. STEERING FILE 166
3.20.7. FIGURES 169
3.21. POINT DISCHARGE WITHOUT DIFFUSION 171
3.21.1. PURPOSE 171
3.21.2. LINKED CLAIMS 171
3.21.3. APPROACH 171
3.21.4. RESULTS 172
3.21.5. CONCLUSIONS 172
3.21.6. STEERING FILE 173
3.21.7. FIGURES 175
3.22. POINT DISCHARGE WITH DIFFUSION 178
3.22.1. PURPOSE 178
3.22.2. LINKED CLAIMS 178
3.22.3. APPROACH 178
3.22.4. RESULTS 180
3.22.5. CONCLUSIONS 180
3.22.6. STEERING FILE 180
3.22.7. FIGURES 182

LIST OF FIGURES

FIGURE 3.1.1 : MESH AND INITIAL STATE. 36
FIGURE 3.1.2 : EVOLUTION OF THE SHAPE OF THE FREE SURFACE FROM 0.0 SEC. TO 1.2 SEC. 37
FIGURE 3.1.3 : EVOLUTION OF THE SHAPE OF THE FREE SURFACE FROM 1.8 SEC. TO 3.0 SEC. 38
FIGURE 3.1.4 : EVOLUTION OF THE WATER DEPTH FROM 0.0 SEC. TO 4.0 SEC. 39
FIGURE 3.2.1 : MESH AND INITIAL STATE. 45
FIGURE 3.2.2 : EVOLUTION OF THE SHAPE OF THE FREE SURFACE FROM 0.0 SEC. TO 1.2 SEC. 46
FIGURE 3.2.3 : EVOLUTION OF THE SHAPE OF THE FREE SURFACE FROM 1.8 SEC. TO 3.0 SEC. 47
FIGURE 3.2.4 : EVOLUTION OF THE WATER DEPTH FROM 0.0 SEC. TO 4.0 SEC 48
FIGURE 3.3.1 : MESH AND TOPOGRAPHY. 53
FIGURE 3.3.2 : VELOCITY FIELD AND COMPARISON BETWEEN ANALYTICAL SOLUTION AND TELEMAC-2D SOLUTION FOR THE FREE SURFACE ELEVATION AND THE FROUDE NUMBER 54
FIGURE 3.4.1 : MESH AND TOPOGRAPHY. 60
FIGURE 3.4.2 : VELOCITY FIELD AND COMPARISON BETWEEN ANALYTICAL SOLUTION AND TELEMAC-2D SOLUTION FOR THE FREE SURFACE ELEVATION AND THE FROUDE NUMBER 61
FIGURE 3.5.1 : MESH AND TOPOGRAPHY. 67
FIGURE 3.5.2 : VELOCITY FIELD AND TELEMAC-2D SOLUTION FOR THE FREE SURFACE ELEVATION AND THE FROUDE NUMBER. 68
FIGURE 3.6.1 : MESH AND TOPOGRAPHY. 73
FIGURE 3.6.2 : EVOLUTION OF THE FREE SURFACE ELEVATION IN TIME. 74
FIGURE 3.7.1 : MESH AND INITIAL STATE. 80
FIGURE 3.7.2 : COMPARISON BETWEEN ANALYTICAL SOLUTION AND TELEMAC-2D SOLUTION FOR THE FREE SURFACE ELEVATION. 81
FIGURE 3.8.1 : MESH AND TOPOGRAPHY. 85
FIGURE 3.8.2 : VELOCITY FIELD AT TIME 240 SEC. AND EVOLUTION OF THE VELOCITY BEHIND ONE PIER 86
FIGURE 3.10.1 : MESH FOR THE PROBLEM OF SILLS IN A CHANNEL. 92
FIGURE 3.10.2 : EVOLUTION OF VELOCITY FIELD IN TIME. 93
FIGURE 3.10.3 : EVOLUTION OF THE FREE SURFACE ELEVATION IN TIME. 94
FIGURE 3.11.1 : MESH AND INITIAL STATE. 99
FIGURE 3.11.2 : EVOLUTION OF THE TRACER CONCENTRATION IN THE SOURCE TANK, AND EVOLUTION OF THE FREE SURFACE ELEVATION IN TIME IN BOTH TANKS. 100
FIGURE 3.11.3 : EVOLUTION OF VELOCITY FIELD IN TIME IN BOTH TANKS 101
FIGURE 3.12.1 : MESH AND INITIAL STATE. 107
FIGURE 3.12.2 : EVOLUTION OF THE WATER DEPTH FROM 0.1 SEC. TO 1.0 SEC. 108
FIGURE 3.12.3 : EVOLUTION OF THE WATER DEPTH PROFILE FROM 0.9 SEC. TO 1.5 SEC. 109
FIGURE 3.13.1 : MESH AND INITIAL STATE. 115
FIGURE 3.13.2 : EVOLUTION OF THE WATER DEPTH FROM 0 SEC. TO 1500 SEC. AND COMPARISON BETWEEN ANALYTICAL SOLUTION AND TELEMAC-2D SOLUTION. 116
FIGURE 3.14.1 : MESH AND FREE SURFACE ELEVATION. 122
FIGURE 3.15.1: RIVER CONFLUENCE, GEOMETRY AND USED MESH. 127
FIGURE 3.15.2 : TOPOGRAPHY, FREE SURFACE ELEVATION AND VELOCITY FIELD. 128
FIGURE 3.16.1 : MESH AND TOPOGRAPHY. 134
FIGURE 3.16.2 : VELOCITY FIELD AND FREE SURFACE IN THE CAVITY AT TIME 15225 SEC. 135
FIGURE 3.16.3 : EVOLUTION OF THE VELOCITY FIELD IN TIME – SMALL EDDY PATTERN. 136
FIGURE 3.16.4 : EVOLUTION OF THE VELOCITY FIELD IN TIME – LARGE EDDY PATTERN. 137
FIGURE 3.17.1 : MESH AND TOPOGRAPHY. 142
FIGURE 3.17.2 : EVOLUTION OF THE FREE SURFACE ELEVATION IN TIME. 143
FIGURE 3.18.1 : MESH AND TOPOGRAPHY. 148
FIGURE 3.18.2 : EVOLUTION OF THE WATER DEPTH IN TIME. 149
FIGURE 3.18.3 : EVOLUTION OF THE VELOCITY FIELD IN TIME. 150
FIGURE 3.19.1 : MESH AND TOPOGRAPHY. 155
FIGURE 3.19.2 : EVOLUTION OF THE FREE SURFACE ELEVATION IN TIME AT THE DOWNSTREAM POINT AND EVOLUTION OF THE WATER DEPTH IN TIME. 156

FIGURE 3.20.1 : MESH AND TOPOGRAPHY........ 161
FIGURE 3.20.2 : WATER DEPTH AT HIGH TIDE AND LOW TIDE AND SURFACE ELEVATION AT DIFFERENT LOCATIONS. 162
FIGURE 3.20.3 : VELOCITY FIELDS AT TWO TIMES OF THE TIDE DURING THE FLOW AND THE EBB........ 163
FIGURE 3.21.1 : MESH AND TOPOGRAPHY........ 169
FIGURE 3.21.2 : AMPLITUDE AND PHASE OF THE M2 CONSTITUENT. 170
FIGURE 3.22.1 : MODEL MESH AND HYDRODYNAMIC FIELD........ 175
FIGURE 3.22.2 : CONCENTRATIONS. 176
FIGURE 3.22.3 : ALONG AND CROSS FLOW CONCENTRATION PROFILES........ 177
FIGURE 3.23.1 : MODEL MESH AND HYDRODYNAMIC FIELD........ 182
FIGURE 3.23.2 : CONCENTRATIONS. 183
FIGURE 3.23.3 : ALONG AND CROSS FLOW CONCENTRATION PROFILES........ 184

oOo

INTRODUCTION

CONVENTIONS USED IN THIS MANUAL

All the figures of computational results included in this manual were produced with FUDAA-PREPRO v1.1RC29 post-processor, except the 3D views on Figure 3.1.2, Figure 3.1.3, Figure 3.2.2 and Figure 3.2.3 which were produced with Blue Kenue.

BLUEKENUE is the property of the Canadian Hydraulics Centre, Ottawa, Ontario, Canada

http://www.nrc-cnrc.gc.ca/eng/ibp/chc/software/kenue/blue-kenue.html

ABSTRACT

TELEMAC-2D solves the two-dimensional shallow water flow equations using Finite Element Method or Finite Volume Method operating on non structured grids of triangular elements. The software is imbedded in an integrated and user-friendly software environment, the TELEMAC system.

This report is part of the user documentation of TELEMAC-2D. It substantiates a number of claims that validate version 6.1 of TELEMAC-2D with different respects.

22 validation test cases and sub-cases are presented.

The document conforms to a standard system supported by the International Association for Hydraulic Research (IAHR).

oOo

PREFACE

The subject of this document is the validation of a computational model. The term *computational model* refers to software whose primary function is to model a certain class of physical systems, and may include pre- and post-processing components and other necessary ancillary programmes. *Validation* applies primarily to the theoretical foundation and to the computational techniques that form the basis for the numerical and graphical results produced by the software. In the context of this document, validation of the model is viewed as the formulation and substantiation of explicit claims about applicability and accuracy of the computational results.

This preface explains the approach that has been adopted in organising and presenting the information contained in this document.

Standard Validation Documents

This document conforms to a standard system for validation documentation (see (1) in Appendix B). This system, the Standard Validation Document, has been developed by the hydraulic research industry in order to address the need for useful and explicit information about the validity of computational models. Such information is summarised in a validation document, which accompanies the technical reference documentation associated with a computational model.

In conforming to the Standard, this validation document meets the following requirements:

1) It has a prescribed table of contents, based on a framework that allows separate quality issues to be clearly distinguished and described.
2) It includes a comprehensive list of the assumptions and approximations that were made during the design and implementation of the model.
3) It contains claims about the performance of the model, together with statements that point to the available substantiating evidence for these claims.
4) Claims about the model made in this document are substantiated and bounded: they can be tested, justified, or supported by means of physical or computational experiments, theoretical analysis, or case studies.
5) Claims are substantiated by evidence contained within this document, or by specific reference to accessible publications.
6) Results of validation studies included or referred to in this document are reproducible. Consequently the contents of this document are consistent with the current version of the software.
7) This document will be updated as the process of validating the model progresses.

Organisation of this document

Chapter 1 contains a short overview of the computational model and introduces the main issues to be addressed by the validation process. The model overview includes information about the purpose of the model, about pre- and post-processing options and other software features, and about reference versions of the software. Validation priorities and approaches are briefly described, and a list of related documents is included.

Chapter 2 summarises the available information about the validity of the computational core of the model. In this chapter, claims are made about the range of applicability of the model and about the accuracy of computational results. Each claim is followed by a brief statement regarding its substantiation. This statement indicates the extent to which the claim has in fact been substantiated and points to the available evidence.

Chapter 3 contains such evidence, in the form of brief descriptions of relevant validation studies. Each description includes information about the purpose and approach of the study, and a summary of main results and implications.

Validation studies are numbered 1 to 22. They are presented respectively in sections 3.1 to 3.22 of this document.

A glossary and complete list of references are contained in Appendix A and Appendix B respectively.

Caution

This document contains information about the quality of a complex modelling tool. Its purpose is to assist the user in assessing the reliability and accuracy of computational results, and to provide guidelines with respect to the applicability and judicious employment of this tool. This document does not, however, provide mathematical proof of the correctness of results for a specific application. The reader is referred to the License Agreement for pertinent legal terms and conditions associated with the use of the software.

The contents of this validation document attest to the fact that computational modelling of complex physical systems requires great care and inherently involves a number of uncertain factors. In order to obtain useful and accurate results for a particular application, the use of high-quality modelling tools is necessary but not sufficient. Ultimately, the quality of the computational results that can be achieved will depend upon the adequacy of available data as well as a suitable choice of model and modelling parameters.

1. INTRODUCTION

TELEMAC-2D is a computational software that calculates free surface flows in natural water bodies. It solves the two-dimensional shallow water flow equations thanks to finite element or finite volume techniques operating on non structured grids of triangular elements.

The software is imbedded in an integrated and user-friendly software environment, the TELEMAC system. TELEMAC is developed by the Laboratoire National d'Hydraulique et d'Environnement, a department of the Division for Research and Development of the French Electricity Board (EDF-R&D). The software system is owned by EDF-R&D.

TELEMAC-2D, as all software of the TELEMAC system, conforms to the EDF-R&D Quality Assurance Plan for Scientific and Technical Software. As provided for in this plan, the present document validates the TELEMAC-2D conceptual model, algorithmic and software implementations through a number of claims. These are substantiated through reference to 22 test cases presented in this document and to the appropriate bibliography.

Other computational models and technical software of the TELEMAC system are not validated in the present document.

1.1. MODEL OVERVIEW

1.1.1. PURPOSE

TELEMAC is a software system designed to study environmental processes in free surface transient flows. It is therefore applicable to seas and coastal domains, estuaries, rivers and lakes. Its main fields of application are in hydrodynamics, water quality, sedimentology and water waves.

All computational models of the TELEMAC system are based on a common library of finite element numerical solvers. The spatial discretization of two-dimensional domains is based on grids composed of triangular elements of various size and shape. Three-dimensional domains are discretized by grids of prismatic elements. The uniform finite element approach common to all TELEMAC scientific software facilitates multidisciplinary studies. For instance it is easy to shift from a 2D to a 3D computation, the 3D grid having the same structure as the 2D one in the horizontal plane. The effort required from a TELEMAC's user for learning the manipulation of a new simulation module is also reduced.

Inside the TELEMAC processing line, the primary purpose of TELEMAC-2D is to calculate the dynamics of flows taking place in a river, a lake, an estuary or a coastal area. The distribution of currents and the water surface elevation are computed at the different times of a simulated hydro-climatic event.

1.1.2. PRE- AND POST-PROCESSING AND OTHER SOFTWARE FEATURES

Several pre- and post-processing modules are supplied as part of the TELEMAC package in addition to the simulation modules. These pre- and post-processors allow models to be set up quickly and easily and the model results to be displayed and analysed.

The program SINUSX (see reference (1)) receives input of the coastlines and bathymetry from a digitizing tablet or from files, allows these to be manipulated and outputs them in a form suitable to the mesh generator MATISSE (see ref. (2)) or to the interface module FUDAA-PREPRO. The integrated grid generator MATISSE is then used for space discretisation and boundary conditions specification, generating the geometry file (including bathymetry) and the boundary conditions file necessary for the simulation modules.

In addition, FUDAA-PREPRO accepts the file formats from the other following grid generators:

- SIMAIL, produced by SIMULOG,
- IDEAS-MS 2 (SUPERTAB), produced by SDRC,
- TRIGRID, produced by Ocean Fisheries of Canada,
- FASTTABS, produced by the Brigham Young University, U.S.A.

If necessary, the interface module FUDAA-PREPRO combines the data concerning the geometry of the region to be modelled with the mesh of triangles generated to build up a file which is input to the main TELEMAC-2D program. FUDAA-PREPRO interpolates at grid nodes the bed elevations provided by SINUSX or external files at arbitrary locations.

In the case of the use of MATISSE, the generated files are directly used as input to the TELEMAC-2D program.

The interface module FUDAA-PREPRO also enables the interactive creation of the steering file required for running a simulation.

Post-processing of TELEMAC-2D results is performed with FUDAA-PREPRO, a graphics and data analysis package incorporating a point-and-click user interface.

Simulation modules of TELEMAC including TELEMAC-2D are developed in FORTRAN 90. SINUSX, and MATISSE are developed in C++ language; they are based on the use of X-Window, OSF/MOTIF and ILOGViews. The TELEMAC system is therefore portable on any Unix environment and on Windows NT 4 or upper.

TELEMAC-2D as well as other modules of the TELEMAC system have been developed in agreement with Quality Assurance procedures of the Scientific and Technical software of EDF-R&D. The present document only deals with the validation of the version 6.1 of TELEMAC-2D, the validation of other simulation modules being the subject of separate validation documents.

All modules of the TELEMAC processing line, including TELEMAC-2D, are available in French and English versions.

1.1.3. VERSION INFORMATION

The information in this document refers to Version 6.1 of TELEMAC-2D, released in August 2011.

1.2. VALIDATION PRIORITIES AND APPROACHES

This document addresses validation issues in three steps: *concepts*, *algorithmic* implementation and *software* aspects. Questions raised by the various stages of the modelling process can be affected to one of these three steps.

The *conceptual issue* addresses the question of whether the Shallow Water Equations (SWE) which are solved, provide an accurate description of the physical processes for the particular situation of interest and whether the solution of these equations provides the information required. If the SWE is not appropriate for that situation then it is not even necessary to consider algorithmic and software issues since TELEMAC-2D is inapplicable and cannot be expected to supply appropriate results. In particular, basic hypothesis necessary to the application of this equation should be satisfied for the use of TELEMAC-2D in any case.

This document explains which kinds of problems can be tackled with TELEMAC-2D and gives examples which illustrate a range of possible applications.

Any computer program based on a mechanistic modelling of physical processes solves differential equations by discretizing the equations in space and time. The values of the dependent variables (the water depth h and the two velocity components u and v along x and y respectively) are calculated only at certain locations and certain times. Space discretization transforms the original differential equation into a set of algebraic equations which are solved in the program to give an approximate solution to the original differential equations. The way in which this discretization is carried out is very important in determining the ability of a program to model physical reality. These are the main *algorithmic issues***.**

The *software issues* are concerned with the way in which the solution algorithm is implemented and ways in which the input and output of information can be effected.

1.3. RELATED DOCUMENTS

A complete list of references, including those mentioned in this section, is given in Appendix B. The main TELEMAC-2D documents to which the reader will refer for complements are:

- TELEMAC-2D • Version 6.0 - User Manual.

  This manual describes all the necessary informations needed to use TELEMAC-2D, whose validation is the subject of the present report.

- “Hydrodynamics of Free Surface Flows modelling with the finite element method” by Jean-Michel Hervouet (Wiley, 2007)

  This book describes in detail the set of equations solved by TELEMAC-2D and the numerical method used.

- TELEMAC system – Guide for Programming in the TELEMAC system version 6.0 Fortran 90. This report describes the structure of the BIEF library and its subroutines. It provides recommendations to the programmer of TELEMAC scientific modules. The BIEF library contains all the functions concerning finite elements operations.

2. MODEL VALIDITY

2.1. PHYSICAL SYSTEM

TELEMAC-2D is designed to simulate transient free surface flow of water or any Newtonian fluid. It is particularly adapted to study water movements in shallow aquatic domains: coastal areas, estuaries, rivers and lakes. It can also represent float tracking and the movement of a dissolved substance possibly having and action on the vertically averaged fluid density.

In order to set the model up for hydrodynamic simulations of a particular physical system, the following data must be collected, analysed and digitized:

- System layout and geometry (coastline, topography, bathymetry, geometrical characteristics of hydraulic structures);
- Information on roughness conditions: nature of the bottom in coastal domains, river beds or lakes, vegetation cover or land use in river valleys;

- Hydrodynamic or hydraulic conditions on the open boundaries of the study domain: discharge, velocity, surface elevation or two of these (prescribed velocity with approximate information on the velocity for instance);
- Characteristics of any additional forcing (meteorological forcing, outfalls, etc.)

For calibration and validation of the model, one or more sets of simultaneous observed time series at internal locations are required (water levels, velocities), corresponding to hydro-climatological events with known effects on the open boundaries of the computational domain. Roughness and viscosity coefficients can be defined node by node or globally over an area or the whole computational domain.

TELEMAC-2D solves, through a Finite Element Method (FEM) or Finite Volume Method (FVM) over non-structured grids consisting of triangles, the shallow water flow equations (SWE) in two horizontal space dimensions. These equations express the conservation of water mass (continuity equation) and the conservation of momentum in both horizontal space directions (dynamic equations) in every node of the computational domain.

Furthermore, Telemac can compute simultaneously, and in a coupled way, the dispersion of any dissolved substance (tracer conservation equation). The average vertical density of water can depend upon the local value of that tracer, as it happens, for instance, with salinity in the case of a study in an estuarial environment.

2.2. MODEL FUNCTIONALITY

2.2.1. APPLICATIONS

TELEMAC-2D is applicable as a principal tool for hydrodynamic analysis to river and coastal engineering studies. As any pluri-disciplinary study of environmental water bodies must be funded on a good knowledge of water movements, TELEMAC-2D is the more commonly used of the different TELEMAC computational programmes. It can be used in a wide range of applications, a number of which are listed below along with references to appropriate case studies and reference to previous studies.

In the **maritime sphere**, particular mention may be made of the following studies:

- sizing of port and coastal structures;
- impact of waste discharged from coastal outfalls;
- dispersion of thermal plumes, recirculation of cooling water of a thermal power plant;
- navigational and environmental impacts of dredgings.

In the study of **continental waters**, particular mention may be made of:

- detailed analysis of flow conditions in channels and river beds, in the bends of meandering rivers, close to locks: such analysis can be requested in the frame of navigation or dredging studies;
- studies relating to the impact of construction works: road and railway embankments, groynes, bridges, weirs, culverts, locks, barrages, etc.;
- in association with morphological and sedimentological expertise, assessment of morphological evolution of a river bed;
- flooding of river valleys;
- dam break waves;
- transport of decaying or non-decaying substances dissolved in the water.

TELEMAC-2D has also been used for a number of **special hydrodynamic applications**: avalanches of rocks or snow falling into reservoirs, bursting of industrial tanks.

Claim 2.2.1.1 TELEMAC-2D can be used to set up a database of tidal constituents on a continental shelf.

Substantiation: See validation study 3.20 (Western European Continental shelf).

Claim 2.2.1.2 TELEMAC-2D can be used for an accurate prediction and for the analysis of tidal currents and water surface elevations in coastal or estuarial zones due to long period tidal wave forcing, with covering/uncovering flats.

Substantiation: See validation studies 3.6 (Uncovering of a beach), 3.7 (Propagation of a surface wave) and 3.19 (Mersey Estuary).

Claim 2.2.1.3 TELEMAC-2D can be used to investigate the hydrodynamic impact of engineering works in coastal or estuarine environment such as breakwaters, piers, navigation channels, submersible dikes, etc.

Substantiation: Validation study 3.16 (Breakwater) shows the-flow over a breakwater. Studies 3.9 (Sills in a channel treated as singularities) and 3.10 (Culverts) are relative to other types of engineering works in coastal environment.

Claim 2.2.1.4 TELEMAC-2D can be used to study the impact of a storm surge on depth mean currents and coastal sea level variations.

Substantiation: See study 3.6 (Uncovering of a beach).

Claim 2.2.1.5 TELEMAC-2D can be used to study the long term environmental impact of a release of contaminant in a maritime domain.

Substantiation: See study 3.7 (Propagation of a surface wave).

Claim 2.2.1.6 TELEMAC-2D can accurately represent currents and water surface elevations in canals and in river beds.

Claim 2.2.1.7 TELEMAC-2D can be used to investigate the hydrodynamic impact of engineering works in rivers.

Substantiation: Validation study 3.3 (Bump - Subcritical condition), 3.4 (Bump - Transcritical condition) and 3.5 (Bump - transcritical condition with hydraulic jump) shows the flow over a bump

Validation study 3.8 (Flow around bridge piers) shows Karman vortices generated behind two cross-flow cylindrical piers

Validation studies 3.9 (Sills in a channel treated as singularities) and 3.10 (Culverts) illustrate the treatment of sills and culverts as singularities

Studies 3.11 (Theoretical dam break wave - Initially dry bed), 3.12 (Theoretical dam break wave - Initially wet bed), 3.13 (Wind set-up), 3.14 (River confluence) and 3.15 (Cavity) are relative to other types of engineering works in river environment.

Claim 2.2.1.8 TELEMAC-2D can be used as a tool for assessing the impact of river flooding.

Substantiation: See validation study 3.18 (River Culm).

Claim 2.2.1.9 TELEMAC-2D can appropriately be used to analyse the consequences of the water wave following a dam break. Situation with dry or wet beds in the valley downstream the dam can both be examined.

Substantiation: See validation studies 3.11 (Theoretical dam break wave - Initially dry bed), 3.12 (Theoretical dam break wave - Initially wet bed) and 3.17 (Malpasset dam break wave).

Claim 2.2.1.10 TELEMAC-2D can be used for detailed analysis of flow conditions and head-loss generated at a river confluence.

Substantiation: See validation study 3.14 (River confluence).

Claim 2.2.1.11 TELEMAC-2D can be used to follow the behaviour of tracers (either conservative or decaying).

Substantiation: See study 3.21 (Point discharge without diffusion) and 3.22 (Point discharge with diffusion).

Claim 2.2.1.12 TELEMAC-2D can efficiently be used to analyse rapid flows following total or partial bursting of an industrial reservoir.

Substantiation: See study 3.16 (Breakwater).

2.2.2. PROCESSES

TELEMAC-2D is capable of representing a large number of physical processes. Some of these are presented below along with references to suitable examples.

Claim 2.2.2.1 TELEMAC-2D can consider domains with complex geometries: changing bed slopes, teeth-shaped coastline, islands, flow contractions and expansions, etc.

Substantiation: Many validation studies in the present document illustrate this property, see for instance validation studies 3.3 (Bump - Subcritical condition), 3.4 (Bump - Transcritical condition), 3.5 (Bump - transcritical condition with hydraulic jump), 3.17 (Malpasset dam break wave), 3.18 (River Culm), 3.19 (Mersey Estuary) and 3.20 (Western European Continental shelf).

Claim 2.2.2.2 TELEMAC-2D is capable of modelling flows in a network of open channels and rivers joined together in any topological configuration.

Substantiation: Validation study 3.14 (River confluence) illustrates the flow at a river confluence.

Claim 2.2.2.3 TELEMAC-2D can compute steady and transient flow regimes. Steady flow is achieved by integrating through time until a steady condition is obtained.

Substantiation: Validation studies 3.3 (Bump - Subcritical condition), 3.4 (Bump - Transcritical condition), 3.5 (Bump - transcritical condition with hydraulic jump) and 3.14 (River confluence) illustrate this procedure.

Note that validation study 3.8 (Flow around bridge piers) demonstrates that unsteady shedding vortices may be the flow resulting from imposition of steady boundary conditions.

Claim 2.2.2.4 TELEMAC-2D can consider conditions on open boundaries of a study domain corresponding to a wide range of physical phenomena encountered in the nature: tides (maritime applications), floods, river water inflow, dam failure (river applications).

Substantiation: Validation studies 3.19 (Mersey Estuary) and 3.20 (Western European Continental shelf) illustrate tide forcing;

Validation studies 3.18 (River Culm), and 3.19 (Mersey Estuary) illustrate the inflow of a river;

Validation study 3.11 (Theoretical dam break wave - Initially dry bed) and 3.12 (Theoretical dam break wave - Initially wet bed) illustrates a dam break boundary condition.

Claim 2.2.2.5 TELEMAC-2D can simulate the effect of meteorological conditions such as surface layer motion generated by the wind blowing at the water surface and hydrodynamics induced by variations in atmospheric pressure, provided a depth integration of these processes is adequate.

Substantiation: See validation study 3.13 (Wind set-up).

Claim 2.2.2.6 TELEMAC-2D can simulate the propagation of long waves;

Substantiation: This is treated in validation study 3.7 (Propagation of a surface wave).

Claim 2.2.2.7 TELEMAC-2D is capable of representing the hydrodynamic influence of bed friction as a bottom boundary condition in the vertical averaging process. Roughness options available are quadratic (Chezy, Strickler, Manning), Nikuradse or linear friction laws.

Substantiation: Validation studies 3.14 (River confluence), 3.16 (Breakwater) and 3.18 (River Culm) illustrate this feature on river applications and validation study 3.19 (Mersey Estuary) illustrates it on a maritime application.

Claim 2.2.2.8 TELEMAC can include the impact of the Coriolis force arising from Earth rotation.

Substantiation: See validation study 3.20 (Western European Continental shelf).

Claim 2.2.2.9 TELEMAC-2D is capable to represent areas of injection or extraction of fluid in the flow domain. This feature considers a source/sink of mass and of momentum in option. It is also possible to simulate a source/sink of tracer in the same location.

Substantiation: See validation studies 3.10 (Culverts), 3.21 (Point discharge without diffusion) and 3.22 (Point discharge with diffusion).

Claim 2.2.2.10 Turbulence effect can be modelled through different options of increasing complexity: constant viscosity, Elder model (linear viscosity), k-ε model (two equations) or Smagorinski options are available.

Substantiation: A constant viscosity model is adequate in most case, which explains why the major part or validation studies have been performed with this option. Validations study 3.15 (Cavity) illustrates the properties of the Smagorinski model.

Claim 2.2.2.11 TELEMAC-2D can accurately simulate sub-critical flows when the boundary conditions are time dependent (boundary driven flow).

Substantiation: See validation studies 3.18 (River Culm), 3.19 (Mersey Estuary) and 3.20 (Western European Continental shelf).

Claim 2.2.2.12 TELEMAC-2D can be used to give a good representation of transcritical flows. Regime transitions consisting in critical flow conditions or hydraulic jumps are computed and located appropriately.

Substantiation: See validation study 3.4 (Bump - Transcritical condition) and 3.5 (Bump - transcritical condition with hydraulic jump).

Claim 2.2.2.13 TELEMAC-2D can represent the impact of temperature and salinity horizontal gradients on water density. Horizontal density gradient existing in mixed waters of large estuaries may be of importance in the accurate prediction of water levels in these estuaries.

Substantiation: This feature is not illustrated in the present document. It has been used in some real applications to estuarine domains.

Claim 2.2.2.14 TELEMAC-2D can solve the flow equations written in spherical co-ordinates as needed by the study of large maritime domains, where earth sphericity has to be taken into account. A Cartesian co-ordinate system is the option traditionally employed for smaller domains.

Substantiation: Validation study 3.20 (Western European Continental shelf) makes use of spherical co-ordinates.

Claim 2.2.2.15 TELEMAC-2D can represent internal forcing induced by the tide raising potential of heavenly bodies for large maritime domains.

Substantiation: Validation study 3.20 (Western European Continental shelf) and 3.6 (Uncovering of a beach) include the tide raising potential of heavenly bodies.

Claim 2.2.2.16 TELEMAC-2D can represent the wetting and drying of intertidal zones (maritime applications) and of inundated plains (river applications). The numerical treatment of this process is very robust.

Substantiation: See validation studies 3.6 (Uncovering of a beach), 3.11 (Theoretical dam break wave - Initially dry bed), 3.16 (Breakwater), 3.18 (River Culm) and 3.19 (Mersey Estuary).

Claim 2.2.2.17 TELEMAC-2D is able to compute the dispersion by currents and diffusion of a tracer, with source or sink terms; this feature is employed only when water density is dependent upon the concentration in tracer.

Substantiation: See validation study 3.10 (Culverts), 3.21 (Point discharge without diffusion) and 3.22 (Point discharge with diffusion).

Claim 2.2.2.18 TELEMAC-2D can compute drogue monitoring and Lagrangian drifts.

Substantiation: Study 3.7 (Propagation of a surface wave) makes use of this possibility.

Claim 2.2.2.19 TELEMAC-2D is able to treat a number of flow singularities as internal boundary conditions: sills may be considered under the traditional approach made in open channel hydraulics through a relation between two superimposed open boundaries, short culverts may be treated as a couple of nodes with respective source and sink terms.

Substantiation: Validation studies 3.9 (Sills in a channel treated as singularities) and 3.10 (Culverts) illustrate these properties.

2.3. CONCEPTUAL MODEL

TELEMAC-2D is a depth-averaged model which solves the shallow water equations. (SWE). An in-depth description of the equations and their implementations is given in the book of Jean-Michel Hervouet (3).

2.3.1. ASSUMPTIONS AND APPROXIMATIONS

The basic equations of fluid mechanics (including free surface flows in natural domains) are the Navier-Stokes equations. To simplify them up to the depth-averaged SWE which are solved by TELEMAC-2D, several assumptions are necessary:

- The fluid is *newtonian*. The stress tensor can therefore be split into a spherical part (pressure component) and a shear (viscous terms);
- The fluid is *incompressible* and *homogeneous upon the vertical*. Horizontal density variations are considered through the hypothesis of Boussinesq, in which the fluid density $\rho(x,y)$ is approximated by a reference value ρ_o everywhere excepted in the vertical pressure gradient term. Under this assumption, we must have:

$$\frac{\rho-\rho_o}{\rho_o} << 1$$

- The *long wave approximation* is adopted; therefore, the pressure distribution is hydrostatic (the vertical pressure gradient is balanced by the gravity acceleration);
- The three-dimensional set of equations is *depth-averaged*, thus resulting in two-dimensional horizontal equations. In this operation, bottom and free surface boundary condition are used: they represent the effects of bed roughness and surface shear stress under wind effect respectively;
- A *Reynolds decomposition* and stochastic averaging (equivalent to time-averaging over a short time step under the principle of ergodicity) is applied in order to model turbulence. Reynolds shear stress resulting from this process are modelled as proposed by Boussinesq by using a *turbulent viscosity*;

- *Bottom friction* is modelled generally using non-linear laws versus velocity such as the Chezy, Strickler or Nikuradse friction laws;
- Substitution of the continuity equation in the two momentum equations leads to the non-divergent form of the momentum equation. This is the form of the SWE considered by TELEMAC-2D.

TELEMAC 2D therefore processes the following four equations simultaneously:

$$\frac{\partial h}{\partial t} + \vec{u}\cdot\vec{\nabla}(h) + h\,div(\vec{u}) = S_h \qquad \textit{continuity}$$

$$\frac{\partial u}{\partial t} + \vec{u}\cdot\vec{\nabla}(u) = -g\frac{\partial Z}{\partial x} + S_x + \frac{1}{h}div(h\nu_t\vec{\nabla}u) \qquad \textit{x-momentum}$$

$$\frac{\partial v}{\partial t} + \vec{u}\cdot\vec{\nabla}(v) = -g\frac{\partial Z}{\partial y} + S_y + \frac{1}{h}div(h\,\nu_t\vec{\nabla}v) \qquad \textit{y-momentum}$$

$$\frac{\partial T}{\partial t} + \vec{u}\cdot\vec{\nabla}(T) = S_T + \frac{1}{h}div(h\,\nu_T\vec{\nabla}T) \qquad \textit{tracer conservation}$$

in which:	h	(m)	depth of water
	u,v	(m/s)	velocity components
	T	(g/l or °c)	non-buoyant tracer or temperature
	g	(m/s^2)	gravity acceleration
	ν_t, ν_T	(m^2/s)	momentum and tracer diffusion coefficients
	Z	(m)	elevation of the free surface
	t	(s)	time
	x,y	(m)	horizontal space co-ordinates
	S_h	(m/s)	point injection or removal of fluid
	S_x, S_y	(m/s^2)	momentum source terms including the Coriolis force, bottom friction, surface wind shear and source/sink of momentum in the domain
	S_T	(g/l/s)	source/sink of tracer or heat

The main results at each node of the computational mesh are the depth of water h and the horizontal components of the depth-averaged velocity (u,v), and possibly a tracer T (temperature, salinity or other). The equations are given here in Cartesian coordinates. They can be processed in spherical co-ordinates as well when continental margins or large sea domains are considered.

The various terms of these equations are processed in one or several steps (if the method of characteristics is used for advection):

1) Advection of h, u, v and T,
2) Propagation, diffusion and source terms of dynamic equations,
3) Diffusion and source term of the tracer or heat transport equation.

It is possible not to process any of these steps and different equations are then solved. In addition, each of the h, u, v and T variables can be advected separately. For example, an equation of tracer dispersion can then be solved with a fixed advection field.

In TELEMAC-2D, a great care has been given to the treatment of boundary conditions. No flow boundaries and those where the velocity is prescribed are considered directly in the governing equations. Besides, conditions on open boundaries where the information is only partially known can be treated thanks to a genuine formulation based on the equations of characteristics.

Areas alternately dry and wet during a simulation can be considered in two different ways: either by masking semi-covered elements or by using a specific computation of the free surface gradient as soon as the free surface intersects with the bottom of an element.

The viscosity, including molecular and turbulent effects, may either be:

- an isotropic depth mean value,
- a non isotropic velocity dependent depth mean value if Elder formulae are used (the viscosity is different along the velocity direction and across this direction),
- the Smagorinski model in which the eddy viscosity is proportional to the square of the characteristic filter length introduced by the grid discretization and the modulus of the large eddy strain rate.
- a value computed thanks to a transport model of turbulent quantities k (turbulent kinetic energy k) and epsilon (turbulent dissipation ε), the equations of which are as follows:

$$\frac{\partial k}{\partial t} + \vec{u} \cdot \vec{\nabla}(k) = \frac{1}{h} div(h \frac{\nu_t}{\sigma_k} \vec{\nabla} k) + P - \varepsilon + P_{kv}$$

$$\frac{\partial \varepsilon}{\partial t} + \vec{u} \cdot \vec{\nabla}(\varepsilon) = \frac{1}{h} div(h \frac{\nu_t}{\sigma_\varepsilon} \vec{\nabla} \varepsilon) + \frac{\varepsilon}{k}(c_{1\varepsilon} P - c_{2\varepsilon}\varepsilon) + P_{\varepsilon v}$$

The right-hand side of these equations are production and dissipation of turbulent quantities.

2.3.2. CLAIMS AND SUBSTANTIATIONS

Claim 2.3.2.1 The Shallow Water Equations (SWE) accurately describe the physics of a large class of free surface flow problems in natural domains.

Substantiation: All assumption on which are based the SWE (see section 2.3.1) are commonly met in rivers, shallow lakes estuaries, coastal waters and even continental shelves for which horizontal dimensions are much larger than depth. A complete presentation of the derivation of these equations and the underlying assumptions is given by Kowalik and Murty (1993) (see reference (4)) or Martin and McCutcheon (see reference (5)).

Claim 2.3.2.2 The distribution of passive contaminant (tracer) can be determined by solution of the advection-diffusion equation.

Substantiation: The reader is referred to Fisher et al. (1979, chapter 2) (see reference (6)).

Studies 3.21 (Point discharge without diffusion) and 3.22 (Point discharge with diffusion) could also illustrate this point

Claim 2.3.2.3 The different treatments of turbulence in TELEMAC-2D are satisfactory for many free surface flow problems in rivers, estuaries and coastal waters.

Substantiation: The reader is referred to Abbott (1997) (see reference (7)) for a presentation of common turbulence modelling approaches used for tidal flow modelling.

Validation study 3.15 (Cavity) also present the interest of the k-ε and Smagorinski turbulence models respectively. It should be noted that shedding eddies formed behind circular piers as shown in validation study 3.8 (Flow around bridge piers) have been obtained by using a constant isotropic eddy viscosity and completely steady state boundary conditions.

The depth-averaging however results in a number of phenomena not being represented at all or only partially e.g. stratification (such as in brackish estuarine waters), rising plumes, wind driven currents (such as in lakes), geostrophic currents. In these cases, the TELEMAC user will have the possibility to use TELEMAC-3D. If the water column is not well mixed the water level will normally be accurately predicted by a depth-averaged model but the vertical distribution of currents cannot be resolved.

Vertical accelerations are neglected in the long wave approximation. This is not completely legitimate when studying bores, tsunamis and in the front of dam break waves, but as long as the violation is only local and vertical accelerations are moderate the model may provide adequate solutions (see validation studies 3.11 (Theoretical dam break wave - Initially dry bed), 3.12 (Theoretical dam break wave - Initially wet bed) and 3.16 (Breakwater) for examples). For this class of problems however, a new option was recently added in TELEMAC-2D in order to solve the Boussinesq equations instead of the SWE. Boussinesq equations are based on a second order wave theory which allows to study short wave phenomena. Vertical accelerations are not approximated to zero in this theory. However, Boussinesq equations should not be used in any situation, first because they are not really adapted to the study of typical small amplitude wave phenomena, and secondly because they need a much larger computational time than the SWE.

In all cases, an accurate computation of the near field of rising plumes where there exist strong vertical accelerations induced by buoyancy effects will not be possible with TELEMAC-2D. Parameterisation of buoyancy effects taking place in salinity-induced or thermal rising plumes is available in TELEMAC-3D.

2.4. ALGORITHMIC IMPLEMENTATION

TELEMAC-2D makes use of the Finite Element Method or Finite Volume Method as all simulation modules of the TELEMAC software system. The computational grid is made up of linear triangles; it is generally unstructured.

For a more detailed description of these issues the reader is referred to the TELEMAC-2D User Manual and Principle Note (see reference (3)).

2.4.1. ASSUMPTIONS AND APPROXIMATIONS

TELEMAC-2D uses the finite element method in which the solution is calculated at a number of nodes within the domain. At points which are not nodes, variables are defined as the sum of nodal values multiplied by interpolation functions: TELEMAC-2D uses a linear interpolation between the nodes of the element surrounding the point in question. This can be a limitation if the solution or the physical data such as the bed elevation is varying rapidly compared with the distance between nodes. In such cases, accuracy can usually be improved by refining the grid.

Some alternative finite element approaches use quadratic or higher order interpolation functions which can represent more detail within an element, but lead to much more complex and time-consuming calculations. It is not clear in general which is more efficient: fewer elements with complex interpolation functions or more elements with linear interpolation. However, the use of linear interpolation functions in TELEMAC-2D means that integrals over elements are calculated from analytical formulae, making for more speed and accuracy: more complex interpolation functions require element integrals to be calculated by Gaussian quadrature. It is also worthwhile to be noted that a so-called quasi-bubble triangular element option exist in TELEMAC-2D: the quasi-bubble triangle is made of one additional central point, interpolation functions being still linear. Use of this element often leads to an improvement in the stability of the solution.

The time discretization of the set of governing equations is semi-implicit, thus resulting in a system of 3N simultaneous algebraic equations to be solved at each time step, N being the number of nodes in the domain (about 7N simultaneous algebraic equations in the case of the quasi-bubble element). This system is solved by conjugated gradient-like iterative techniques or by direct techniques. For a given accuracy imposed to the solution and a given time step, iterative techniques allow an automatic adaptation of the needed computational time to the difficulty to solve each time step. Different options of conjugated gradients with different options for preconditioning are available in TELEMAC-2D.

The difficulty posed by the non-linear advection terms is solved by a number of numerical schemes available in TELEMAC. These include:

- A fractional step method using the method of characteristics for the advection terms and a Galerkin method for the propagation, diffusion and source/sink terms in the governing equations;
- Two variants of the Streamline Upwind Petrov Galerkin (SUPG) scheme;
- The N scheme;
- The PSI (Positive Streamline Invariant) scheme.

Each of these have different advantages in terms of accuracy, monotonicity, mass conservation, numerical diffusion and speed of execution. In some cases, the physical situation being modelled can influence the choice of the most appropriate and efficient solution method, but in general, default selections of methods will give reliable results. For more a detailed description of these issues see “Hydrodynamics of Free Surface Flows modelling with the finite element method” by Jean-Michel Hervouet (Wiley, 2007) (see ref. (3)) and recommendations given in section 13 of the TELEMAC-2D - Version 6.0 - User Manual (see ref. (8)).

In TELEMAC-2D, each dependent variable (h, u, v and optionally T) can be prescribed or set free independently on open boundaries. For solid boundaries, governing equations are written in the weak form and a no normal flow condition is considered. A similar treatment is applied to open boundaries with prescribed velocity. Finally, an option is available in order to deal with over or under-stressed boundary problems (Thompson boundary condition scheme): in this case values given by the user on the open boundaries are target values but they are not prescribed strictly by the code.

2.4.2. CLAIMS AND SUBSTANTIATIONS

Claim 2.4.2.1 The numerical methods used in TELEMAC-2D can provide an accurate solution of the SWE.

<u>Substantiation:</u> Comparison of numerical results with observations on calibrated models (e.g. Western European Continental shelf, validation study 3.20) and comparison of simple test cases with analytical solutions (e.g. Propagation of a surface wave, validation study 3.7) proves it.

Claim 2.4.2.2 The numerical methods used in TELEMAC-2D can provide an accurate solution of the tracer conservation equation.

Substantiation: Validation study 3.10 (Culverts), 3.21 (Point discharge without diffusion) and 3.22 (Point discharge with diffusion) are some examples of tracer transport computed by TELEMAC-2D.

Claim 2.4.2.3 TELEMAC-2D can solve the SWE on an unstructured grid of triangles, which allows grid resolution to vary in space and enables complex coastlines to be represented accurately.

Substantiation: See examples of grids used by TELEMAC-2D in real case studies (validation studies 3.16 to 3.20) where the mesh size can vary between 1 and a few metres (Breakwater, validation study 3.16) or between 5 and 50 km (Western European Continental shelf, validation study 3.20) in the same model.

Claim 2.4.2.4 Flow results are insensitive to the location of the grid points as long as the physical data and processes are adequately resolved.

Substantiation: Validation study 3.13 (Wind set-up) shows a one dimensional problem solved on a grid of triangles where the solution is accurately one dimensional.

Claim 2.4.2.5 TELEMAC-2D can support distorted triangular meshes up to a ratio 1:10. It is recommended to exploit this possibility in places where the flow is mono-directional in order to save memory and CPU time: meshes must be longer in the direction of the flow than in the transverse direction.

Substantiation: River Culm model (validation study 3.18) shows elongated meshes in the direction of the stream.

Claim 2.4.2.6 The solution method displays no preferred direction with respect to the axis orientation or to the mesh orientation.

Substantiation: This property is well illustrated by validation studies 3.1 and 3.2 showing the evolution of an initially Gaussian water surface in the centre of a square domain.

Claim 2.4.2.7 Different options for the numerical solution methods allow the most efficient choice to be made for a particular problem.

Substantiation: Reference is made to the User Manual (see ref. (8)) and Principle Note (see ref. (3)) of TELEMAC-2D.

Claim 2.4.2.8 The solution algorithm allows the calculation to progress faster when the solution is slowly varying because the iterative method converges in fewer iteration.

Substantiation: Validation study 3.17 (Malpasset dam break wave) shows number of iterations at different stages of calculation.

Claim 2.4.2.9 The solution method if efficient in use of memory and CPU time, particularly for large problems.

Substantiation: Preconditioned Conjugated Gradient (PCG) methods are more efficient compared with direct methods for matrix inversion when Element By Element (EBE) techniques are used in finite elements, both for number of operations required and storage space used. This is particularly true for large problems. See J.-M. Hervouet, reference (9).

Claim 2.4.2.10 With appropriate choice of the mesh resolution, TELEMAC-2D produces solutions without wiggles (i.e. oscillations in space at a given time or oscillations in time at a given position).

Substantiation: Validation studies 3.4 (Bump - Transcritical condition) and 3.5 (Bump - transcritical condition with hydraulic jump) shows transcritical flows without the occurrence of wiggles. In validation study 3.20 (Western European Continental shelf), mesh refinement on the Atlantic shelf prevents the formation of space wiggles in the Gulf of Biscay.

Claim 2.4.2.11 With appropriate choice of solution method options, the flow results are mass conservative.

Substantiation: Figures for mass conservation are given for two examples: validation studies 3.17 (Malpasset dam break wave) and 3.19 (Mersey Estuary).

Claim 2.4.2.12 With appropriate choice of solution method options, the calculation of tracer dispersion is mass conservative.

Substantiation: Figures for tracer conservation are given in the validation study 3.10 (Culverts), 3.21 (Point discharge without diffusion) and 3.22 (Point discharge with diffusion).

Claim 2.4.2.13 TELEMAC-2D offers non diffusive solution methods of the advective terms of the flow equations.

Substantiation: This is shown by the transport of a tracer in a uniform velocity field where the diffusion is not considered like in study 3.21 (Point discharge without diffusion).

Claim 2.4.2.14 TELEMAC-2D offers unconditionally stable semi-implicit solution methods. However, it is recommended to adopt a time step such that the Courant number is not larger than 3 in general.

Substantiation: See validation studies 3.16 (Breakwater) and 3.17 (Malpasset dam break wave).

Claim 2.4.2.15 Boundary conditions can be applied as elevations, velocities, discharges or non-reflecting conditions.

Substantiation: In river studies, the discharge is generally imposed upstream and the water surface elevation is imposed downstream, see for instance validation study 3.18 (River Culm).

Evolution of an initially Gaussian surface with permeable boundary is an example of an incident wave boundary condition with zero incident wave (see validation study 3.2 (Gaussian water surface centred in a square domain - open boundaries)).

Claim 2.4.2.16 Boundary condition values can be prescribed exactly or "suggested" in order to deal with under or over-stressed problems (in the cases where too much or insufficient information is given on the domain open boundaries). This option should not be used blindly however.

Substantiation: Validation studies 3.2 (Gaussian water surface centred in a square domain - open boundaries) and 3.7 (Propagation of a surface wave) illustrates the use of the Thompson type of boundary condition which allows to deal with under or over-stressed problems.

Claim 2.4.2.17 The natural treatment made in the governing equations in order to prescribe a no normal flow on solid boundaries is efficient in TELEMAC.

Substantiation: All validation studies having solid boundaries illustrate this property.

Claim 2.4.2.18 TELEMAC-2D can represent eddy formation and shedding behind structure.

Substantiation: Validation study 3.8 (Flow around bridge piers) illustrates this feature. Provided the viscosity is chosen to be sufficiently low then eddies are shed and a time dependent solution is obtained to this problem that has steady state boundary conditions.

Claim 2.4.2.19 TELEMAC-2D propagates surface waves at the celerity given by the shallow water theory.

Substantiation: See validation study 3.7 (Propagation of a surface wave).

Claim 2.4.2.20 The numerical treatment applied to alternately drying and wetting elements in TELEMAC-2D is efficient.

Substantiation: See validation studies 3.18 (River Culm) and 3.19 (Mersey Estuary).

2.5. SOFTWARE IMPLEMENTATION

TELEMAC-2D makes use of the TELEMAC data structure. It can be run on UNIX workstations, micro-computers under Windows (NT4 or upper) or CRAY computers. The code is optimised for use with vector architecture computers such as CRAY. Starting at version 4.0 a parallel option is available in TELEMAC-2D: it makes use of possibilities of parallel computing offered by super-computers (such as CRAY) or by a cluster of workstations.

2.5.1. IMPLEMENTATION TECHNIQUES

TELEMAC-2D, as well as other TELEMAC simulation software, has been written in the FORTRAN 90 programming language. It respects Quality Assurance Procedures (see the Quality Assurance Manual for EDF-R&D Scientific and Technical Software [3]), an approach to construction and quality assessment at various stages in the life of the software, including the preparation of a Quality Plan (the Quality Assurance Plan of TELEMAC-2D is referenced [4]) to guide all of the work. In particular, a product under Quality Assurance is provided with a Validation file (the present document) which describes a set of test cases. This makes it possible to assess both the advantages and drawbacks of the product and to define its range of applications. These test cases are used during the software development and are checked each time the code is modified. The quality assurance scheme also involves the production of a Principle Note and User Manual (see ref. (3) and (8)), the use of source code control and the issuing of controlled documents.

The code has been developed in a highly structured and modular way increasing maintainability and reducing the possibility of errors. The code is well documented and all variables are explicitly typed (using the FORTRAN `implicit none` statement) which also helps to produce maintainable error-free code. All floating point variables are double precision throughout, to increase the accuracy of the matrix solution and also to cater for large variations in grid size over the model.

The code is written using efficient memory management techniques which, combined with the element by element approach which is relatively undemanding of storage space, means that even quite large models do not require a great deal of memory to run. At the highest level of the program, all variables are combined in two large arrays, one integer and one floating point. The overall size of these can be specified at run time as the main subroutine is compiled and linked to libraries containing the bulk of the code each time the program is run.

In addition, certain subroutines are provided for the user to alter if he/she requires to specify complex initial and boundary conditions, to compare the solution provided by TELEMAC-2D with an analytical solution, to store specific outputs, etc. This allows great flexibility in the situations which can be represented, provided that the user has some elementary programming ability. Assistance with these routines can be provided as part of the user support to customers of TELEMAC-2D. These user routines are described in more detail in the User Manual (see reference (8)).

TELEMAC-2D uses input files containing the mesh and bathymetry (binary file), the boundary conditions (ASCII file) and the steering parameters (ASCII file). This last file includes such information as the input/output filenames, the timestep, the friction and viscosity formulations adopted and the solver used for the computation. The information contained in the different input files is generally produced by using user-friendly pre-processors included in the TELEMAC processing line. The user can also define his own binary or ASCII files through user subroutine facilities.

Many functions facilitating the modeller's work in the realisation of a study, such as a hot start function (definition of initial conditions from a previous computation), a verification of mass balance after the computation, statistics on the solver, user accessible definition of the result and output files, user subroutines and input/output files, etc., are implemented in TELEMAC-2D.

Results produced by TELEMAC-2D are stored in a binary file respecting the standard common to all TELEMAC simulation software (named SELAFIN format). In this way, results are treated by FUDAA-PREPRO, the user-friendly post-processor of the TELEMAC processing line allow producing graphical outputs and statistics.

TELEMAC-2D is free software. At the instigation of its User's Club and to meet EDF-R&D internal needs, TELEMAC-2D is currently experiencing rapid development.

2.5.2. CLAIMS AND SUBSTANTIATIONS

Claim 2.5.2.1 TELEMAC-2D is a well tested implementation of its numerical algorithms, developed in a structured way according to quality assurance procedures.

<u>Substantiation:</u> User documentation and the EDF-R&D Quality Assurance Manual for Technical and Scientific Software (see reference [3]).

Claim 2.5.2.2 TELEMAC-2D is inserted in a complete processing line of software, the TELEMAC software system.

Substantiation: The reader is referred to the commercial brochure of the TELEMAC software system (see ref. [7]), or the user manuals of the different modules (references [5] to [17]).

Claim 2.5.2.3 TELEMAC-2D makes use of the TELEMAC data structure and finite element library in common with other simulation software of the TELEMAC processing line.

Substantiation: Existence of software description manuals for the TELEMAC input/output, the finite element library BIEF (see ref. (10)) and a guide for programming in the TELEMAC system (see ref. (11)).

Claim 2.5.2.4 The TELEMAC-2D software is written in a portable programming language and can be installed on a variety of workstations.

Substantiation: TELEMAC-2D has been installed on Hewlett-Packard, Sun Microsystems, Digital, IBM workstations running under Unix, Linux, on Cray and Fujitsu super-computers and on micro-computers under Windows (NT4 or upper).

Claim 2.5.2.5 The TELEMAC-2D computational code does not impose any limitation on the size of the model to be represented.

Substantiation: The code is fully written with a dynamic allocation of the memory.

Claim 2.5.2.6 The graphical front end for TELEMAC-2D can be run on Unix workstations with access to IlogViews libraries.

Substantiation: This property has been verified.

Claim 2.5.2.7 The TELEMAC-2D software is structured so that user defined subroutines can easily be applied instead of the default subroutines.

Substantiation: This is possible by means of a series of well documented blank subroutines (30 in Version 6.0) which can be used in order to overwrite default values.

Claim 2.5.2.8 A TELEMAC User's Club has been meeting every year since 1994. Modelling difficulties and solutions are discussed at this meeting; new functions of TELEMAC-2D are presented.

Substantiation: Minutes of the yearly meetings.

3. VALIDATION STUDIES

3.1. GAUSSIAN WATER SURFACE CENTRED IN A SQUARE DOMAIN - SOLID BOUNDARIES

Title	**Evolution of a Gaussian water surface centred in a square domain with solid boundaries**
Initial study	**J.-M. Hervouet – April 1992**
Last update	**December 2011**
Version	**TELEMAC-2D 6.1**

3.1.1. PURPOSE

To demonstrate that the Telemac-2D solution is not polarised because it can simulate the circular spreading of a wave. Also to show that the no-flow condition is satisfied on solid boundaries and that the solution remains symmetric after reflection of the circular wave on the boundaries.

3.1.2. LINKED CLAIMS

- **Claim 2.4.2.6** The solution method displays no preferred direction with respect to the axis orientation or to the mesh orientation.
- **Claim 2.4.2.15** Boundary conditions can be applied as elevations, velocities, discharges or non-reflecting conditions.

3.1.3. APPROACH

The fluid is initially at rest with a Gaussian free surface in the centre of a square domain (see Figure 3.1.1).

Geometry:

- Size of the model = 20.1 m x 20.1 m
- Water depth at rest = $2.4 \exp \frac{-\left[(x-10)^2 + (y-10)^2\right]}{4}$

Mesh:

The mesh is regular. It is made up with squares split into two triangles (see Figure 3.1.1).

- 8978 triangular elements
- 4624 nodes
- Size of triangles = 0.3 m

Boundaries:

- Solid walls with slip condition

Bottom:

- Strickler formula with K = 40
- Horizontal bed at depth h = 2.40 m along the boundaries

Turbulence:

- Constant viscosity equal to zero

Nota: instead of prescribing a zero viscosity, no diffusion step could have been used (keyword *DIFFUSION OF VELOCITY* prescribed to *NO*)

Algorithm:

- Type of advection:
  - centred semi-implicit scheme + SUPG upwinding on velocities
  - conservative + modified SUPG on depth
- Type of element:
  - P1 triangle for velocities
  - P1 triangle for h
- GMRES solver
- Accuracy = 10^{-4}

Time data:

- Time step = 0,04 sec.
- Simulation duration = 4 sec.

3.1.4. RESULTS

The wave spreads circularly around the initial water surface peak elevation (Figure 3.1.2 to Figure 3.1.4). When it reaches the boundaries, reflection occurs. Interaction between reflected waves issuing from the four walls can be observed after time 1.8 sec. on Figure 3.1.3 and on Figure 3.1.4.

The final volume in the domain is equal to the initial volume.

3.1.5. CONCLUSIONS

Even though the mesh is polarised (along the x and y directions and the main diagonal), the solution is not.

Solid boundaries are treated properly: no bias occurs in the reflected wave.

Water mass is conserved.

3.1.6. STEERING FILE

```
/---------------------------------------------------------------------/
/   TELEMAC-2D       VALIDATION TEST CASE NUMBER 1                    /
/                     GAUSSIAN SOLID BOUNDARIES                       /
/---------------------------------------------------------------------/
/-----------------------------------------------
/ COMPUTER INFORMATIONS
/-----------------------------------------------
/
GEOMETRY FILE                     = geo
FORTRAN FILE                      = princi.f
BOUNDARY CONDITIONS FILE          = cli.txt
RESULTS FILE                      = res1
REFERENCE FILE                    = ref
/
/-----------------------------------------------
/ GENERAL INFORMATIONS - OUTPUTS
/-----------------------------------------------
/
TITLE                             = 'GAUSSIAN WALL'
VARIABLES FOR GRAPHIC PRINTOUTS = 'U,V,H,T'
GRAPHIC PRINTOUT PERIOD           = 10
LISTING PRINTOUT PERIOD           = 10
VALIDATION                        = YES
TIME STEP                         = 0.04
NUMBER OF TIME STEPS              = 100
MASS-BALANCE                      = YES
INFORMATION ABOUT SOLVER          = YES
/
/-----------------------------------------------
/ INITIAL CONDITIONS
/-----------------------------------------------
/
COMPUTATION CONTINUED             = NO
INITIAL CONDITIONS                = 'PARTICULAR'
/
/-----------------------------------------------
/ PHYSICAL PARAMETERS
/-----------------------------------------------
/
LAW OF BOTTOM FRICTION            = 3
FRICTION COEFFICIENT              = 40.
TURBULENCE MODEL                  = 1
VELOCITY DIFFUSIVITY              = 0.
/
/-----------------------------------------------
/  NUMERICAL PARAMETERS
/-----------------------------------------------
/
TYPE OF ADVECTION                 = 2;5;1;1
SOLVER                            = 7
SOLVER OPTION                     = 3
SOLVER ACCURACY                   = 1.E-6
DISCRETIZATIONS IN SPACE          = 12 ; 11
PRECONDITIONING                   = 2
INITIAL GUESS FOR H               = 1
IMPLICITATION FOR DEPTH           = 0.6
IMPLICITATION FOR VELOCITY        = 0.6
```

```
/-----------------------------------------------
/ IN CASE OF USE OF FINITE VOLUME
/-----------------------------------------------
/EQUATIONS                          = 'SAINT-VENANT VF'
/FINITE VOLUME SCHEME               = 6
/VARIABLE TIME-STEP                 = YES
/DESIRED COURANT NUMBER             = 0.8
/DURATION                           = 4.
/-----------------------------------------------

&FIN
```

3.1.7. FIGURES

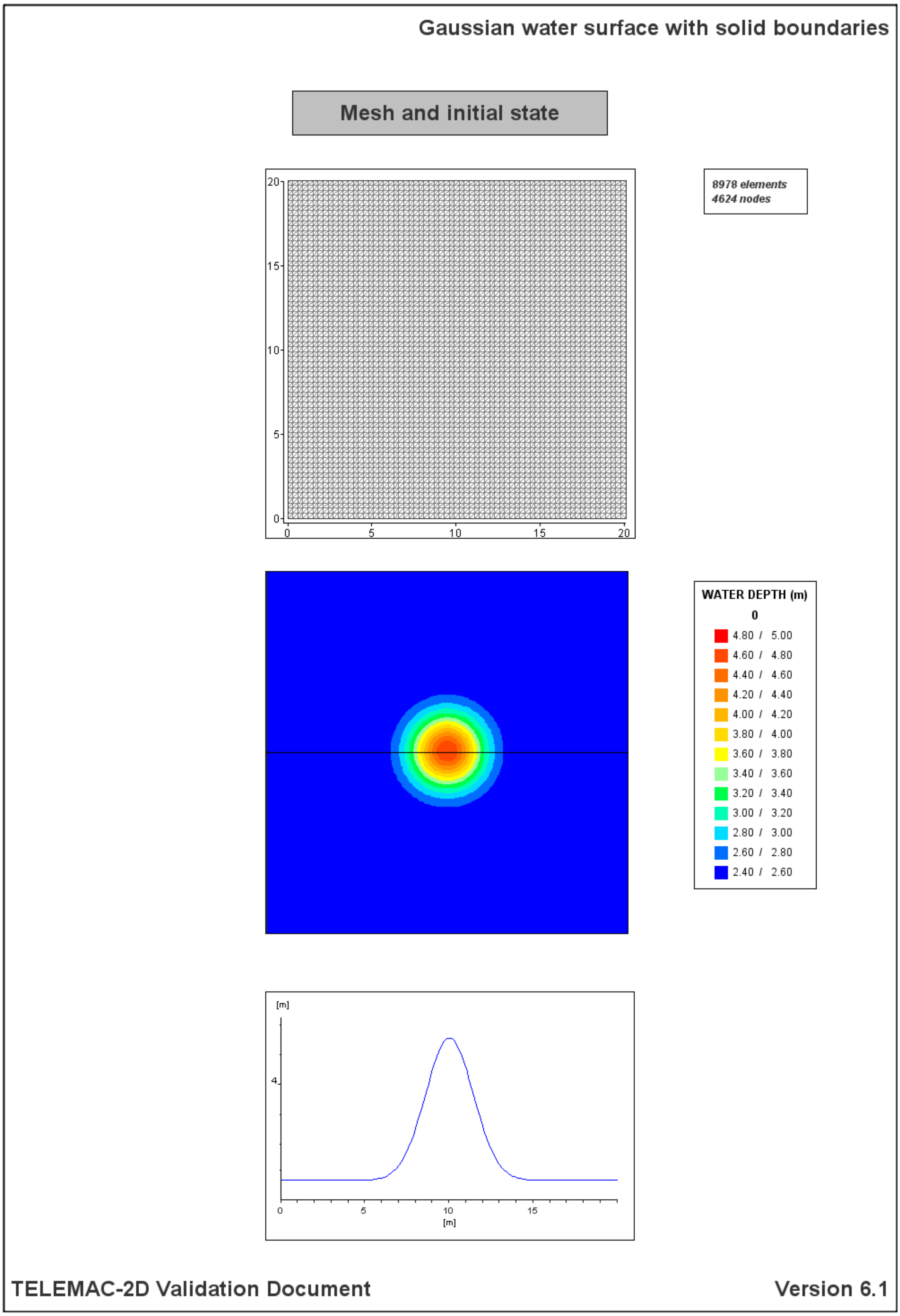


Figure 3.1.1 : Mesh and initial state.

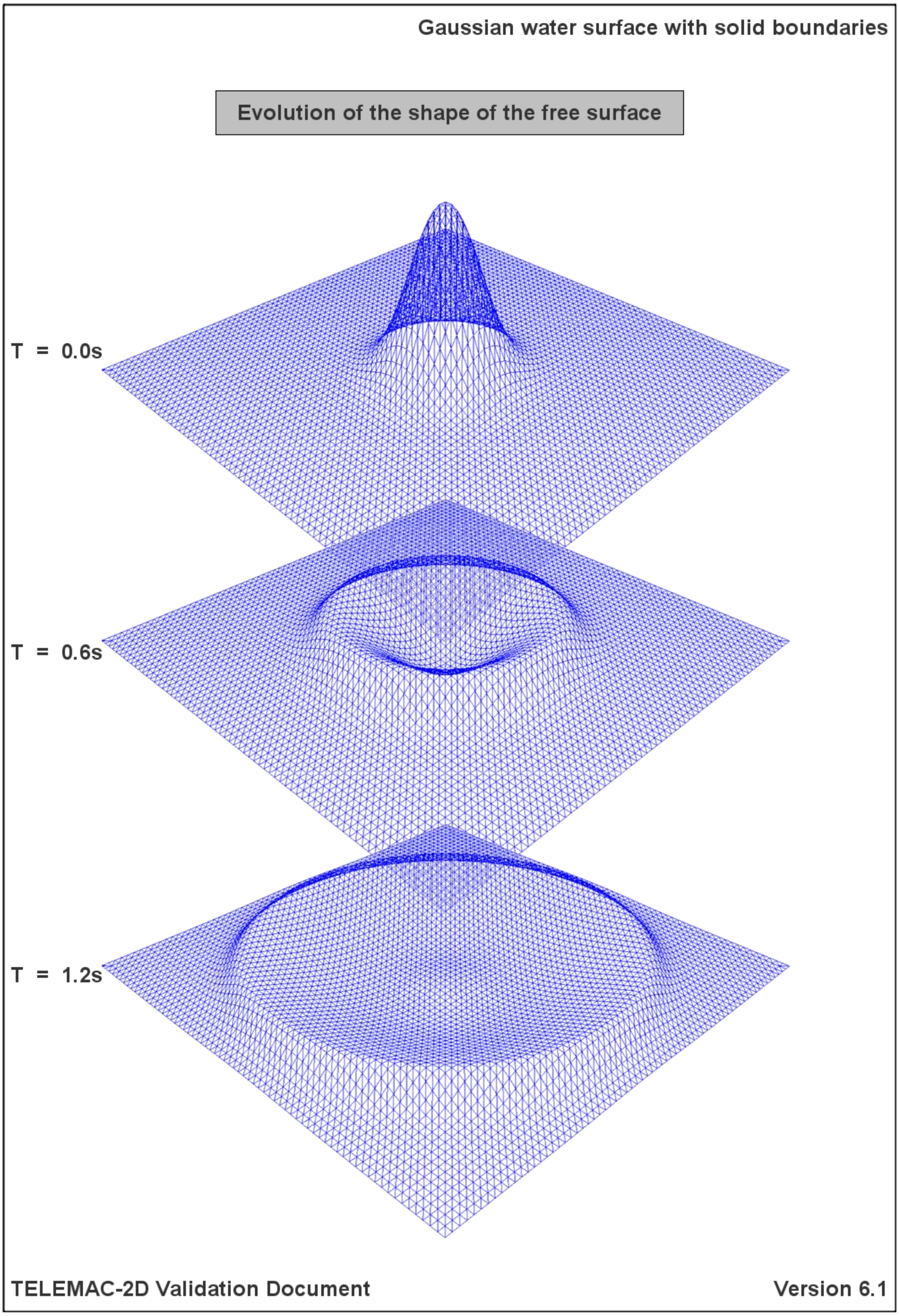


Figure 3.1.2 : Evolution of the shape of the free surface from 0.0 sec. to 1.2 sec.

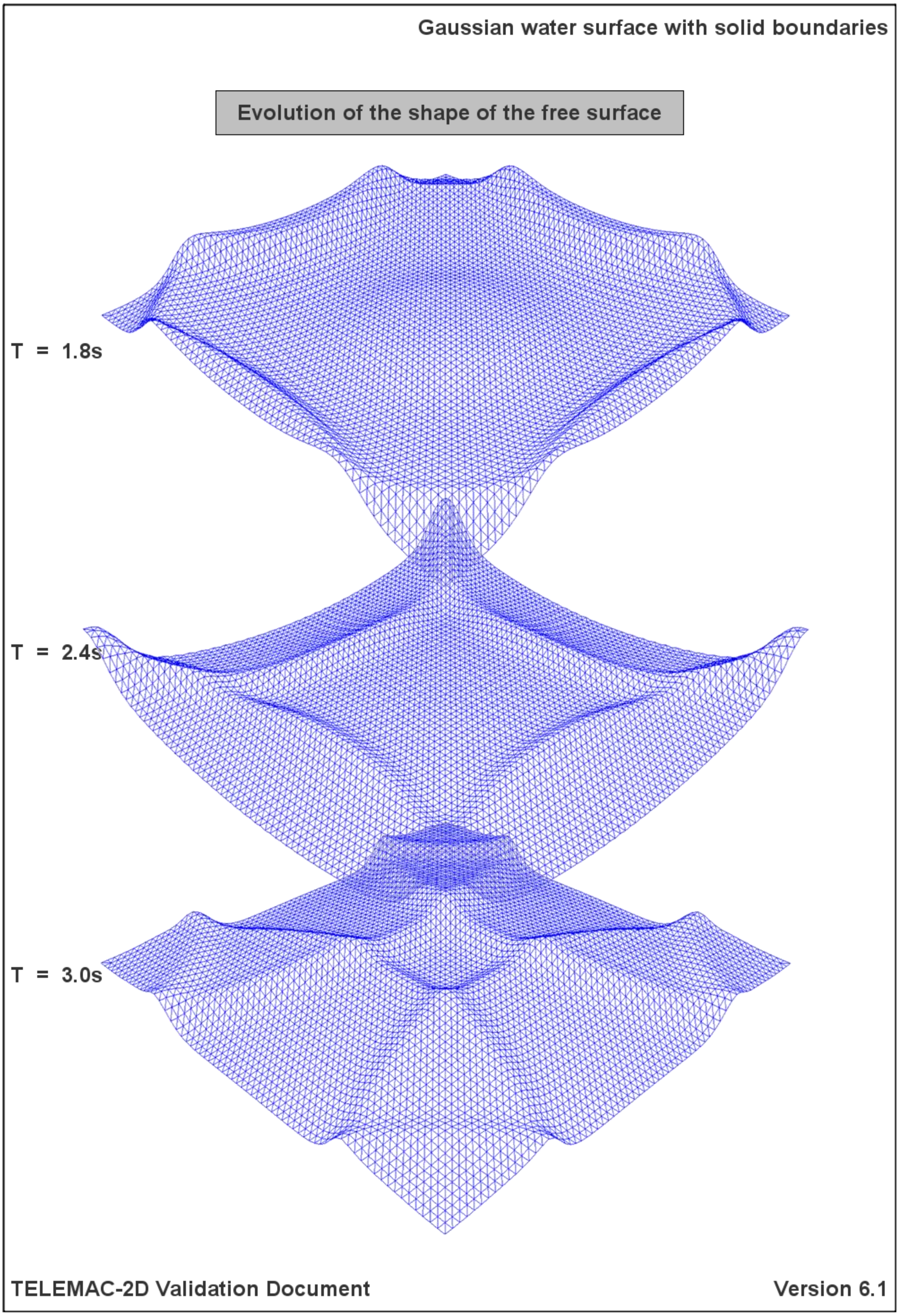


Figure 3.1.3 : Evolution of the shape of the free surface from 1.8 sec. to 3.0 sec.

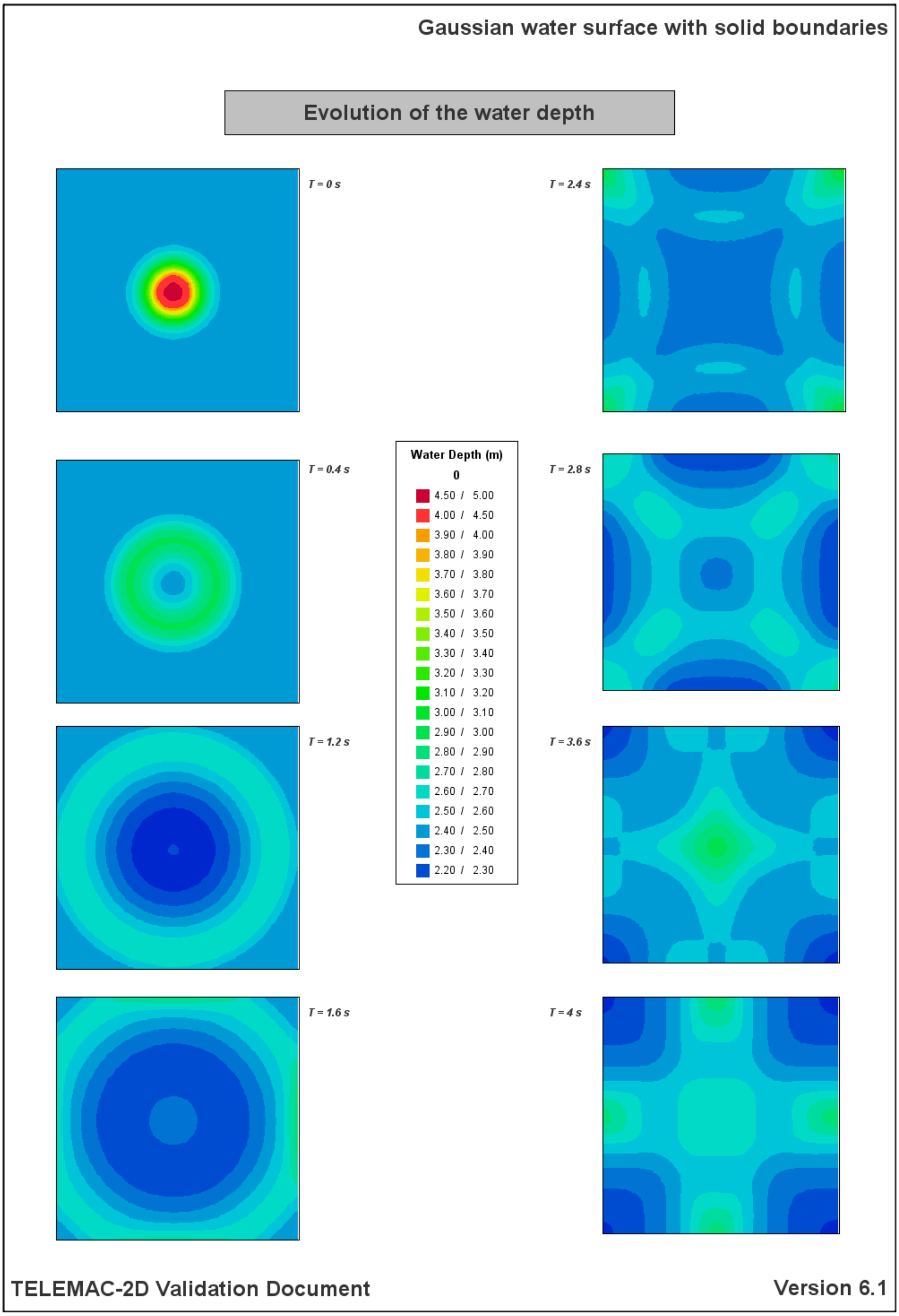
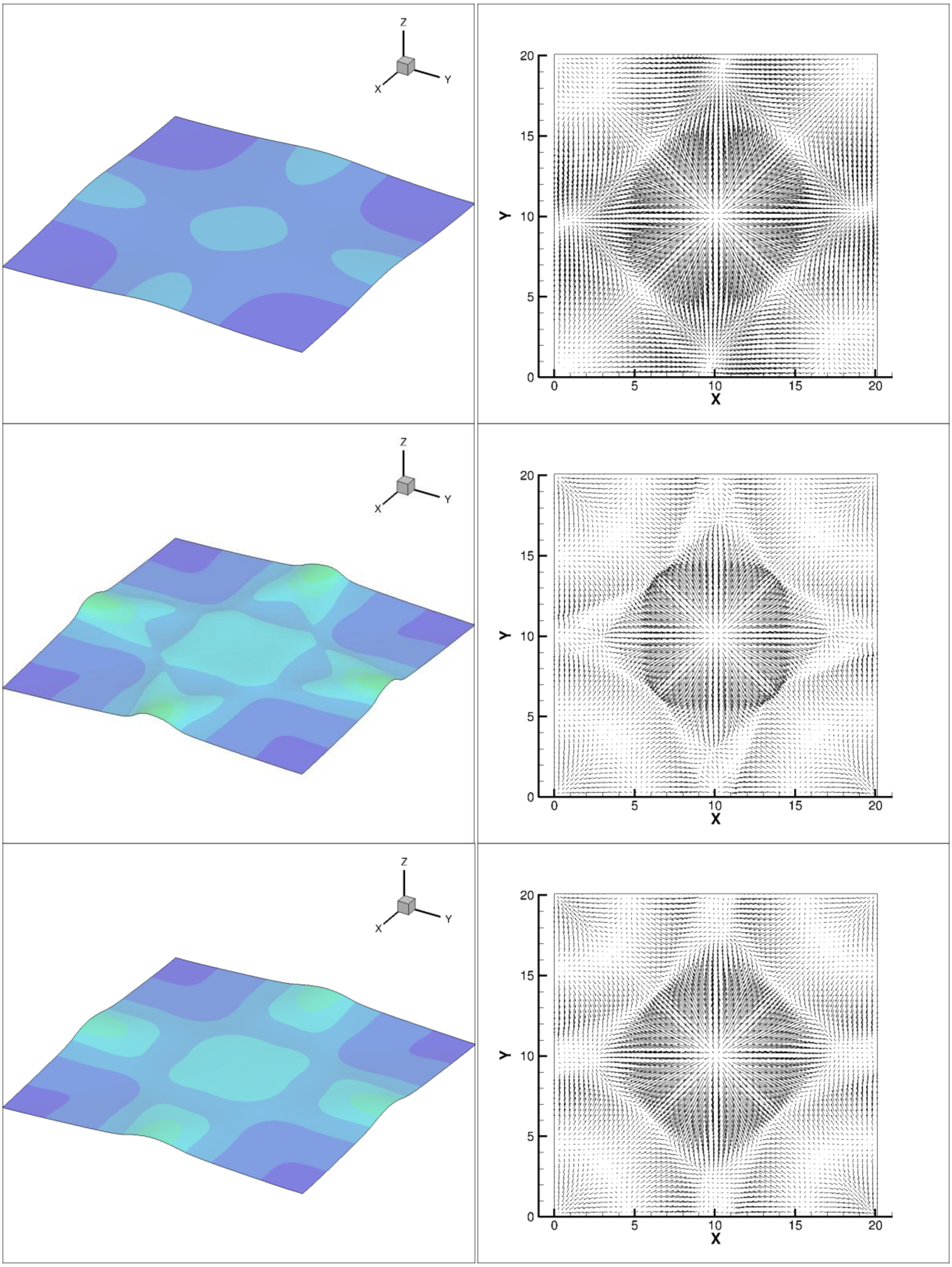


Figure 3.1.4 : Evolution of the water depth from 0.0 sec. to 4.0 sec.



Water depth and velocity field given by finite volume (Kinetic scheme 1st and 2nd order and WAF scheme) at t=4.0 sec

3.2. GAUSSIAN WATER SURFACE CENTRED IN A SQUARE DOMAIN - OPEN BOUNDARIES

Title	**Evolution of a Gaussian water surface centred in a square domain with open boundaries**
Initial study	**J.-M. Hervouet – April 1992**
Last update	**December 2011**
Version	**TELEMAC-2D 6.1**

3.2.1. PURPOSE

To demonstrate that the Telemac-2D solution is not polarised because it can simulate the circular spreading of a wave. Also to show that options available in Telemac-2D can simulate radiation of a wave out of a domain without any spurious reflection on the open boundaries.

3.2.2. LINKED CLAIMS

- **Claim 2.4.2.6** The solution method displays no preferred direction with respect to the axis orientation or to the mesh orientation.
- **Claim 2.4.2.15** Boundary conditions can be applied as elevations, velocities, discharges or non-reflecting conditions.
- **Claim 2.4.2.16** Boundary condition values can be prescribed exactly or “suggested” in order to deal with under or over-stressed problems (in the cases where too much or insufficient information is given on the domain open boundaries). This option should not be used blindly however.

3.2.3. APPROACH

The fluid is initially at rest with a Gaussian free surface in the centre of a square domain (see Figure 3.2.1).

<u>Geometry:</u>

– Size of the model = 20.1 m x 20.1 m

– Water depth at rest = $2.4\exp\dfrac{-\left[(x-10)^2+(y-10)^2\right]}{4}$

<u>Mesh:</u>

The mesh is regular. It is made up with squares split into two triangles (see Figure 3.2.1).

– 8978 triangular elements

– 4624 nodes

– Size of triangles ≈ 0.30 m

<u>Boundaries:</u>

- Free sea surface and velocity components
- Thompson scheme for open boundaries

Bottom:

- No friction
- Horizontal bed at depth 2.40 m along the boundaries

Turbulence:

- Constant viscosity equal to zero

Nota: instead of prescribing a zero viscosity, no diffusion step could have been used (keyword *DIFFUSION OF VELOCITY* prescribed to *NO*)

Algorithm:

- Type of advection:
  - Characteristics on velocities
  - conservative + modified SUPG on depth
- Type of element:
  - quasi-bubble triangle for velocities
  - P1 triangle for h
- GMRES solver
- Accuracy = 10^{-6}

Time data:

- Time step = 0,04 sec.
- Simulation duration = 4 sec.

3.2.4. RESULTS

The wave spreads circularly around the initial water surface peak elevation (Figure 3.2.2 to Figure 3.2.4). The velocity field is radial. No reflection occurs on the open boundaries.

The initial volume of water in the domain is 2000 m^3. The volume of water lost is 0.41 m^3, i.e. 0,02 %.

Long wave celerity is 4.85 m/s for h = 2.40 m and 6.86 m/s for h = 4.80 m, which means the peak of the wave should reach the boundary after 1.46 to 2 seconds in the long wave hypothesis. The computed value is 1.6 sec.

3.2.5. CONCLUSIONS

Even though the mesh is polarised (along the x and y directions and the main diagonal), the solution is not. Open boundaries are treated properly thanks to the Thompson scheme for open boundaries (keyword `OPTION FOR LIQUID BOUNDARIES`).

3.2.6. STEERING FILE

```
/---------------------------------------------------------------------/
/   TELEMAC-2D       VALIDATION TEST CASE NUMBER 2                    /
/                     GAUSSIAN OPEN BOUNDARIES                        /
/---------------------------------------------------------------------/
/
/----------------------------------------------
/ COMPUTER INFORMATIONS
/----------------------------------------------
/
GEOMETRY FILE                      = geo
FORTRAN FILE                       = princi.f
BOUNDARY CONDITIONS FILE           = cli.txt
RESULTS FILE                       = res
REFERENCE FILE                     = ref
/
/----------------------------------------------
/ GENERAL INFORMATIONS - OUTPUTS
/----------------------------------------------
/
TITLE                              = 'GAUSSIAN-OPEN'
VARIABLES FOR GRAPHIC PRINTOUTS = 'U,V,H,T'
GRAPHIC PRINTOUT PERIOD            = 10
LISTING PRINTOUT PERIOD            = 10
VALIDATION                         = YES
TIME STEP                          = 0.04
NUMBER OF TIME STEPS               = 100
MASS-BALANCE                       = YES
INFORMATION ABOUT SOLVER           = YES
/
/----------------------------------------------
/ INITIAL CONDITIONS
/----------------------------------------------
/
COMPUTATION CONTINUED              = NO
INITIAL CONDITIONS                 = 'PARTICULAR'
/
/----------------------------------------------
/ PHYSICAL PARAMETERS
/----------------------------------------------
/
LAW OF BOTTOM FRICTION             = 0
FRICTION COEFFICIENT               = 0.
TURBULENCE MODEL                   = 1
VELOCITY DIFFUSIVITY               = 0.
/
/----------------------------------------------
/ NUMERICAL PARAMETERS
/----------------------------------------------
/
TYPE OF ADVECTION                  = 1;5
SUPG OPTION                        = 1;1
SOLVER                             = 7
SOLVER OPTION                      = 3
SOLVER ACCURACY                    = 1.E-6
DISCRETIZATIONS IN SPACE           = 12 ; 11
PRECONDITIONING                    = 2
```

```
INITIAL GUESS FOR H                = 1
IMPLICITATION FOR DEPTH            = 0.6
IMPLICITATION FOR VELOCITY         = 0.6
OPTION FOR LIQUID BOUNDARIES       = 2
/-----------------------------------------------
/ IN CASE OF USE OF FINITE VOLUME
/-----------------------------------------------
/EQUATIONS                            = 'SAINT-VENANT VF'
/FINITE VOLUME SCHEME                 = 6
/VARIABLE TIME-STEP                   = YES
/DESIRED COURANT NUMBER               = 0.8
/DURATION                             = 4.
/-----------------------------------------------
/
&FIN
```

3.2.7. FIGURES

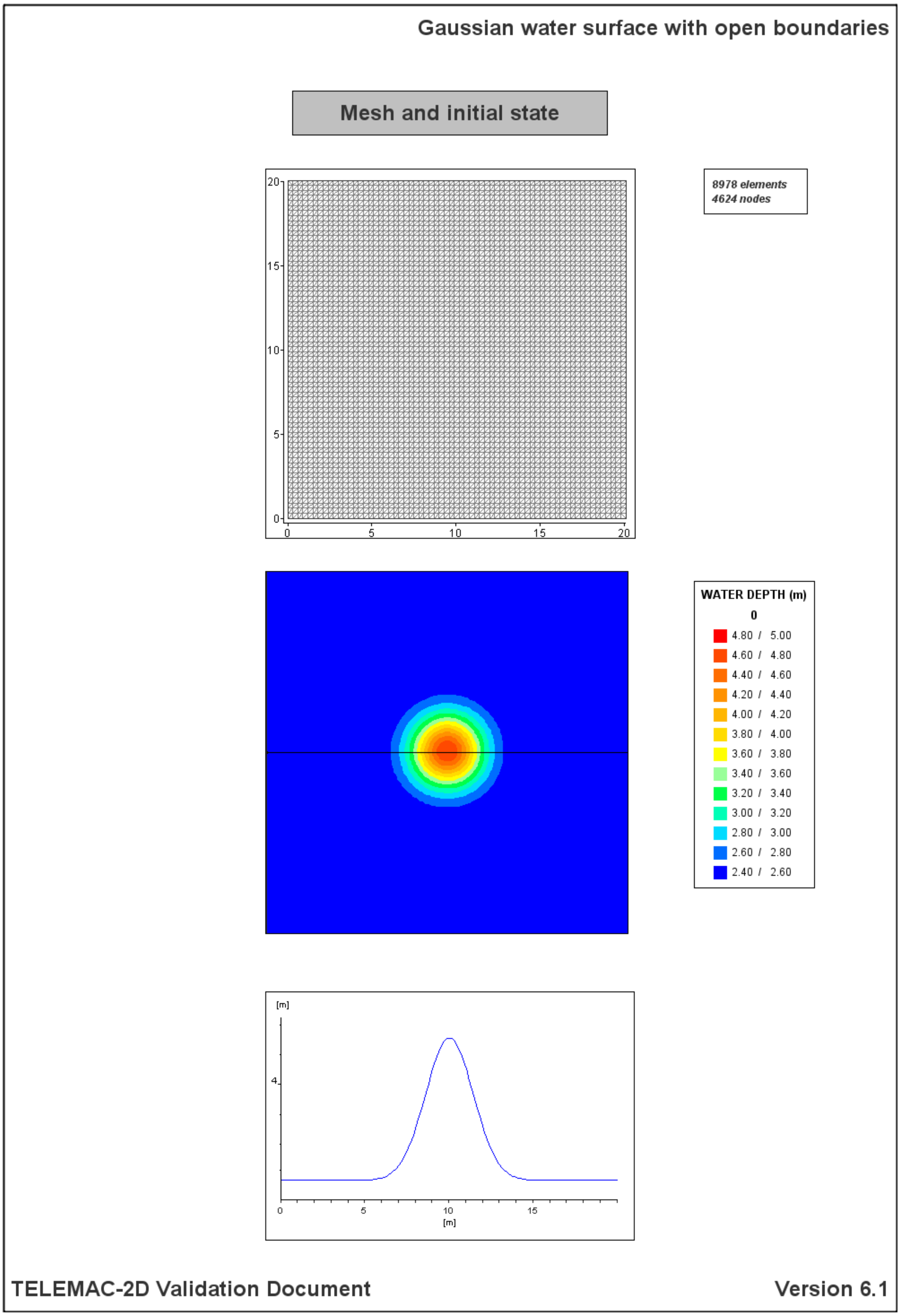


Figure 3.2.1 : Mesh and initial state.

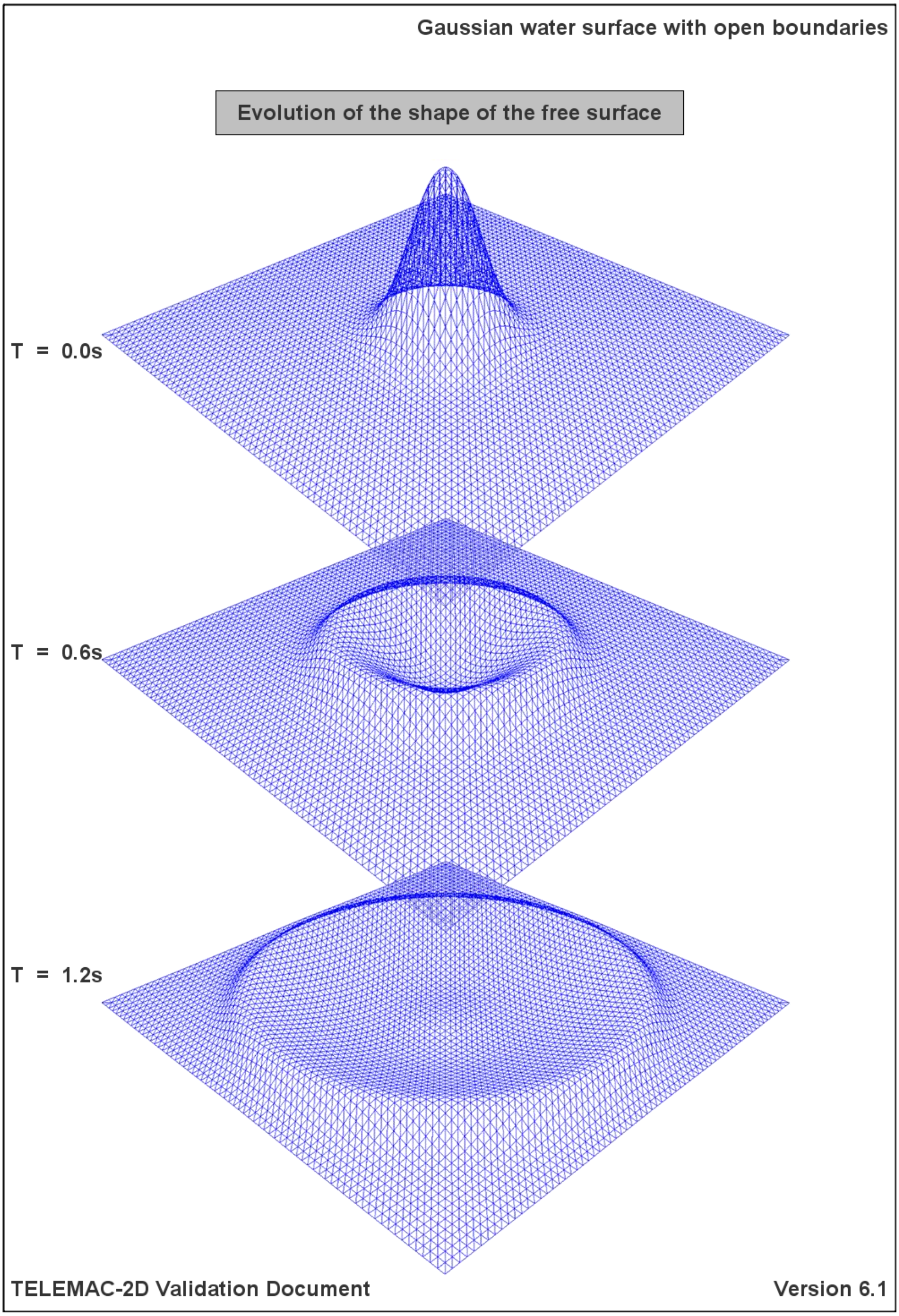


Figure 3.2.2 : Evolution of the shape of the free surface from 0.0 sec. to 1.2 sec.

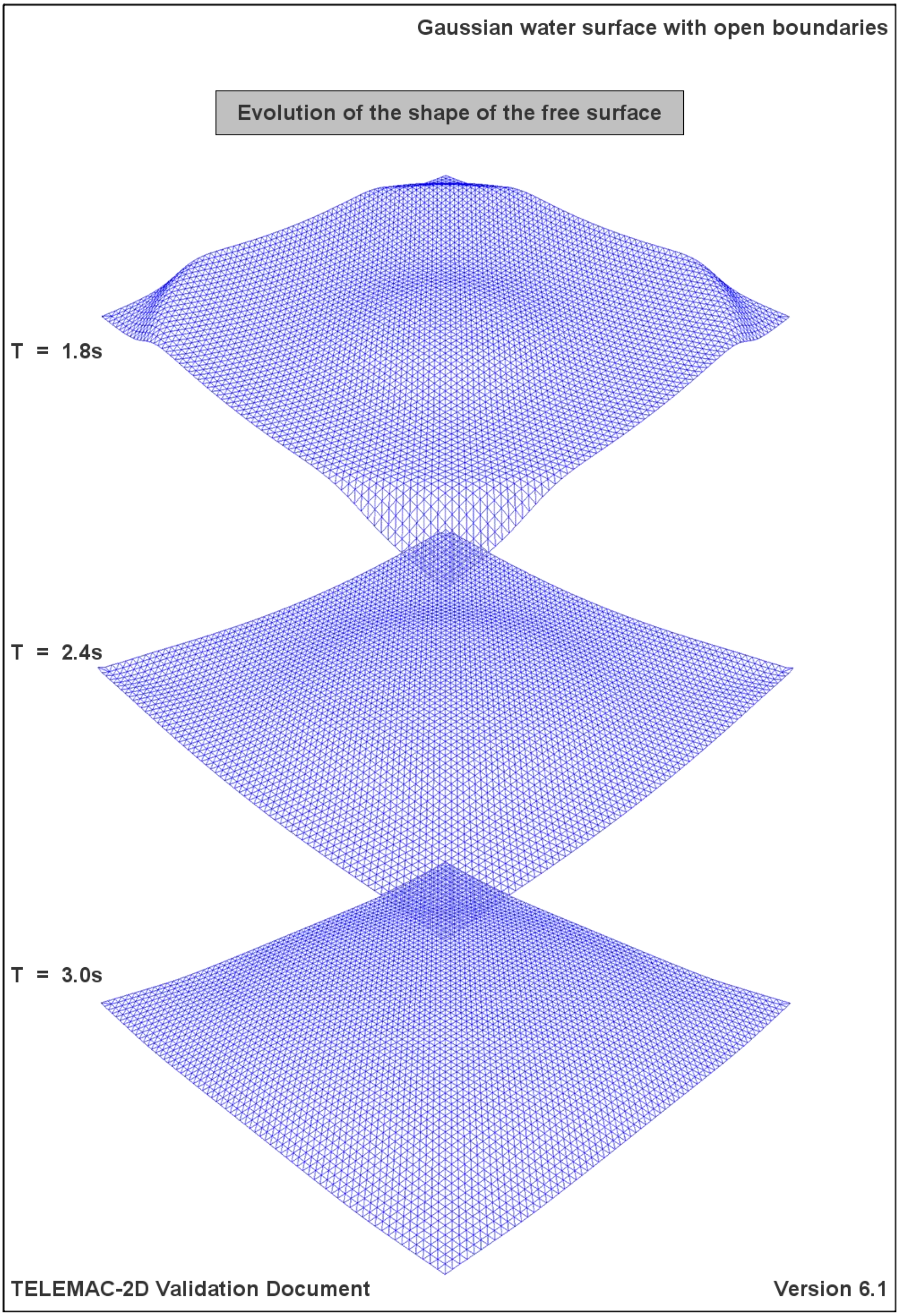


Figure 3.2.3 : Evolution of the shape of the free surface from 1.8 sec. to 3.0 sec.

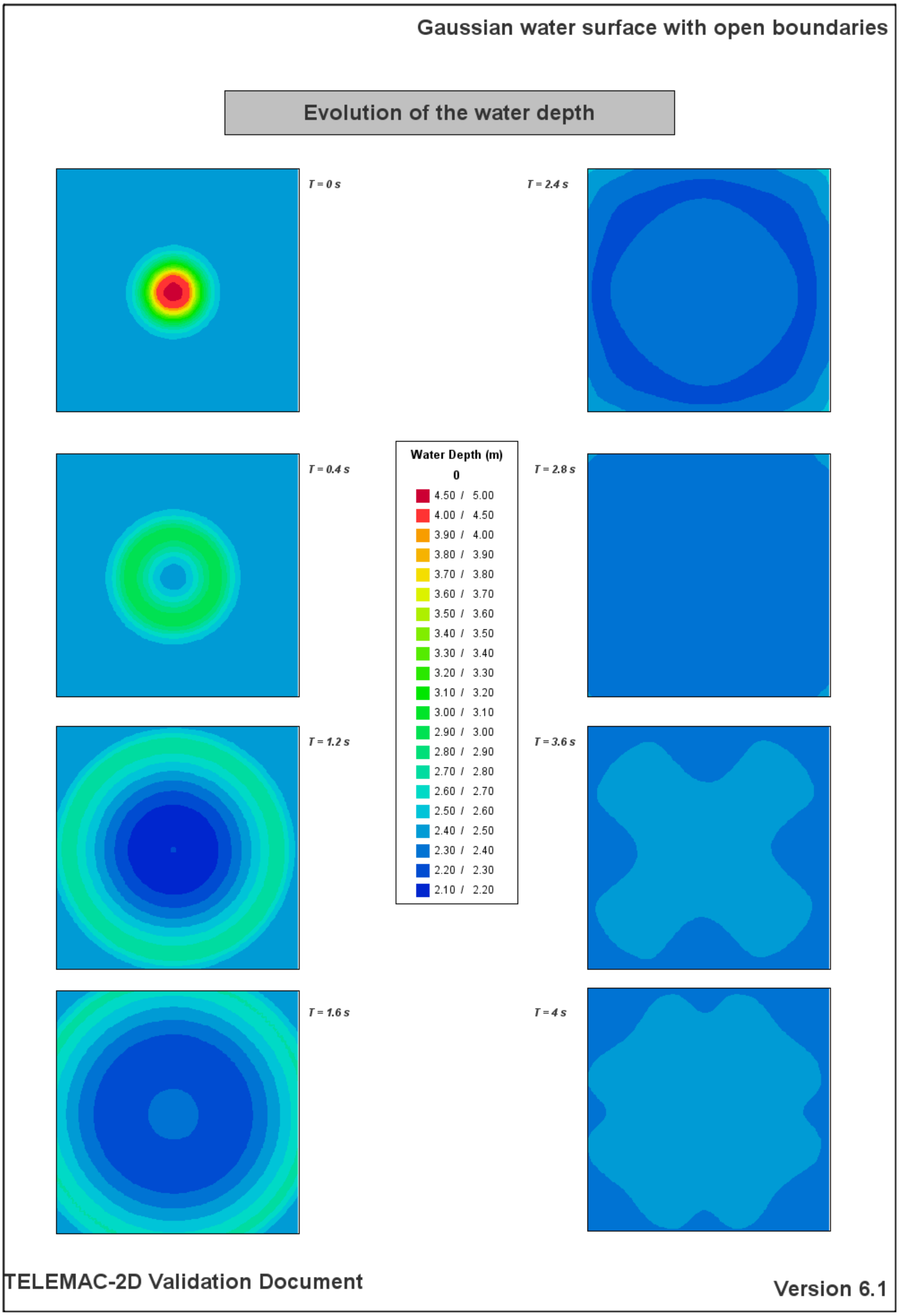


Figure 3.2.4 : Evolution of the water depth from 0.0 sec. to 4.0 sec

3.3. BUMP - SUBCRITICAL CONDITION

Title	**Flow over a bump on the bed - subcritical condition**
Initial study	**J.-M. Hervouet – March 1992**
Last update	**MARCH 2012**
Version	**TELEMAC-2D 6.1**

3.3.1. PURPOSE

To compare the solution produced by TELEMAC in a frictionless channel presenting an idealised bump on the bottom with the analytical solution to this problem.

The flow regime is sub-critical.

3.3.2. LINKED CLAIMS

- **Claim 2.2.1.7** TELEMAC-2D can be used to investigate the hydrodynamic impact of engineering works in rivers.
- **Claim 2.2.2.1** TELEMAC-2D can consider domains with complex geometries: changing bed slopes, teeth-shaped coastline, islands, flow contractions and expansions, etc.
- **Claim 2.2.2.3** TELEMAC-2D can compute steady and transient flow regimes. Steady flow is achieved by integrating through time until a steady condition is obtained.
- **Claim 2.2.2.12** TELEMAC-2D can be used to give a good representation of transcritical flows. Regime transitions consisting in critical flow conditions or hydraulic jumps are computed and located appropriately.

3.3.3. APPROACH

The channel is horizontal with a 4 metres long bump in its middle. The maximum elevation of the bump is 20 cm (see Figure 3.3.1).

<u>Geometry:</u>

– Size of the model: channel = 20.5 m x 2 m

– If 8 < x < 12 m, $z_f = -0.05(x-10)^2$,

– z_f = -0,20 m elsewhere

– Initial water depth: analytical solution

<u>Mesh:</u>

The mesh is regular, with a higher resolution in the middle of the channel. It is made up with quadrangles split into two triangles (see Figure 3.3.1).

- 1180 triangular elements
- 660 nodes
- Size of triangles ranges between 0.28 and 0.54 metres

Boundaries:

- Channel entrance: discharge Q = 8.86 m^3/s imposed
- (note that $q = \frac{Q}{B} = \sqrt{gh}$, B being the channel width)
- Channel outlet: water level y = 1.8 imposed. Therefore $h = y - z_f = 2$ m
- Lateral boundaries: Solid walls without friction

Bottom:

- No bottom friction

Turbulence:

- Constant viscosity equal to zero

Nota: instead of prescribing a zero viscosity, no diffusion step could have been used (keyword *DIFFUSION OF VELOCITY* prescribed to *NO*)

Algorithm:

- Type of advection:
  - Characteristics on velocities
  - centred semi-implicit scheme + SUPG decentring for depth
- Type of element:
  - quasi-bubble triangle for velocities
  - P1 triangle for h
- Implicitation coefficient for depth 0.6
- Implicitation coefficient for velocities = 0.6
- Accuracy = 10^{-6}

Time data:

- Time step = 0,01 sec.
- Simulation duration = 6 sec.

3.3.4. RESULTS

No friction on the bottom allows to write the Bernoulli equation between the entrance of the channel E (where $q_E = 4.43$ m^2/s and $h_E = 2$ m) and point A of abscissa x_A:

$$z_{fA} + h_A + \frac{q^2}{2gh_A^2} = z_{fE} + h_E + \frac{q^2}{2gh_E^2} = z_{fE} + 2.25m$$

Therefore, water depth h_A is given by: $h_A^3 + (z_{fA} - 2.05)h_A^2 + \frac{q^2}{2g} = 0$

TELEMAC-2D is run forward in time until a steady state flow is obtained. Numerical results are compared with the above analytical solution on Fig. 2.2. The computed Froude number is also compared with the analytical solution $F_A = \frac{q}{h_A \sqrt{gh_A}}$.

The solution produced by TELEMAC is in close agreement with the analytical solution as shown on Figure 3.3.2. The maximum drought in the water surface is reproduced perfectly. However, a small shift of this drought upstream of the bump maximum is observed.

3.3.5. CONCLUSIONS

This type of channel flow is driven by advection and pressure gradient terms. It is adequately reproduced by TELEMAC-2D in sub-critical flow regime.

3.3.6. STEERING FILE

```
/---------------------------------------------------------------------/
/    TELEMAC-2D  -  VALIDATION TEST CASE NUMBER 3  -  03/2012        /
/                            BUMP CASE 3                             /
/---------------------------------------------------------------------/
/
/----------------------------------------------
/  COMPUTER INFORMATIONS
/----------------------------------------------
/
FORTRAN FILE            = princi.f
GEOMETRY FILE           = geo
BOUNDARY CONDITIONS FILE = cli.txt
REFERENCE FILE          = ref
RESULTS FILE            = res
PARALLEL PROCESSORS     = 0
/
/----------------------------------------------
/  GENERAL INFORMATIONS - OUTPUTS
/----------------------------------------------
/
COMPUTATION CONTINUED                       = NO
TITLE                                       = 'BUMP CASE 3'
VARIABLES FOR GRAPHIC PRINTOUTS  = 'U,V,S,B,Q,F,H,N,O,R'
NUMBER OF PRIVATE ARRAYS                    = 4
GRAPHIC PRINTOUT PERIOD                     = 100
LISTING PRINTOUT PERIOD                     = 100
VALIDATION                                  = YES
TIME STEP                                   = 0.01
NUMBER OF TIME STEPS                        = 1000
INFORMATION ABOUT SOLVER                    = YES
MASS-BALANCE                                = YES
/
/----------------------------------------------
/  INITIAL CONDITIONS
```

```
/---------------------------------------------
/
INITIAL CONDITIONS                             = 'Particular'
/
/---------------------------------------------
/  BOUNDARY CONDITIONS
/---------------------------------------------
/
PRESCRIBED FLOWRATES                           = 0.;8.85889
PRESCRIBED ELEVATIONS                          = 1.8;0.
/
/---------------------------------------------
/  PHYSICAL PARAMETERS
/---------------------------------------------
/
LAW OF BOTTOM FRICTION                         = 0
FRICTION COEFFICIENT                           = 0
TURBULENCE MODEL                               = 1
VELOCITY DIFFUSIVITY                           = 0.
/
/---------------------------------------------
/  NUMERICAL PARAMETERS
/---------------------------------------------
/
TYPE OF ADVECTION                              = 1;5
SUPG OPTION                                    = 1;2
DISCRETIZATIONS IN SPACE                       = 13;11
SOLVER                                         = 7
SOLVER OPTION                                  = 3
SOLVER ACCURACY                                = 1.E-5
IMPLICITATION FOR DEPTH                        = 0.6
IMPLICITATION FOR VELOCITY                     = 0.6
/
MATRIX STORAGE : 1
TREATMENT OF THE LINEAR SYSTEM : 1
/
/---------------------------------------------
/ IN CASE OF USE OF FINITE VOLUME
/---------------------------------------------
/EQUATIONS                          = 'SAINT-VENANT VF'
/FINITE VOLUME SCHEME               = 6
/VARIABLE TIME-STEP                 = YES
/DESIRED COURANT NUMBER             = 0.8
/DURATION                           = 10.
/---------------------------------------------

&ETA
```

3.3.7. FIGURES

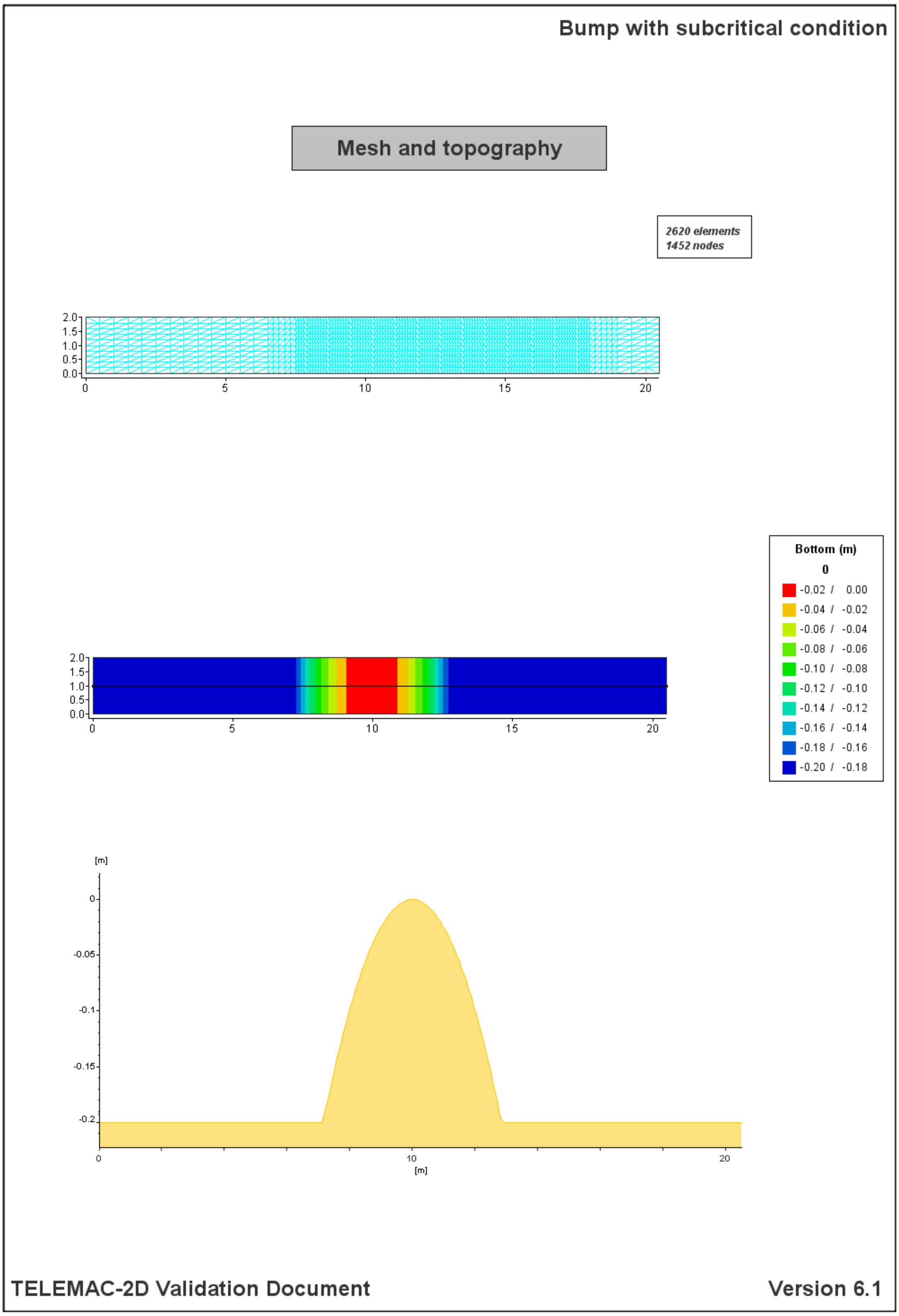


Figure 3.3.1 : Mesh and topography.

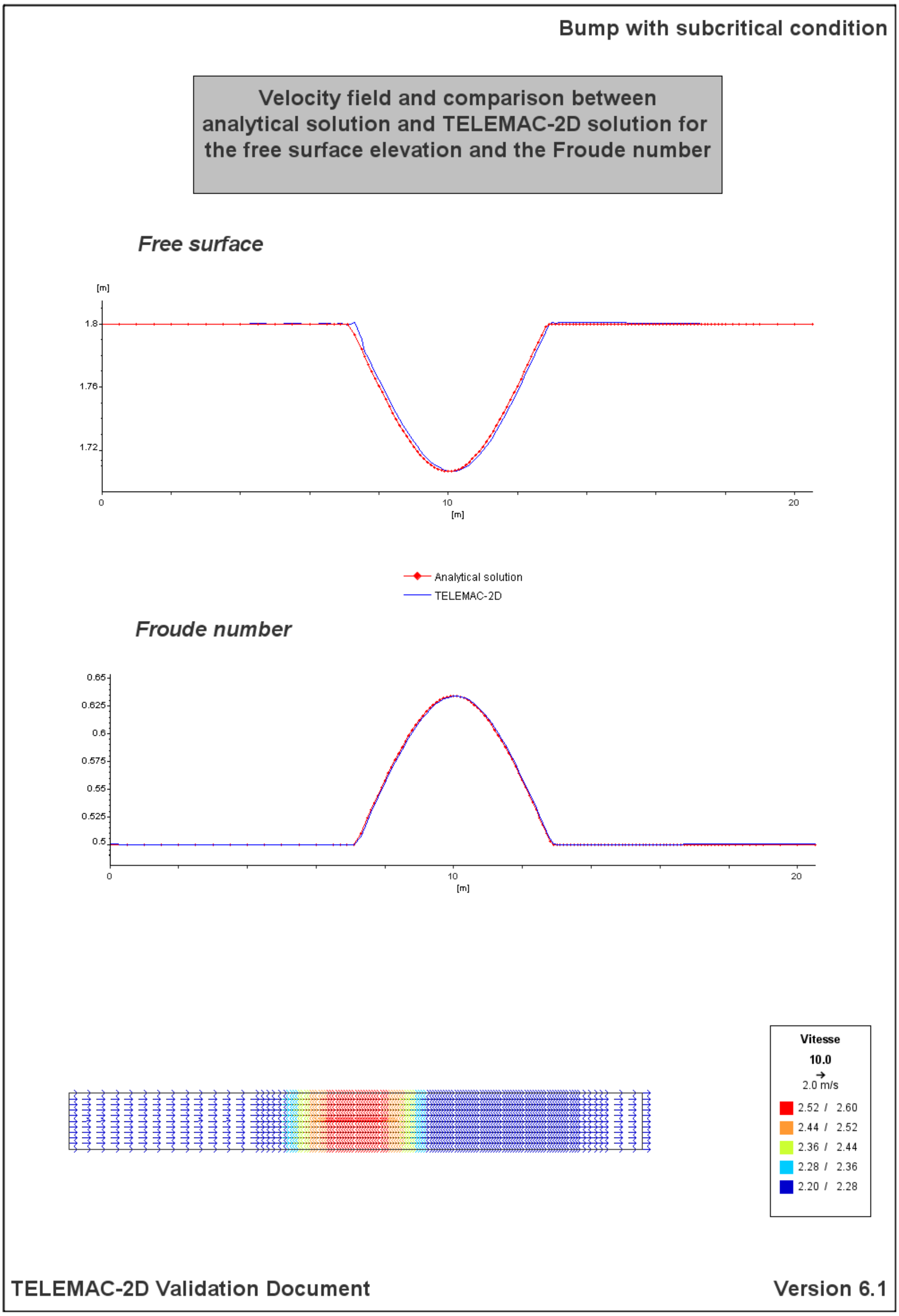
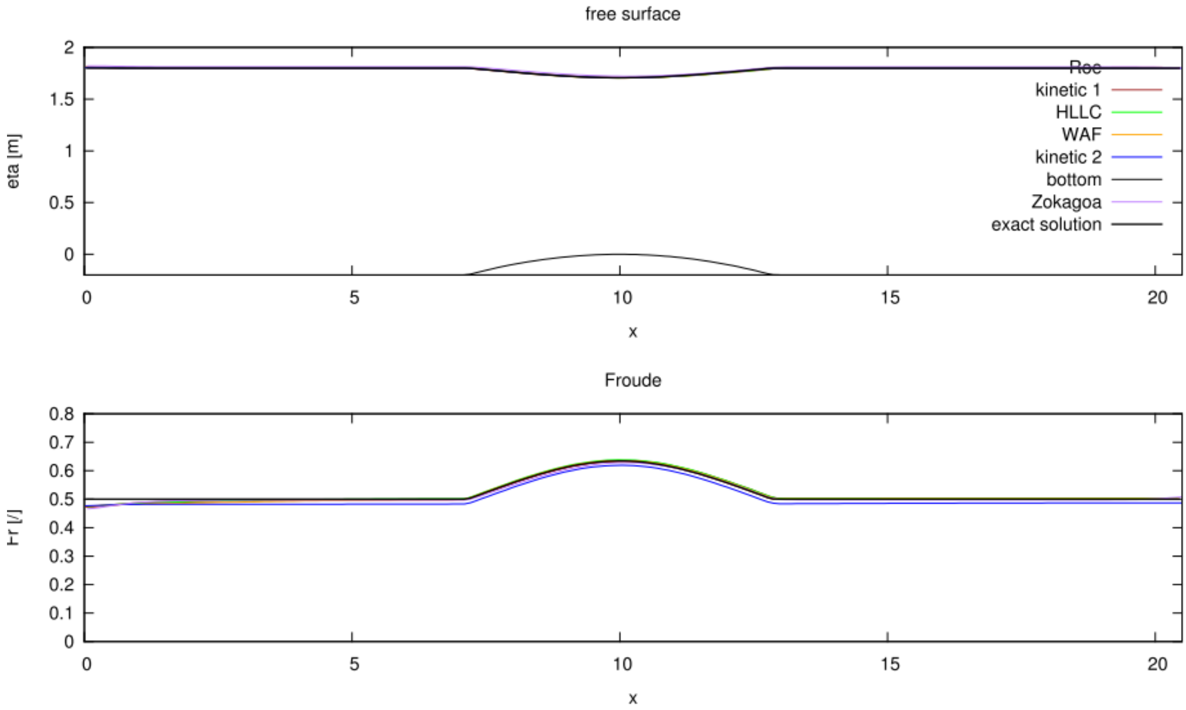


Figure 3.3.2 : Velocity field and comparison between analytical solution and TELEMAC-2D solution for the free surface elevation and the Froude number



Flow over a bump: subcritical conditions – Finite volumes results

3.4. BUMP - TRANSCRITICAL CONDITION

Title	**Flow over a bump on the bed – Super-critical condition**
Initial study	**J.-M. Hervouet – April 1992**
Last update	**December 2011**
Version	**TELEMAC-2D 6.1**

3.4.1. PURPOSE

To compare the solution produced by TELEMAC in a frictionless channel presenting an idealised bump on the bottom with the analytical solution to this problem.

Flow conditions are such that the bump generates a transition from sub-critical to super-critical flow.

3.4.2. LINKED CLAIMS

- **Claim 2.2.1.7** TELEMAC-2D can be used to investigate the hydrodynamic impact of engineering works in rivers.
- **Claim 2.2.2.1** TELEMAC-2D can consider domains with complex geometries: changing bed slopes, teeth-shaped coastline, islands, flow contractions and expansions, etc.
- **Claim 2.2.2.3** TELEMAC-2D can compute steady and transient flow regimes. Steady flow is achieved by integrating through time until a steady condition is obtained.
- **Claim 2.2.2.12** TELEMAC-2D can be used to give a good representation of transcritical flows. Regime transitions consisting in critical flow conditions or hydraulic jumps are computed and located appropriately.

3.4.3. APPROACH

The channel is horizontal with a 4 metres long bump in its middle. The maximum elevation of the bump is 20 cm (see Figure 3.4.1).

Geometry:

- Size of the model: channel = 20.5 m x 2 m
- If 8 < x < 12 m, $z_f = -0.05(x-10)^2$,
- z_f = -0,20 m elsewhere
- Initial water depth: analytical solution

Mesh:

The mesh is regular, with a higher resolution in the middle of the channel. It is made up with quadrangles split into two triangles (see Figure 3.4.1).

- 2620 triangular elements
- 1452 nodes
- Size of triangles ranges between 0.3 and 0.5 metres

Boundaries:

- Channel entrance: discharge Q = 0.6 m^3/s imposed
- Channel outlet: liquid boundary with free velocity and free water level
- Lateral boundaries: Solid walls without friction

Bottom:

- No bottom friction

Turbulence:

- Constant viscosity equal to zero

Nota: instead of prescribing a zero viscosity, no diffusion step could have been used (keyword *DIFFUSION OF VELOCITY* prescribed to *NO*)

Algorithm:

- Type of advection:
  - Characteristics on velocities
  - centred semi-implicit scheme + SUPG decentring for depth
- Type of element:
  - P1 triangle for h and for velocity
- Implicitation for depth = 0.6
- Implicitation for velocity = 0.6
- Solver: Conjugate gradient
- SUPG option:
  - upwinding equal to 1 for velocity
  - upwinding equal to courant number for H
- Accuracy = 10^{-5}

Time data:

- Time step = 0,03 sec.
- Simulation duration = 30 sec.

3.4.4. RESULTS

The critical depth is $h_c = \left(\frac{q^2}{g}\right)^{1/3} = 0.20934m$. This value is obtained above the bump. The corresponding specific energy is $E_c = h_c + \frac{q^2}{2gh_c^2} = \frac{3}{2}h_c$.

Writing the Bernoulli equation between the bump and any point A located in the channel, it comes:

$$z_A + h_{fA} + \frac{q^2}{2gh_A^2} = z_c + E_c$$

$$\text{i.e.: } h_A^3 + (z_A - z_c - E_c)h_A^2 + \frac{q^2}{2g} = 0.$$

At the foot of the bump, positive solutions are $h_{upstream} = 0.49535m$ and $h_{downstream} = 0.106m$. These solutions are the upstream sub-critical and downstream super-critical water depths. Corresponding free surface elevations are $y_{upstream}$ = 0.29535 m and $y_{downstream}$ = - 0.094 m.

TELEMAC-2D is run forward in time until a steady state flow is obtained. The simulation reproduces very well these different free surface elevations and the overall analytical solution, as shown on Figure 3.4.2.

3.4.5. CONCLUSIONS

This transition from a sub-critical to a super-critical flow regime over a bump in a channel without friction is adequately reproduced by TELEMAC-2D, provided the mesh resolution is fine enough. This transition occurs at critical depth as announced by channel flow hydraulics.

3.4.6. STEERING FILE

```
/---------------------------------------------------------------------/
/    TELEMAC-2D  -  VALIDATION TEST CASE NUMBER 4  -  04/2012        /
/                              BUMP CASE 4                           /
/---------------------------------------------------------------------/
/----------------------------------------------
/  COMPUTER INFORMATIONS
/----------------------------------------------
/
FORTRAN FILE            = princi.f
GEOMETRY FILE           = geo
BOUNDARY CONDITIONS FILE = cli.txt
REFERENCE FILE          = ref
RESULTS FILE            = res
PARALLEL PROCESSORS     = 0
/
/----------------------------------------------
/  GENERAL INFORMATIONS - OUTPUTS
/----------------------------------------------
/
COMPUTATION CONTINUED                        = NO
TITLE                                        = 'BUMP CASE 4'
VARIABLES FOR GRAPHIC PRINTOUTS  = 'U,V,S,B,Q,F,H,N,O,R,O2,R2'
```

```
NUMBER OF PRIVATE ARRAYS                          = 6
GRAPHIC PRINTOUT PERIOD                           = 50
LISTING PRINTOUT PERIOD                           = 100
VALIDATION                                        = YES
TIME STEP                                         = 0.03
NUMBER OF TIME STEPS                              = 1000
INFORMATION ABOUT SOLVER                          = YES
MASS-BALANCE                                      = YES
/
/-----------------------------------------------
/  INITIAL CONDITIONS
/-----------------------------------------------
/
INITIAL CONDITIONS       = 'PARTICULAR'
/
/-----------------------------------------------
/  BOUNDARY CONDITIONS
/-----------------------------------------------
/
PRESCRIBED FLOWRATES                              = 0.;6.
/-----------------------------------------------
/  PHYSICAL PARAMETERS
/-----------------------------------------------
/
LAW OF BOTTOM FRICTION                            = 0
FRICTION COEFFICIENT                              = 0.
TURBULENCE MODEL                                  = 1
VELOCITY DIFFUSIVITY                              = 0.
/
/-----------------------------------------------
/  NUMERICAL PARAMETERS
/-----------------------------------------------
/
TYPE OF ADVECTION                                 = 1;5
SUPG OPTION                                       = 1;2
DISCRETIZATIONS IN SPACE                          = 13;11
SOLVER ACCURACY                                   = 1.E-5
IMPLICITATION FOR DEPTH                           = 0.6
IMPLICITATION FOR VELOCITY                        = 0.6
SOLVER                                            = 7
SOLVER OPTION                                     = 3
/
MATRIX STORAGE : 1
TREATMENT OF THE LINEAR SYSTEM : 1
/
/-----------------------------------------------
/ IN CASE OF USE OF FINITE VOLUME
/-----------------------------------------------
/EQUATIONS                          = 'SAINT-VENANT VF'
/FINITE VOLUME SCHEME               = 6
/VARIABLE TIME-STEP                 = YES
/DESIRED COURANT NUMBER             = 0.8
/DURATION                           = 10.
/-----------------------------------------------

&ETA
/----------------------------------------------------------------------
```

3.4.7. FIGURES

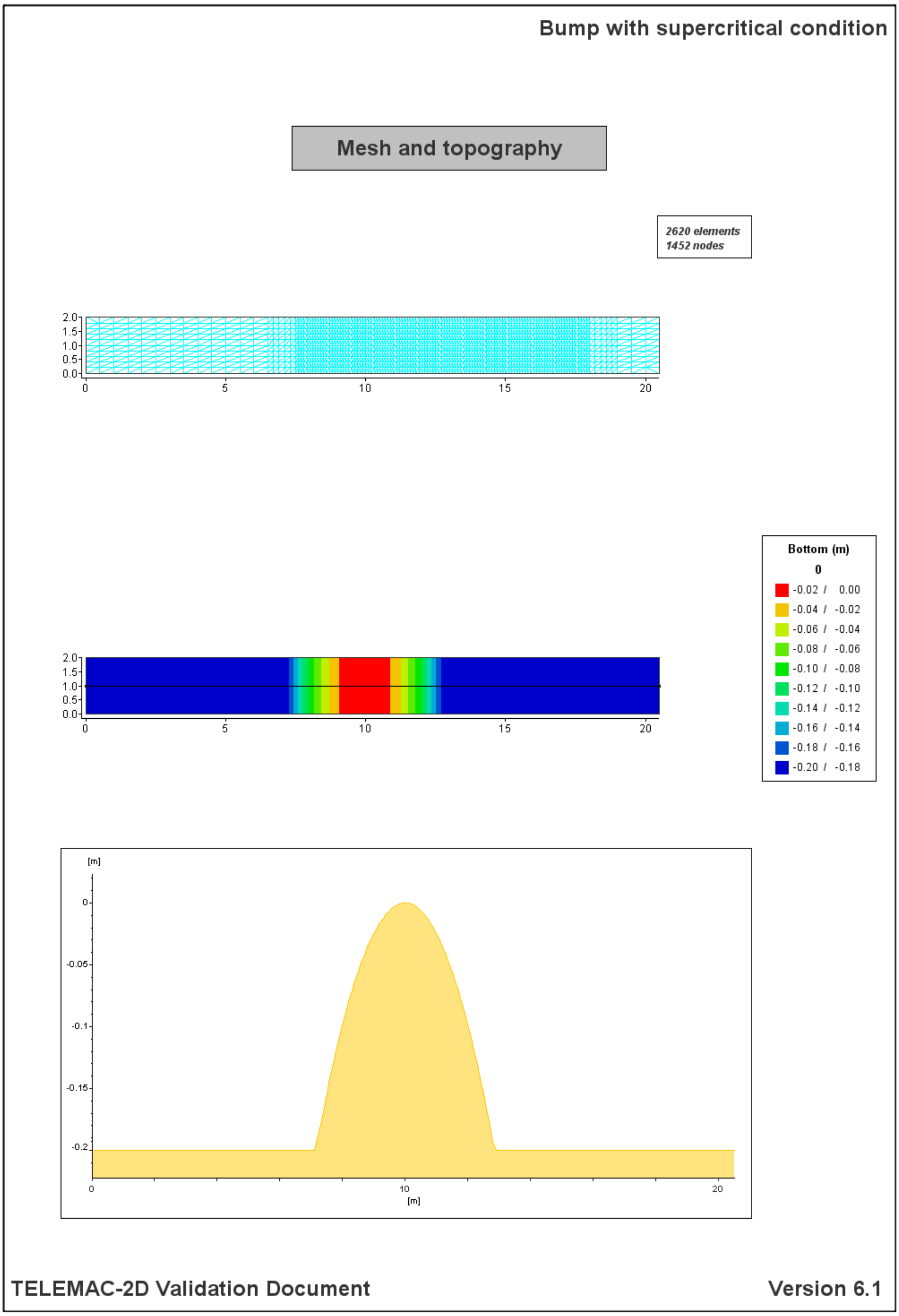


Figure 3.4.1 : Mesh and topography.

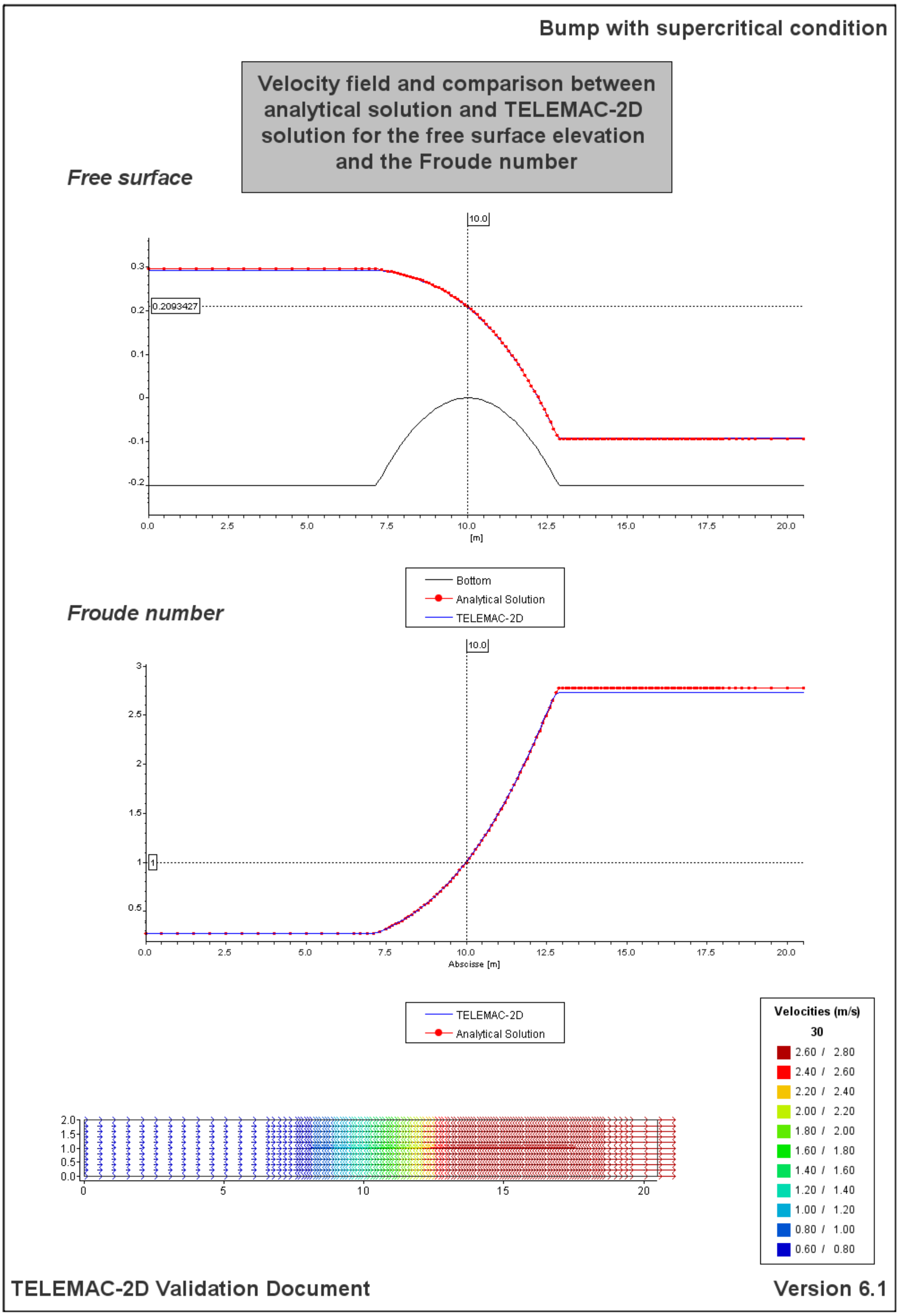
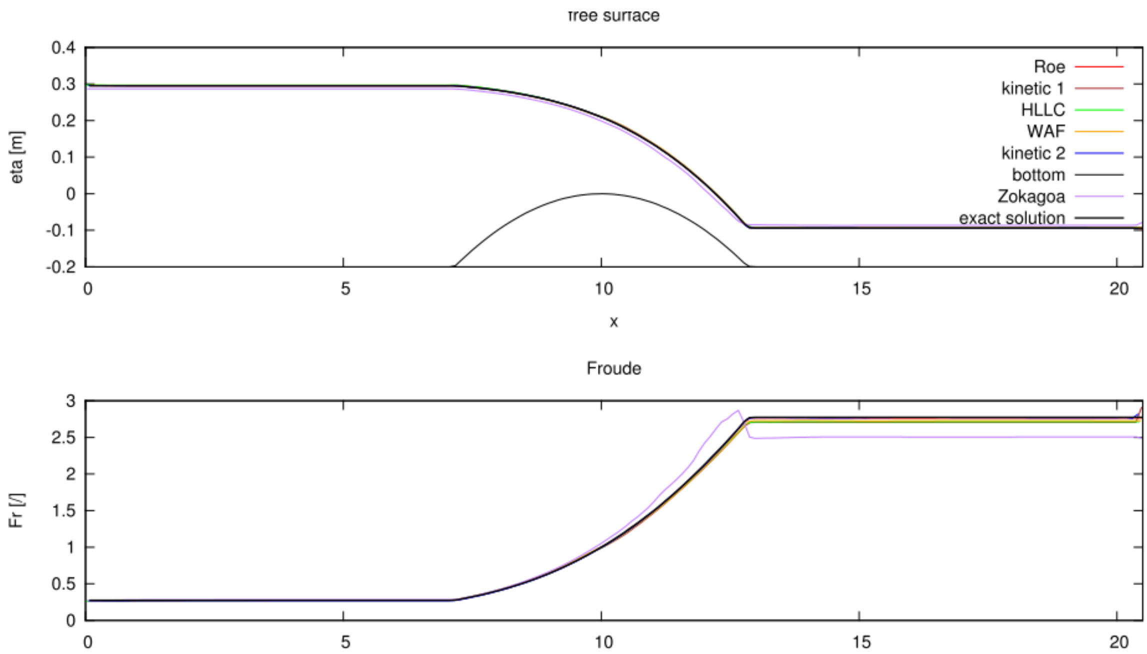


Figure 3.4.2 : Velocity field and comparison between analytical solution and TELEMAC-2D solution for the free surface elevation and the Froude number



Transcritical flow over a bump – Finite volume results.

3.5. BUMP - TRANSCRITICAL CONDITION WITH HYDRAULIC JUMP

Title	**Flow over a bump on the bed – Super-critical condition with hydraulic jump**
Initial study	**J.-M. Hervouet – April 1992**
Last update	**April 2012**
Version	**TELEMAC-2D 6.1**

3.5.1. PURPOSE

To compare the solution produced by TELEMAC in a channel with friction presenting an idealised bump on the bottom with the analytical solution to this problem.

The flow regime is sub-critical upstream; a transition to super-critical flow conditions is located on the bump, and the downstream water level imposes the presence of an hydraulic jump in the downstream reach of the channel.

3.5.2. LINKED CLAIMS

- **Claim 2.2.1.7** TELEMAC-2D can be used to investigate the hydrodynamic impact of engineering works in rivers.
- **Claim 2.2.2.1** TELEMAC-2D can consider domains with complex geometries: changing bed slopes, teeth-shaped coastline, islands, flow contractions and expansions, etc.
- **Claim 2.2.2.3** TELEMAC-2D can compute steady and transient flow regimes. Steady flow is achieved by integrating through time until a steady condition is obtained.
- **Claim 2.2.2.12** TELEMAC-2D can be used to give a good representation of transcritical flows. Regime transitions consisting in critical flow conditions or hydraulic jumps are computed and located appropriately.
- **Claim 2.4.2.10** With appropriate choice of the mesh resolution, TELEMAC-2D produces solutions without wiggles (i.e. oscillations in space at a given time or oscillations in time at a given position).

3.5.3. APPROACH

The channel is horizontal with a 4 metres long bump in its middle. The maximum elevation of the bump is 20 cm (see Figure 3.5.1).

Geometry:

- Size of the model: channel = 20.5 m x 2 m
- If 8 < x < 12 m, $z_f = -0.05(x-10)^2$,
- z_f = -0,20 m elsewhere

- Initial water depth: analytical solution

Mesh:

The mesh is regular, with a higher resolution in the middle of the channel. It is made up with quadrangles split into two triangles (see Figure 3.5.1).

- 2620 triangular elements
- 1452 nodes
- Maximum triangle size is 0.25 metres

Boundaries:

- Channel entrance: discharge Q = 2 m^3/s imposed
- Channel outlet: h = 0.4 m imposed
- Lateral boundaries: Solid walls without friction

Bottom:

- Strickler formula with friction coefficient = 40

Turbulence:

- Constant viscosity equal to zero

Nota: instead of prescribing a zero viscosity, no diffusion step could have been used (keyword *DIFFUSION OF VELOCITY* prescribed to *NO*)

Algorithm:

- Type of advection:
  - Characteristics on velocities
  - Conservative + modified SUPG on depth
- Type of element:
  - Quasi-bubble triangle for velocities
  - P1 triangle for h and for velocity
- Implicitation for depth and for velocity = 0.6
- SUPG option:
  - decentring equal to 1 for velocity
  - decentring equal to courant number for H
- Accuracy = 10^{-5}

Time data:

- Time step = 0,01 sec.
- Simulation duration = 50 sec.

3.5.4. RESULTS

As in the previous test case 3.2, the sub-critical flow upstream the bump and the transition are correctly reproduced.

The prescribed downstream elevation defines a backwater line which takes into account roughness effects on the downstream reach. The super-critical flow issuing from the downstream foot of the bump also generates a water line. The hydraulic jump should be located in a position where the water depths of these two water lines are conjugate, i.e. they are related by the hydraulic jump equation:

$$\frac{h_2}{h_1} = \frac{1}{2}(\sqrt{1+8Fr_1^2} - 1),$$

where index 1 relates to the position upstream the jump and index 2 relates to the position downstream the jump. Let us notice that it is necessary to have bottom friction to have a well-posed problem in this test case. In the contrary, the water lines issuing from the foot of the bump and from the downstream end respectively would be parallel to the bed, and the jump could not occur in the downstream reach.

The hydraulic jump computed by TELEMAC occurs in a position where $h_1 = 0.18\,m$, $h_2 = 0.41m$ and $Fr_1 = 1.64$ (see Figure 3.5.2). These values satisfy with more or less accuracy the hydraulic jump formula.

In addition, the followings other results are obtained:

- The output flowrate is equal to the input flowrate
- The Froude number is equal to 1 at the position of the bump.

3.5.5. CONCLUSIONS

The transition from a super-critical to a sub-critical flow regime through a hydraulic jump is properly reproduced by TELEMAC-2D, provided the mesh resolution is fine enough. The hydraulic jump formula is satisfied and the jump is located in the right position.

3.5.6. STEERING FILE

```
/----------------------------------------------------------------------
/ TELEMAC2D Version v6p1 5 dec. 2011
/ Validation test case 5
/----------------------------------------------------------------------

/----------------------------------------------------------------------
/ EQUATIONS
/----------------------------------------------------------------------

FRICTION COEFFICIENT   =40.
LAW OF BOTTOM FRICTION =3
VELOCITY DIFFUSIVITY   =0.

/----------------------------------------------------------------------
/ EQUATIONS, BOUNDARY CONDITIONS
/----------------------------------------------------------------------

PRESCRIBED FLOWRATES  =0.;2.
PRESCRIBED ELEVATIONS =0.4;0.

/----------------------------------------------------------------------
/ EQUATIONS, INITIAL CONDITIONS
/----------------------------------------------------------------------

INITIAL ELEVATION  =0.4
INITIAL CONDITIONS ='CONSTANT ELEVATION'
```

```
/---------------------------------------------------------------------
/ INPUT-OUTPUT, FILES
/---------------------------------------------------------------------

FORTRAN FILE             ='princi.f'
GEOMETRY FILE            ='geo'
REFERENCE FILE           ='ref'
STEERING FILE            ='cas.txt'
BOUNDARY CONDITIONS FILE ='cli.txt'
RESULTS FILE             ='res'

/---------------------------------------------------------------------
/ INPUT-OUTPUT, GRAPHICS AND LISTING
/---------------------------------------------------------------------

LISTING PRINTOUT PERIOD         =500
VARIABLES FOR GRAPHIC PRINTOUTS ='U,V,S,B,Q,F,H,N,O,R,O2,R2'
INFORMATION ABOUT SOLVER        =YES
MASS-BALANCE                    =YES
GRAPHIC PRINTOUT PERIOD         =500

/---------------------------------------------------------------------
/ INPUT-OUTPUT, INFORMATION
/---------------------------------------------------------------------

VALIDATION =YES
TITLE      ='Validation test case 5'

/---------------------------------------------------------------------
/ NUMERICAL PARAMETERS
/---------------------------------------------------------------------

NUMBER OF PRIVATE ARRAYS       =6
NUMBER OF TIME STEPS           =2500
TREATMENT OF THE LINEAR SYSTEM =2
SUPG OPTION                    =0;2;2;2
TIME STEP                      =0.02

/---------------------------------------------------------------------
/ NUMERICAL PARAMETERS, SOLVER
/---------------------------------------------------------------------
SOLVER          =1
SOLVER ACCURACY =1.E-5
/---------------------------------------------------------------------
/ NUMERICAL PARAMETERS, VELOCITY-CELERITY-HIGHT
/---------------------------------------------------------------------
IMPLICITATION FOR DEPTH    =0.6
IMPLICITATION FOR VELOCITY =0.6
/----------------------------------------------
/ IN CASE OF USE OF FINITE VOLUME
/----------------------------------------------
/EQUATIONS                         = 'SAINT-VENANT VF'
/FINITE VOLUME SCHEME              = 6
/VARIABLE TIME-STEP                = YES
/DESIRED COURANT NUMBER            = 0.8
/DURATION                          = 10.
/----------------------------------------------
```

3.5.7. FIGURES

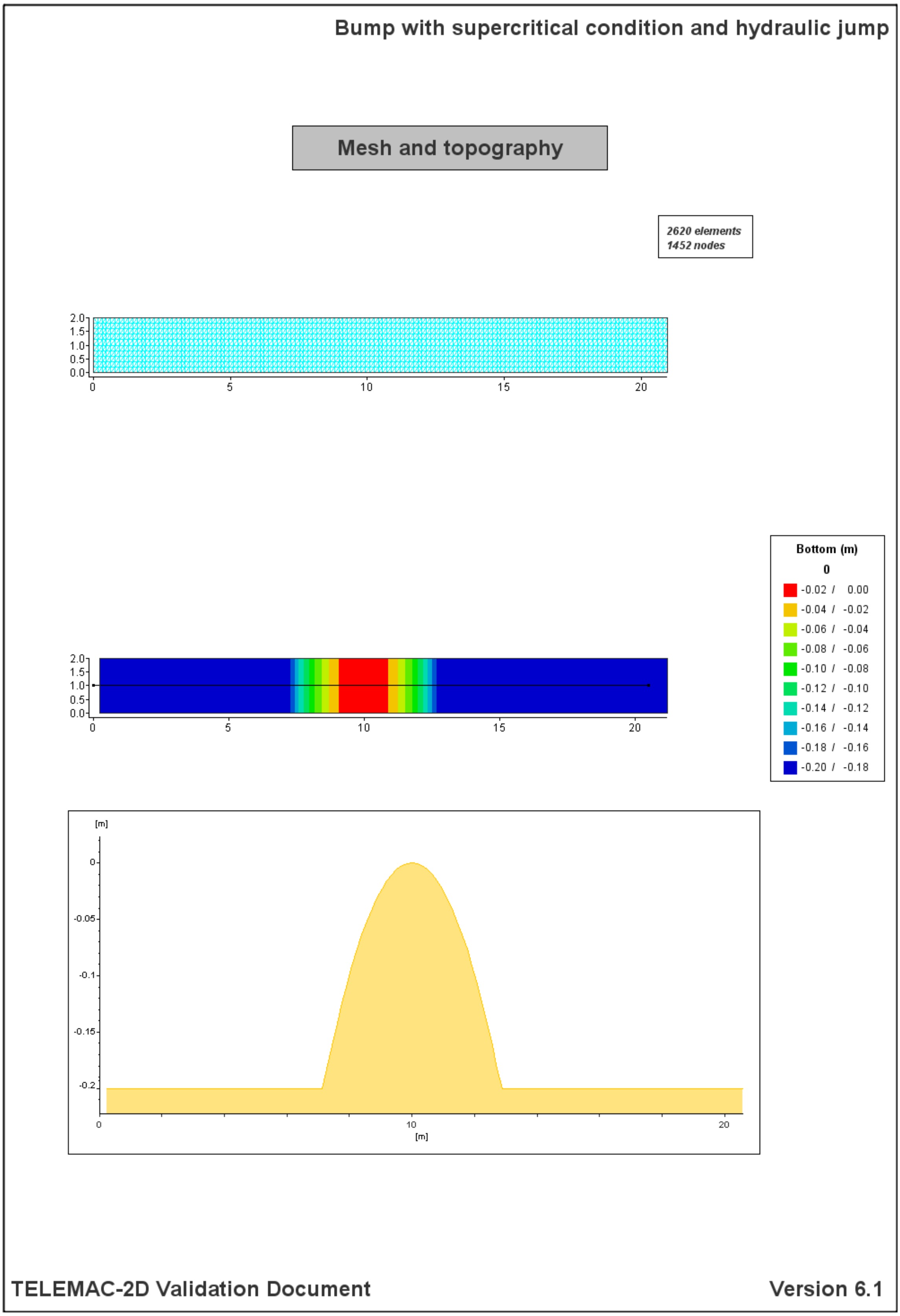


Figure 3.5.1 : Mesh and topography.

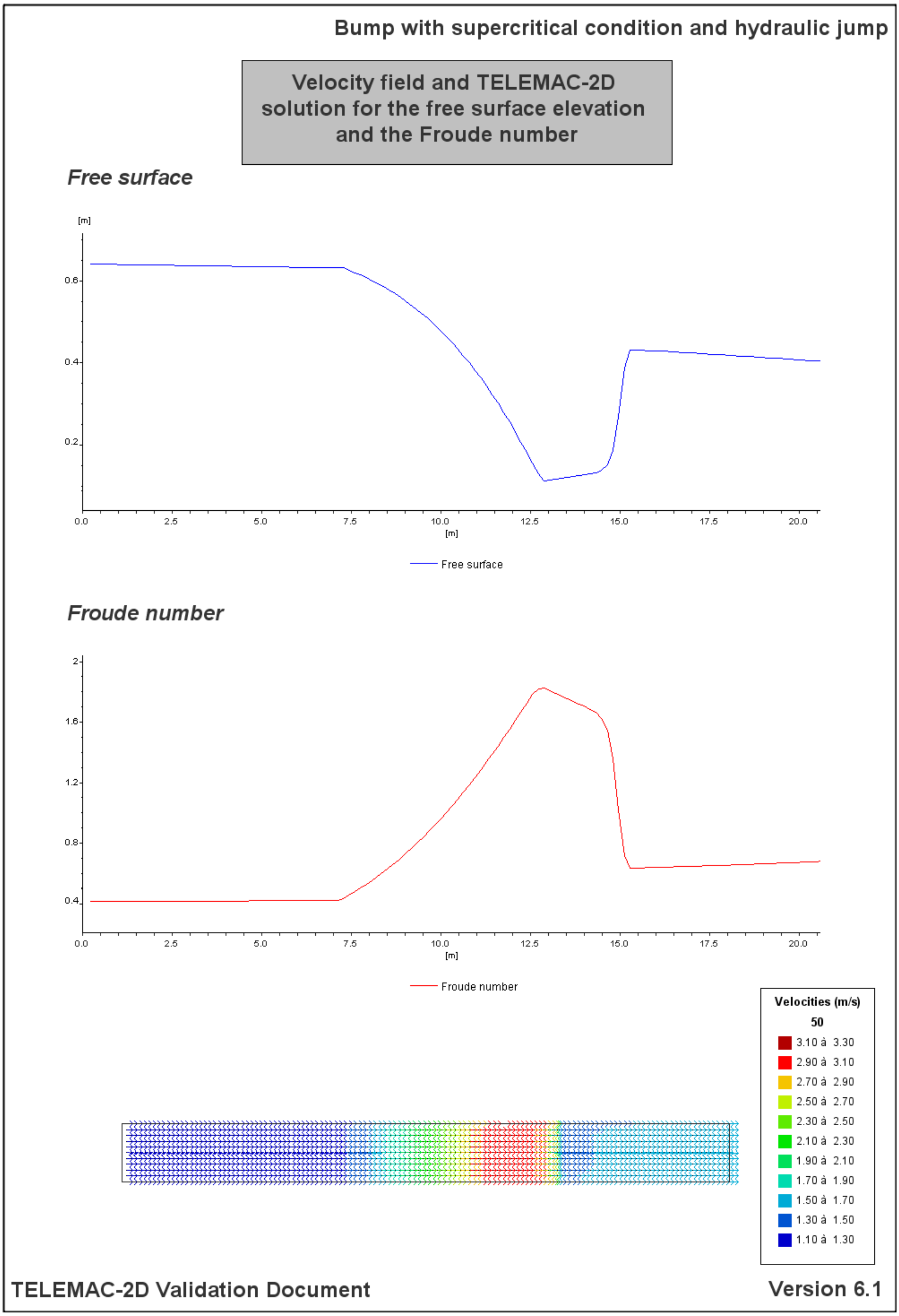
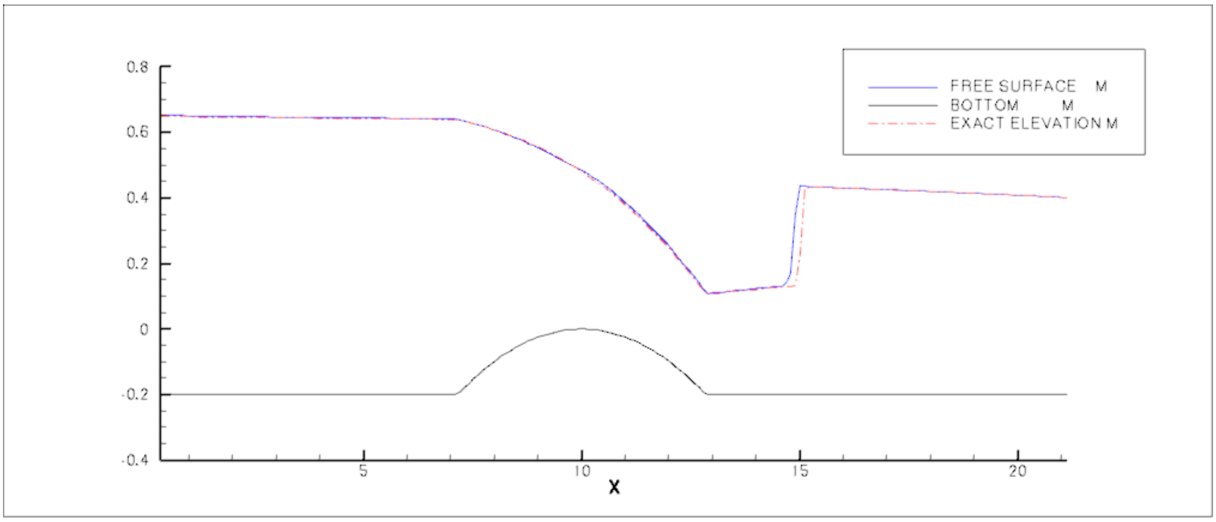


Figure 3.5.2 : Velocity field and TELEMAC-2D solution for the free surface elevation and the Froude number.



Transcritic flow over a bump with hydraulic jump – Finite volume (WAF Scheme) result

3.6. UNCOVERING OF A BEACH

Title	**Uncovering of a cylindrical beach**
Initial study	**J.-M. Hervouet – April 1992**
Last update	**December 2011**
Version	**TELEMAC-2D 6.1**

3.6.1. PURPOSE

To demonstrate that TELEMAC-2D is capable of representing the drying of a cylindrical beach presenting a pool.

Also to show that TELEMAC works properly when disconnected water bodies exist within the study domain.

3.6.2. LINKED CLAIMS

- **Claim 2.2.1.2** TELEMAC-2D can be used for an accurate prediction and for the analysis of tidal currents and water surface elevations in coastal or estuarial zones due to long period tidal wave forcing, with covering/uncovering flats.
- **Claim 2.2.2.16** TELEMAC-2D can represent the wetting and drying of intertidal zones (maritime applications) and of inundated plains (river applications). The numerical treatment of this process is very robust.

3.6.3. APPROACH

A grid made of squares split into two triangles is used to represent a cylindrical beach presenting a pool. The water initially covers the entire domain. As the tide falls by 0,60 m, a 46 m cross-shore length of the beach is uncovered, thus leaving the pool full of water at rest.

Geometry:

- Size of the model: channel = 46 m x 9 m
- Initial water level = 0 m

Mesh:

The mesh is regular. It is made up with squares split into two triangles.

- 828 triangular elements
- 470 nodes
- Maximum size range: √2 metres

Boundaries:

- Shoreline: solid wall with slip condition

- Offshore boundary: Q = - 5 H / 0.6 imposed
- Lateral boundaries: solid walls with slip condition in the channel

Bottom:

- Defined by: $z_B = -0.6 + 0.01x + 0.1e^{\frac{-(x-19)^2}{20}}$
- Strickler friction formula with friction coefficient = 40
- The mesh and topography are shown on Figure 3.6.1.

Turbulence:

- Constant viscosity = 10^{-2} m^2/s

Algorithm:

- Type of advection:
  - Characteristics on velocities
  - Centred Semi-implicit scheme, SUPG decentring
- Solver accuracy = 10^{-4}
- Implicitation for depth and for velocity : 0.6

Time data:

- Time step = 1 sec.
- Simulation duration = 300 sec.

3.6.4. RESULTS

The water level decreases regularly over the beach. At the end of the calculation, the water depth is equal to zero upstream and downstream of the pool, and the water level in the pool is completely horizontal (see Figure 3.6.2).

3.6.5. CONCLUSIONS

The uncovering of tidal flats is properly represented by TELEMAC-2D.

3.6.6. STEERING FILE

```
/---------------------------------------------------------------------
/ TELEMAC2D Version v6p1 May 5. 2012
/ Validation test case 6
/---------------------------------------------------------------------

/---------------------------------------------------------------------
/ INPUT-OUTPUT FILES
/---------------------------------------------------------------------

RESULTATS FILES            ='res'
BOUNDARY CONDITIONS FILE  ='cli.txt'
REFERENCE FILE             ='ref'
FORTRAN FILE               ='princi.f'
GEOMETRY FILE              ='geo'
```

```
/---------------------------------------------------------------------
/ INPUT-OUTPUT
/---------------------------------------------------------------------

VALIDATION = YES
TITRE      = 'Validation test case 6'
LISTING PRINTOUT PERIOD         =100
VARIABLES FOR GRAPHIC PRINTOUTS ='U,V,H,B,S'
GRAPHIC PRINTOUT PERIOD         =100
MASS-BALANCE                    =OUI

/---------------------------------------------------------------------
/ EQUATIONS
/---------------------------------------------------------------------

FRICTION COEFFICIENT            =40.
VELOCITY DIFFUSIVITY            =1.E-2
LOW OF BOTTOM FRICTION          =3

/---------------------------------------------------------------------
/ BOUNDARY CONDITIONS
/---------------------------------------------------------------------

PRESCRIBED ELEVATIONS =0.
/---------------------------------------------------------------------
/ NUMERICAL PARAMETERS
/---------------------------------------------------------------------
SUPG OPTION                =1;2;2;2
NUMBER OF TIME STEPS       =300
TIME STEP                  =1.
IMPLICITATION FOR DEPTH    =0.6
IMPLICITATION FOR VELOCITY =0.6

/---------------------------------------------------------------------
/ SOLVER
/---------------------------------------------------------------------
MAXIMUM NUMBER OF ITERATIONS FOR SOLVER =100
/---------------------------------------------------------------------
/-----------------------------------------------
/ IN CASE OF USE OF FINITE VOLUME
/-----------------------------------------------
/EQUATIONS                           = 'SAINT-VENANT VF'
/FINITE VOLUME SCHEME                = 6
/VARIABLE TIME-STEP                  = YES
/DESIRED COURANT NUMBER              = 0.8
/DURATION                            = 300.
/-----------------------------------------------
```

3.6.7. FIGURES

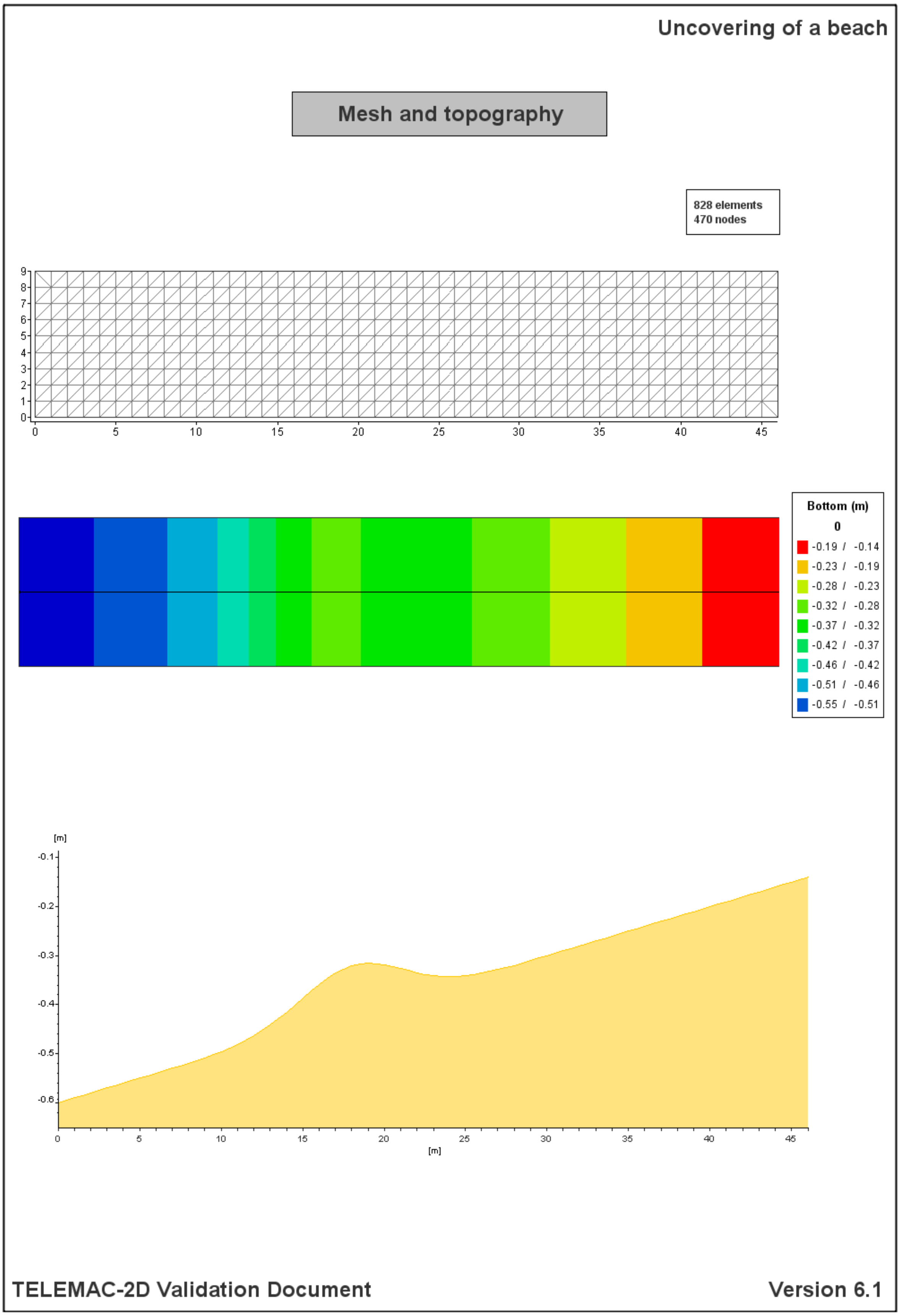


Figure 3.6.1 : Mesh and topography.

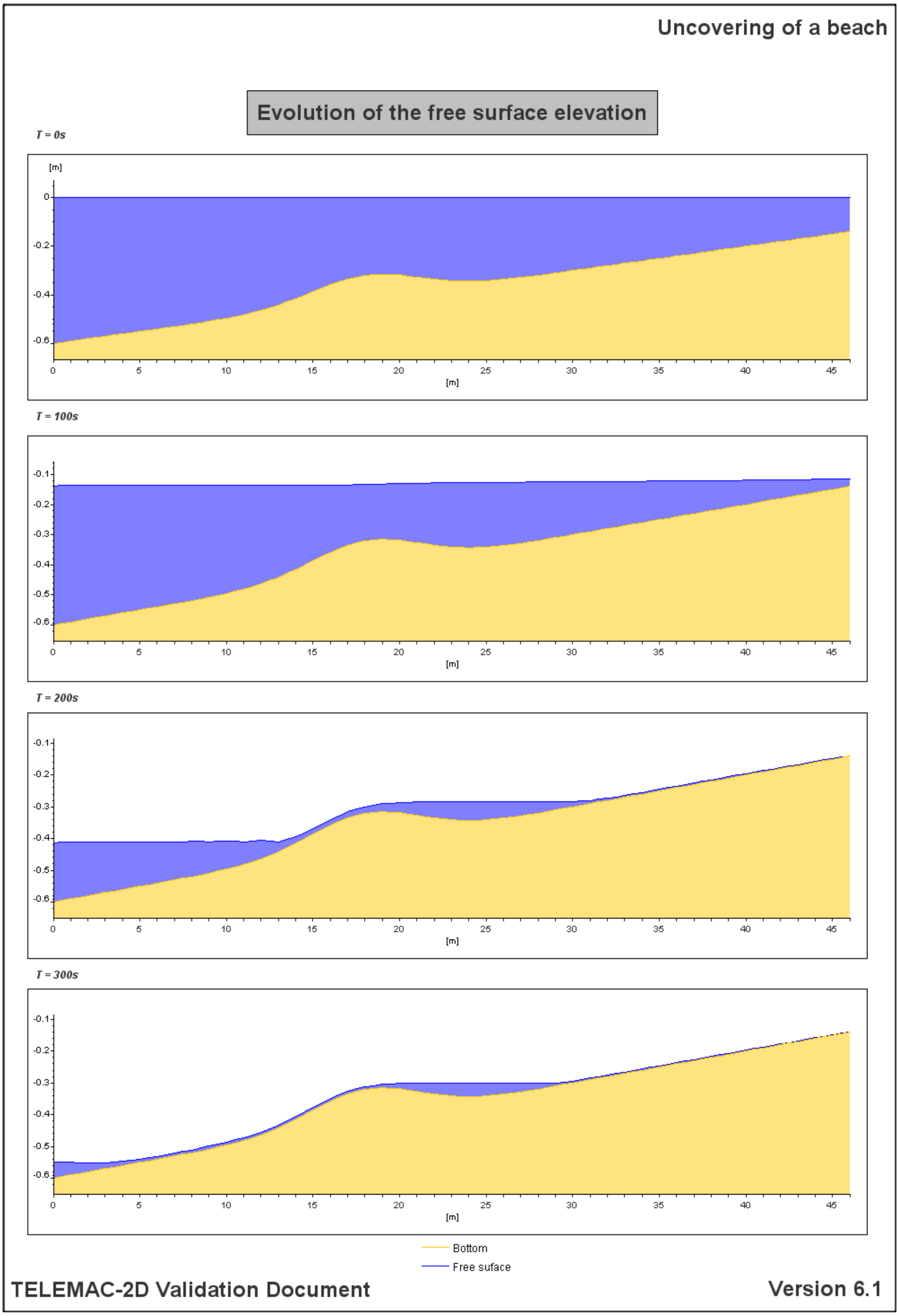


Figure 3.6.2 : Evolution of the free surface elevation in time.

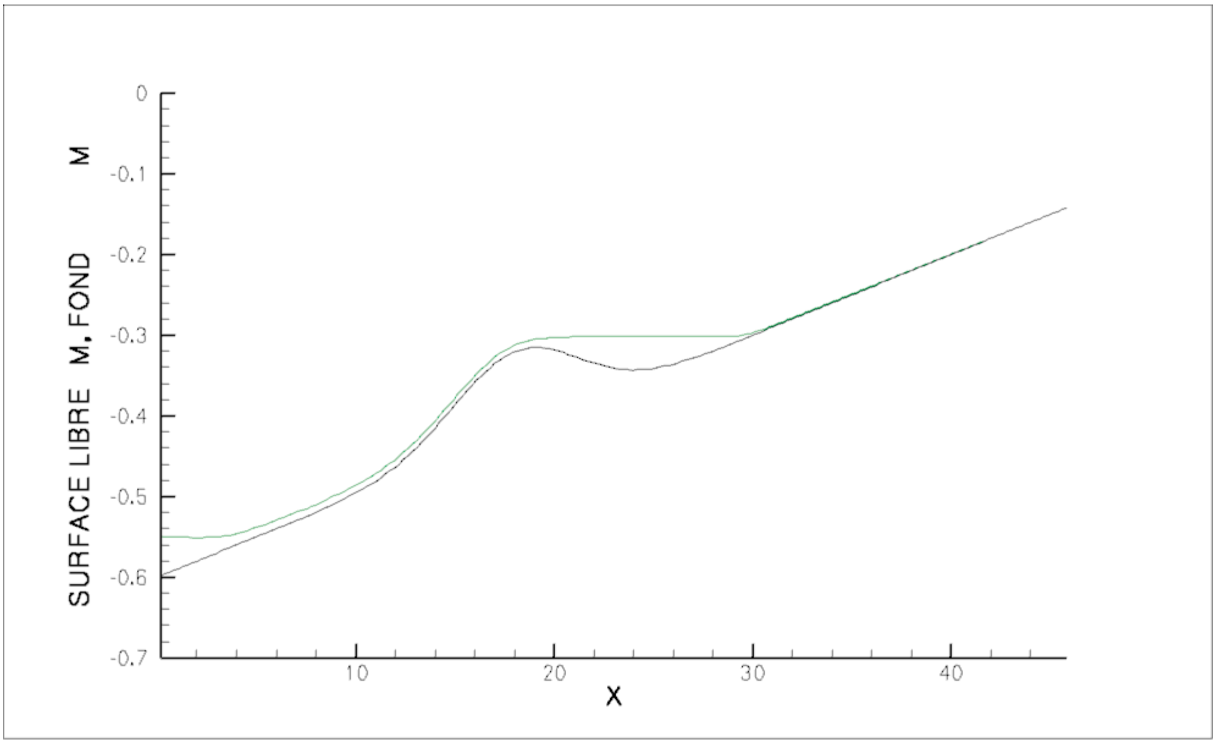


Figure 3.6.3 : Finite volume (WAF scheme) result for validation case 6

3.7. PROPAGATION OF A SURFACE WAVE

Title	**Propagation of a surface wave in a linear channel**
Initial study	**A.Masson – March 1999**
Last update	**December 2011**
Version	**TELEMAC-2D 6.1**

3.7.1. PURPOSE

To assess the properties of TELEMAC-2D for the propagation of a long wave in a rectilinear channel without resistance effects.

3.7.2. LINKED CLAIMS

- **Claim 2.2.1.2** TELEMAC-2D can be used for an accurate prediction and for the analysis of tidal currents and water surface elevations in coastal or estuarial zones due to long period tidal wave forcing, with covering/uncovering flats.
- **Claim 2.2.2.6** TELEMAC-2D can simulate the propagation of long waves;
- **Claim 2.4.2.1** The numerical methods used in TELEMAC-2D can provide an accurate solution of the SWE.
- **Claim 2.4.2.16** Boundary condition values can be prescribed exactly or “suggested” in order to deal with under or over-stressed problems (in the cases where too much or insufficient information is given on the domain open boundaries). This option should not be used blindly however.
- **Claim 2.4.2.19** TELEMAC-2D propagates surface waves at the celerity given by the shallow water theory.

3.7.3. APPROACH

A 16 metres long and 0.3 metres wide channel with constant depth of 10 m is fed by a constant flow. At the channel entrance, a sinusoïdal water surface elevation is imposed. No bed resistance occurs and the advection step of TELEMAC-2D is skipped in order to examine pure propagation effects.

Geometry:

- Size of the model: channel = 16 m x 0.3 m
  - Horizontal bottom
  - Water depth at rest = 10 m

Mesh:

The mesh is regular. It is made up with squares split into two triangles (see Figure 3.7.1).

- 3840 triangular elements

- 2247 nodes
- Maximum size range: 0.07 metres

Boundaries:

- Channel entrance: incident water wave imposed such as : $H = 10 + 0.05 \sin\left(\frac{2\pi t}{0.25}\right)$

Where t is time (sec.) and H is the water depth at the channel entrance (m).

- Channel outlet: free surface elevation
- Thompson scheme for open boundaries (see keyword `OPTION FOR LIQUID BOUNDARIES`)
- Lateral boundaries: solid walls with slip condition

Bottom:

- No bottom friction.

Turbulence:

- Constant viscosity = 10^{-6} m^2/s

Algorithm:

- Simulation:
  - with propagation
  - without advection
- Type of element:
  - quasi-bubble triangle for velocities
- GMRES solver
- Accuracy = 10^{-6}

Time data:

- Time step = 0,0025 sec.
- Simulation duration = 5 sec.

3.7.4. RESULTS

The solution produced by TELEMAC-2D shows very good agreement with the exact solution (see Figure 3.7.2). The incident water wave is properly propagated in the channel.

The celerity of the wave is exactly $c = \sqrt{g.H_0}$. The phase of the wave is correct. The amplitude of the wave is nearly the same at the channel entrance and at the outlet. A very small difference between the surface elevation computed by TELEMAC-2D and the exact solution is observed. The maximum error is lower than 3 % on the amplitude of the wave.

There is no reflection of the wave on the open boundary at the outlet of the channel thanks to the Thompson scheme.

3.7.5. CONCLUSIONS

TELEMAC reproduces accurately the propagation of surface long waves in terms of celerity, phase and amplitude.

3.7.6. STEERING FILE

```
/----------------------------------------------------------------------
/ TELEMAC2D Version v6p1 5 déc. 2011
/ Validation test case 7
/----------------------------------------------------------------------

/----------------------------------------------------------------------
/ EQUATIONS
/----------------------------------------------------------------------

DIFFUSION OF VELOCITY =NO
ADVECTION             =NO

/----------------------------------------------------------------------
/ EQUATIONS, BOUNDARY CONDITIONS
/----------------------------------------------------------------------

PRESCRIBED ELEVATIONS        =0.;10.
OPTION FOR LIQUID BOUNDARIES =2;2

/----------------------------------------------------------------------
/ EQUATIONS, INITIAL CONDITIONS
/----------------------------------------------------------------------

INITIAL ELEVATION  =10.
INITIAL CONDITIONS ='CONSTANT ELEVATION'

/----------------------------------------------------------------------
/ INPUT-OUTPUT, FILES
/----------------------------------------------------------------------

FORTRAN FILE             ='princi.f'
GEOMETRY FILE            ='geo'
REFERENCE FILE           ='ref'
STEERING FILE            ='cas.txt'
BOUNDARY CONDITIONS FILE ='cli.txt'
RESULTS FILE             ='res'

/----------------------------------------------------------------------
/ INPUT-OUTPUT, GRAPHICS AND LISTING
/----------------------------------------------------------------------

LISTING PRINTOUT PERIOD         =100
VARIABLES FOR GRAPHIC PRINTOUTS ='U,V,S,B,H,L,C,N'
GRAPHIC PRINTOUT PERIOD         =100

/----------------------------------------------------------------------
/ INPUT-OUTPUT, INFORMATION
/----------------------------------------------------------------------

VALIDATION =YES
TITLE      ='Validation test case 7'
```

```
/----------------------------------------------------------------------
/ NUMERICAL PARAMETERS
/----------------------------------------------------------------------

TIME STEP                =0.0025
NUMBER OF PRIVATE ARRAYS =1
NUMBER OF TIME STEPS     =2000

/----------------------------------------------------------------------
/ NUMERICAL PARAMETERS, SOLVER
/----------------------------------------------------------------------

SOLVER                                   =7
SOLVER ACCURACY                          =1.E-6
MAXIMUM NUMBER OF ITERATIONS FOR SOLVER =200

/----------------------------------------------------------------------
/ NUMERICAL PARAMETERS, VELOCITY-CELERITY-HIGHT
/----------------------------------------------------------------------

IMPLICITATION FOR VELOCITY =0.6
INITIAL GUESS FOR H        =2

/-----------------------------------------------
/ IN CASE OF USE OF FINITE VOLUME
/-----------------------------------------------
/EQUATIONS                          = 'SAINT-VENANT VF'
/FINITE VOLUME SCHEME               = 6
/VARIABLE TIME-STEP                 = YES
/DESIRED COURANT NUMBER             = 0.8
/DURATION                           = 5.
/-----------------------------------------------
```

3.7.7. FIGURES

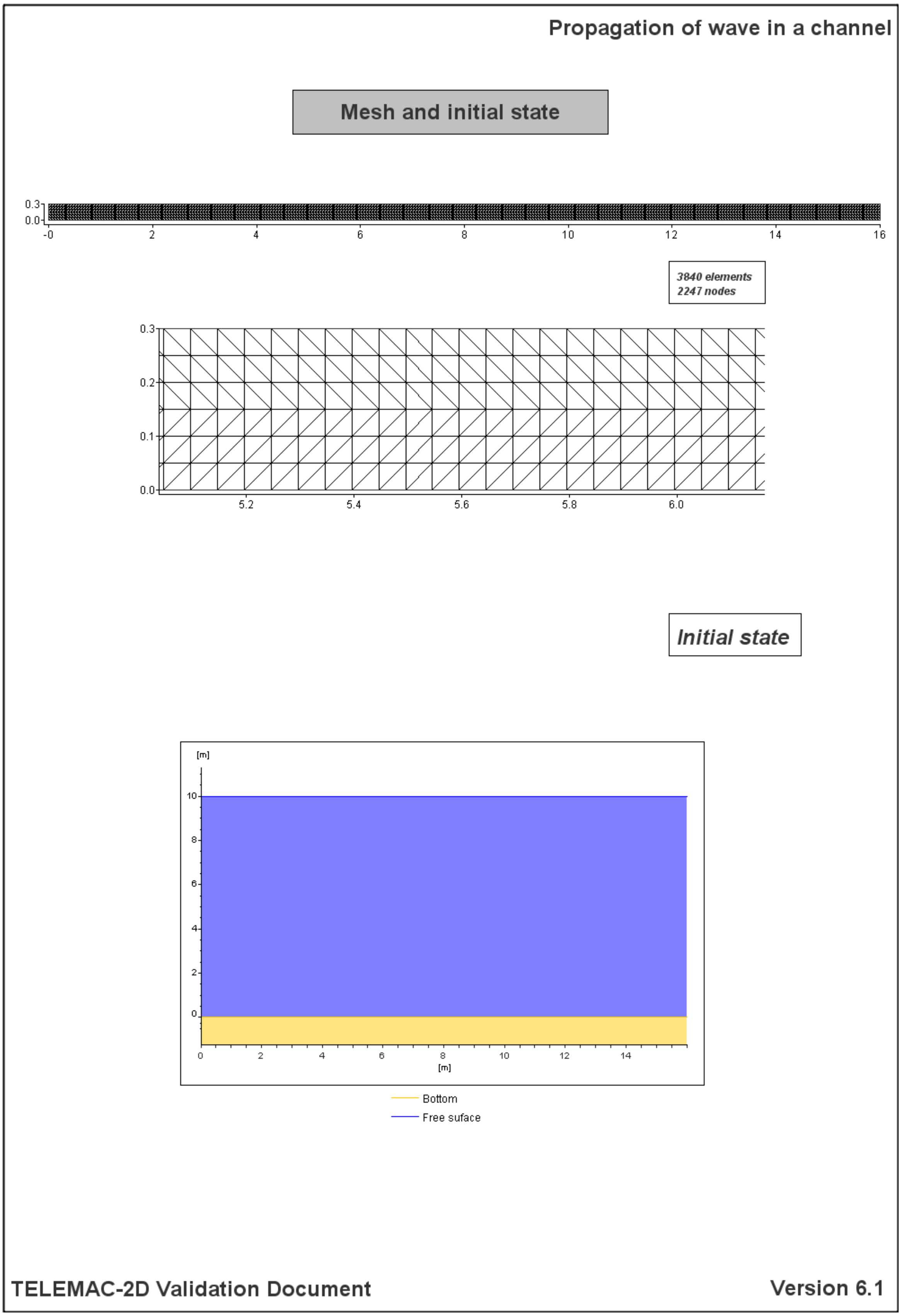


Figure 3.7.1 : Mesh and initial state.

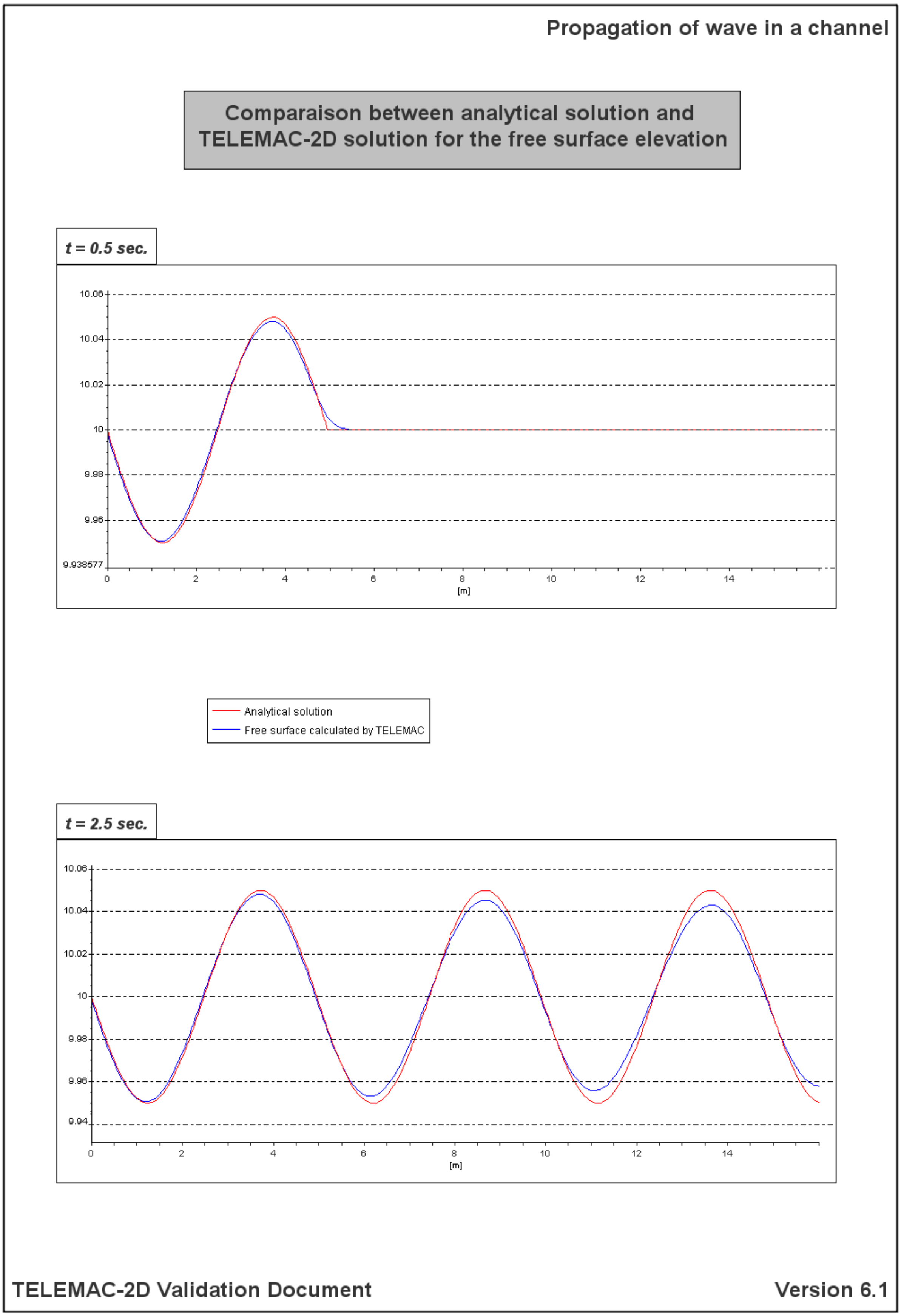


Figure 3.7.2 : Comparison between analytical solution and TELEMAC-2D solution for the free surface elevation.

3.8. FLOW AROUND BRIDGE PIERS

Title	**Karman vortices behind two cross-flow cylindrical piers**
Initial study	**J.-M. Hervouet – March 1992**
Last update	**October 2011**
Version	**TELEMAC-2D 6.1**

3.8.1. PURPOSE

To study the impact of a channel flow around obstacles. Also to demonstrate that TELEMAC-2D can simulate flows with unsteady eddies even with steady state boundary conditions.

3.8.2. LINKED CLAIMS

- **Claim 2.2.1.7** TELEMAC-2D can be used to investigate the hydrodynamic impact of engineering works in rivers.
- **Claim 2.3.2.3** The different treatments of turbulence in TELEMAC-2D are satisfactory for many free surface flow problems in rivers, estuaries and coastal waters.
- **Claim 2.4.2.18** TELEMAC-2D can represent eddy formation and shedding behind structure.

3.8.3. APPROACH

A 20 m wide prismatic channel with trapezoidal cross-section contains bridge-like obstacles in one cross-section made of two abutments and two circular 4 m diameter piles. The flow resulting from steady state boundary conditions is studied.

Geometry:

- Size of the model: channel = 28.5 m x 20 m
- Zero elevation

Mesh:

The mesh is symmetric and built with 4 obstacles: 2 piers, and 2 abutments (see Figure 3.8.1).

- 4304 triangular elements
- 2280 nodes

Boundaries:

- Channel entrance: Q = 62 m^3/s imposed

- Channel outlet: h = 0 m imposed
- Lateral boundaries: solid walls with slip condition in the channel

Bottom:

- Strickler formula with friction coefficient = 40
- Bed isolines are shown on Figure 3.8.1.

Turbulence:

- Model of constant viscosity equal to 0.005 m^2/s

Algorithm:

- Type of advection:
  - characteristics on velocities
  - conservative + modified SUPG on depth
- Type of element:
  - quasi-bubble triangle for velocities
  - P1 triangle for h
- Direct solver
- Mass lumping on depth = 1
- Implicitation for velocity and for depth = 0.6

Time data:

- Time step = 0,1 sec.
- Simulation duration = 80 sec.

3.8.4. RESULTS

The obstacles create a contraction of the streamlines, and Karman vortices are observed behind the piers. The Karman vortices produce an asymmetry of the velocity field. This velocity field is unsteady behind the piers (in the Karman vortices) even though everywhere else the flow is steady (see Figure 3.8.2).

3.8.5. CONCLUSIONS

TELEMAC can be used to study the hydrodynamic impact of engineering works (like bridge piers), and to analysed unsteady flow, such as the Karman vortices.

3.8.6. STEERING FILE

```
/---------------------------------------------------------------------
/ TELEMAC2D Version v6p1 5 may. 2011
/ Validation test case 8
/---------------------------------------------------------------------

/---------------------------------------------------------------------
/ INPUT-OUTPUTS
/---------------------------------------------------------------------

FORTRAN FILE            ='princi.f'
```

```
GEOMETRY FILE             ='geo'
REFERENCE FILE            ='ref'
BOUNDARY CONDITIONS FILE ='cli.txt'
RESULTS FILE              ='res'
/---------------------------------------------------------------------
VALIDATION  = YES
TITLE       = 'Validation test case 8'
/---------------------------------------------------------------------
LISTING PRINTOUT PERIOD         =100
VARIABLES FOR GRAPHIC PRINTOUTS ='U,V,S,B,H,L,C,N'
GRAPHIC PRINTOUT PERIOD         =10
MASS-BALANCE                    =YES

/---------------------------------------------------------------------
/ EQUATIONS
/---------------------------------------------------------------------

FRICTION COEFFICIENT    =40.
VELOCITY DIFFUSIVITY    =0.005
BOTTOM FRICTION LOW     =3

/---------------------------------------------------------------------
/ NUMERICAL PARAMETERS
/---------------------------------------------------------------------

TREATMENT OF THE LINEAR SYSTEM =2
TIME STEP                      =0.4
SUPG OPTION                    =1;2;2;2
NUMBER OF TIME STEPS           =750
CONTINUITY CORRECTION          =YES
DISCRETISATION IN SPACE        =12;11;11;11

/---------------------------------------------------------------------
/ SOLVER
/---------------------------------------------------------------------

SOLVER          =1
SOLVER ACCURACY =1.E-5

/---------------------------------------------------------------------
/ NUMERCIAL PARMETERS
/---------------------------------------------------------------------

MASS-LUMPING ON VELOCITY   =1.
MASS-LUMPING ON DEPTH      =1.
IMPLICITATION FOR DEPTH    =0.6
IMPLICITATION FOR VELOCITY =0.6
```

3.8.7. FIGURES

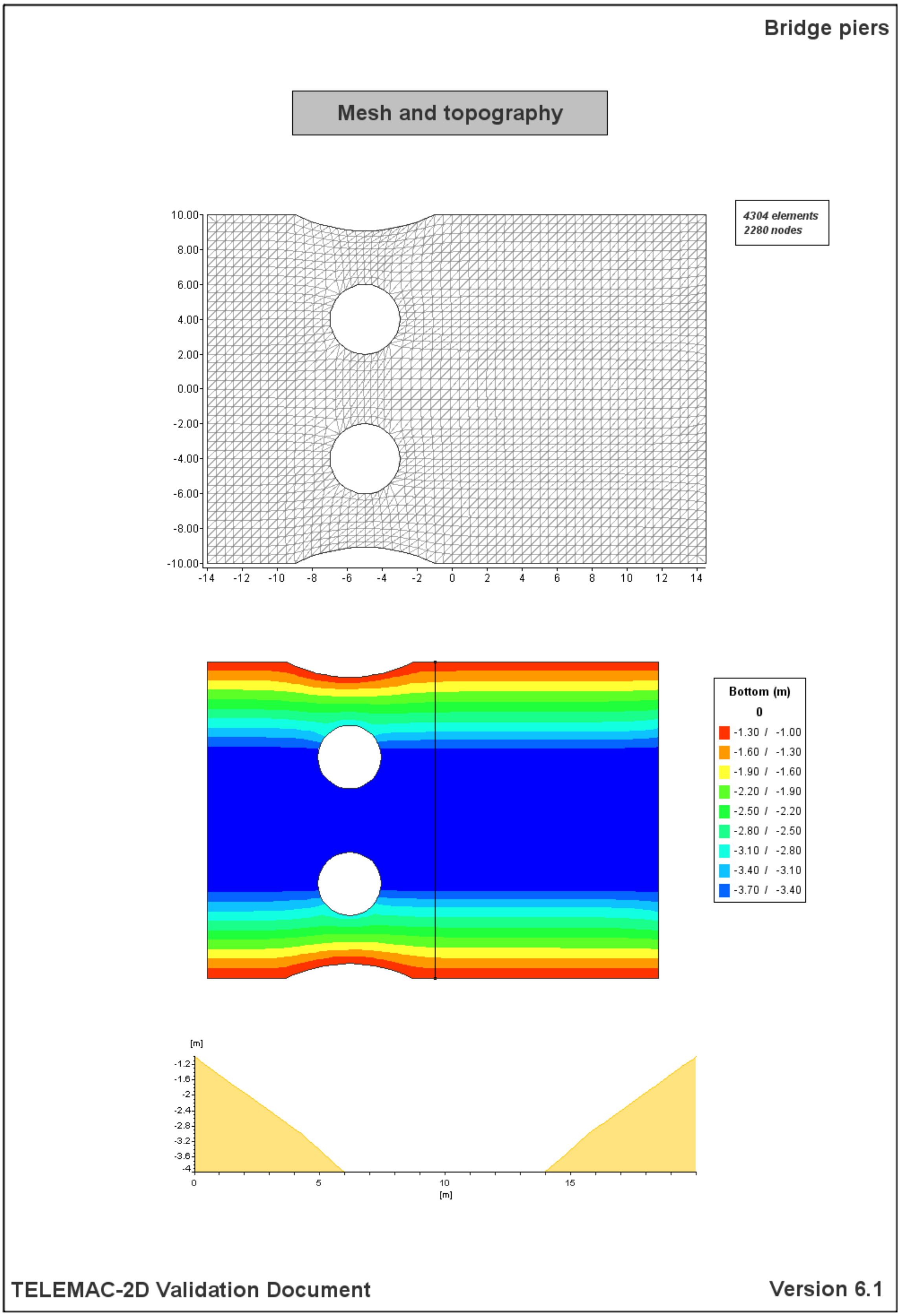


Figure 3.8.1 : Mesh and topography.

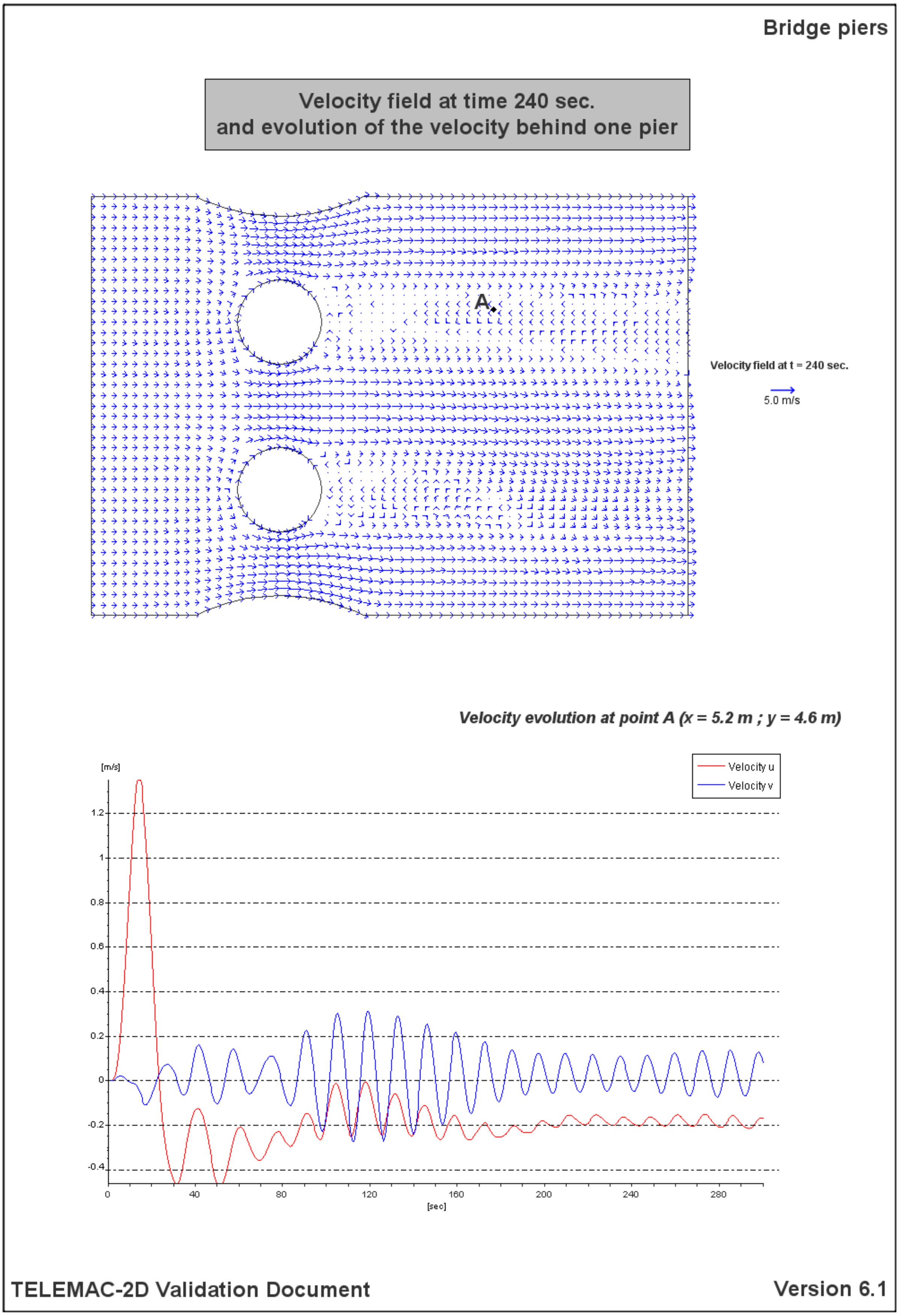


Figure 3.8.2 : Velocity field at time 240 sec. and evolution of the velocity behind one pier

3.9. SILLS IN A CHANNEL TREATED AS SINGULARITIES

Title	**Flow over three successive sills in a rectangular channel treated as singularities**
Initial study	**V. Guinot – July 1996**
Last update	**December 2011**
Version	**TELEMAC-2D 6.0**

3.9.1. PURPOSE

This study tests the hydraulic behaviour of TELEMAC-2D when representing sills in a channel as “internal singularities”. This functionality was introduced in TELEMAC-2D in order to avoid the multiplication of computational nodes and associated reduction in time step when a sill is represented thanks to variations in the bathymetry (see for instance the breakwater test case 3.16).

The weir law as traditionally used in channel hydraulics is prescribed through two boundary conditions: one upstream the weir and one downstream. It should be mentioned however that this option gives satisfactory results only if the flow is relatively perpendicular to the weir, which is the case in the present test.

3.9.2. LINKED CLAIMS

- **Claim 2.2.1.7** TELEMAC-2D can be used to investigate the hydrodynamic impact of engineering works in rivers.
- **Claim 2.2.2.19** TELEMAC-2D is able to treat a number of flow singularities as internal boundary conditions: sills may be considered under the traditional approach made in open channel hydraulics through a relation between two superimposed open boundaries, short culverts may be treated as a couple of nodes with respective source and sink terms.

3.9.3. APPROACH

A rectangular channel with flat bottom is made of four reaches 848 m long. The 3 upstream reaches are limited at their downstream end by 3 sills with crest heights 1.8 (upstream sill), 1.6 and 1.4 m (downstream sill).

Geometry:

– Size of the model: channel = 3522 m x 848.5 m

– Water depth at rest = 1.35 m

Mesh:

The mesh is generally made up with squares split into 2 triangles. It is irregular in the upstream reach in order to test the sensitivity on this feature of the flow results above the sill (see Figure 3.9.1).

- 870 triangular elements
- 519 nodes
- Maximum size range: from 53 to 120 metres

Boundaries:

- Channel entrance: Q = 600 m^3/s imposed
- Channel outlet: H = 1.35 m imposed
- Lateral boundaries: solid walls with slip condition in the channel

Bottom:

- Strickler formula with friction coefficient = 30

Turbulence:

- Model of constant viscosity with velocity diffusivity = 1 m^2/s

Algorithm:

- Type of advection:
  - characteristics on velocities
  - conservative + modified SUPG on depth
- Type of element:
  - quasi-bubble triangle for velocities
  - P1 triangle for h
- GMRES solver and Direct solver for Tracer
- Accuracy = 10^{-5}

Time data:

- Time step = 150 sec.
- Simulation duration = 6000 sec.

3.9.4. RESULTS

The velocity field remains regular in the different reaches of the channel (see Figure 3.9.2).

The water level increases progressively as expected in the three upstream reaches during the simulated period (see Figure 3.9.3). The relations between the discharge (per unit of width) on the sill, the water levels upstream and downstream and the sill crest elevation are respected. They are:

$$q = \mu\sqrt{2g}\left(Z_{up} - Z_{sill}\right)^{3/2}$$ free overflow weir

$$q = \frac{2}{3\sqrt{3}}\mu\sqrt{2g}\left(Z_{down} - Z_{sill}\right)\sqrt{\left(Z_{up} - Z_{down}\right)}$$ drowned weir

The transition from free overflow to drowned condition is defined by:

$$Z_{down} \geq Z_{sill} + \frac{2}{3}\left(Z_{up} - Z_{sill}\right)$$

Where:

Z_{up} = water level upstream (m)

Z_{down} = water level downstream (m)

Z_{sill} = sill crest elevation (m)

q = discharge per unit width (m^2/s)

μ = coefficient of discharge (usually between 0.4 and 0.5)

3.9.5. CONCLUSIONS

TELEMAC-2D computes adequately weir flows as given by channel hydraulics. This type of flow is represented as an internal singularity in the model.

3.9.6. STEERING FILE

```
/---------------------------------------------------------------------
/ TELEMAC2D Version v6p1 5 déc. 2011
/ Validation test case 10
/---------------------------------------------------------------------

/---------------------------------------------------------------------
/ EQUATIONS
/---------------------------------------------------------------------

FRICTION COEFFICIENT      =30.
LAW OF BOTTOM FRICTION    =3
VELOCITY DIFFUSIVITY      =1.
NUMBER OF TRACERS         =1
INITIAL VALUES OF TRACERS =50.

/---------------------------------------------------------------------
/ EQUATIONS, BOUNDARY CONDITIONS
/---------------------------------------------------------------------

PRESCRIBED FLOWRATES      =600.;0.
PRESCRIBED ELEVATIONS     =0.;1.35
PRESCRIBED TRACERS VALUES =100.;0.

/---------------------------------------------------------------------
/ EQUATIONS, INITIAL CONDITIONS
/---------------------------------------------------------------------

INITIAL ELEVATION  =1.35
INITIAL CONDITIONS ='CONSTANT ELEVATION'

/---------------------------------------------------------------------
/ INPUT-OUTPUT, FILES
/---------------------------------------------------------------------

FORMATTED DATA FILE 1    ='format1.txt'
GEOMETRY FILE            ='geo'
REFERENCE FILE           ='ref'
```

```
STEERING FILE             ='cas.txt'
BOUNDARY CONDITIONS FILE ='cli.txt'
RESULTS FILE              ='res'

/----------------------------------------------------------------------
/ INPUT-OUTPUT, GRAPHICS AND LISTING
/----------------------------------------------------------------------

LISTING PRINTOUT PERIOD          =10
GRAPHIC PRINTOUT PERIOD          =10
VARIABLES FOR GRAPHIC PRINTOUTS ='U,V,S,T1,H'
MASS-BALANCE                     =YES

/----------------------------------------------------------------------
/ INPUT-OUTPUT, INFORMATION
/----------------------------------------------------------------------

VALIDATION =YES
TITLE      ='Validation test case 10'

/----------------------------------------------------------------------
/ NUMERICAL PARAMETERS
/----------------------------------------------------------------------

CONTINUITY CORRECTION             =YES
NUMBER OF WEIRS                   =3
NUMBER OF TIME STEPS              =40
DISCRETIZATIONS IN SPACE          =12;11;11;11
SUPG OPTION                       =1;0;2;2
COMPATIBLE COMPUTATION OF FLUXES =YES
TIME STEP                         =150.
TIDAL FLATS                       =NO
TYPE OF ADVECTION                 =1;5;4;1

/----------------------------------------------------------------------
/ NUMERICAL PARAMETERS, SOLVER
/----------------------------------------------------------------------

SOLVER FOR DIFFUSION OF TRACERS =8
SOLVER                           =7
SOLVER ACCURACY                  =1.E-10
SOLVER OPTION                    =5

/----------------------------------------------------------------------
/ NUMERICAL PARAMETERS, VELOCITY-CELERITY-HIGHT
/----------------------------------------------------------------------

IMPLICITATION FOR VELOCITY =0.6
```

3.9.7. FILE OF DATA CONCERNING THE SILLS

```
Nb of culverts Option for tangential velocity
          3             0
---------------------------- Singularity 1
Nb of points for 1side
11
Points side 1
71 72 73 74 75 76 77 78 79 80 41
Points side 2
21 20 19 18 17 16 15 14 13 12 11
Levels of the dike
1.8 1.8 1.8 1.8 1.8 1.8 1.8 1.8 1.8 1.8 1.8
Flowrate coefficients
 .4  .4  .4  .4  .4  .4  .4  .4  .4  .4  .4
---------------------------- Singularity 2
Nb of points side 1
11
Points side 1
111 112 113 114 115 116 117 118 119 120 81
Points side 2
61 60 59 58 57 56 55 54 53 52 51
Levels of the dike
1.6 1.6 1.6 1.6 1.6 1.6 1.6 1.6 1.6 1.6 1.6
Flowrate coefficients
 .4  .4  .4  .4  .4  .4  .4  .4  .4  .4  .4
---------------------------- Singularity 3
Nb of points
11
Points side 1
151 152 153 154 155 156 157 158 159 160 121
Points side 2
101 100  99  98  97  96  95  94  93  92  91
Levels of the dike
1.4 1.4 1.4 1.4 1.4 1.4 1.4 1.4 1.4 1.4 1.4
Flowrate coefficients
 .4  .4  .4  .4  .4  .4  .4  .4  .4  .4  .4
```

3.9.8. FIGURES

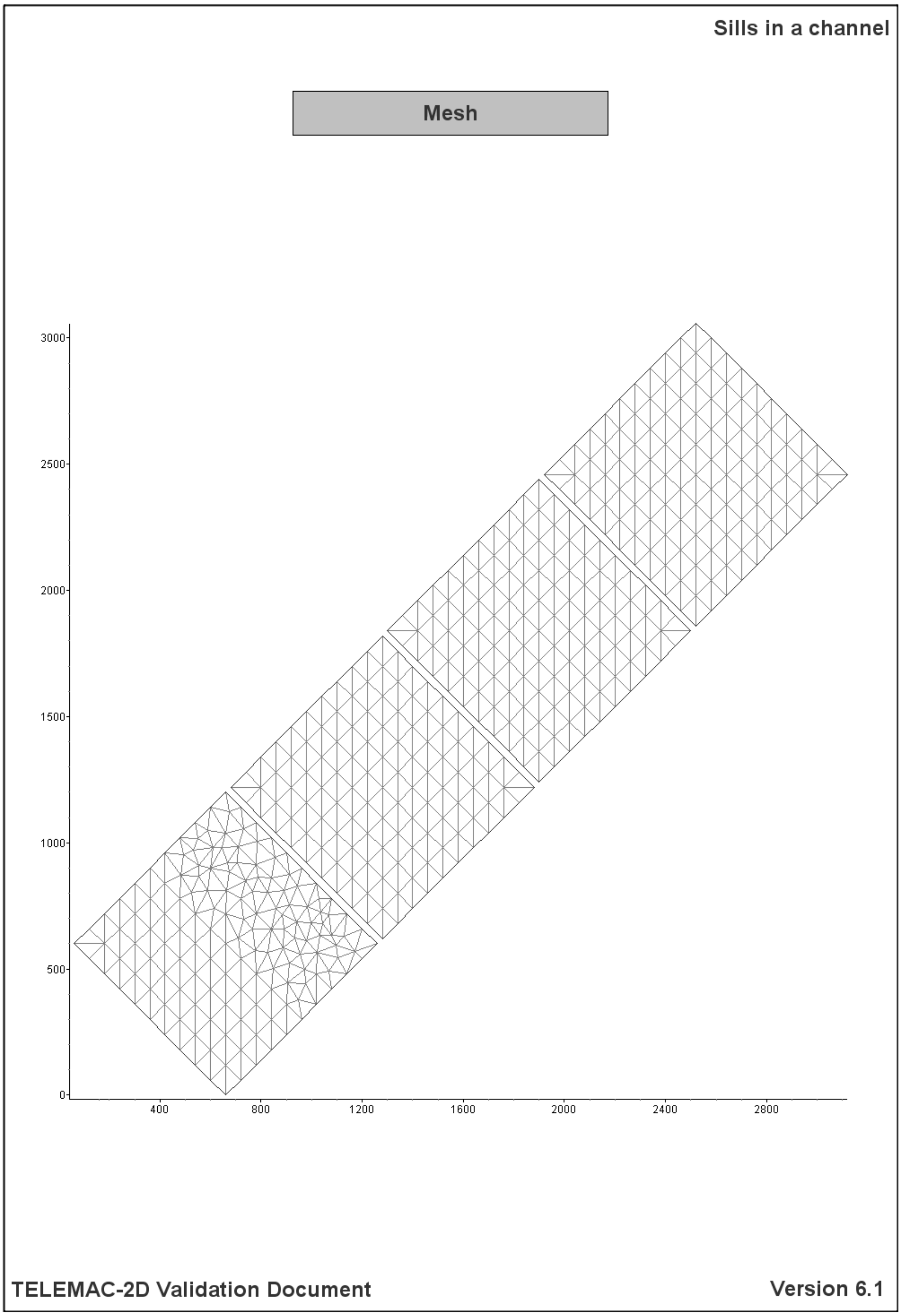


Figure 3.9.1 : Mesh for the problem of sills in a channel.

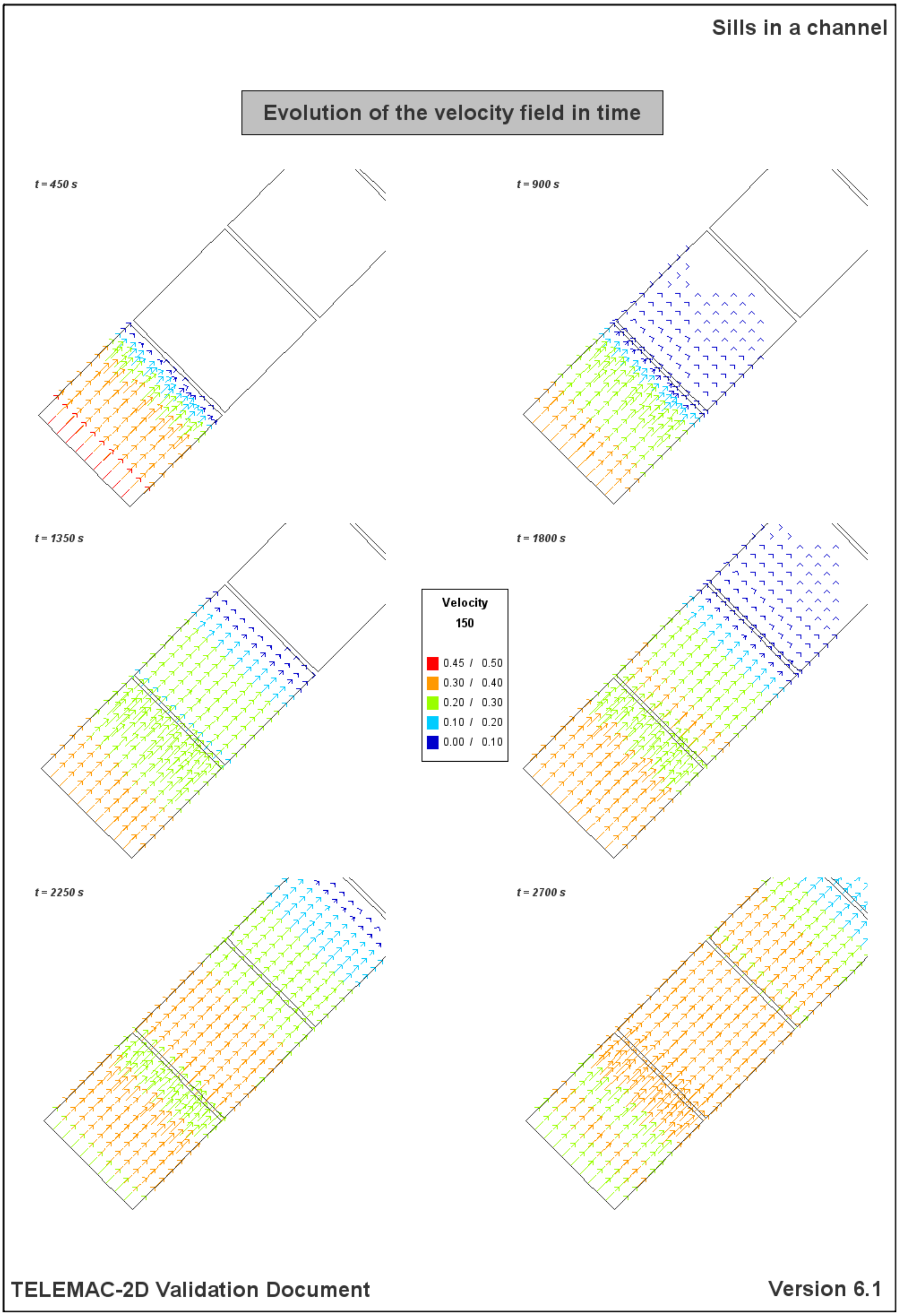


Figure 3.9.2 : Evolution of velocity field in time.

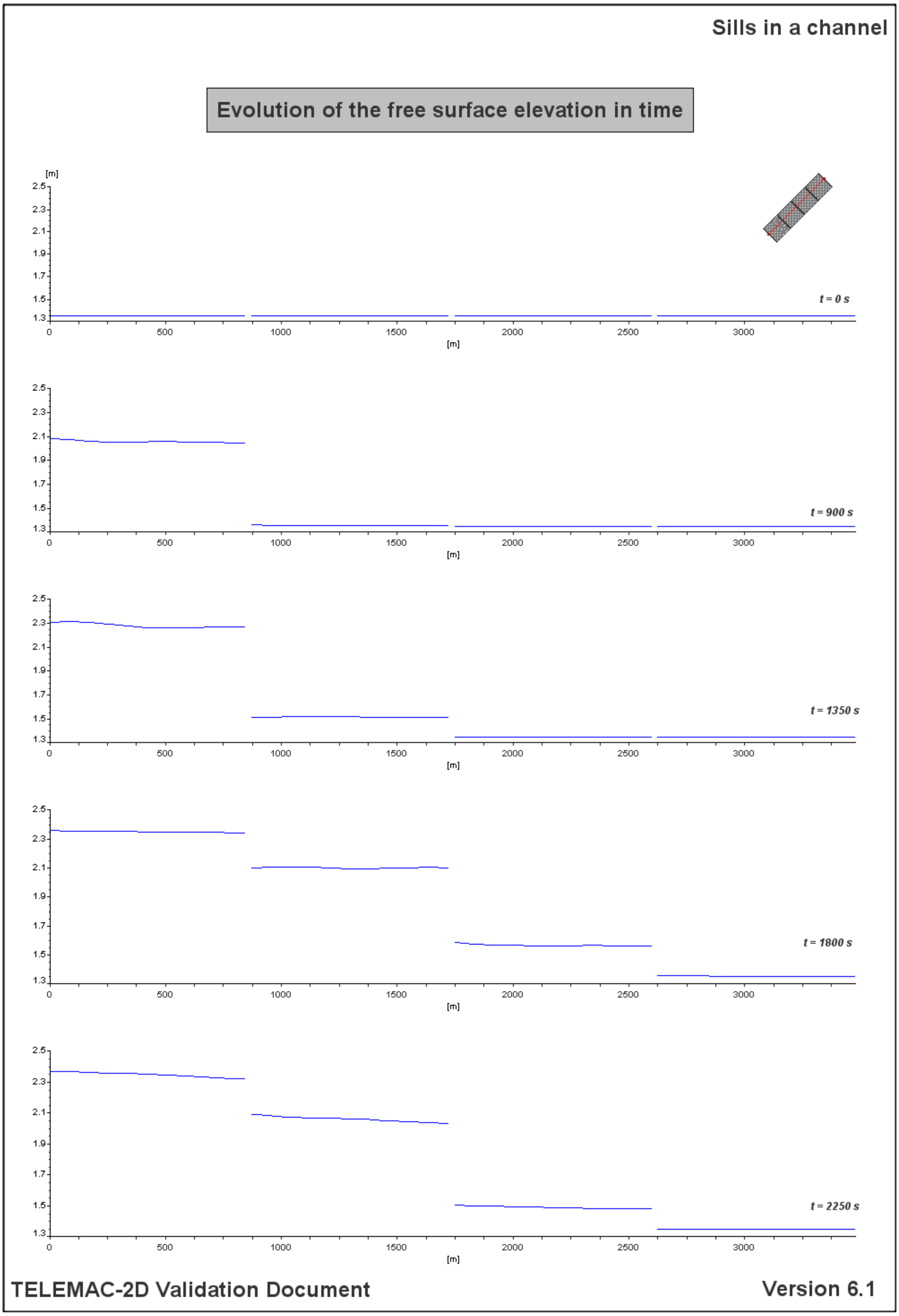


Figure 3.9.3 : Evolution of the free surface elevation in time.

3.10. CULVERTS

Title	**Flow in a culvert treated as a couple of source/sink nodes**
Initial study	**V. Guinot – July 1996**
Last update	**May 2012**
Version	**TELEMAC-2D 6.1**

3.10.1. PURPOSE

To demonstrate that TELEMAC-2D can solve the flow in a culvert considered as an internal singularity, under the form of a couple of source and sink nodes. Also to show that TELEMAC-2D computes tracer dispersion.

3.10.2. LINKED CLAIMS

- **Claim 2.2.1.7** TELEMAC-2D can be used to investigate the hydrodynamic impact of engineering works in rivers.
- **Claim 2.2.2.9** TELEMAC-2D is capable to represent areas of injection or extraction of fluid in the flow domain. This feature considers a source/sink of mass and of momentum in option. It is also possible to simulate a source/sink of tracer in the same location.
- **Claim 2.2.2.17** TELEMAC-2D is able to compute the dispersion by currents and diffusion of a tracer, with source or sink terms; this feature is employed only when water density is dependent upon the concentration in tracer.
- **Claim 2.2.2.19** TELEMAC-2D is able to treat a number of flow singularities as internal boundary conditions: sills may be considered under the traditional approach made in open channel hydraulics through a relation between two superimposed open boundaries, short culverts may be treated as a couple of nodes with respective source and sink terms.
- **Claim 2.4.2.2** The numerical methods used in TELEMAC-2D can provide an accurate solution of the tracer conservation equation.
- **Claim 2.4.2.12** With appropriate choice of solution method options, the calculation of tracer dispersion is mass conservative.

3.10.3. APPROACH

Two square tanks are connected hydraulically by a culvert. The culvert is represented by a couple of source / sink nodes, one in each tank. The water level and the tracer concentration are initially higher in the left tank.

<u>Geometry</u>:

– Two identical square tanks are located 100 metres apart.

– The dimension of each square tank is 200 m x 200 m

- The water depth at rest is 4 m in the left tank, and 2 m in the right tank

Mesh:

The mesh is regular. It is made up with squares split into 2 triangles.

- 1600 triangular elements
- 882 nodes
- Maximum size range: $\sqrt{200} = 14.14$ metres

Boundaries:

- Solid walls with slip condition in the domain.

Bottom:

- Strickler formula with friction coefficient = 20

Turbulence:

- Model of constant viscosity with velocity diffusivity = 1 m^2/s

Algorithm:

- Type of advection:
  - characteristics on velocities
  - conservative + modified SUPG on depth
  - centred semi-implicit scheme + SUPG decentring on tracer
- Type of element:
  - quasi-bubble triangle for velocities
  - P1 triangle for h
- GMRES solver
- Accuracy = 10^{-5}

Tracer:

- initial concentrations : 100% in the left square, and 50% in the right square
- co-ordinates of source / sink nodes : left = (100;100) and right = (400;100)
- no water discharge of sources
- GMRES solver

Time data:

- Time step = 5 sec.
- Simulation duration = 640 sec.

Mesh and initial state are shown on Figure 3.10.1.

3.10.4. RESULTS

The water flows from the left tank to the right tank through the culvert. The water level decreases regularly in the left tank and increases regularly in the right tank (Figure 3.10.2). Simultaneously, a spot of tracer with concentration 100 arrives in the right tank and disperses.

In the left tank, the culvert is vertical. Therefore, the flow is regular and symmetric around the sink node. In the right tank, the culvert is horizontal in the direction of y-axis. Therefore, the flow takes this direction from the source node and the velocity field forms two eddies around the source (Figure 3.10.3).

Water mass and tracer mass are conserved: no water mass is lost whereas the cumulated loss of tracer mass at the end of the simulation is below 0.9 % of the initial mass.

3.10.5. CONCLUSIONS

TELEMAC-2D can be used for the treatment of an internal singularity, such as a culvert, and also for the treatment of dispersion by currents and diffusion of a tracer.

3.10.6. STEERING FILE

```
/---------------------------------------------------------------------
/ TELEMAC2D Version v6p1 5 may 2011
/ Validation test case 11
/---------------------------------------------------------------------
/---------------------------------------------------------------------
/ INPUT-OUTPUT, FILES
/---------------------------------------------------------------------

GEOMETRY FILE            ='geo'
REFERENCE FILE           ='ref'
BOUNDARY CONDITIONS FILE ='cli.txt'
FORTRAN FILE             ='princi.f'
RESULTS FILE             ='res'
FORMATTED DATA FILE 1    ='format1.txt'

/---------------------------------------------------------------------
/ VALIDATION
/---------------------------------------------------------------------

VALIDATION = YES
TITLE      ='Validation test case 11'

/---------------------------------------------------------------------
/ INPUT-OUTPUT, GRAPHICS AND LISTING
/---------------------------------------------------------------------

LISTING PRINTOUT PERIOD         =10
GRAPHIC PRINTOUT PERIOD         =10
VARIABLES FOR GRAPHIC PRINTOUTS =' U,V,H,S,B,T1,T2'
MASS-BALANCE                    =YES

/---------------------------------------------------------------------

FRICTION COEFFICIENT      =20.
LAW OF BOTTOM FRICTION    =3
VELOCITY DIFFUSIVITY      =1.
```

```
/----------------------------------------------------------------------
/ TRACERS
/----------------------------------------------------------------------

NUMBER OF TRACERS                                             =2
INITIAL VALUES OF TRACERS                                     =100.;200.
MAXIMUM NUMBER OF ITERATIONS FOR DIFFUSION OF TRACERS =40
IMPLICITATION COEFFICIENT OF TRACERS                          =1.
COEFFICIENT FOR DIFFUSION OF TRACERS                          =1.
ACCURACY FOR DIFFUSION OF TRACERS                             =1.E-10

/----------------------------------------------------------------------
/ EQUATIONS, SOURCES
/----------------------------------------------------------------------

VELOCITIES OF THE SOURCES ALONG X     =0.;0.
VELOCITIES OF THE SOURCES ALONG Y     =0.;0.
ABSCISSAE OF SOURCES                  =100.;400.
ORDINATES OF SOURCES                  =100.;100.
WATER DISCHARGE OF SOURCES            =0.;0.
VALUES OF THE TRACERS AT THE SOURCES  =0.;0.;0.;0.

/----------------------------------------------------------------------
/ INITIAL CONDITIONS
/----------------------------------------------------------------------

INITIAL CONDITIONS       ='PARTICULAR'
INITIAL ELEVATION        =2.

/----------------------------------------------------------------------
/ NUMERICAL PARAMETERS
/----------------------------------------------------------------------

NUMBER OF CULVERTS                          =1
TIME STEP                                   =2.5
OPTION FOR THE TREATMENT OF TIDAL FLATS     =1
SUPG OPTION                                 =1;2;0;2
NUMBER OF TIME STEPS                        =240
TYPE OF ADVECTION                           =1;5;2;1
DISCRETIZATIONS IN SPACE                    =12;11;11;11
MASS-LUMPING ON H =1.

/----------------------------------------------------------------------
/ SOLVER
/----------------------------------------------------------------------

SOLVER                               =7
SOLVER FOR DIFFUSION OF TRACERS      =7
SOLVER OPTION                        =5
SOLVER ACCURACY                      =1.E-8
```

3.10.7. FILE OF DATA CONCERNING THE CULVERT

```
Relaxation,  NUMBER OF CULVERTS
    0.2              1
I1  I2  d1  d2  Ce1  Ce2  Cs1  Cs2   S12   L12   z1    z2    a1   a2
 1   2  90. 0.  0.5  0.5   1.   1.    5.   0.2   0.3   0.1   0.   90.
```

3.10.8. FIGURES

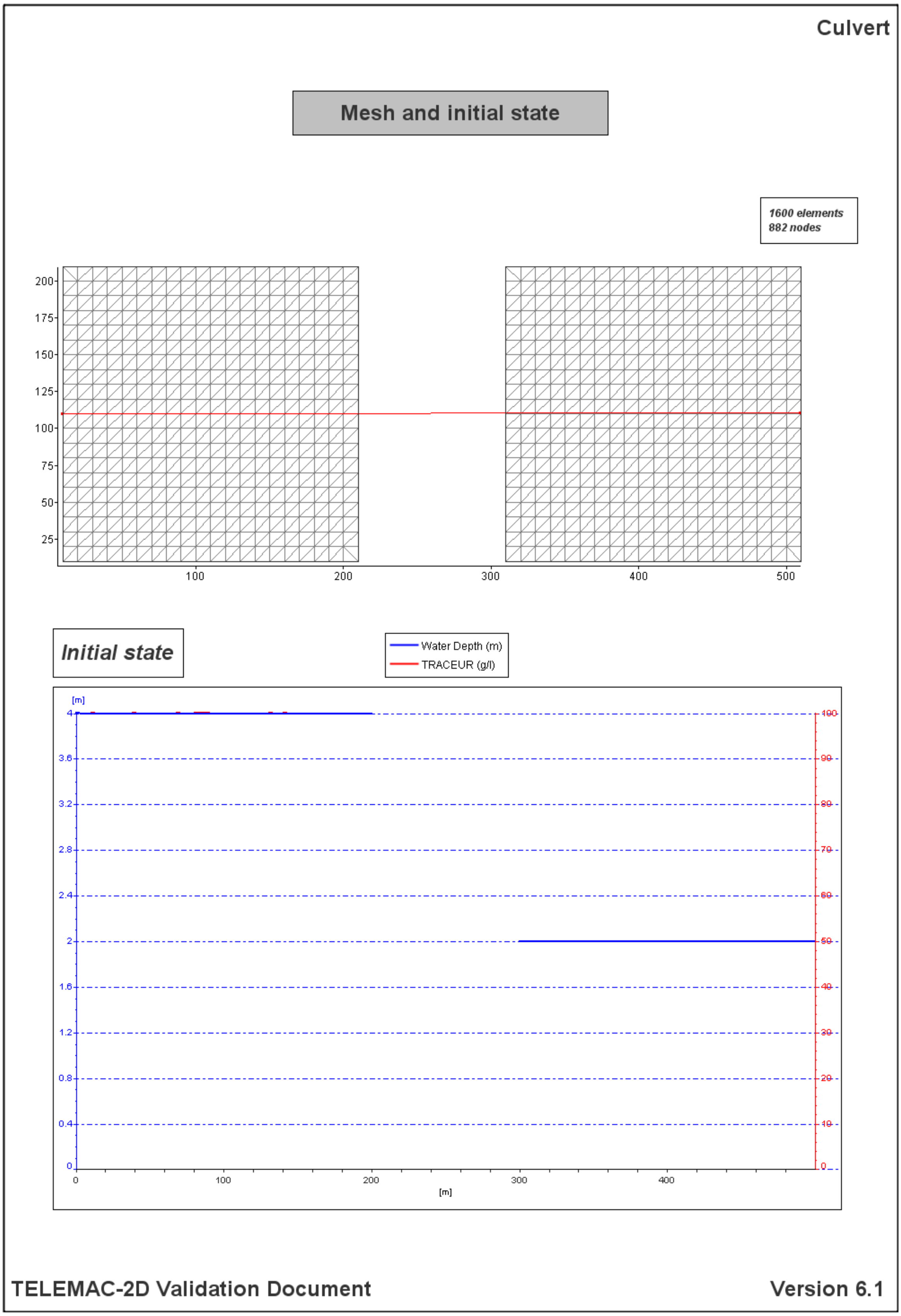


Figure 3.10.1 : Mesh and initial state.

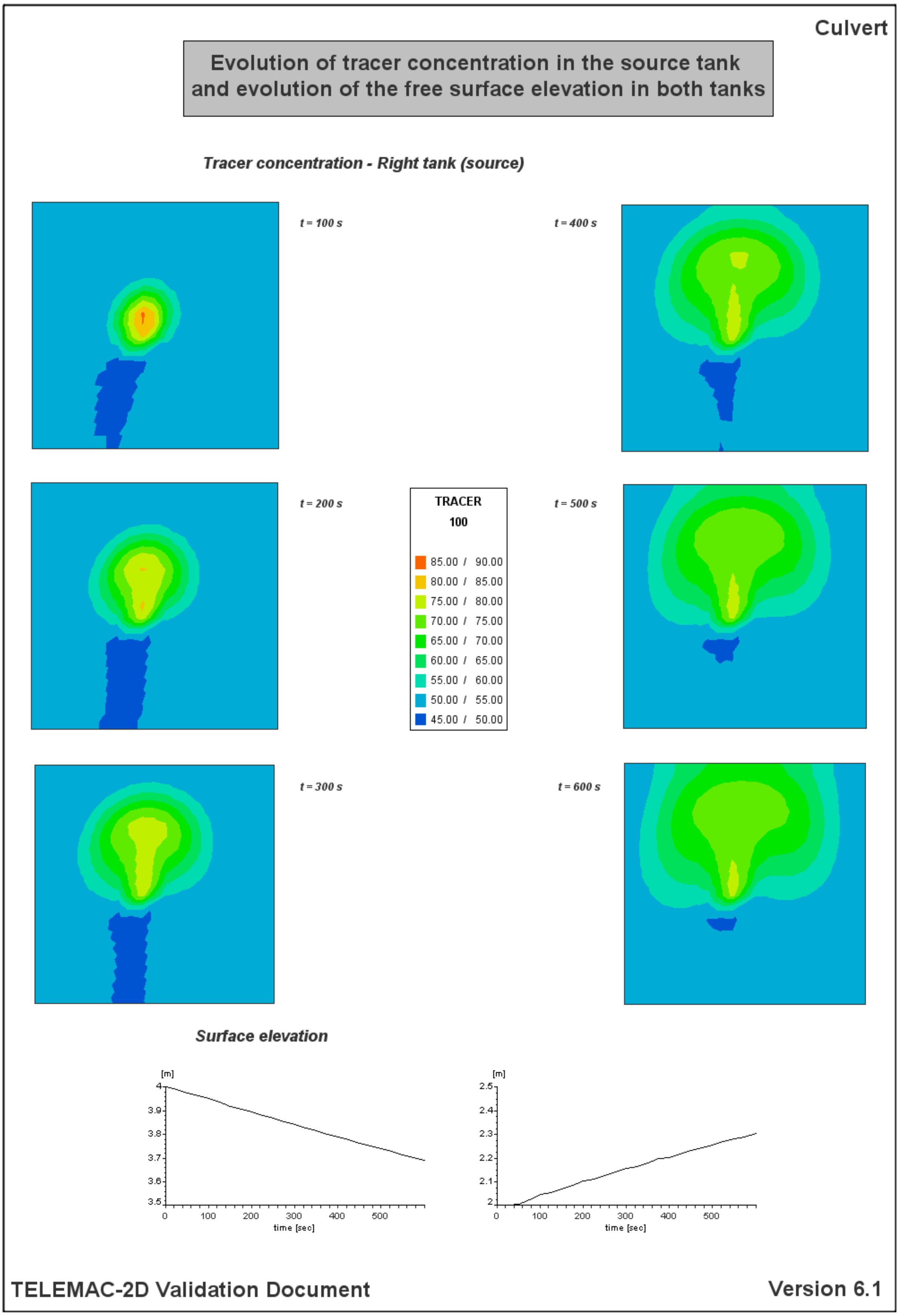


Figure 3.10.2 : Evolution of the tracer concentration in the source tank, and evolution of the free surface elevation in time in both tanks.

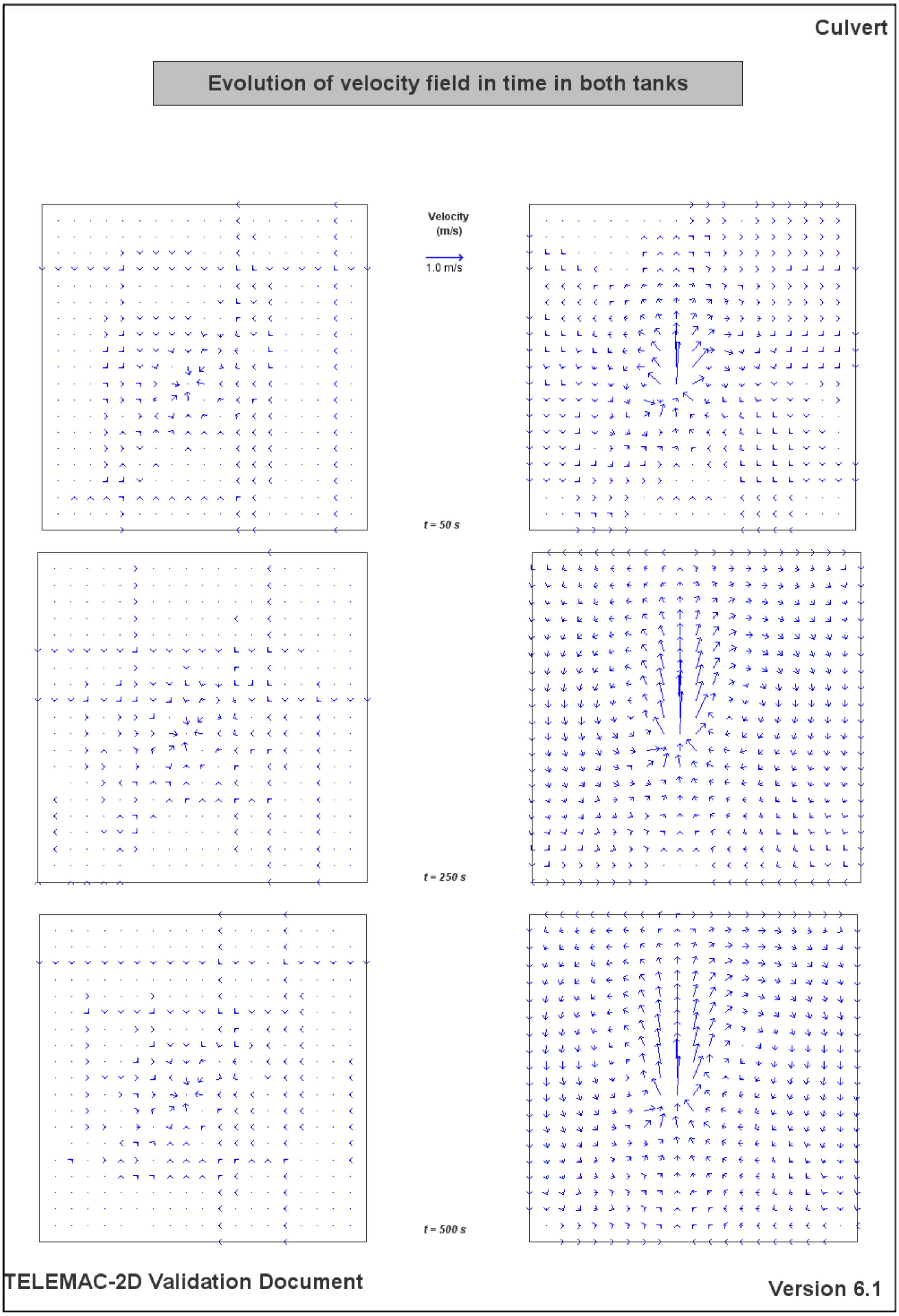


Figure 3.10.3 : Evolution of velocity field in time in both tanks.

3.11. THEORETICAL DAM BREAK WAVE - INITIALLY DRY BED

Title	**Theoretical dam break wave in a rectangular channel – Initially dry bed**
Initial study	**J.M.Hervouet – March 1992**
Last update	**December 2011**
Version	**TELEMAC-2D 6.1**

3.11.1. PURPOSE

To demonstrate that TELEMAC-2D can solve the problem of a dam break wave propagating on an initially dry bed. Also to compare the water depth evolution versus time as computed by TELEMAC-2D with the analytic solution to this problem.

3.11.2. LINKED CLAIMS

- **Claim 2.2.1.9** TELEMAC-2D can appropriately be used to analyse the consequences of the water wave following a dam break. Situation with dry or wet beds in the valley downstream the dam can both be examined.
- **Claim 2.2.2.4** TELEMAC-2D can consider conditions on open boundaries of a study domain corresponding to a wide range of physical phenomena encountered in the nature: tides (maritime applications), floods, river water inflow, dam failure (river applications).
- **Claim 2.2.2.16** TELEMAC-2D can represent the wetting and drying of intertidal zones (maritime applications) and of inundated plains (river applications). The numerical treatment of this process is very robust.

3.11.3. APPROACH

The computational domain is a square basin. The mesh is made up with squares split into two triangles. Initially, the free surface elevation presents a discontinuity at abscissae x = 10.5 m, hereafter called “dam location”: the left side of the basin is covered with water at rest, the water depth being 4 metres, whereas the right side is dry. After the start of the simulation, the dam location becomes a critical flow section where the velocity and the depth are constant in time. The flow is subcritical to the left of the dam location (upstream part), and supercritical to the right (downstream part).

Geometry:

– Size of the model: square = 20.1 m x 20.1 m

– Water depth at rest: 0<x<10.5, H=4 m; 10.5<x<20.1, H=0m

Mesh:

- 8978 triangular elements
- 4624 nodes
- Maximum size range: 0.4 metres

Boundaries:

- solid walls with slip condition in the domain

Bottom:

- No bottom friction

The mesh and the initial state are shown on Figure 3.11.1.

Turbulence:

- Constant viscosity = 0 m^2/s

Algorithm:

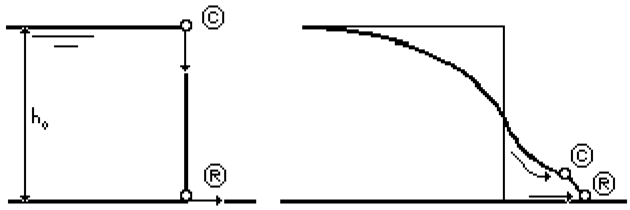
- Type of advection:
  - Edge-based N-Scheme on velocities
  - Conservative on depth
- Type of element:
  - P1 triangle for h and for velocity
- Solver: Conjugate gradient
- Solver accuracy = 10^{-4}
- Implicitation for depth and for velocity = 0.6
- Finite volumes resolution

Time data:

- Time step = 0.01 sec.
- Simulation duration = 1.6 sec.

3.11.4. RESULTS

Left : Initial phase of dambreak wave with surface particle © and bottom particle ® at the



dam section ; Right : Formation of the initial wave ® and the dynamic wave ©.

According to Lauber and Hager (12), particle © behaves initially as a free falling body under the influence of gravity, such that:

The fall velocity is $\mathrm{w}_C = gt$ and the fall distance is $s_C = (1/2)gt^2$,

Where:

w is the velocity along the z-axis ($m.s^{-1}$),

g is the gravity acceleration ($m.s^{-2}$),

t is time (s).

The particle reaches the bottom when $s_C = h_0$, i.e. $t = \sqrt{\dfrac{2h_0}{g}} = t_0$,

where h_o is the initial water depth in the reservoir.

For $t < t_0$, particles move mainly in the vertical direction, and they propagate mainly in the horizontal direction for $t > t_0$.

In this test case, t_0 is equal to 0.90305 sec.

From 0 sec. to 0.9 sec., the solution given by TELEMAC-2D shows a positive front wave like a step (see Figure 3.11.2).

Ritter's solution (13) is currently used for initiating dam break waves due to sudden dam removal. It is however demonstrated that detailed laboratory experiments are in contradiction with this procedure. On the other hand, Ritter's solution reproduces the general features of a dam break wave quite well, at least for wave propagation on a dry tailwater channel with a semi-infinite reservoir extension. However, deviations are significant at both wave fronts and during the wave initial phase.

$$h = \frac{4}{9g}\left(\sqrt{gh_0} - \frac{x}{2t}\right)^2$$

$$u = \frac{2}{3}\left(\sqrt{gh_0} + \frac{x}{t}\right)$$

Where:
- h is the water depth (m)
- x is the downstreal location from the dam section (m)
- t is the time (s)
- g is the gravity acceleration ($m.s^{-2}$)
- h_0 is the initial reservoir water depth (m)
- u is the velocity along the axis ($m.s^{-1}$)

From time t_0, TELEMAC-2D shows good agreements with the Ritter's solution (see Figure 3.11.3). The free surface profiles at different times cross each other at the critical flow section, i.e. at the dam location.

3.11.5. CONCLUSIONS

TELEMAC-2D reproduces well the Ritter's solution to the dam break wave problem on initially dry bed.

3.11.6. STEERING FILE

```
/----------------------------------------------------------------------
/ TELEMAC2D Version v6p1 5 déc. 2011
/ Validation test case 12
/----------------------------------------------------------------------

/----------------------------------------------------------------------
/ INPUT-OUTPUT, FILES
/----------------------------------------------------------------------

FORTRAN FILE             ='princi.f'
GEOMETRY FILE            ='geo'
REFERENCE FILE           ='refpos'
BOUNDARY CONDITIONS FILE ='cli.txt'
RESULTS FILE             ='respos'

/----------------------------------------------------------------------
/ INPUT-OUTPUT, GRAPHICS AND LISTING
/----------------------------------------------------------------------

LISTING PRINTOUT PERIOD         =15
GRAPHIC PRINTOUT PERIOD         =10
VARIABLES FOR GRAPHIC PRINTOUTS ='U,V,H,F,L,N'
MASS-BALANCE                    =YES

/----------------------------------------------------------------------
/ INPUT-OUTPUT, INFORMATION
/----------------------------------------------------------------------

VALIDATION =YES
TITLE      ='Validation test case 12'

/----------------------------------------------------------------------
/ INITIAL CONDITIONS
/----------------------------------------------------------------------

INITIAL CONDITIONS ='SPECIAL'

/----------------------------------------------------------------------
/ NUMERICAL PARAMETERS
/----------------------------------------------------------------------

TREATMENT OF NEGATIVE DEPTHS         =2
CONTINUITY CORRECTION                =YES
NUMBER OF TIME STEPS                 =150
NUMBER OF PRIVATE ARRAYS             =1
FREE SURFACE GRADIENT COMPATIBILITY =0.9
TREATMENT OF THE LINEAR SYSTEM       =2
SUPG OPTION                          =0;0;2;2
```

```
TIME STEP                           =0.01
DURATION                            =1.5
TYPE OF ADVECTION                   =14;5;1;1

/----------------------------------------------------------------------
/ NUMERICAL PARAMETERS, SOLVER
/----------------------------------------------------------------------

SOLVER =1

/----------------------------------------------------------------------
/ NUMERICAL PARAMETERS, VELOCITY-CELERITY-HIGHT
/----------------------------------------------------------------------

MASS-LUMPING ON H          =1.
IMPLICITATION FOR DEPTH    =1.
IMPLICITATION FOR VELOCITY =0.6

/----------------------------------------------
/ IN CASE OF USE OF FINITE VOLUME
/----------------------------------------------
/EQUATIONS                         = 'SAINT-VENANT VF'
/FINITE VOLUME SCHEME              = 6
/VARIABLE TIME-STEP                = YES
/DESIRED COURANT NUMBER            = 0.8
/DURATION                          = 1.5
/----------------------------------------------
```

3.11.7. FIGURES

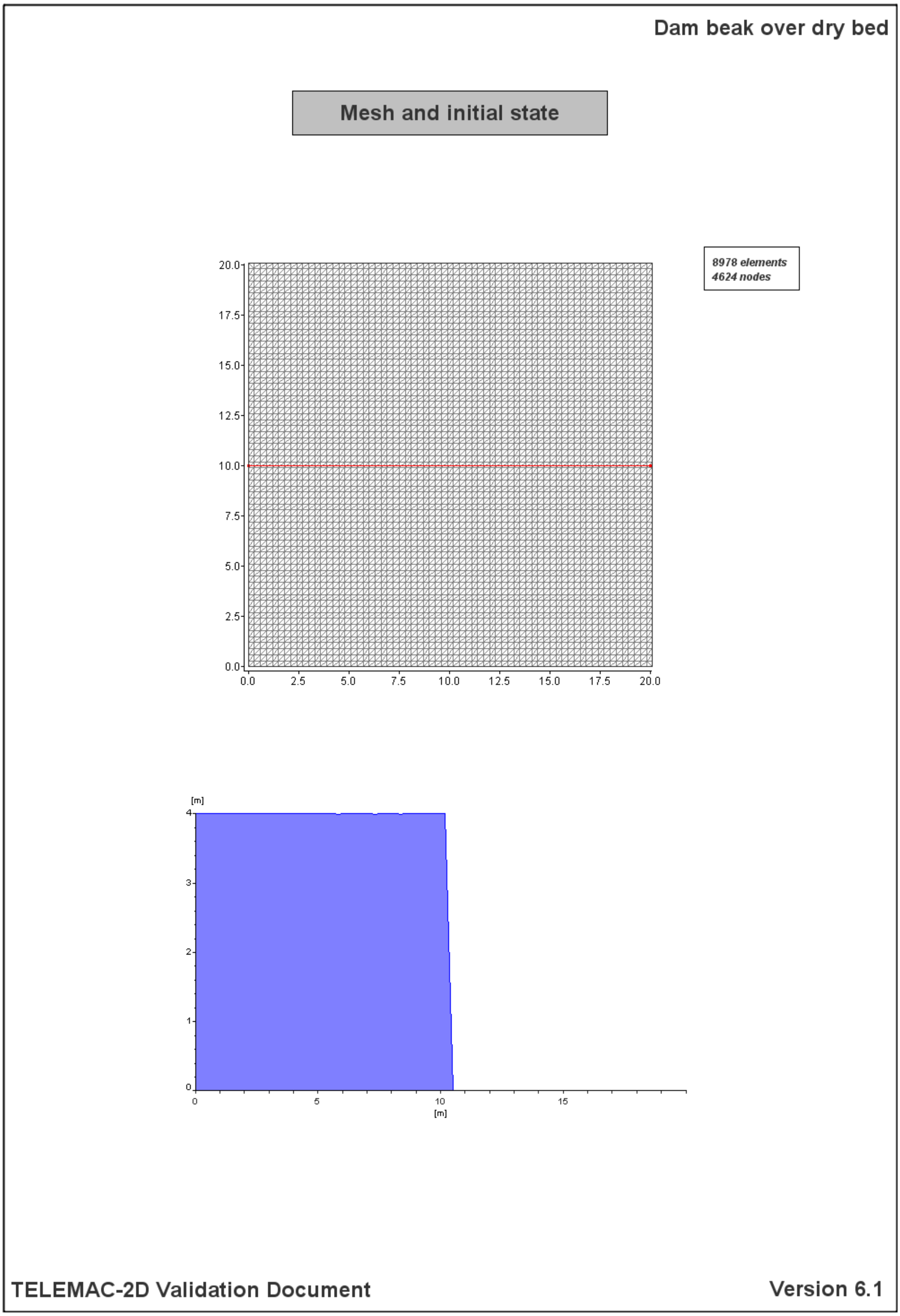


Figure 3.11.1 : Mesh and initial state.

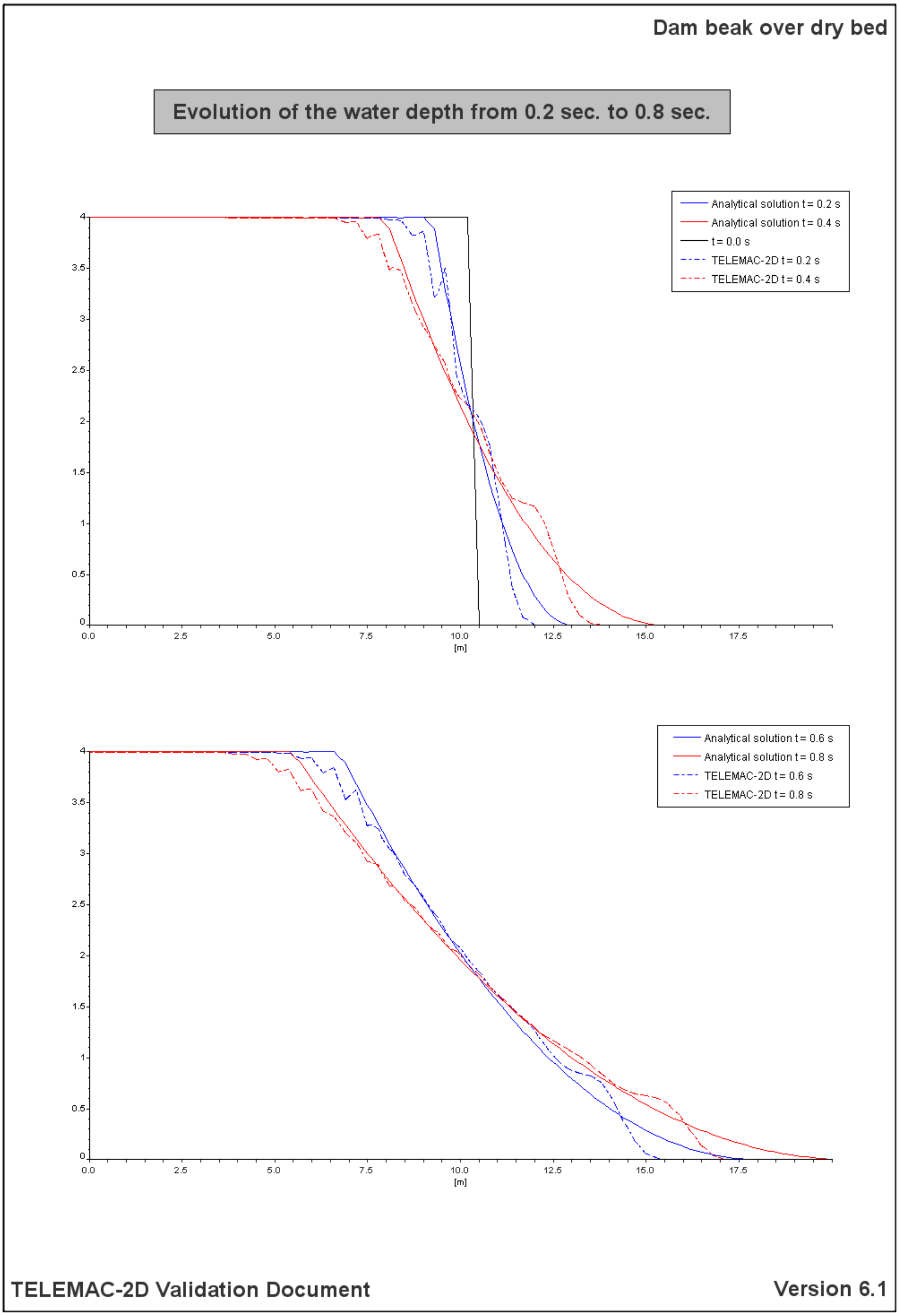


Figure 3.11.2 : Evolution of the water depth from 0.1 sec. to 1.0 sec.

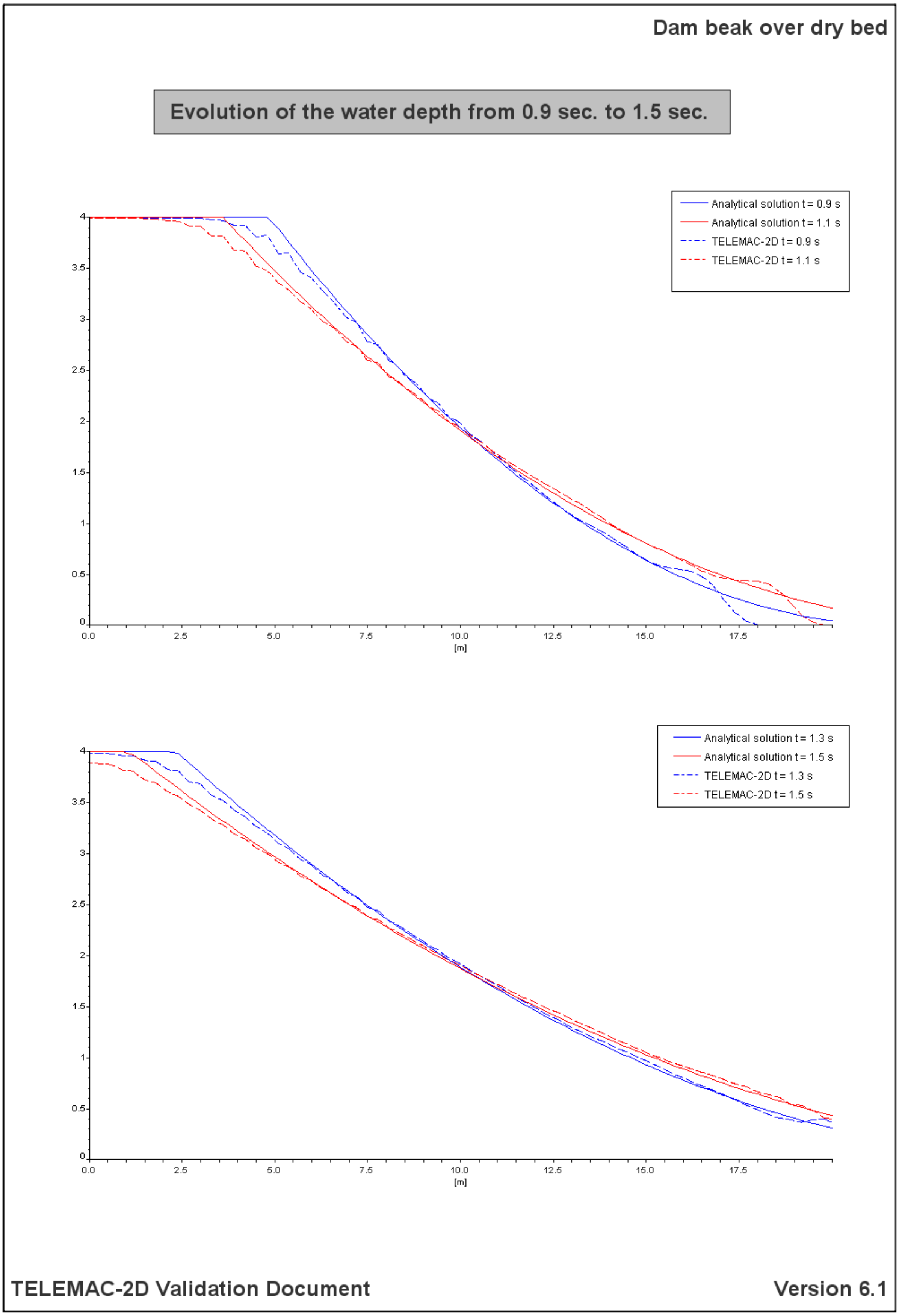
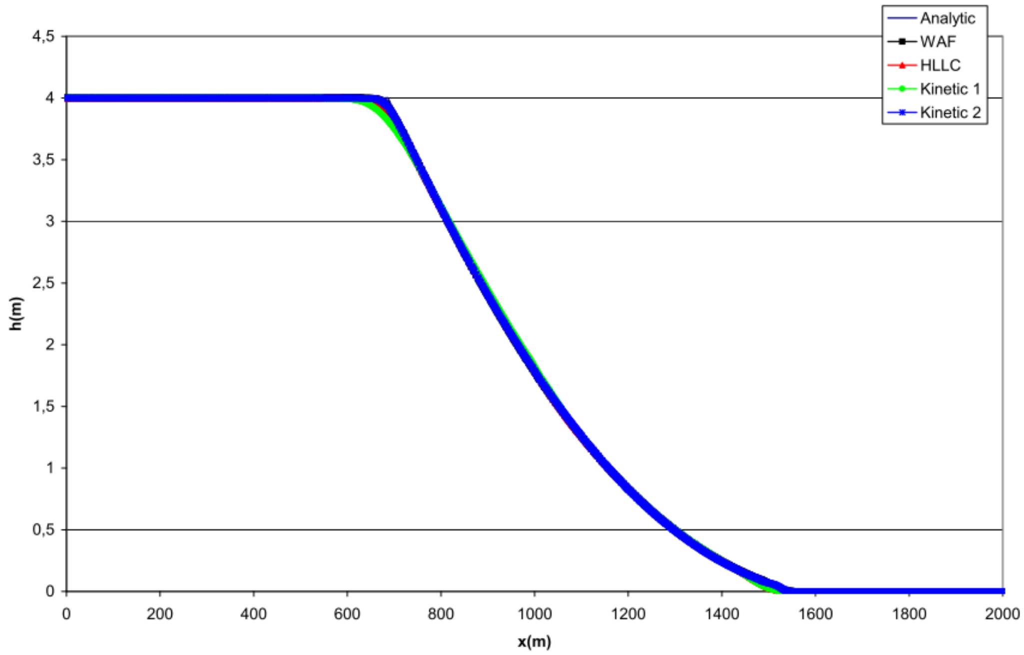
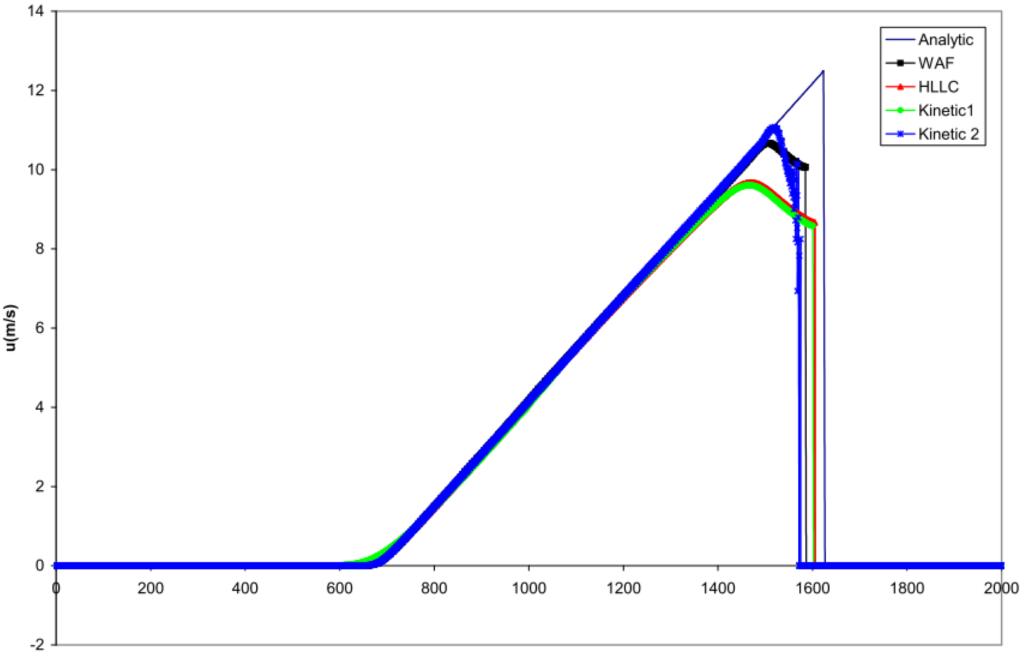


Figure 3.11.3 : Evolution of the water depth profile from 0.9 sec. to 1.5 sec.





Ritter Problem- Telemac2D Finite volumes results with several schemes (Water depth (up) and velocity (down)).

3.12. THEORETICAL DAM BREAK WAVE - INITIALLY WET BED

Title	**Theoretical dam break wave in a rectangular channel – Initially wet bed**
Initial study	**J.M.Hervouet – March 1992**
Last update	**December 2011**
Version	**TELEMAC-2D 6.1**

3.12.1. PURPOSE

To demonstrate that TELEMAC-2D can solve the problem of a dam break wave on an initially wet bed. Also to compare the water depth evolution versus time as computed by TELEMAC-2D with the analytic solution to this problem.

3.12.2. LINKED CLAIMS

- **Claim 2.2.1.9** TELEMAC-2D can appropriately be used to analyse the consequences of the water wave following a dam break. Situation with dry or wet beds in the valley downstream the dam can both be examined.
- **Claim 2.2.2.4** TELEMAC-2D can consider conditions on open boundaries of a study domain corresponding to a wide range of physical phenomena encountered in the nature: tides (maritime applications), floods, river water inflow, dam failure (river applications).
- **Claim 2.2.2.16** TELEMAC-2D can represent the wetting and drying of intertidal zones (maritime applications) and of inundated plains (river applications). The numerical treatment of this process is very robust.

3.12.3. APPROACH

The computational domain is a 6.000 m wide and 24.000 m long rectangular channel. The mesh is made up with squares split into two triangles (see Figure 3.12.1). Initially, the free surface elevation presents a discontinuity located at abscissa y = 0 m, hereafter called “dam location”: the flow domain is initially at rest, the water depth in the “upstream” part of the channel being 6 metres whereas it is only 2 metres downstream the dam location.

Geometry:

Size of the model: rectangle = 6000 m x 24000 m

- Water depth at rest : y<0, H=6 m ; y>0, H=2m

Mesh:

The mesh is regular. It is made up with squares split into 2 triangles.

- 3200 triangular elements
- 1701 nodes
- Maximum size range: 224 metres

Boundaries:

- Solid walls with slip conditions in the domain

Bottom:

- No bottom friction

The mesh and the initial state are shown on Figure 3.12.1.

Turbulence:

- Model of constant viscosity = 0 m^2/s

Algorithm:

- Type of advection:
  - Edge-based N-Scheme on velocities
  - Conservative on depth
- Type of element:
  - quasi-bubble triangle for velocities
  - P1 triangle for h
- Solver accuracy = 10^{-4}
- SUPG option :
  - decentring equal to 1 for velocity
  - decentring equal to Courant number
- Implicitation for depth and for velocity: 1

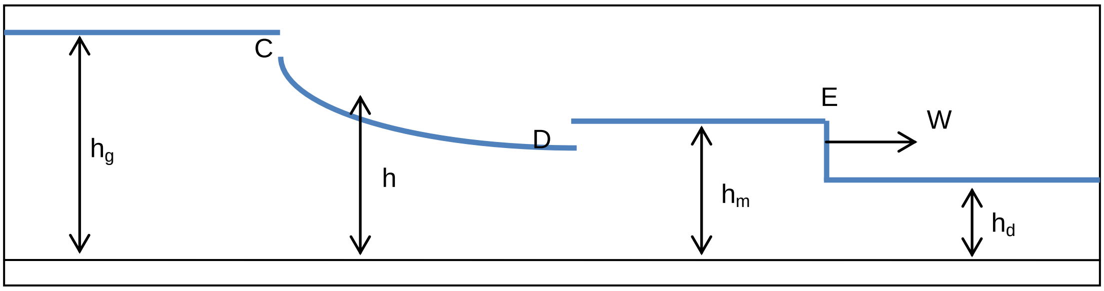
Time data:

- Time step = 25 sec.
- Simulation duration = 1500 sec.

3.12.4. RESULTS

The Stoker solution (see ref. (14)) is a 1D solution that reproduce a dam break when the depth on the downstream is constant and velocity is zero.

In the Stoker solution, there are 4 distinct zones of solutions, separated by 3 points C, D and E define like this:



$c_g = \sqrt{gh_g}$ $\qquad$ $c_d = \sqrt{gh_d}$

$x_C = -c_g t$ $\qquad$ $x_D = (u_m - c_m)t$ $\qquad$ $x_E = Wt = \frac{h_m u_m}{h_m - h_d} t$

$u_m = 2(c_g - c_m)$ $\qquad$ $h_m = \frac{{c_m}^2}{g}$

c_m is the solution of ${c_m}^6 - 9{c_d}^2{c_m}^4 + 16c_g{c_d}^2{c_m}^3 - {c_d}^2({c_d}^2 + 8{c_g}^2){c_m}^2 + {c_d}^6 = 0$

Between Upstream and C: $h=h_g$ and u=0;

Between C and D: $h = \frac{4}{9g}\left(\sqrt{gh_0} - \frac{x}{2t}\right)^2$ and $u = \frac{2}{3}\left(\sqrt{gh_0} + \frac{x}{t}\right)$ (Ritter solution)

Between D and E: $h=h_m$ and $u=u_m$

Between E and Downstream : $h=h_d$ and u=0.

Very good agreement is shown between the TELEMAC-2D solution and the analytical solution (see Figure 3.12.2). The wave is propagated forward with the correct height. The propagation celerity is also in good agreement with the analytical solution, the calculated wave front occurring slightly later than the one given by the analytical solution.

3.12.5. CONCLUSIONS

TELEMAC-2D reproduces well the propagation of a dam break wave on an initially wet bed without friction as compared to the analytical solution to this problem.

3.12.6. STEERING FILE

```
/---------------------------------------------------------------------
/ TELEMAC2D Version v6p1 5 déc. 2011
/ Validation test case 13
/---------------------------------------------------------------------

/---------------------------------------------------------------------
/ INPUT-OUTPUT, FILES
/---------------------------------------------------------------------

FORTRAN FILE             ='princi.f'
GEOMETRY FILE            ='geo'
REFERENCE FILE           ='ref'
BOUNDARY CONDITIONS FILE ='cli.txt'
RESULTS FILE             ='res'

/---------------------------------------------------------------------
/ INPUT-OUTPUT, GRAPHICS AND LISTING
```

```
/-------------------------------------------------------------------------

LISTING PRINTOUT PERIOD         =10
GRAPHIC PRINTOUT PERIOD         =30
VARIABLES FOR GRAPHIC PRINTOUTS ='U,V,H,F,L,N'
MASS-BALANCE                    =YES
VALIDATION =YES
TITLE      ='Validation test case 13'

/-------------------------------------------------------------------------
/ INITIAL CONDITIONS
/-------------------------------------------------------------------------

INITIAL CONDITIONS ='SPECIAL'

/-------------------------------------------------------------------------
/ NUMERICAL PARAMETERS
/-------------------------------------------------------------------------

/-------------------------------------------------------------------------
/ PARAMETRES NUMERIQUES
/-------------------------------------------------------------------------
NUMBER OF SUB-ITERATIONS FOR NON-LINEARITIES =2
DISCRETIZATIONS IN SPACE                     =12;11;11;11
TREATMENT OF NEGATIVE DEPTHS                 =2
CONTINUITY CORRECTION                        =YES
NUMBER OF TIME STEPS                         =300
NUMBER OF PRIVATE ARRAYS                     =2
FREE SURFACE GRADIENT COMPATIBILITY          =0.9
TREATMENT OF THE LINEAR SYSTEM               =2
SUPG OPTION                                  =2;0;2;2
TIME STEP                                    =5.
DURATION                                     =1.5
TYPE OF ADVECTION                            =14;5;1;1

/-------------------------------------------------------------------------
/ NUMERICAL PARAMETERS, SOLVER
/-------------------------------------------------------------------------

SOLVER =1

/-------------------------------------------------------------------------
/ NUMERICAL PARAMETERS, VELOCITY-CELERITY-HIGHT
/-------------------------------------------------------------------------

MASS-LUMPING ON H           =1.
IMPLICITATION FOR DEPTH     =1.
IMPLICITATION FOR VELOCITY =0.6
/----------------------------------------------
/ IN CASE OF USE OF FINITE VOLUME
/----------------------------------------------
/EQUATIONS                        = 'SAINT-VENANT VF'
/FINITE VOLUME SCHEME             = 6
/VARIABLE TIME-STEP               = YES
/DESIRED COURANT NUMBER           = 0.8
/DURATION                         = 1.5
/----------------------------------------------
```

3.12.7. FIGURES

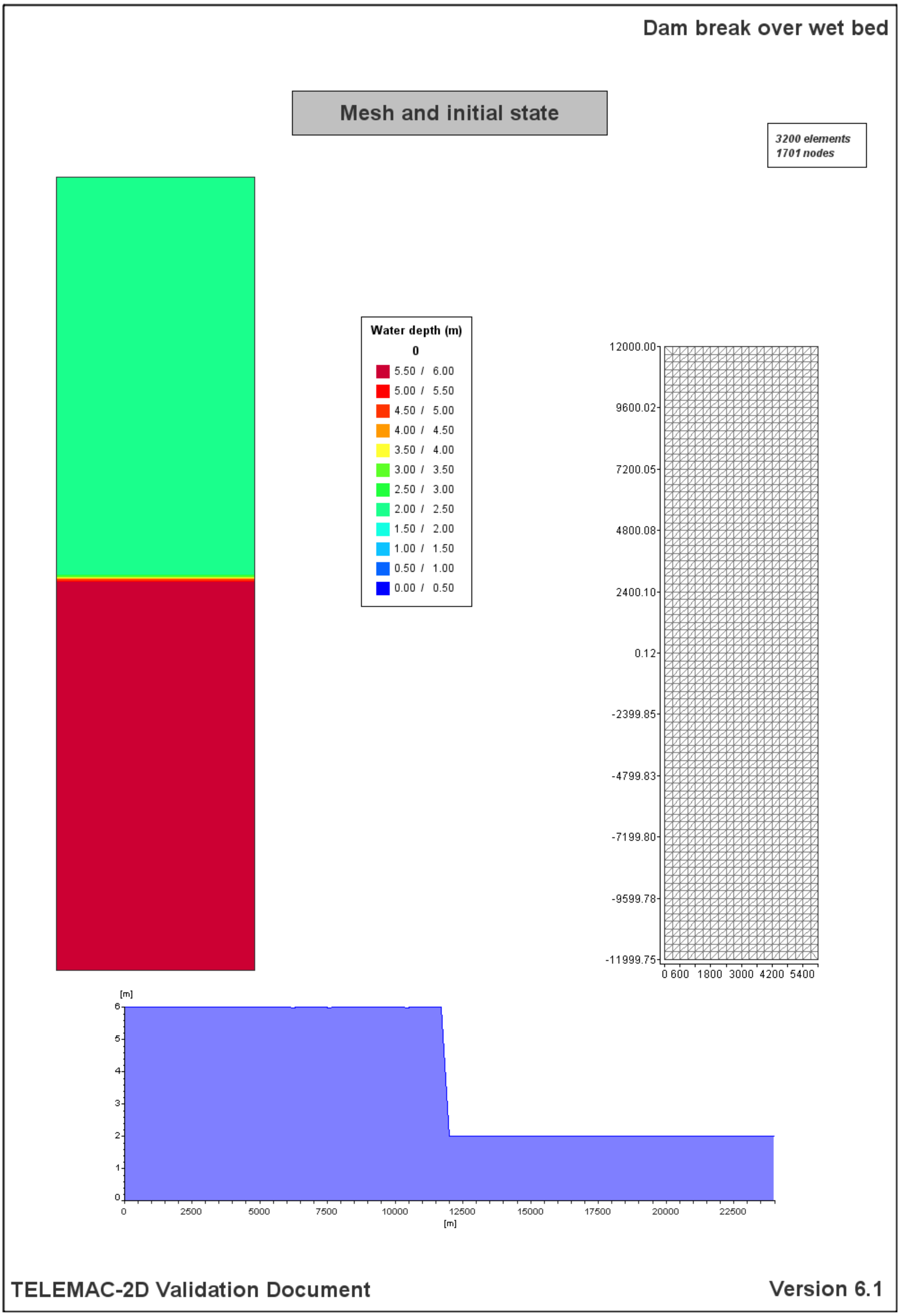


Figure 3.12.1 : Mesh and initial state.

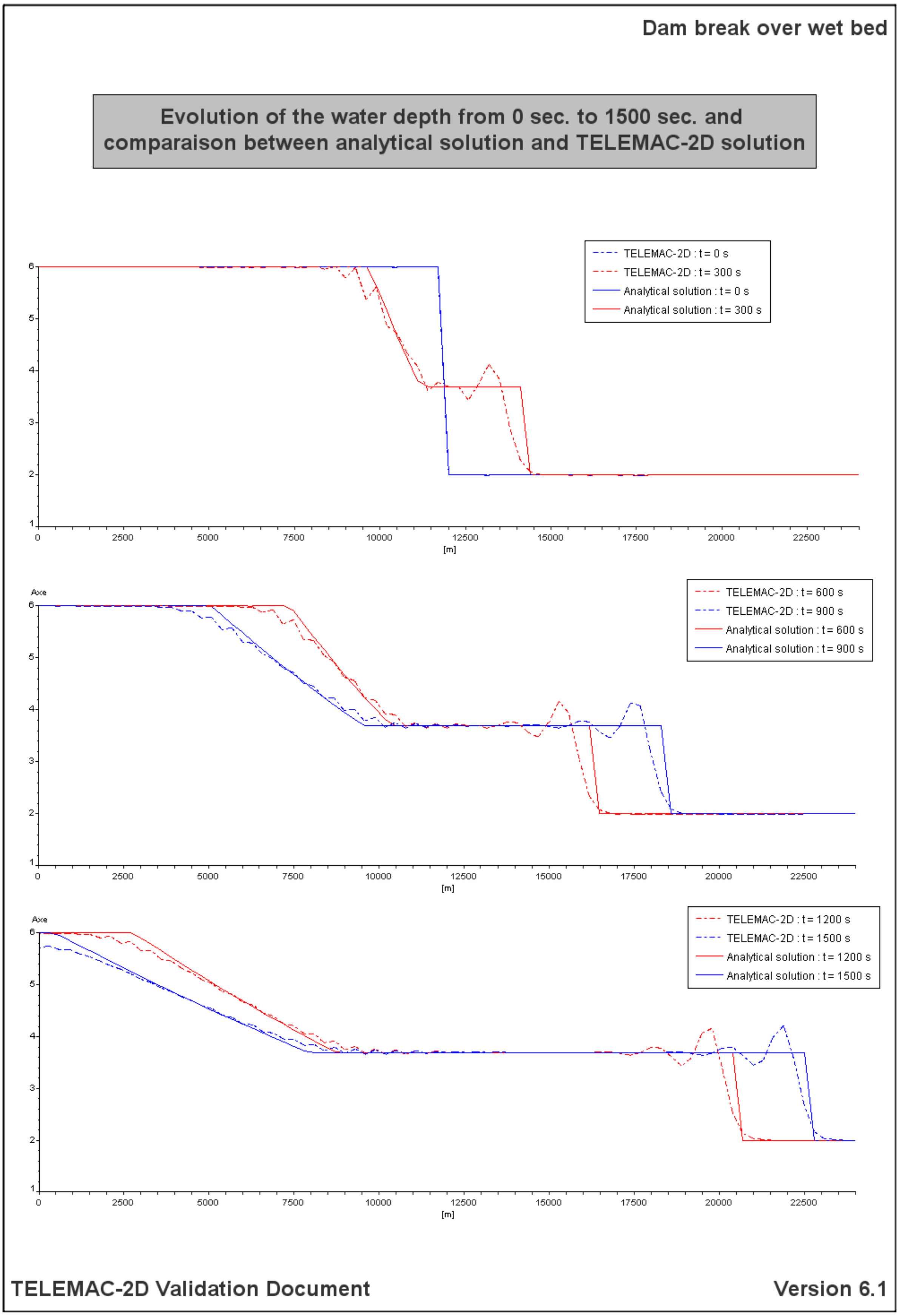
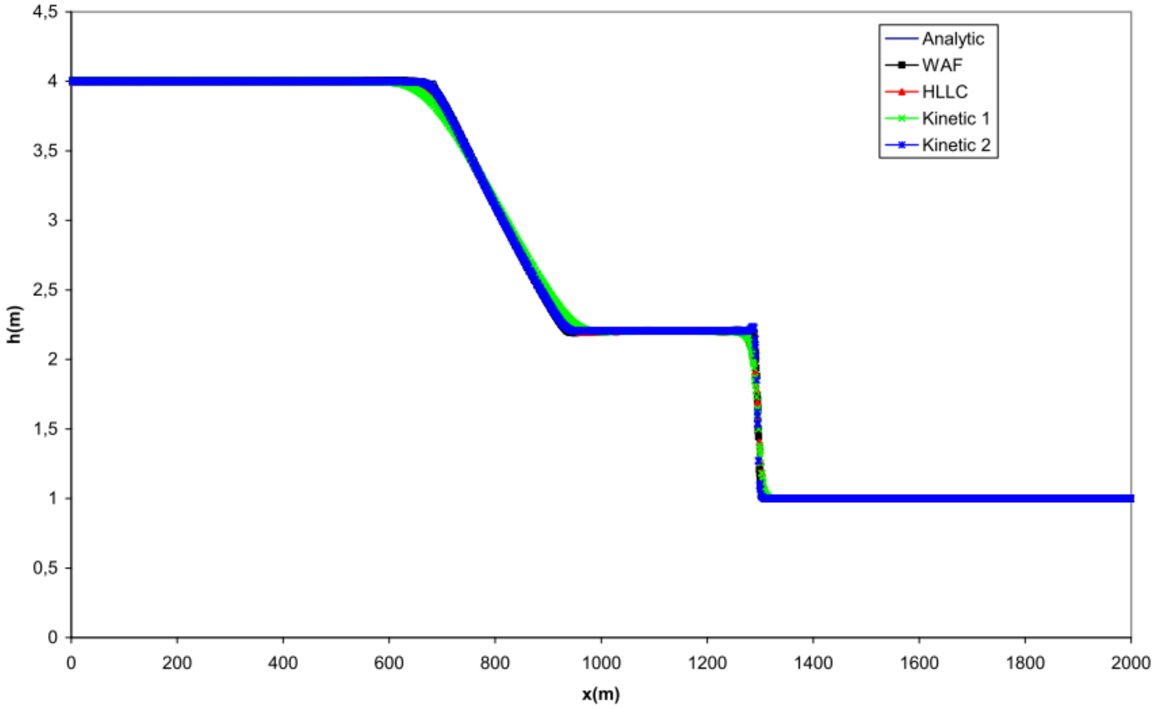
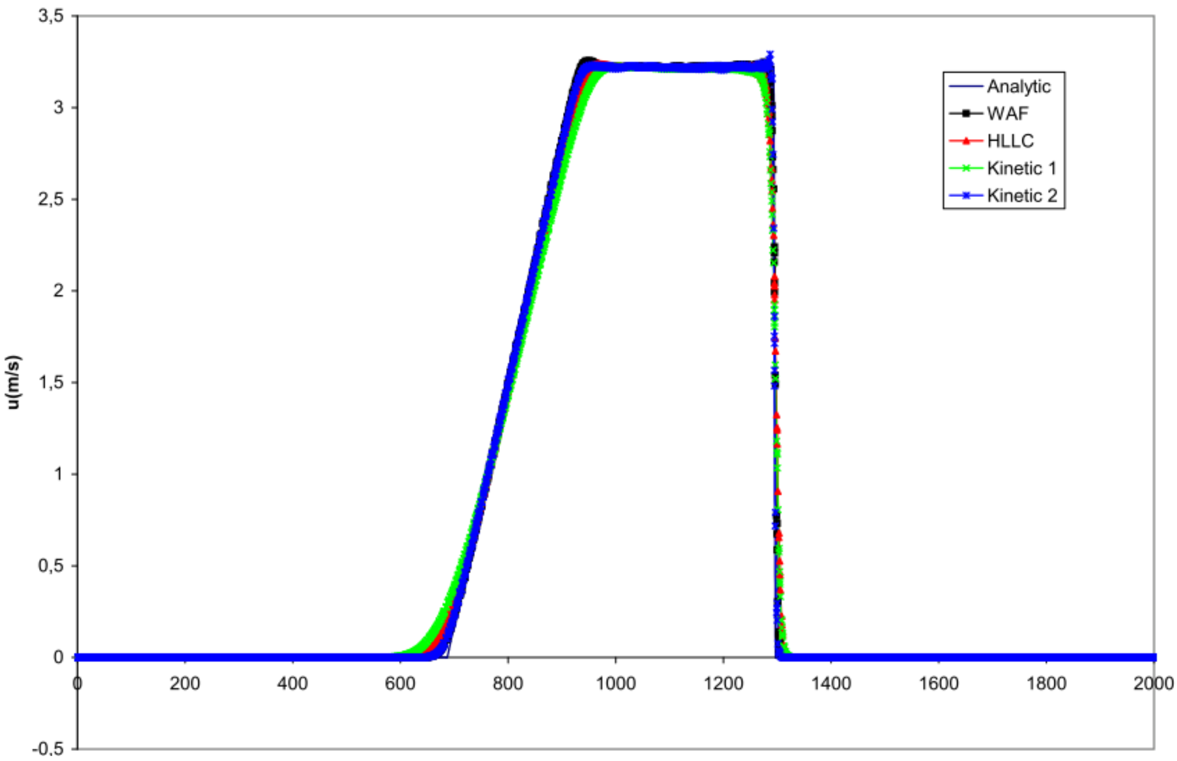


Figure 3.12.2 : Evolution of the water depth from 0 sec. to 1500 sec. and comparison between analytical solution and TELEMAC-2D solution.





Stoker problem: dambreak on a wet bed – results of some finite volumes schemes of Telemac2D.

3.13. WIND SET-UP

Title	**Wind set-up in a closed rectangular basin**
Initial study	**J.M.Hervouet – April 1992**
Last update	**MAY 2012**
Version	**TELEMAC-2D 6.1**

3.13.1. PURPOSE

To study the hydrodynamics resulting from wind set-up in a closed rectangular channel. Also to demonstrate that TELEMAC-2D produces the expected one-dimensional solution even though the grid of triangles is irregular (various sizes and orientations of triangles).

3.13.2. LINKED CLAIMS

- **Claim 2.2.2.5** TELEMAC-2D can simulate the effect of meteorological conditions such as surface layer motion generated by the wind blowing at the water surface and hydrodynamics induced by variations in atmospheric pressure, provided a depth integration of these processes is adequate.
- **Claim 2.4.2.4** Flow results are insensitive to the location of the grid points as long as the physical data and processes are adequately resolved.

3.13.3. APPROACH

The wind blowing on a 100 m x 500 m rectangular basin produces a surface current in the direction of the wind and a bottom current in the opposite direction. The total discharge in each cross-section is nil. The wind shear stress is balanced by the slope of the induced free surface.

The initial water depth is 2 m.

The wind is applied progressively during the 10 000 first time step and the result is observed after 20 000 time step.

An unrealistic gravity acceleration (10% of real g) is prescribed in order to amplify the wind effect.

Geometry:

- Size of the model: channel = 500 m x 100 m
- Water depth at rest: 2 m

Mesh:

The mesh is irregular.

- 551 triangular elements
- 319 nodes
- Maximum size range: from 14 to 24 meters

Boundaries:

- Solid walls with slip condition in the channel

Bottom:

- No bottom friction

The mesh is shown on Figure 3.13.1.

Turbulence:

- Constant viscosity = 0 m^2/s

Algorithm:

- Type of advection:
  - Characteristics on velocities
  - Conservative + modified SUPG on depth
- Type of element:
  - P1 triangle for h and for velocities
  - Implicitation for depth and velocities = 0.5
- SUPG option:
  - decentring equal to 1 for velocity
  - decentring equal to Courant number
- Solver accuracy = 10^{-6}
- Gravity = 0,981

Wind:

- Wind velocity = 20 m/s (west-east wind)
- Coefficient of wind influence $\left(= \frac{\rho_{air}}{\rho_{water}} a_{wind} \right)$ = 3,178206 x 10^{-6} with notations as given in the Principle Note (3).

Time data:

- Time step = 10 sec.
- Simulation duration = 20000 sec.

3.13.4. RESULTS

A balance between the wind stress and the surface slope occurs.

The analytical solution of this problem is given by the equation

$$H(x) = \sqrt{{H_0}^2 + \frac{2}{g} . \frac{\rho_{air}}{\rho_{water}} . a_{wind} . \|Wind\|^2 . x}$$

with H_0 solution of

$$F(x) = \left(\frac{2}{g} . \frac{\rho_{air}}{\rho_{water}} . a_{wind} . \|Wind\|^2 . L + x^2\right)^{3/2} - x^3 - \frac{3}{g} . \frac{\rho_{air}}{\rho_{water}} . a_{wind} . \|Wind\|^2 . L . H_{initial} = 0$$

In our case, $H_0 \approx 1.8332523$ and consequently $H_L \approx 2.1579431$.

The solution is one-dimensional (independent of the y-axis) as shown on Figure 3.13.1. The water surface elevation difference between the two extremities of this 500 m long channel is 32.45558 cm whereas it should be 32.46909 cm according to the analytical solution.

3.13.5. CONCLUSIONS

TELEMAC-2D is able to compute wind generated flows on the basis of the empirical wind shear stress formulation presented in the Principle Note. The solution computed by TELEMAC-2D in this one-dimensional test case is independent of the computational grid characteristics although the grid meshes are very irregular.

3.13.6. STEERING FILE

```
/---------------------------------------------------------------------
/ TELEMAC2D Version v6p1 5 déc. 2011
/ Validation test case 14
/---------------------------------------------------------------------

/---------------------------------------------------------------------
/ INPUT-OUTPUT, FILES
/---------------------------------------------------------------------

FORTRAN FILE             ='princi.f'
GEOMETRY FILE            ='geo'
REFERENCE FILE           ='ref'
STEERING FILE            ='cas.txt'
BOUNDARY CONDITIONS FILE ='cli.txt'
RESULTS FILE             ='res'

/---------------------------------------------------------------------
/ WIND
/---------------------------------------------------------------------

WIND VELOCITY ALONG X         =20.
WIND VELOCITY ALONG Y         =0.
COEFFICIENT OF WIND INFLUENCE =3.178206E-6
WIND                          =YES
VELOCITY DIFFUSIVITY          =0.

/---------------------------------------------------------------------
/ EQUATIONS, INITIAL CONDITIONS
/---------------------------------------------------------------------
```

```
INITIAL DEPTH      =2.
INITIAL CONDITIONS ='CONSTANT DEPTH'

/----------------------------------------------------------------------
/ INPUT-OUTPUT, GRAPHICS AND LISTING
/----------------------------------------------------------------------

LISTING PRINTOUT PERIOD                                =10
INFORMATION ABOUT SOLVER                               =YES
GRAPHIC PRINTOUT PERIOD                                =100
VARIABLES FOR GRAPHIC PRINTOUTS                        ='U,V,H,S,X,Y,N'
MASS-BALANCE                                           =YES
NUMBER OF FIRST TIME STEP FOR GRAPHIC PRINTOUTS =10000

/----------------------------------------------------------------------
/ INPUT-OUTPUT, INFORMATION
/----------------------------------------------------------------------

VALIDATION =YES
TITLE      ='Validation test case 14'

/----------------------------------------------------------------------
/ NUMERICAL PARAMETERS
/----------------------------------------------------------------------

NUMBER OF TIME STEPS  =20000
MATRIX STORAGE        =3
SUPG OPTION           =1;2;2;2
TIME STEP             =10.
MATRIX-VECTOR PRODUCT =2

/----------------------------------------------------------------------
/ NUMERICAL PARAMETERS, SOLVER
/----------------------------------------------------------------------

SOLVER ACCURACY                         =1.E-6
MAXIMUM NUMBER OF ITERATIONS FOR SOLVER =200

/----------------------------------------------------------------------
/ NUMERICAL PARAMETERS, VELOCITY-CELERITY-HIGHT
/----------------------------------------------------------------------

IMPLICITATION FOR DEPTH    =0.5
INITIAL GUESS FOR H        =0
IMPLICITATION FOR VELOCITY =0.5

/----------------------------------------------------------------------
/ PHYSICAL CONSTANTS
/----------------------------------------------------------------------

GRAVITY ACCELERATION =0.981
```

3.13.7. FIGURES

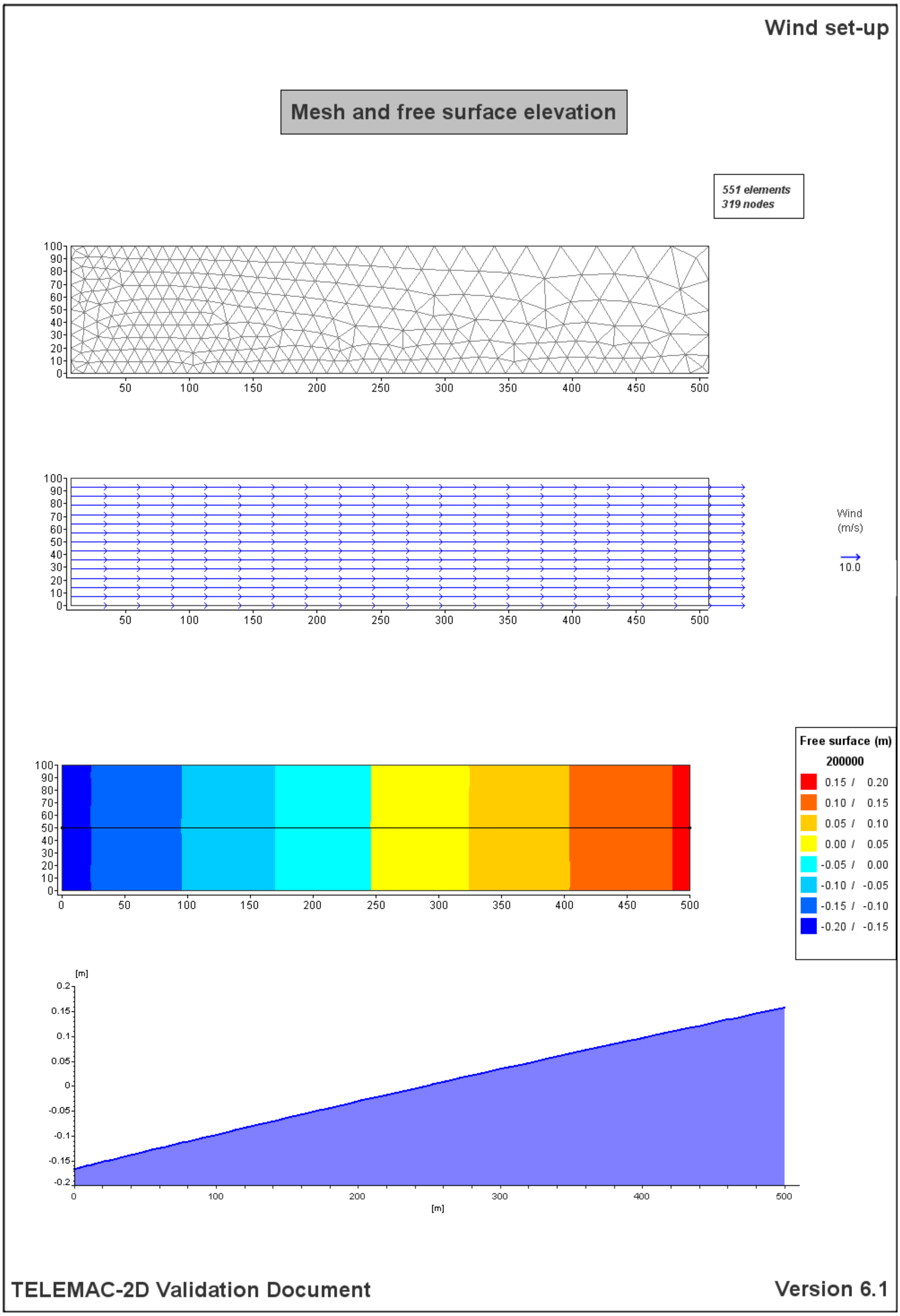


Figure 3.13.1 : Mesh and free surface elevation.

3.14. RIVER CONFLUENCE

Title	**Flow at a river confluence**
Initial study	**J.-M. Hervouet – April 1992**
Last update	**MAY 2012**
Version	**TELEMAC-2D 6.1**

3.14.1. PURPOSE

To demonstrate that TELEMAC-2D can model the flow that occurs at a river confluence.

3.14.2. LINKED CLAIMS

- **Claim 2.2.1.10** TELEMAC-2D can be used for detailed analysis of flow conditions and head-loss generated at a river confluence.
- **Claim 2.2.2.2** TELEMAC-2D is capable of modelling flows in a network of open channels and rivers joined together in any topological configuration.
- **Claim 2.2.2.3** TELEMAC-2D can compute steady and transient flow regimes. Steady flow is achieved by integrating through time until a steady condition is obtained.
- **Claim 2.2.2.7** TELEMAC-2D is capable of representing the hydrodynamic influence of bed friction as a bottom boundary condition in the vertical averaging process. Roughness options available are quadratic (Chezy, Strickler, Manning), Nikuradse or linear friction laws.

3.14.3. APPROACH

The model represents the junction between two rectilinear laboratory channels with rectangular cross-sections and constant slope. The main channel is 0.8 m broad whereas its influent is 0.5 m broad. Both have a slope of 10^{-3} m/m. The two channels join with an angle of 55°.

<u>Geometry:</u>

- Size of the model :
  - main channel: 10.8 m x 0.8 m
  - influent: 3.2 m x 0.5 m
- Free surface at rest = 0.2852 m

<u>Mesh:</u>

The mesh is refined near the confluence as shown on Figure 3.14.1.

- 6168 triangular elements

- 3303 nodes
- Maximum size range: from 0.03 to 0.1 metres

Boundaries:

- main channel:
  - channel entrance: Q = 0.07 m^3/s
  - channel outlet: H = 0.2852 m
- Influent:
  - channel entrance: Q = 0.035 m^3/s
- Lateral boundaries: solid walls with slip condition in the channel

Bottom:

- Strickler formula with friction coefficient = 62

The mesh is shown on Figure 3.14.1 and the topography on Figure 3.14.2.

Turbulence:

- Constant viscosity equal to 10^{-3} m^2/s

Algorithm:

- Type of advection:
  - Characteristics on velocities and on depth
- Type of element:
  - P1 triangle for h and for velocities
- Solver : Conjugate gradient
- Solver accuracy = 10^{-4}
- Implicitation for depth and for velocity = 0.6

Time data:

- Time step = 0.1 sec.
- Simulation duration = 100 sec.

3.14.4. RESULTS

Initially the water level is horizontal. In the main channel and in the lateral channel, the free surface increases with time. At the end of the calculation the water surface profile is constant in time downstream and upstream from the confluence which shows that the computation has converged. The water depths in both channels (upstream and downstream) tend to be uniform. The water level upstream from the confluence is 0.30 m higher than the water level downstream. Close to the confluence, the water surface is rapidly varying. The velocity field is regular in the whole domain (see Figure 3.14.2). No back eddy is computed at the junction of the two rivers with the turbulence model used in this test case despite mesh refinement in this area; such back eddy has been observed on physical model experiments (see ref. (15)).

3.14.5. CONCLUSIONS

TELEMAC-2D reproduces appropriately free surface variations at a river confluence. However, in order to simulate in detail the flow pattern in such conditions, more sophisticated turbulence model should be used (see e.g; test case n# 12 showing a cavity flow).

3.14.6. STEERING FILE

```
/---------------------------------------------------------------------
/ TELEMAC2D Version v6p1 May 2012
/ Validation test case 15
/---------------------------------------------------------------------
/---------------------------------------------------------------------
/ INPUT-OUTPUT, FILES
/---------------------------------------------------------------------

FORTRAN FILE             ='princi.f'
GEOMETRY FILE            ='geo'
REFERENCE FILE           ='ref'
BOUNDARY CONDITIONS FILE ='cli.txt'
RESULTS FILE             ='res'

/---------------------------------------------------------------------
/ INPUT-OUTPUT, GRAPHICS AND LISTING
/---------------------------------------------------------------------

LISTING PRINTOUT PERIOD         =100
VARIABLES FOR GRAPHIC PRINTOUTS ='U,V,H,S,B'
MASS-BALANCE                    =YES
GRAPHIC PRINTOUT PERIOD         =100

/---------------------------------------------------------------------
/ PARAMETERS
/---------------------------------------------------------------------

FRICTION COEFFICIENT   =62.
LAW OF BOTTOM FRICTION =3
VELOCITY DIFFUSIVITY   =1.E-3

/---------------------------------------------------------------------
/ EQUATIONS, BOUNDARY CONDITIONS
/---------------------------------------------------------------------

VELOCITY PROFILES     =2;2;2
PRESCRIBED FLOWRATES  =0.;0.035;0.070
PRESCRIBED ELEVATIONS =0.2852;0.;0.

/---------------------------------------------------------------------
/ EQUATIONS, INITIAL CONDITIONS
/---------------------------------------------------------------------

INITIAL ELEVATION  =0.2852
INITIAL CONDITIONS ='CONSTANT ELEVATION'

/---------------------------------------------------------------------
/ INPUT-OUTPUT, INFORMATION
/---------------------------------------------------------------------

VALIDATION =YES
TITLE      ='Validation test case 15'
```

```
/----------------------------------------------------------------------
/ NUMERICAL PARAMETERS
/----------------------------------------------------------------------

TIME STEP                      =0.1
NUMBER OF TIME STEPS           =1000
TREATMENT OF THE LINEAR SYSTEM =2

/----------------------------------------------------------------------
/ NUMERICAL PARAMETERS, SOLVER
/----------------------------------------------------------------------

SOLVER =1
MASS-LUMPING ON H          =1.
IMPLICITATION FOR DEPTH    =0.6
IMPLICITATION FOR VELOCITY =0.6
```

3.14.7. FIGURES

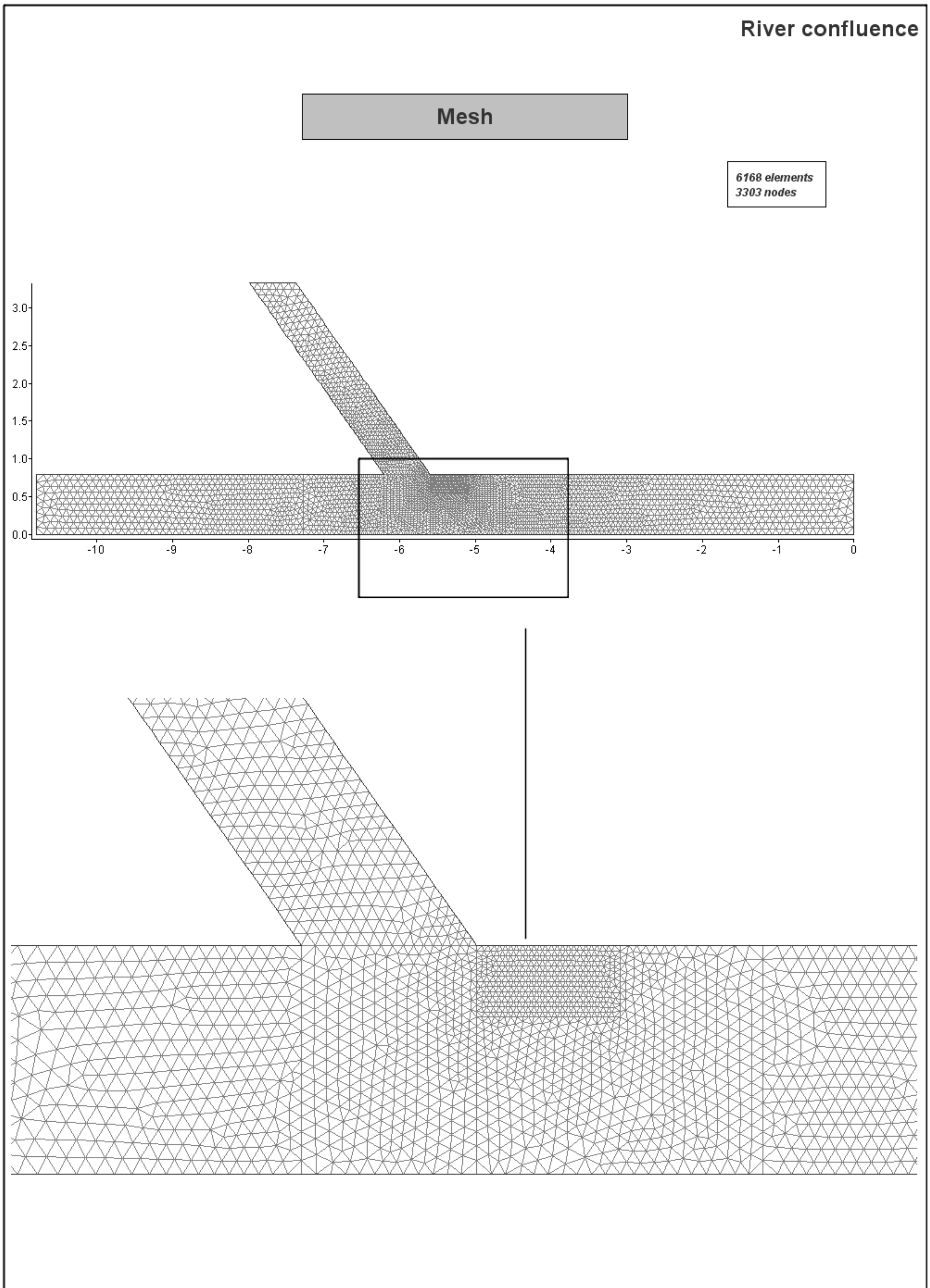


Figure 3.14.1: River confluence, geometry and used mesh.

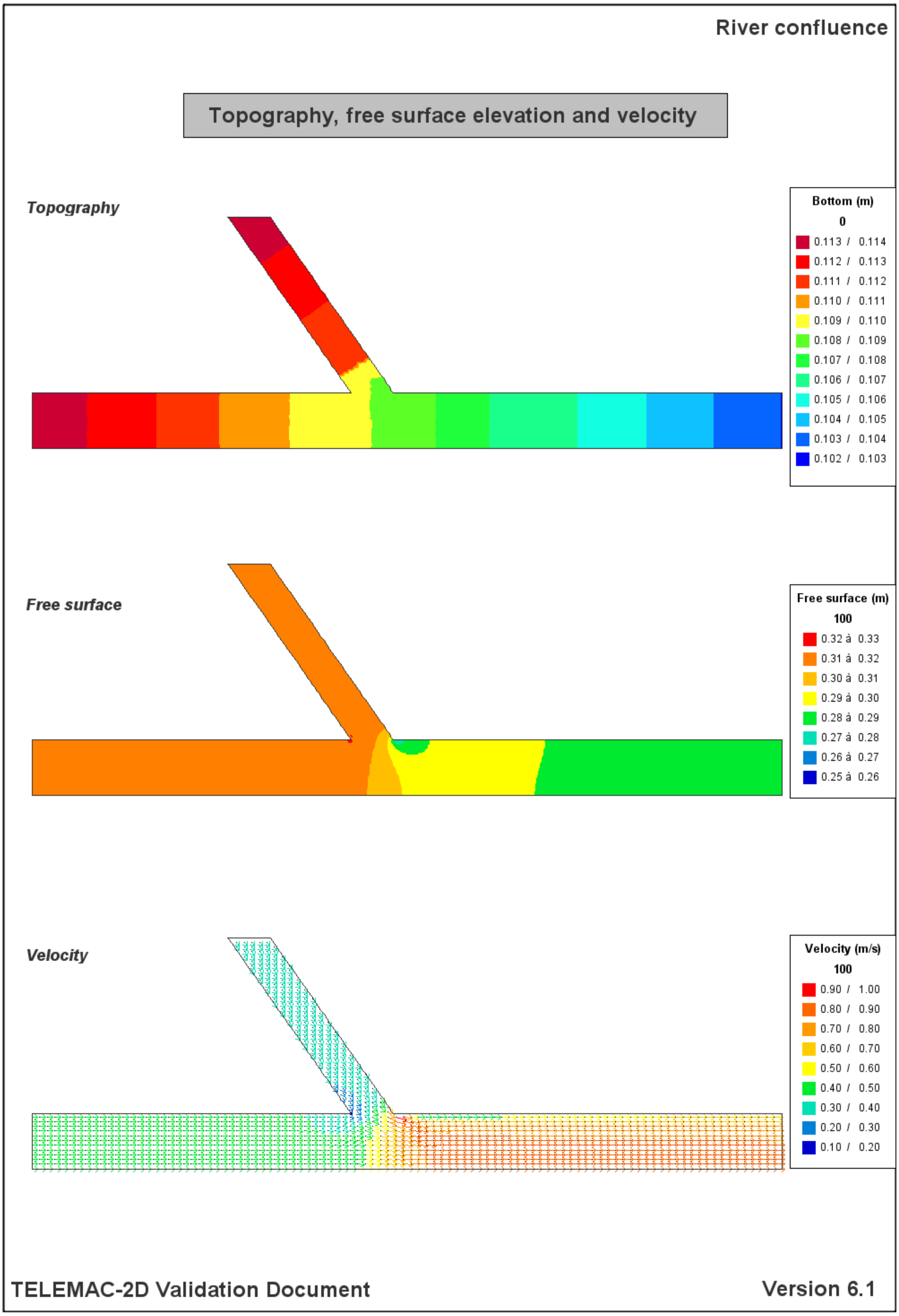
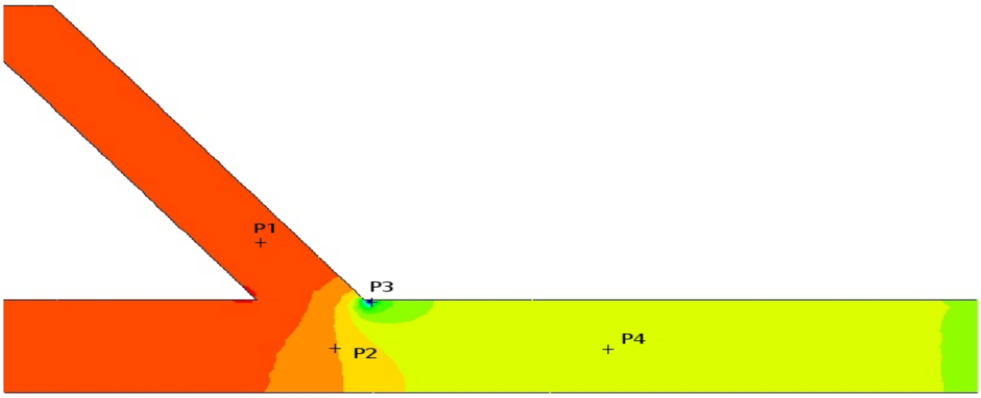


Figure 3.14.2 : Topography, free surface elevation and velocity field.



River confluence – Finite volume result (WAF scheme)

3.15. CAVITY

Title	**Vortices generated in a cavity along a channel, the cavity presenting a shore in its cross-section**
Initial study	**A. Krings – October 1997**
Last update	**May 2011**
Version	**TELEMAC-2D 6.1**

3.15.1. PURPOSE

This test case is inspired from the paper from Faure (1995) (16) where the constant viscosity and k-ε turbulence models are compared to tests performed in a physical model. The Smagorinski model has been added to TELEMAC-2D after this comparison because it proved to be more efficient on this class of applications.

3.15.2. LINKED CLAIMS

- **Claim 2.2.2.10** Turbulence effect can be modelled through different options of increasing complexity: constant viscosity, Elder model (linear viscosity), k-ε model (two equations) or Smagorinski options are available.
- **Claim 2.3.2.3** The different treatments of turbulence in TELEMAC-2D are satisfactory for many free surface flow problems in rivers, estuaries and coastal waters.

3.15.3. APPROACH

A 23 metres long and 3 metres wide channel is fed by a constant flow. The fluid is well-mixed over the 0.43 metres depth: no stratification is present. On the left bank of the channel is located a harbour cavity. The cavity has a cross-section (perpendicular to the channel axis) terminating with a shallow beach having a mild slope (see Figure 3.15.1).

This three-dimensional configuration induces the formation of large size eddies. Bijvelds et al. (18) have shown that vortices induced in a similar harbour with constant depth are better reproduced with 3-D simulation. In our case, a 2-D simulation is possible since the beach where large size vortices are present is very shallow. The flow is typically unsteady although boundary conditions are steady. The Smagorinski turbulence model is tested on this configuration.

Geometry:

- Size of the model: channel = 23 m x 3 m; cavity = 6 m x 3 m
- Water depth at rest = 0.43 m

Mesh:

The mesh is dense, particularly in the cavity where the vortices take place and on the steep junction slope between the cavity and the beach. It is made up with quadrangles split into two triangles (see Figure 3.15.1).

- 8160 triangular elements
- 4233 nodes
- Size range: from 0.033 to 0.34 metres

Boundaries:

- Channel entrance : Q = 0.155 m^3/s imposed
- Channel outlet: h = 0.445 m imposed
- Lateral boundaries: solid walls with slip condition in the channel and no-slip condition in the cavity

Bottom:

- In order to simulate small size bottom variations, the bed is uneven.
- Manning friction law with bottom roughness = 0.015

Bed isolines and a cross-section of the cavity are shown on Figure 3.15.1.

Turbulence:

- Model of Smagorinski with coefficient of Smagorinski = 0,1

Algorithm:

- Type of advection:
  - Characteristics on velocities
  - Conservative + modified SUPG on depth
- Type of element:
  - Quasi-bubble triangle for velocities
  - P1 triangle for h
- GMRES solver
- Accuracy = 10^{-3}

Time data:

- Time step = 0,2 sec.
- Simulation duration = 200 sec.

3.15.4. RESULTS

Due to the strong shearing between the channel and the cavity, a normalised velocity $u/\|u\|^{0.75}$ is used for the presentation of velocity patterns.

The velocity pattern and the free surface elevation in the cavity are shown on Figure 3.15.2.

The flow is unsteady particularly in the cavity even though the boundary conditions are steady. Great and small eddies appear and move periodically in the cavity (see Figure 3.15.3 & Figure 3.15.4). The period of great eddies is longer than the one of small eddies.

3.15.5. CONCLUSIONS

The phenomena observed on physical model are successfully reproduced by TELEMAC-2D with the Smagorinski model, although hydrodynamics are three dimensional in reality. In particular, the small and great eddies structures in the cavity are well represented.

3.15.6. STEERING FILE

```
/---------------------------------------------------------------------
/ TELEMAC2D Version v6p1 May 2012
/ Validation test case 16
/---------------------------------------------------------------------

/---------------------------------------------------------------------
/ INPUT-OUTPUT, FILES
/---------------------------------------------------------------------

FORTRAN FILE              ='princi.f'
PREVIOUS COMPUTATION FILE ='prev'
GEOMETRY FILE             ='geo'
REFERENCE FILE            ='ref'
BOUNDARY CONDITIONS FILE  ='cli.txt'
RESULTS FILE              ='res'

/---------------------------------------------------------------------
/ INPUT-OUTPUT, GRAPHICS AND LISTING
/---------------------------------------------------------------------

LISTING PRINTOUT PERIOD         =25
VARIABLES FOR GRAPHIC PRINTOUTS ='U,V,H,B,S,D'
MASS-BALANCE                    =YES
GRAPHIC PRINTOUT PERIOD         =25

/---------------------------------------------------------------------
/ EQUATIONS
/---------------------------------------------------------------------

FRICTION COEFFICIENT                  =0.015D0
TURBULENCE MODEL FOR SOLID BOUNDARIES =1
LAW OF BOTTOM FRICTION                =4
TURBULENCE MODEL                      =4

/---------------------------------------------------------------------
/ EQUATIONS, BOUNDARY CONDITIONS
/---------------------------------------------------------------------

PRESCRIBED FLOWRATES  =0.; 0.155
PRESCRIBED ELEVATIONS =0.445; 0.

/---------------------------------------------------------------------
/ INPUT-OUTPUT, INFORMATION
/---------------------------------------------------------------------

VALIDATION            =YES
COMPUTATION CONTINUED =YES
TITLE                 ='Validation test case 16'
/---------------------------------------------------------------------
/ NUMERICAL PARAMETERS
/---------------------------------------------------------------------
```

```
NUMBER OF TIME STEPS      =1000
DISCRETIZATIONS IN SPACE =12;11;11;11
TIME STEP                 =0.2

/----------------------------------------------------------------------
/ NUMERICAL PARAMETERS, SOLVER
/----------------------------------------------------------------------

SOLVER          =7
SOLVER ACCURACY =1.E-3
SOLVER OPTION   =3

/----------------------------------------------------------------------
/ NUMERICAL PARAMETERS, VELOCITY-CELERITY-HIGHT
/----------------------------------------------------------------------

MASS-LUMPING ON H           =1.
IMPLICITATION FOR DEPTH     =0.6
IMPLICITATION FOR VELOCITY =0.6
```

3.15.7. FIGURES

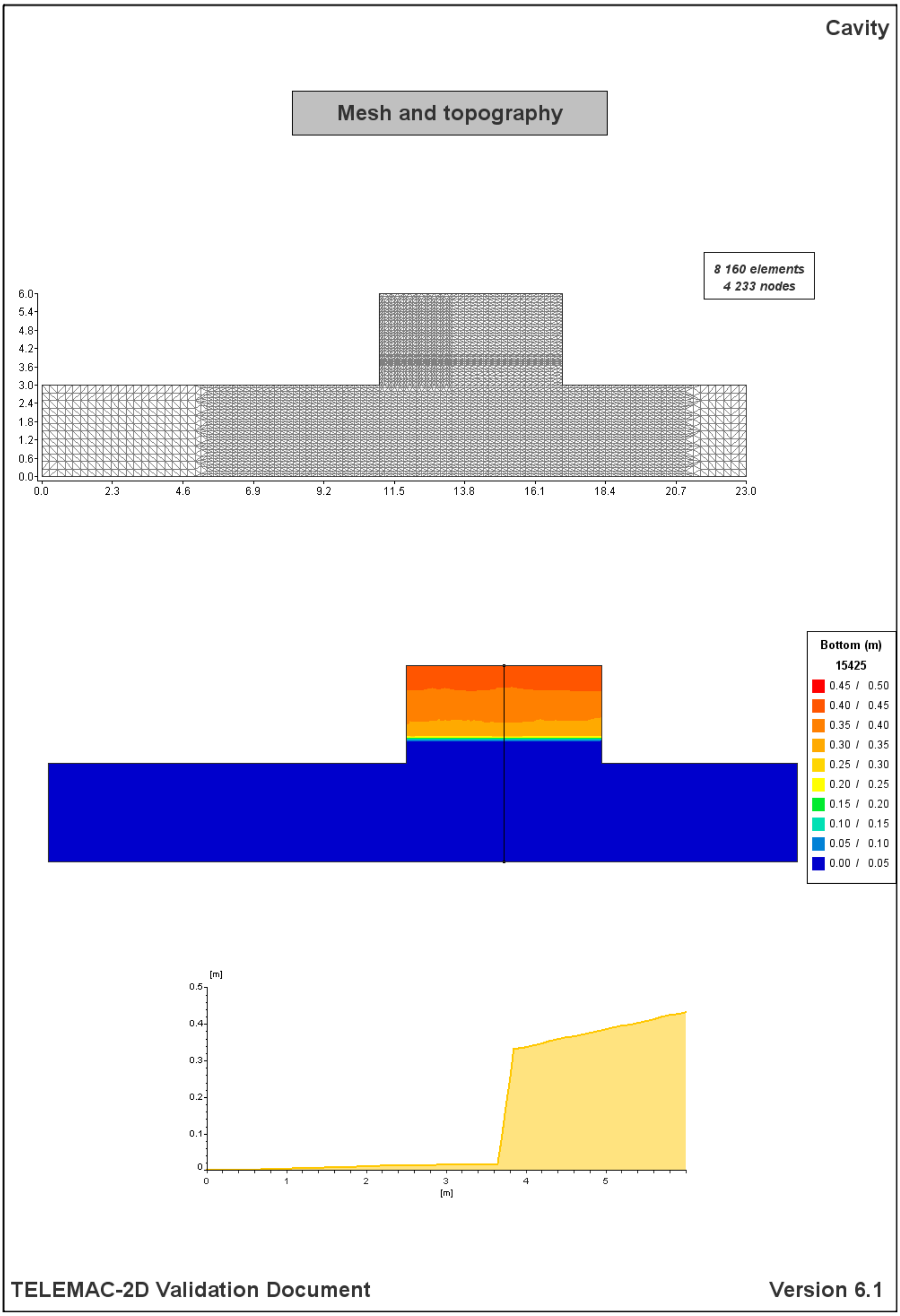


Figure 3.15.1 : Mesh and topography.

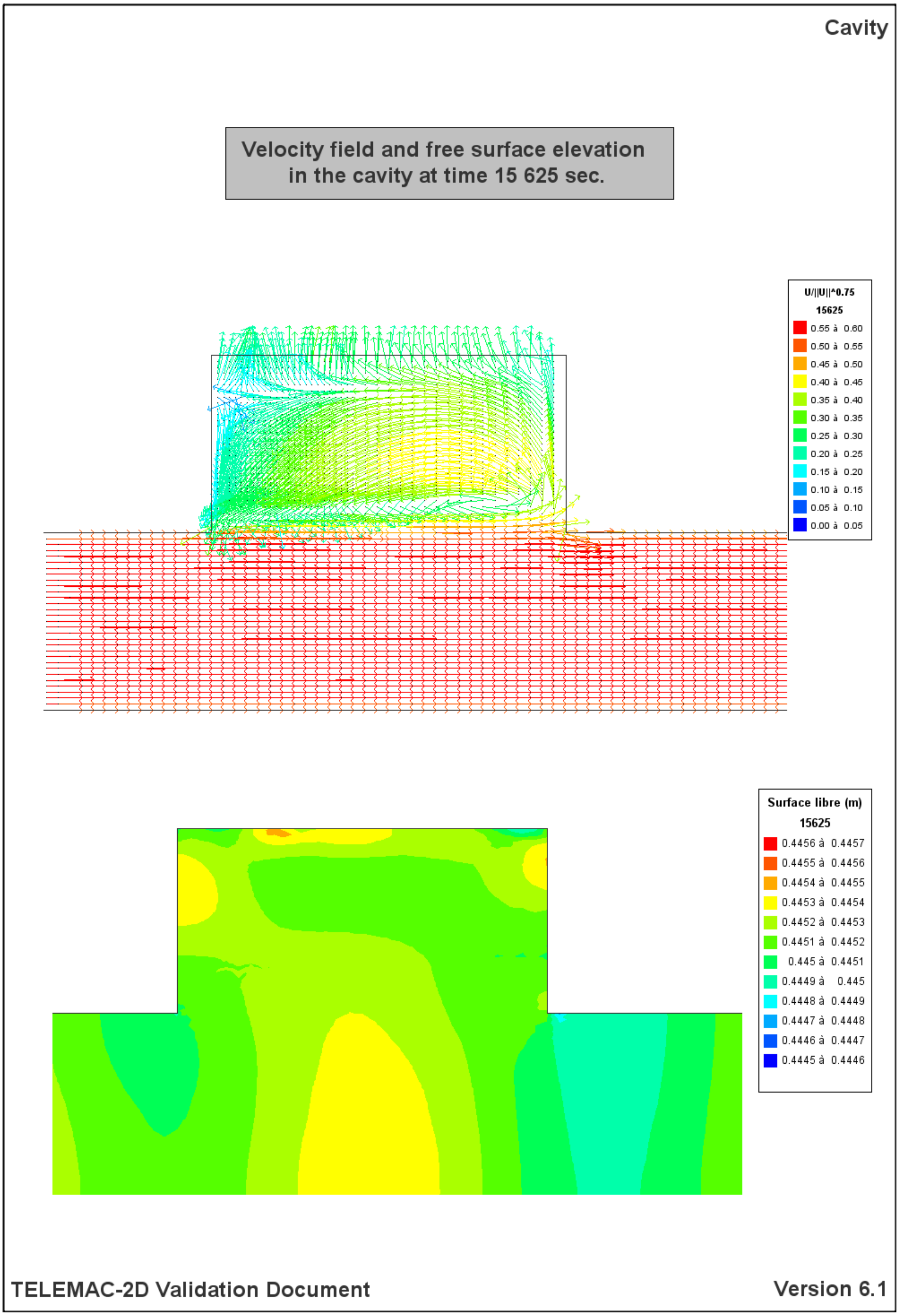


Figure 3.15.2 : Velocity field and free surface in the cavity at time 15225 sec.

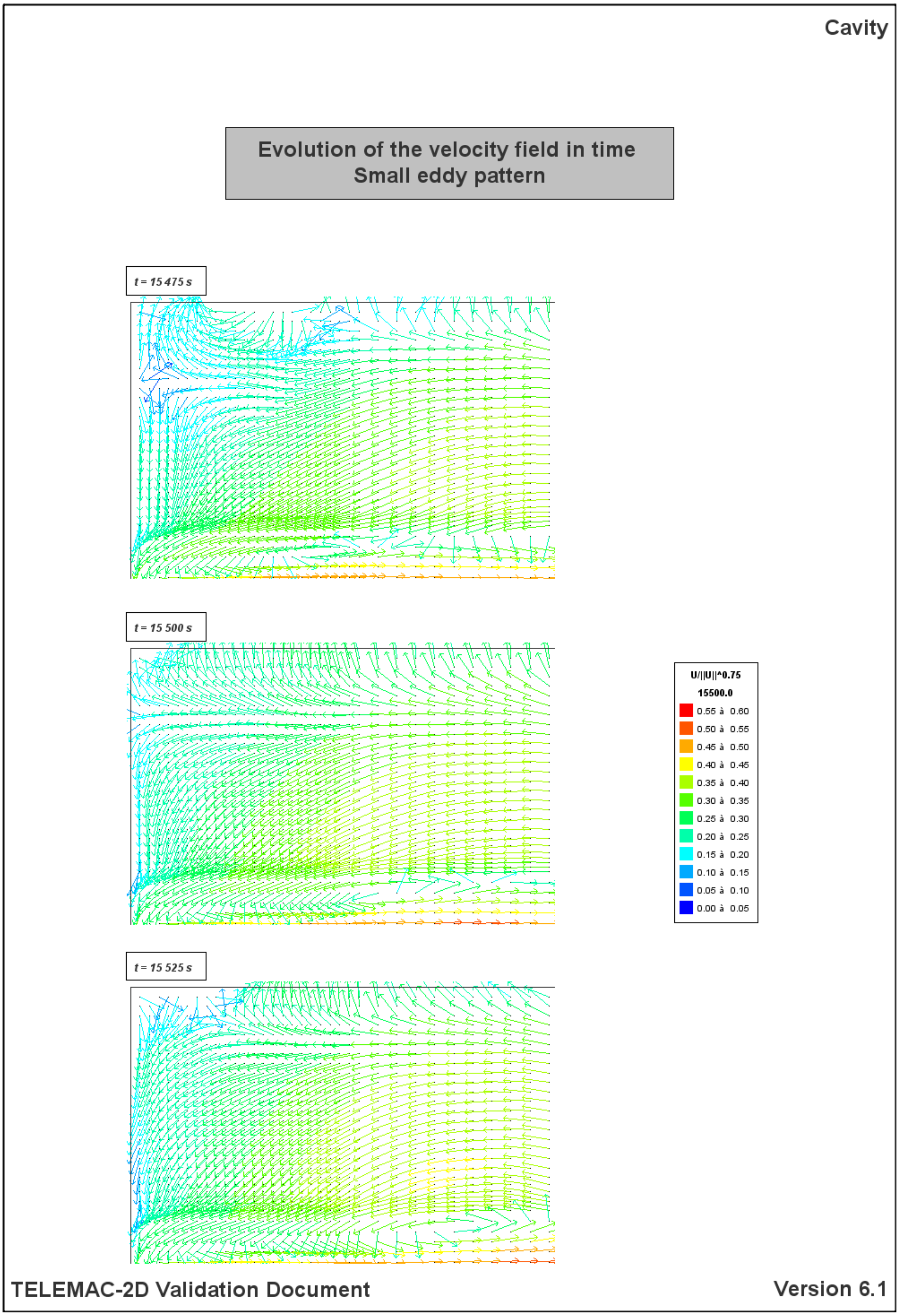


Figure 3.15.3 : Evolution of the velocity field in time – Small eddy pattern.

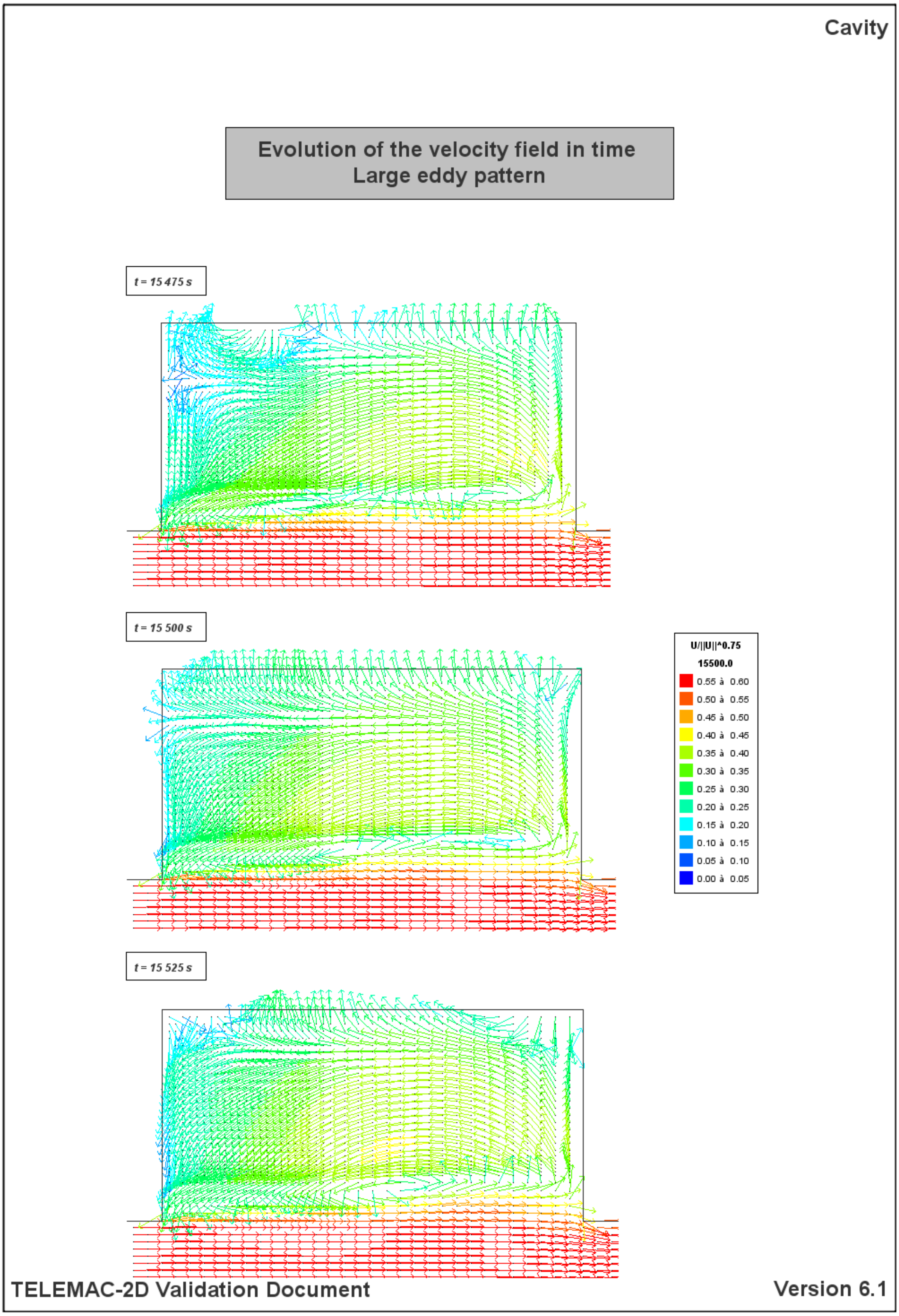


Figure 3.15.4 : Evolution of the velocity field in time – Large eddy pattern.

3.16. BREAKWATER

Title	**Flow over a breakwater**
Initial study	**J.-M. Hervouet – April 1992**
Last update	**May 2012**
Version	**TELEMAC-2D 6.1**

3.16.1. PURPOSE

To demonstrate the capability of TELEMAC-2D to model a situation with a very steep bed thanks to a refined mesh. Two pools are located downstream of the breakwater. They induce flow regime transitions from super-critical to sub-critical flow.

3.16.2. LINKED CLAIMS

- **Claim 2.2.1.3** TELEMAC-2D can be used to investigate the hydrodynamic impact of engineering works in coastal or estuarine environment such as breakwaters, piers, navigation channels, submersible dikes, etc.
- **Claim 2.2.2.7** TELEMAC-2D is capable of representing the hydrodynamic influence of bed friction as a bottom boundary condition in the vertical averaging process. Roughness options available are quadratic (Chezy, Strickler, Manning), Nikuradse or linear friction laws.
- **Claim 2.2.2.16** TELEMAC-2D can represent the wetting and drying of intertidal zones (maritime applications) and of inundated plains (river applications). The numerical treatment of this process is very robust.
- **Claim 2.4.2.3** TELEMAC-2D can solve the SWE on an unstructured grid of triangles, which allows grid resolution to vary in space and enables complex coastlines to be represented accurately.
- **Claim 2.4.2.14** TELEMAC-2D offers unconditionally stable semi-implicit solution methods. However, it is recommended to adopt a time step such that the Courant number is not larger than 3 in general.

3.16.3. APPROACH

The computational domain is a rectangle with flat bottom and initially dry bed in which a 4 metres high breakwater with trapezoidal cross-section splits the domain into two zones. Water is introduced in the area on the one side of the breakwater until this one overflows. The area protected by the breakwater, which includes two pools, is suddenly flooded.

Geometry:

- Size of the model: domain = 1232.5 m x 1000 m
- Water depth at rest = 5 m

Mesh:

The mesh is irregular and refined near the breakwater.

- 19202 triangular elements
- 9734 nodes
- Maximum size range: from 3 to 37 metres

Boundaries:

- Domain entrance: free depth and discharges imposed
- Domain outlet: free depth and incident wave
- Lateral boundaries: solid walls with slip condition

Bottom:

- Strickler formula with friction coefficient = 55

The mesh and the topography are shown on Figure 3.16.1.

Turbulence:

- Model of constant viscosity
- Velocity diffusivity = 2 m^2/s

Algorithm:

- Type of advection:
  - Characteristics on velocities
  - Conservative + modified SUPG on depth
- Type of element:
  - P1 triangle for h and for velocities
- Implicitation for depth and velocities = 0.6
- Solver: Conjugate gradient
- Solver accuracy = 10^{-6}

Time data:

- Time step = 1 sec.
- Simulation duration = 2000 sec.

3.16.4. RESULTS

Even though the slopes of the breakwater are steep, progressive flooding of the breakwater and overflowing with mixed super-critical/sub-critical flow conditions are reproduced as expected. The time when overflowing occurs is 1500 sec (see Figure 3.16.2).

3.16.5. CONCLUSIONS

TELEMAC-2D is capable of simulating the flooding of a breakwater including the different flow regimes and regime transitions it induces provided that the mesh is sufficiently refined. Hydrodynamics are appropriately reproduced at each stage of the flooding event (free overfall or flooded breakwater).

3.16.6. STEERING FILE

```
/---------------------------------------------------------------------
/ TELEMAC2D Version v6p1 May 2012
/ Validation test case 17
/---------------------------------------------------------------------

/---------------------------------------------------------------------
/ INPUT-OUTPUT, FILES
/---------------------------------------------------------------------
TITLE      ='Validation test case 17'
VALIDATION =YES

FORTRAN FILE              ='princi.f'
GEOMETRY FILE             ='geo'
REFERENCE FILE            ='ref'
BOUNDARY CONDITIONS FILE  ='cli.txt'
RESULTS FILE              ='res'

/---------------------------------------------------------------------
/ INPUT-OUTPUT, GRAPHICS AND LISTING
/---------------------------------------------------------------------

LISTING PRINTOUT PERIOD         =20
GRAPHIC PRINTOUT PERIOD         =50
VARIABLES FOR GRAPHIC PRINTOUTS ='U,V,H,B,S'
MASS-BALANCE                    =YES
NUMBER OF FIRST TIME STEP FOR GRAPHIC PRINTOUTS = 12000

/---------------------------------------------------------------------
/ EQUATIONS
/---------------------------------------------------------------------

FRICTION COEFFICIENT                  =55.
LAW OF BOTTOM FRICTION                =3
VELOCITY DIFFUSIVITY                  =2.

/---------------------------------------------------------------------
/ INITIAL CONDITIONS
/---------------------------------------------------------------------

INITIAL CONDITIONS ='PARTICULAR'

/---------------------------------------------------------------------
/ BOUNDARY CONDITIONS
/---------------------------------------------------------------------

PRESCRIBED FLOWRATES  =0.; 100.

/---------------------------------------------------------------------
/ NUMERICAL PARAMETERS
/---------------------------------------------------------------------

NUMBER OF TIME STEPS           =14000
TIME STEP                      =1;
TREATMENT OF THE LINEAR SYSTEM =2

/---------------------------------------------------------------------
/ NUMERICAL PARAMETERS, SOLVER
/---------------------------------------------------------------------
```

```
SOLVER          =1
SOLVER ACCURACY =1.E-6

/----------------------------------------------------------------------
/ NUMERICAL PARAMETERS, VELOCITY-CELERITY-HIGHT
/----------------------------------------------------------------------

MASS-LUMPING ON H           =1.
IMPLICITATION FOR DEPTH     =0.6
IMPLICITATION FOR VELOCITY =0.6
```

3.16.7. FIGURES

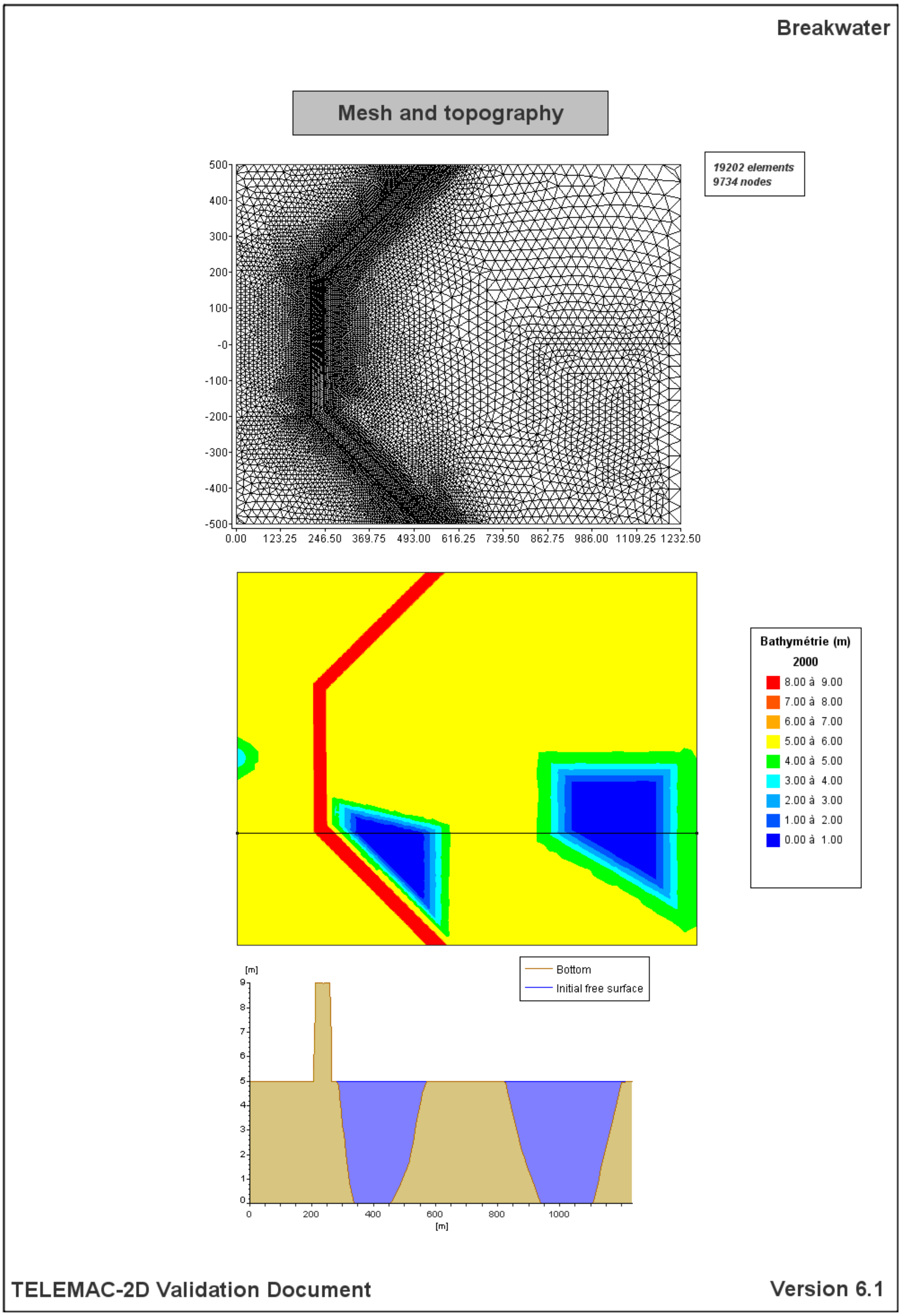


Figure 3.16.1 : Mesh and topography.

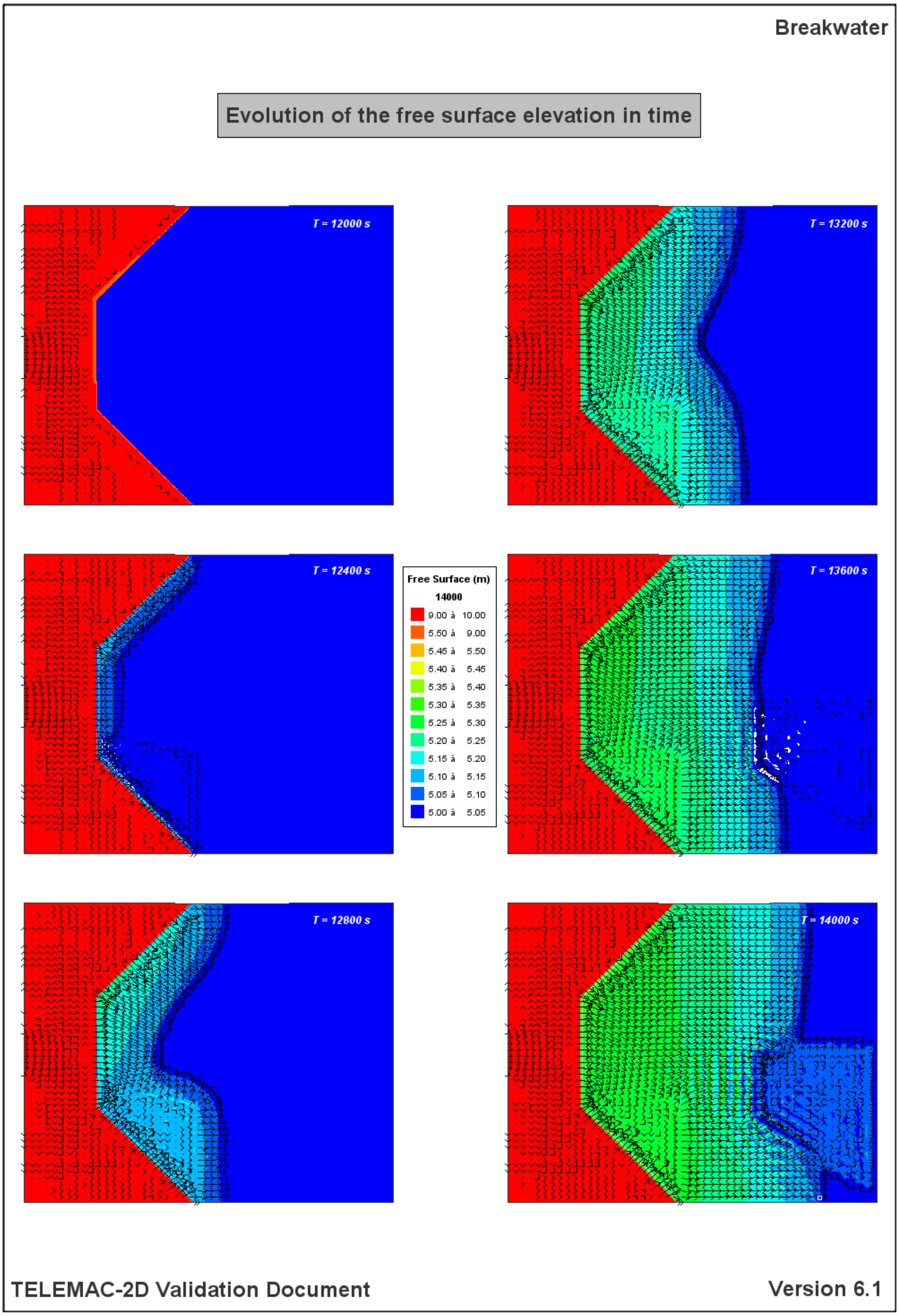


Figure 3.16.2 : Evolution of the free surface elevation in time.

3.17. MALPASSET DAM BREAK WAVE

Title	**Simulation of the dam break wave that occurred after failure of the Malpasset dam (France)**
Initial study	**J.-M. Hervouet – April 1992**
Last update	**December 2011**
Version	**TELEMAC-2D 6.1**

3.17.1. PURPOSE

To illustrate that TELEMAC-2D is capable of simulating a real dam break flow on an initially dry domain. Also to show the propagation of the wave front and the evolution in time of the water surface and velocities in the valley downstream.

3.17.2. LINKED CLAIMS

- **Claim 2.2.1.9** TELEMAC-2D can appropriately be used to analyse the consequences of the water wave following a dam break. Situation with dry or wet beds in the valley downstream the dam can both be examined.
- **Claim 2.2.2.1** TELEMAC-2D can consider domains with complex geometries: changing bed slopes, teeth-shaped coastline, islands, flow contractions and expansions, etc.
- **Claim 2.4.2.3** TELEMAC-2D can solve the SWE on an unstructured grid of triangles, which allows grid resolution to vary in space and enables complex coastlines to be represented accurately.
- **Claim 2.4.2.8** The solution algorithm allows the calculation to progress faster when the solution is slowly varying because the iterative method converges in fewer iteration.
- **Claim 2.4.2.11** With appropriate choice of solution method options, the flow results are mass conservative.
- **Claim 2.4.2.14** TELEMAC-2D offers unconditionally stable semi-implicit solution methods. However, it is recommended to adopt a time step such that the Courant number is not larger than 3 in general.

3.17.3. APPROACH

This case is the simulation of the propagation of the wave following the break of the Malpasset dam (South-East of France). Such accident really occurred in 1959. The model represents the reservoir upstream from the dam and the valley and flood plain downstream. The entire valley is approximately 18 km long and between 200 m (valley) and 7 km wide (flood plain).

Geometry:

– Size of the model: domain = 17700 m x 6800 m

Mesh:

The mesh is refined in the river valley (downstream from the dam) and on the banks. It is made up with irregular triangles.

- 26000 triangular elements
- 13541 nodes
- Maximum size range: from 17 to 313 metres

Boundaries:

- Solid walls with slip conditions

Bottom:

- Strickler formula with friction coefficient = 30

The mesh is shown on Figure 3.17.1.

Turbulence:

- Model of constant viscosity = 1 m^2/s

Algorithm:

- Type of advection:
  - N scheme
  - Conservative + modified SUPG on depth
- Type of element:
  - Quasi-bubble triangle for velocities
  - P1 triangle for H
- SUPG option:
  - No decentring
  - Decentring equal to the Courant number for H
- GMRES solver
- Solver accuracy = 10^{-4}

Time data:

- Time step = 0.5 sec.
- Simulation duration = 4000 sec.

3.17.4. RESULTS

Figure 3.17.2 illustrates the progression of the flood wave after the dam break. The propagation of the wave front is very fast. The water depths increase rapidly in the valley downstream from the dam location. The wave spreads in the plain when arriving to the sea. Figure 3.17.3 represents the velocity patterns at successive times as given by the computation. Maximum velocities are close to 14 m/s in sharp and narrow meanders of the river valley in which the dam break wave propagates. Water mass balance shows that the mass conservation is very good: the relative error cumulated on volume is $–0.354x10^{-14}$.

A complete comparison between simulation results produced by TELEMAC-2D and in-situ data available collected immediately after the catastrophe has been reported by Hervouet (19).

3.17.5. CONCLUSIONS

TELEMAC-2D is capable of simulating the propagation of a dam break wave in a river valley initially dry.

3.17.6. STEERING FILE

```
/----------------------------------------------------------------------
/ TELEMAC2D Version v6p1 may. 2012
/ Validation test case 18
/----------------------------------------------------------------------

/----------------------------------------------------------------------
/ INPUT-OUTPUT, FILES
/----------------------------------------------------------------------
TITLE      ='Validation test case 18'
VALIDATION =YES

FORTRAN FILE              ='princi.f'
GEOMETRY FILE             ='geo'
REFERENCE FILE            ='refprim'
BOUNDARY CONDITIONS FILE  ='cli.txt'
RESULTS FILE              ='resprim'

/----------------------------------------------------------------------
/ INPUT-OUTPUT, GRAPHICS AND LISTING
/----------------------------------------------------------------------

LISTING PRINTOUT PERIOD         =100
GRAPHIC PRINTOUT PERIOD         =200
VARIABLES FOR GRAPHIC PRINTOUTS ='U,V,H,B,S'
MASS-BALANCE                    =YES

/----------------------------------------------------------------------
/ EQUATIONS
/----------------------------------------------------------------------

FRICTION COEFFICIENT                  =30.
LAW OF BOTTOM FRICTION                =3
VELOCITY DIFFUSIVITY                  =1.

/----------------------------------------------------------------------
/ NUMERICAL PARAMETERS
/----------------------------------------------------------------------

SUPG OPTION               =0;2;2;2
NUMBER OF TIME STEPS      =8000
TIME STEP                 =0.5;
TYPE OF ADVECTION         =7;5;1;1
DISCRETIZATIONS IN SPACE  =12;11;11;11

/----------------------------------------------------------------------
/ NUMERICAL PARAMETERS, SOLVER
/----------------------------------------------------------------------
```

```
SOLVER          =7
SOLVER OPTION   =3
SOLVER ACCURACY =1.E-4

/----------------------------------------------------------------------
/ NUMERICAL PARAMETERS, VELOCITY-CELERITY-HIGHT
/----------------------------------------------------------------------

MASS-LUMPING ON H           =1.
IMPLICITATION FOR DEPTH     =0.6
IMPLICITATION FOR VELOCITY =0.6
```

3.17.7. FIGURES

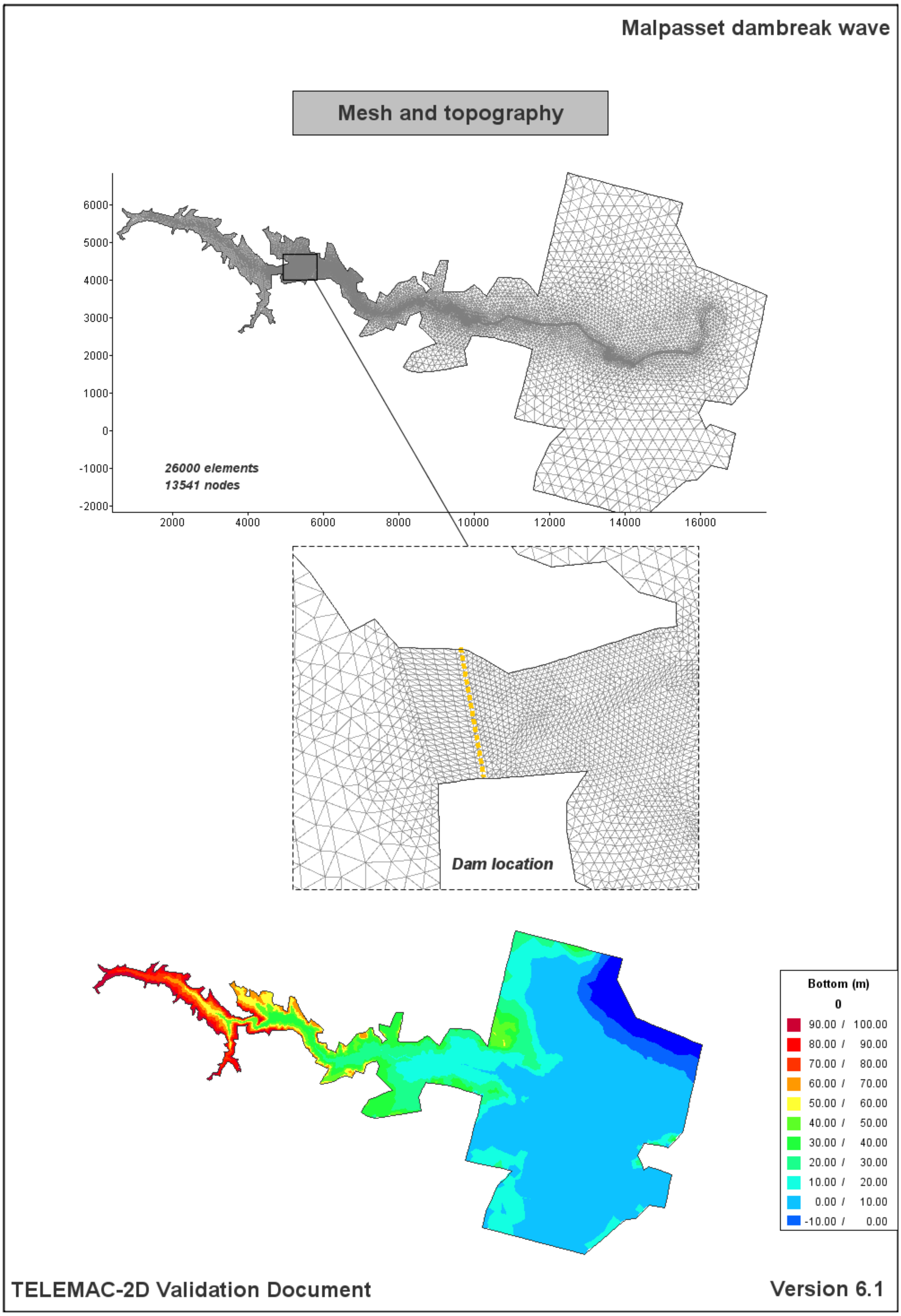


Figure 3.17.1 : Mesh and topography.

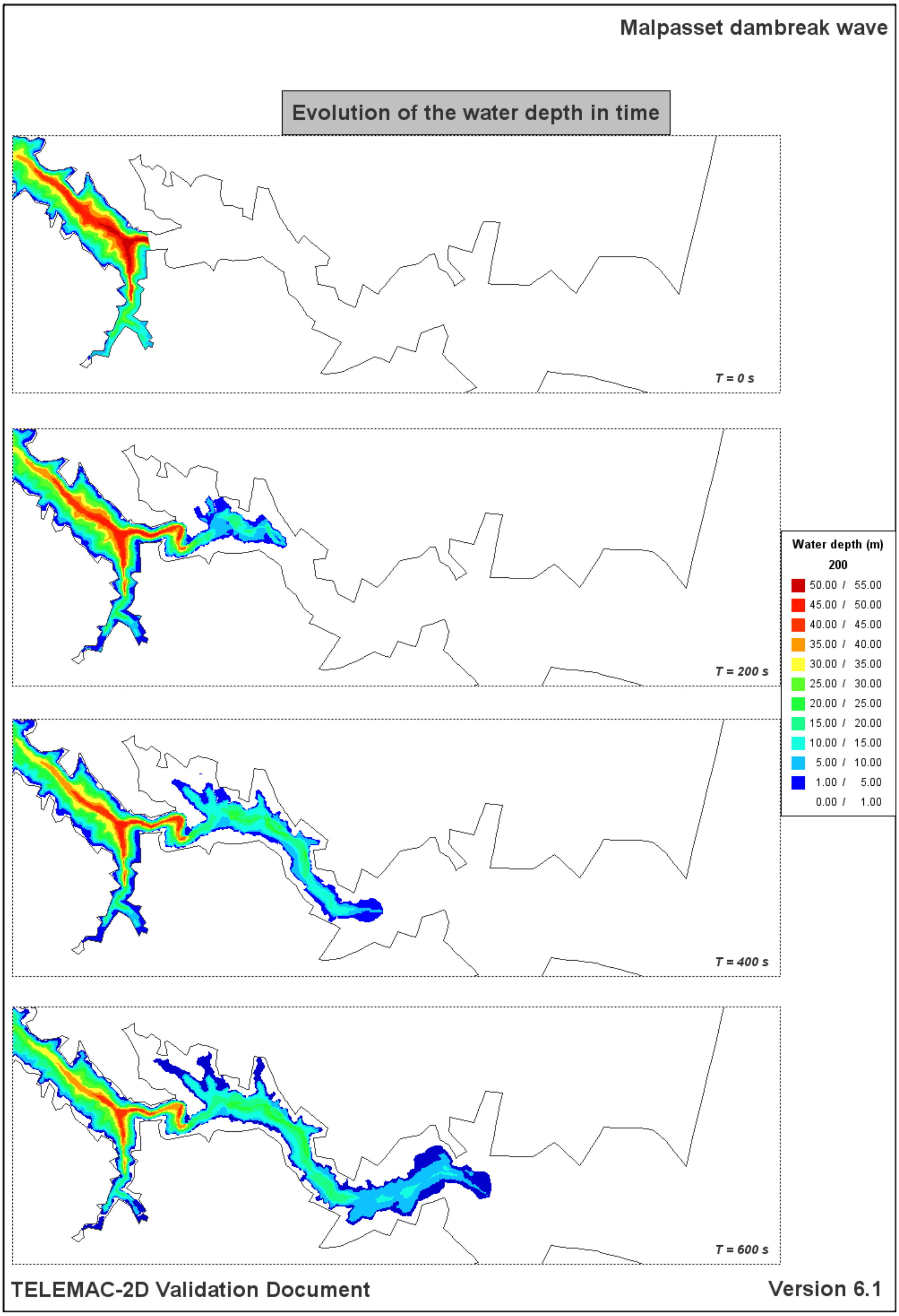


Figure 3.17.2 : Evolution of the water depth in time.

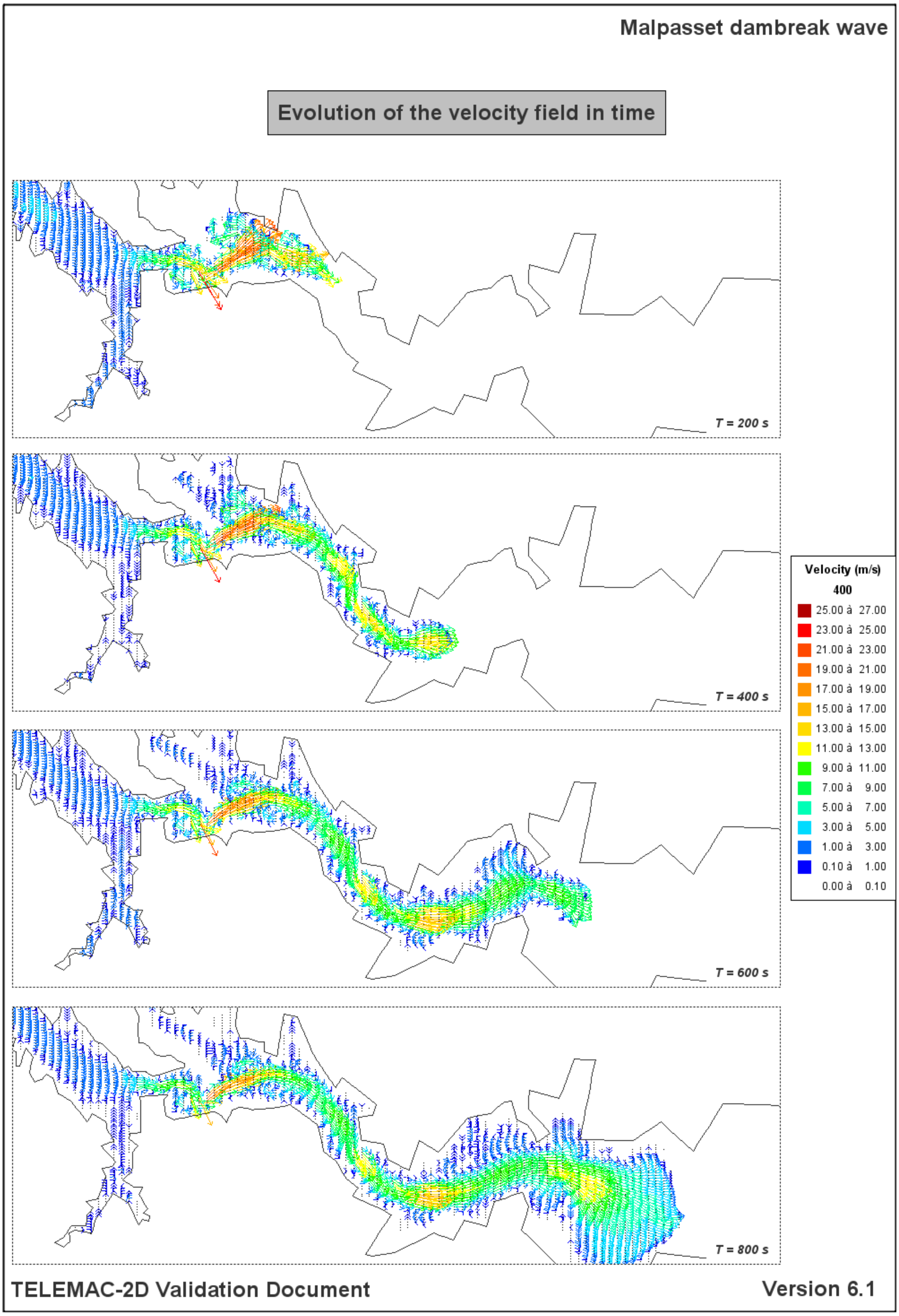


Figure 3.17.3 : Evolution of the velocity field in time.

3.18. RIVER CULM

Title	**Transient flood flow in the valley of river Culm**
Initial study	**P. Bates – 1994**
Last update	**May 2012**
Version	**TELEMAC-2D 6.1**

3.18.1. PURPOSE

To illustrate that TELEMAC-2D is capable of simulating the real inundation of a flood plain. Also to show the evolution in time of the water surface and velocity patterns in the flood plain.

3.18.2. LINKED CLAIMS

- **Claim 2.2.1.8** TELEMAC-2D can be used as a tool for assessing the impact of river flooding.
- **Claim 2.2.2.1** TELEMAC-2D can consider domains with complex geometries: changing bed slopes, teeth-shaped coastline, islands, flow contractions and expansions, etc.
- **Claim 2.2.2.4** TELEMAC-2D can consider conditions on open boundaries of a study domain corresponding to a wide range of physical phenomena encountered in the nature: tides (maritime applications), floods, river water inflow, dam failure (river applications).
- **Claim 2.2.2.7** TELEMAC-2D is capable of representing the hydrodynamic influence of bed friction as a bottom boundary condition in the vertical averaging process. Roughness options available are quadratic (Chezy, Strickler, Manning), Nikuradse or linear friction laws.
- **Claim 2.2.2.11** TELEMAC-2D can accurately simulate sub-critical flows when the boundary conditions are time dependent (boundary driven flow).
- **Claim 2.2.2.16** TELEMAC-2D can represent the wetting and drying of intertidal zones (maritime applications) and of inundated plains (river applications). The numerical treatment of this process is very robust.
- **Claim 2.4.2.3** TELEMAC-2D can solve the SWE on an unstructured grid of triangles, which allows grid resolution to vary in space and enables complex coastlines to be represented accurately.
- **Claim 2.4.2.5** TELEMAC-2D can support distorted triangular meshes up to a ratio 1:10. It is recommended to exploit this possibility in places where the flow is mono-directional in order to save memory and CPU time: meshes must be longer in the direction of the flow than in the transverse direction.
- **Claim 2.4.2.15** Boundary conditions can be applied as elevations, velocities, discharges or non-reflecting conditions.

- **Claim 2.4.2.20** The numerical treatment applied to alternately drying and wetting elements in TELEMAC-2D is efficient.

3.18.3. APPROACH

This test case represents the plain of the river Culm (Devon, UK) over a 11 km reach. The river is meandering in its valley. A narrowing concentrates flow in the middle of the reach represented. The main channel is approximately 19 m wide. The downstream elevation and the upstream discharge prescribed vary in time to simulate one flood of the river.

Field data concerning discharge and flooded areas were available for this event.

Geometry:

- Size of the model: river length = approximately 10 km
- Water depth at rest = 0 m

Mesh:

The mesh is unstructured. It is denser in the bed of the river than in the flood plain. Triangles are elongated in the direction of flow.

- 2019 triangular elements
- 1130 nodes
- Maximum size range: from 18 to 258 metres

Boundaries:

- Initially :
  - River entrance: Q = 28.32 m^3/s imposed
  - River outlet: H = 23.77 m imposed
- During the simulation, the discharge at the entrance and the water level at the outlet are varying with time and prescribed by the formatted data files.
- Lateral boundaries: solid walls with slip condition in the domain

Bottom:

- Strickler formula with friction coefficient = 30

The mesh and the topography are shown on Figure 3.18.1.

Turbulence:

- Model of constant viscosity equal to 2 m^2/s

Algorithm:

- Type of advection:
  - Characteristics on velocities
  - Conservative + modified SUPG on depth
- Type of element:
  - P1 triangle for H and for velocity

- Implicitation for depth and for velocity = 0.6
- Solver: Conjugate gradient
- Solver accuracy = 10^{-4}
- Tidal flats

Time data:

- Time step = 2 sec.
- Simulation duration = 54000 sec.

3.18.4. RESULTS

The water profile in the river varies according to the flood conditions prescribed at the boundaries (water elevation downstream and discharge upstream). Flooding of the plain occurs during the simulation. At the peak of the inundation, the water depth is approximately equal to 0.40 m in the plain close to the river.

Detailed results given by TELEMAC-2D (see Figure 3.18.2) and comparison with field data are presented in the two papers referenced (20) and (21). They show good agreement on the downstream discharge and inundated areas.

3.18.5. CONCLUSIONS

TELEMAC-2D can simulate the flow resulting from the inundation of a river valley by natural floods.

3.18.6. STEERING FILE

```
/----------------------------------------------------------------------
/ TELEMAC2D Version v6p1 May 2012
/ Validation test case 19
/----------------------------------------------------------------------

/----------------------------------------------------------------------
/ INPUT-OUTPUT, FILES
/----------------------------------------------------------------------

VALIDATION             =YES
COMPUTATION CONTINUED =YES
TITLE                  ='Validation test case 19'

LIQUID BOUNDARIES FILE    ='qsl'
PREVIOUS COMPUTATION FILE ='prev'
GEOMETRY FILE             ='geo'
REFERENCE FILE            ='ref'
BOUNDARY CONDITIONS FILE  ='cli.txt'
RESULTS FILE              ='res'

/----------------------------------------------------------------------
/ INPUT-OUTPUT, GRAPHICS AND LISTING
/----------------------------------------------------------------------

LISTING PRINTOUT PERIOD         =250
GRAPHIC PRINTOUT PERIOD         =250
VARIABLES FOR GRAPHIC PRINTOUTS ='U,V,S,B,H'
MASS-BALANCE                    =YES
```

```
/----------------------------------------------------------------------
/ EQUATIONS
/----------------------------------------------------------------------

FRICTION COEFFICIENT   =30.
LAW OF BOTTOM FRICTION =3
VELOCITY DIFFUSIVITY   =2.

/----------------------------------------------------------------------
/ EQUATIONS, BOUNDARY CONDITIONS
/----------------------------------------------------------------------

PRESCRIBED FLOWRATES  =28.32;  0.
PRESCRIBED ELEVATIONS =0.; 23.77

/----------------------------------------------------------------------
/ NUMERICAL PARAMETERS
/----------------------------------------------------------------------

NUMBER OF TIME STEPS           =6750
TREATMENT OF THE LINEAR SYSTEM =2
SUPG OPTION                    =1;0;2;2
TIME STEP                      =8.
MATRIX-VECTOR PRODUCT          =2
INITIAL TIME SET TO ZERO       =YES

/----------------------------------------------------------------------
/ NUMERICAL PARAMETERS, SOLVER
/----------------------------------------------------------------------

SOLVER =1

/----------------------------------------------------------------------
/ NUMERICAL PARAMETERS, VELOCITY-CELERITY-HIGHT
/----------------------------------------------------------------------

MASS-LUMPING ON H          =1.
IMPLICITATION FOR DEPTH    =0.6
IMPLICITATION FOR VELOCITY =0.6
```

3.18.7. FIGURES

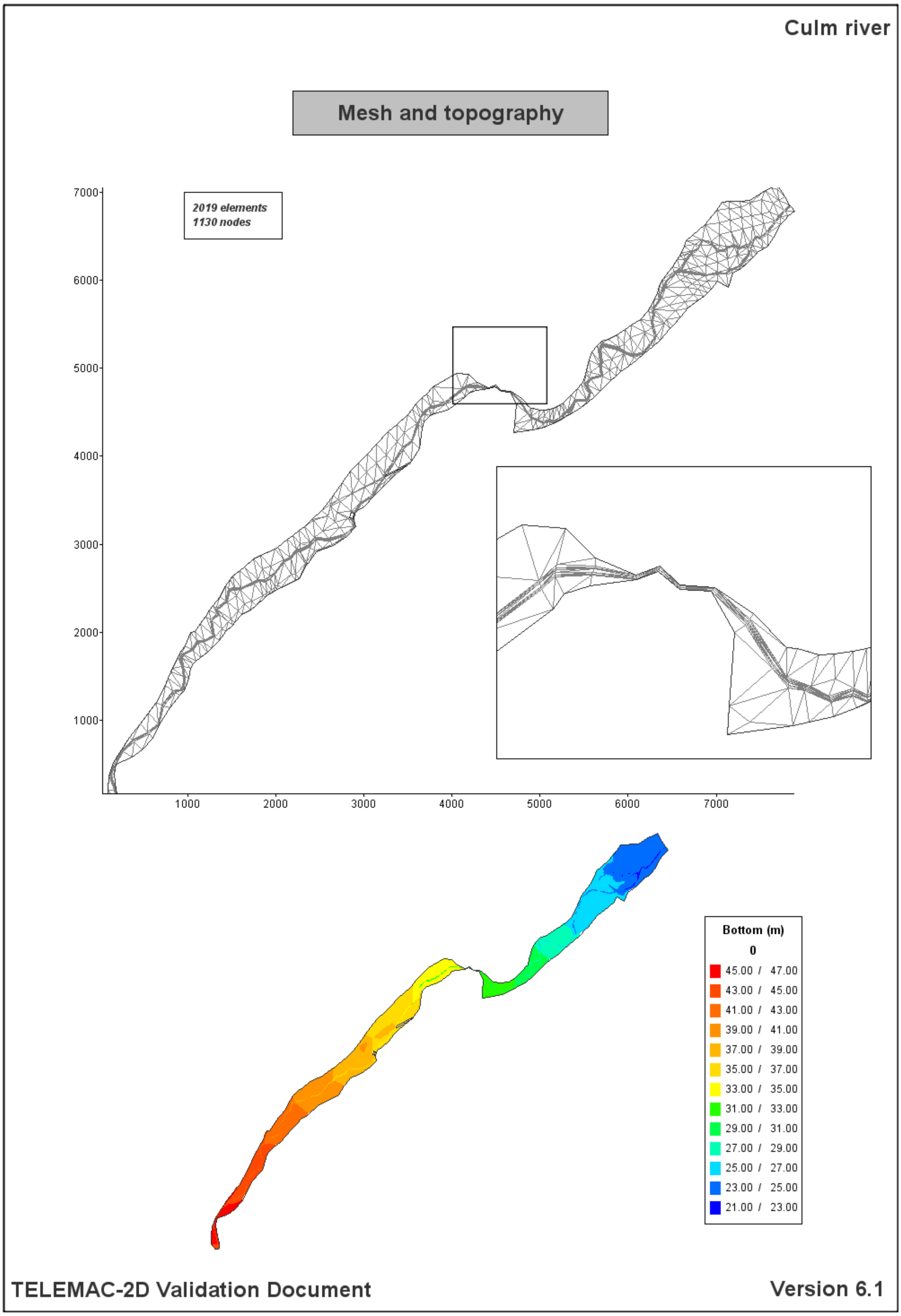


Figure 3.18.1 : Mesh and topography.

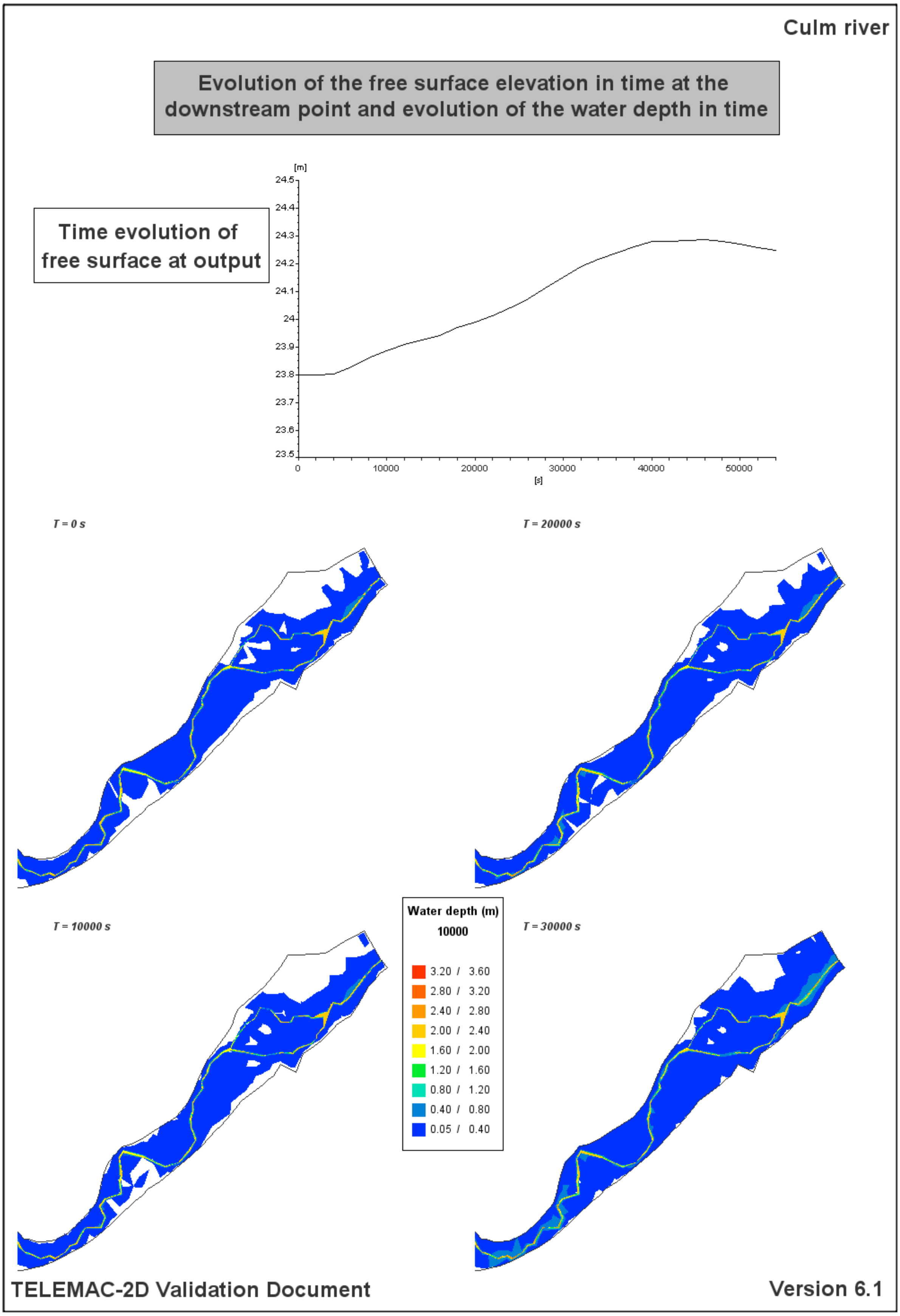


Figure 3.18.2 : Evolution of the free surface elevation in time at the downstream point and evolution of the water depth in time.

3.19. MERSEY ESTUARY

Title	**Tidal flow in the Mersey estuary**
Initial study	**A.-J. Cooper – 1996**
Last update	**May 2012**
Version	**TELEMAC-2D 6.1**

3.19.1. PURPOSE

To illustrate that TELEMAC-2D is capable of simulating the hydrodynamics (tidal currents and water elevations) in an estuarial zone due to long period tidal wave forcing, with the covering/uncovering of tidal flats.

3.19.2. LINKED CLAIMS

- **Claim 2.2.1.2** TELEMAC-2D can be used for an accurate prediction and for the analysis of tidal currents and water surface elevations in coastal or estuarial zones due to long period tidal wave forcing, with covering/uncovering flats.
- **Claim 2.2.2.1** TELEMAC-2D can consider domains with complex geometries: changing bed slopes, teeth-shaped coastline, islands, flow contractions and expansions, etc.
- **Claim 2.2.2.4** TELEMAC-2D can consider conditions on open boundaries of a study domain corresponding to a wide range of physical phenomena encountered in the nature: tides (maritime applications), floods, river water inflow, dam failure (river applications).
- **Claim 2.2.2.7** TELEMAC-2D is capable of representing the hydrodynamic influence of bed friction as a bottom boundary condition in the vertical averaging process. Roughness options available are quadratic (Chezy, Strickler, Manning), Nikuradse or linear friction laws.
- **Claim 2.2.2.11** TELEMAC-2D can accurately simulate sub-critical flows when the boundary conditions are time dependent (boundary driven flow).
- **Claim 2.2.2.16** TELEMAC-2D can represent the wetting and drying of intertidal zones (maritime applications) and of inundated plains (river applications). The numerical treatment of this process is very robust.
- **Claim 2.4.2.3** TELEMAC-2D can solve the SWE on an unstructured grid of triangles, which allows grid resolution to vary in space and enables complex coastlines to be represented accurately.
- **Claim 2.4.2.11** With appropriate choice of solution method options, the flow results are mass conservative.
- **Claim 2.4.2.20** The numerical treatment applied to alternately drying and wetting elements in TELEMAC-2D is efficient.

3.19.3. APPROACH

This case simulates the flow in the Mersey estuary on the west coast of England. Tide hydrodynamics in this area is complex, with large tidal range (up to 9 m on spring tide), extensive intertidal areas and an unusual tide curve: in the upper reaches of the estuary, the tide ebbs for a large proportion of the tidal period, and then floods very rapidly.

A sinusoidal water elevation is prescribed at the maritime open boundary to simulate a mean spring tide wave. The river discharge is neglected at the upstream end of the model.

Geometry:

- Size of the model: domain = 356000 x 409500 m
- Water depth at rest = 9.2 m

Mesh:

The mesh is dense in the river and at the mouth.

- 4490 triangular elements
- 2429 nodes
- Maximum size range: from 200 to 1500 metres

Boundaries:

- Domain entrance: $H = 5.15 + 4.05\cos\left(\frac{\pi}{2}\frac{t}{44700}\right)$
- Lateral boundaries: solid walls with slip condition

Bottom:

- Strickler formula with friction coefficient = 50

The mesh and the topography are shown on Figure 3.19.1.

Turbulence:

- Model of constant viscosity equal to 0.2 m^2/s

Algorithm:

- Type of advection:
  - Edge-based N-Scheme on velocities
  - Conservative on depth
- Type of element:
  - Quasi-bubble triangle for velocities
  - P1 triangle for H
- Implicitation for depth and for velocity = 0.6
- Solver: conjugate gradient on a normal equation
- Solver accuracy = 10^{-3}
- Bottom smoothing = 1

- Mass lumping on depth = 1
- Continuity correction = True
- Treatment of negative depth : 2 = flux control

Time data:

- Time step = 50 sec.
- Simulation duration = 44700 sec.

3.19.4. RESULTS

The total volume lost is 0.134x10^{-4} m^3, in comparison to the initial volume in the domain (0.289417x10^{10} m^3). The error represents 4x10^{-15} which is the machine precision. The final balance of water volume is excellent.

Figure 3.19.2 shows the water depths at high tide and low tide in the domain. Tidal flats are represented. On the same figure are shown (bottom) the evolutions of the free surface elevation at four different points:

- at the open boundary (1), where the tide is regular and symmetric,
- at the mouth of the Mersey river (2), where a small asymmetry of the curve and a drift in time is observed,
- in the estuary (3 and 4), where the tide is very asymmetric and a significant drift in time occurs.

Figure 3.19.3 shows the velocity fields at ebb tide and flow tide, when the velocities are maximum. There is an acceleration of the flow induced by the narrowing of the river. At flow tide, the velocities decrease in the estuary and are near to zero 15 km far from the mouth of the river.

3.19.5. CONCLUSIONS

TELEMAC-2D is capable of simulating the hydrodynamics (tidal currents and water surface elevations) in an estuarial zone due to long period tidal wave forcing, with covering and uncovering of tidal flats.

3.19.6. STEERING FILE

```
/----------------------------------------------------------------------
/ TELEMAC2D Version v6p1 May 2012
/ Validation test case 20
/----------------------------------------------------------------------

/----------------------------------------------------------------------
/ INPUT-OUTPUT, FILES
/----------------------------------------------------------------------

TITLE                    ='Validation test case 20'
VALIDATION               =YES

FORTRAN FILE             ='princi.f'
GEOMETRY FILE            ='geo'
REFERENCE FILE           ='ref'
BOUNDARY CONDITIONS FILE ='cli.txt'
RESULTS FILE             ='res'
```

```
/----------------------------------------------------------------------
/ INPUT-OUTPUT, GRAPHICS AND LISTING
/----------------------------------------------------------------------

LISTING PRINTOUT PERIOD         =20
VARIABLES FOR GRAPHIC PRINTOUTS ='U,V,S,B,H'
MASS-BALANCE                    =YES

/----------------------------------------------------------------------
/ EQUATIONS
/----------------------------------------------------------------------

BOTTOM SMOOTHINGS      =1
FRICTION COEFFICIENT   =50.
LAW OF BOTTOM FRICTION =3
VELOCITY DIFFUSIVITY   =0.2

/----------------------------------------------------------------------
/ EQUATIONS, BOUNDARY CONDITIONS
/----------------------------------------------------------------------

PRESCRIBED ELEVATIONS =0.

/----------------------------------------------------------------------
/ EQUATIONS, INITIAL CONDITIONS
/----------------------------------------------------------------------

INITIAL ELEVATION  =9.2
INITIAL CONDITIONS ='CONSTANT ELEVATION'

/----------------------------------------------------------------------
/ NUMERICAL PARAMETERS
/----------------------------------------------------------------------

TREATMENT OF NEGATIVE DEPTHS =2
CONTINUITY CORRECTION        =true
NUMBER OF TIME STEPS         =894
DISCRETIZATIONS IN SPACE     =12;11;11;11
SUPG OPTION                  =1;0;2;2
TIME STEP                    =50.
TYPE OF ADVECTION            =14;5;1;1

/----------------------------------------------------------------------
/ NUMERICAL PARAMETERS, SOLVER
/----------------------------------------------------------------------

SOLVER ACCURACY                        =1.E-3
MAXIMUM NUMBER OF ITERATIONS FOR SOLVER =300

/----------------------------------------------------------------------
/ NUMERICAL PARAMETERS, VELOCITY-CELERITY-HIGHT
/----------------------------------------------------------------------

MASS-LUMPING ON H          =1
IMPLICITATION FOR DEPTH    =0.6
IMPLICITATION FOR VELOCITY =0.6
```

3.19.7. FIGURES

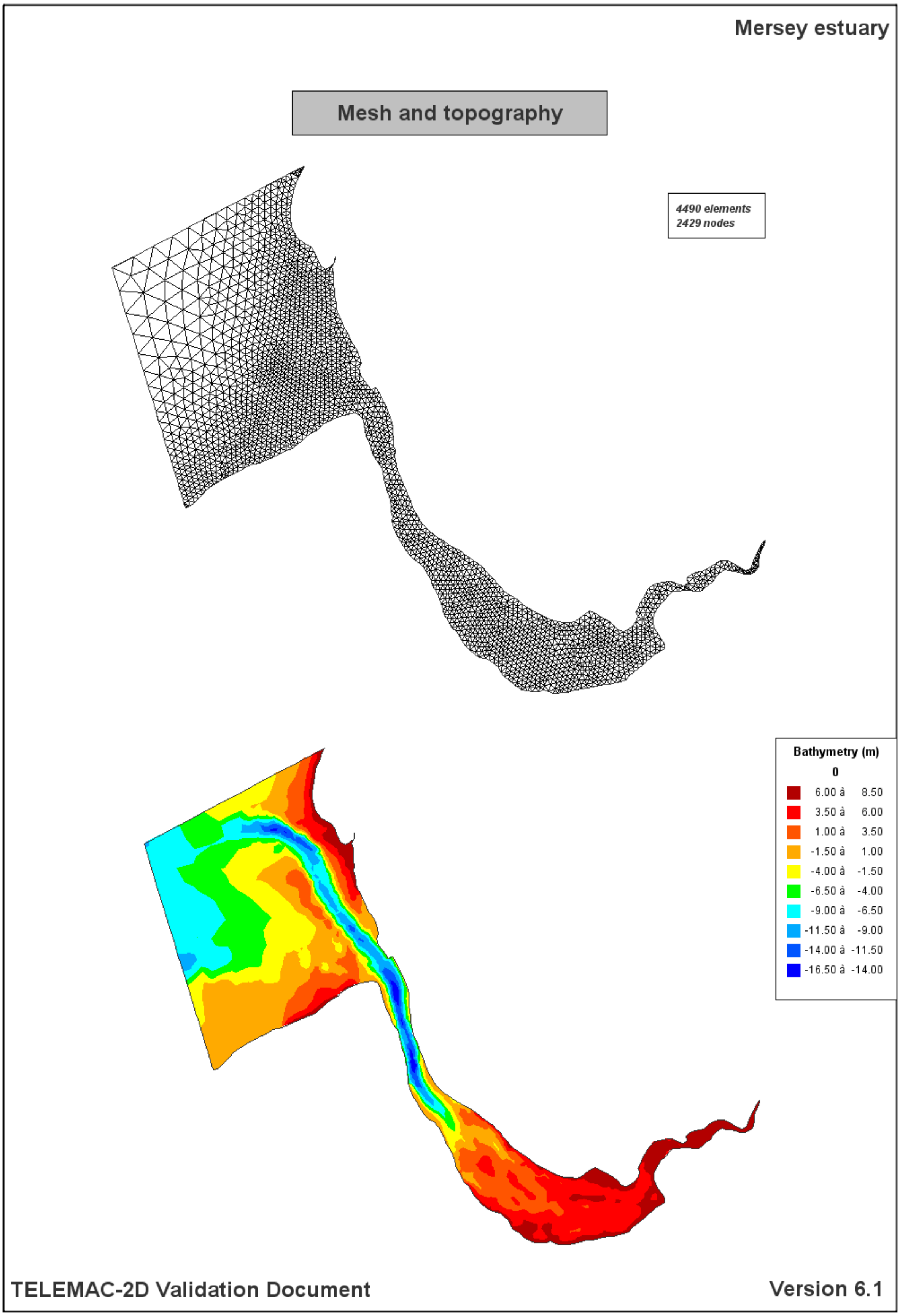


Figure 3.19.1 : Mesh and topography.

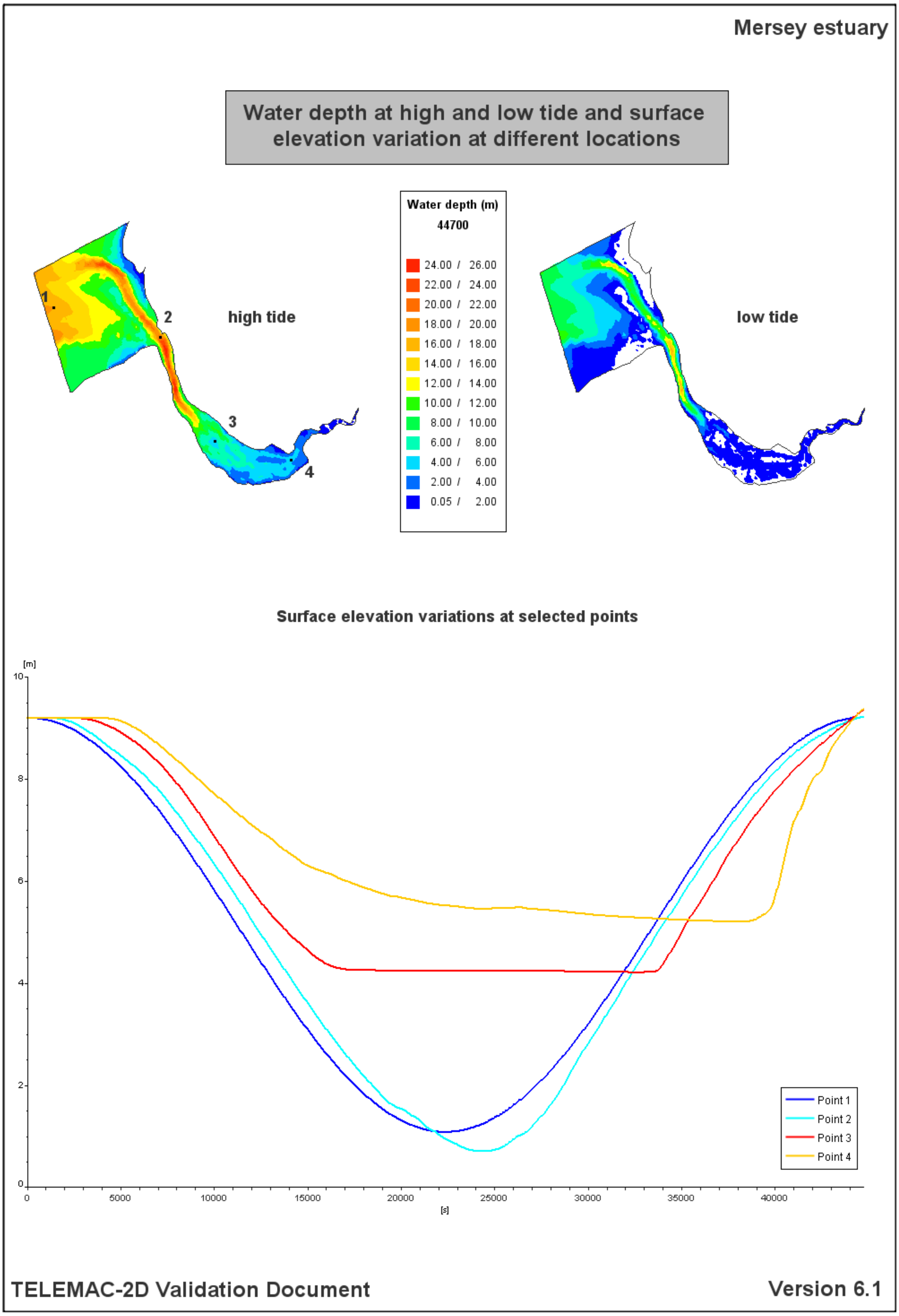


Figure 3.19.2 : Water depth at high tide and low tide and surface elevation at different locations.

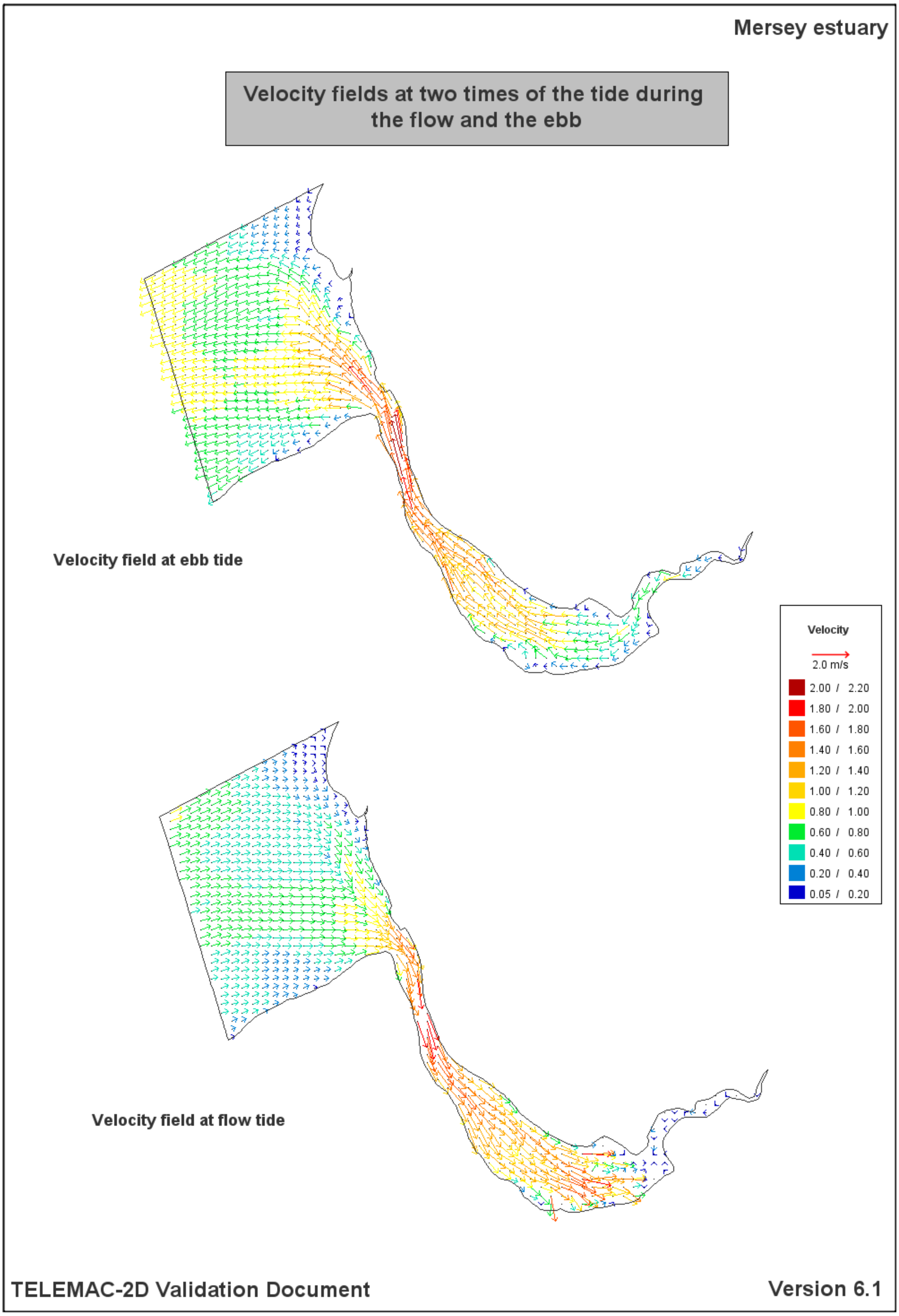


Figure 3.19.3 : Velocity fields at two times of the tide during the flow and the ebb.

3.20. WESTERN EUROPEAN CONTINENTAL SHELF

Title	**Transformation of the M2 tide constituent along the western European Continental shelf**
Initial study	**J.-M. Janin, X. Blanchard – 1992**
Last update	**May 2012**
Version	**TELEMAC-2D 6.1**

3.20.1. PURPOSE

This test case computes the transformation of the main tide constituent (M_2 - semi-diurnal moon generated wave) occurring along the western Europe continental margin.

3.20.2. LINKED CLAIMS

- **Claim 2.2.1.1** TELEMAC-2D can be used to set up a database of tidal constituents on a continental shelf.
- **Claim 2.2.2.1** TELEMAC-2D can consider domains with complex geometries: changing bed slopes, teeth-shaped coastline, islands, flow contractions and expansions, etc.
- **Claim 2.2.2.4** TELEMAC-2D can consider conditions on open boundaries of a study domain corresponding to a wide range of physical phenomena encountered in the nature: tides (maritime applications), floods, river water inflow, dam failure (river applications).
- **Claim 2.2.2.8** TELEMAC can include the impact of the Coriolis force arising from Earth rotation.
- **Claim 2.2.2.11** TELEMAC-2D can accurately simulate sub-critical flows when the boundary conditions are time dependent (boundary driven flow).
- **Claim 2.2.2.14** TELEMAC-2D can solve the flow equations written in spherical co-ordinates as needed by the study of large maritime domains, where earth sphericity has to be taken into account. A Cartesian co-ordinate system is the option traditionally employed for smaller domains.
- **Claim 2.2.2.15** TELEMAC-2D can represent internal forcing induced by the tide raising potential of heavenly bodies for large maritime domains.
- **Claim 2.4.2.1** The numerical methods used in TELEMAC-2D can provide an accurate solution of the SWE.
- **Claim 2.4.2.3** TELEMAC-2D can solve the SWE on an unstructured grid of triangles, which allows grid resolution to vary in space and enables complex coastlines to be represented accurately.
- **Claim 2.4.2.10** With appropriate choice of the mesh resolution, TELEMAC-2D produces solutions without wiggles (i.e. oscillations in space at a given time or oscillations in time at a given position).

3.20.3. APPROACH

The model extends from the Iberian Peninsula up to South England. The whole continental margin of the Atlantic coasts of France is represented as well as the Southern part of the North Sea. The model is forced by the M2 tide component. 4 tide cycles are simulated.

Geometry:

- Size of the model: domain = 520000 x 950000 m

Mesh:

The mesh is denser near the coastline than along the open Atlantic boundaries. However, the mesh is not refined along the slope of the continental shelf in the present model (following studies have shown that this refinement was important for the overall quality of results).

- 9414 triangular elements
- 5007 nodes
- Maximum size range: from 10 to 50 kilometres

Boundaries:

- Ocean open boundaries:
  - H imposed with the tide
- Coastline:
  - solid walls with slip condition

Bottom:

- Chezy formula with friction coefficient = 50

The mesh and the topography are shown on Figure 3.20.1.

Algorithm:

- Type of advection:
  - No advection of U and V
  - Conservative + modified SUPG on depth
- Type of element:
  - Quasi-bubble triangle for velocities
  - P1 triangle for H
- Implicitation for depth and for velocity = 0.6
- GMRES Solver
- Solver accuracy = 10^{-3}
- Initial guess for U = 2
- SUPG option:
  - upwinding equal to 1 for velocity and depth
- Coriolis and spherical coordinates

Time data:

- Time step = 150 sec.
- Simulation duration = 181500 sec.

3.20.4. RESULTS

The resulting amplitude and phase of the M2 tide component are shown in Figure 3.20.2. The comparison with measurements is shown in the two following tables.

- Comparison of M2 amplitude

Reference	Model	Measured
Zeebrugge	1.31	1.59
Calais	2.25	2.46
Boulogne	2.94	2.93
Dieppe	3.10	3.08
Cherbourg	2.02	1.87
Saint-Malo	3.81	3.74
Balise C1	1.29	1.29
Ouessant	1.99	2.06

- Comparison of M2 phase

Reference	Model	Measured
Zeebrugge	15	14
Calais	344	345
Boulogne	335	331
Dieppe	318	311
Cherbourg	228	228
Saint-Malo	183	178
Balise C1	106	103
Ouessant	115	111

More accurate results were found in a subsequent model with a finer mesh along the continental shelf and along the coastline. More tidal constituents were also taken into account. Inclusion of the tide generating potential in basic equations of TELEMAC-2D further improved the results.

3.20.5. CONCLUSIONS

TELEMAC is appropriate for the simulation of tide propagation over large sea domains such as continental margins.

3.20.6. STEERING FILE

```
/----------------------------------------------------------------------
/ TELEMAC2D Version v6p1 5 déc. 2011
/ Validation test case 21
/----------------------------------------------------------------------


/----------------------------------------------------------------------
/ INPUT-OUTPUT, FILES
/----------------------------------------------------------------------

TITLE                     ='Validation test case 21'
VALIDATION                =YES

FORTRAN FILE              ='princi.f'
```

```
FORMATTED DATA FILE 1    ='format1'
GEOMETRY FILE            ='geo'
FORMATTED RESULTS FILE   ='resfor'
REFERENCE FILE           ='ref'
FORMATTED DATA FILE 2    ='format2'
BOUNDARY CONDITIONS FILE ='cli.txt'
RESULTS FILE             ='res'

/----------------------------------------------------------------------
/ INPUT-OUTPUT, GRAPHICS AND LISTING
/----------------------------------------------------------------------

LISTING PRINTOUT PERIOD                                =50
GRAPHIC PRINTOUT PERIOD                                =149
VARIABLES FOR GRAPHIC PRINTOUTS=
'U,V,H,S,B,W,AMPL01,PHAS01,MAXZ,TMXZ,MAXV,TMXV'
NUMBER OF FIRST TIME STEP FOR GRAPHIC PRINTOUTS =894

/----------------------------------------------------------------------
/ EQUATIONS
/----------------------------------------------------------------------

FRICTION COEFFICIENT   =40.
ADVECTION OF U AND V   =NO
SPHERICAL COORDINATES  =YES
LAW OF BOTTOM FRICTION =2
CORIOLIS               =YES
DIFFUSION OF VELOCITY  =NO

/----------------------------------------------------------------------
/ EQUATIONS, SOURCE TERMS
/----------------------------------------------------------------------

FOURIER ANALYSIS PERIODS =44714.

/----------------------------------------------------------------------
/ NUMERICAL PARAMETERS
/----------------------------------------------------------------------
LIST OF POINTS=270; 8 ; 74 ; 15 ; 769 ; 804 ; 4071 ; 2399
NAMES OF POINTS=
'ZEEBRUGGE';'CALAIS';'BOULOGNE';'DIEPPE';'CHERBOURG';
'SAINT-MALO';'C1';'OUESSANT'
NUMBER OF TIME STEPS             =1192
NUMBER OF PRIVATE ARRAYS         =1
DISCRETIZATIONS IN SPACE         =12;11;11;11
SUPG OPTION                      =1;1;2;2
TIME STEP                        =150.
TIDAL FLATS                      =NO
TIME RANGE FOR FOURIER ANALYSIS =44714.;178800.

/----------------------------------------------------------------------
/ NUMERICAL PARAMETERS, SOLVER
/----------------------------------------------------------------------

SOLVER          =7
SOLVER ACCURACY =1.E-3
SOLVER OPTION   =3

/----------------------------------------------------------------------
/ NUMERICAL PARAMETERS, VELOCITY-CELERITY-HIGHT
```

```
/---------------------------------------------------------------------

MASS-LUMPING ON H   =1.
INITIAL GUESS FOR U =2
```

3.20.7. FIGURES

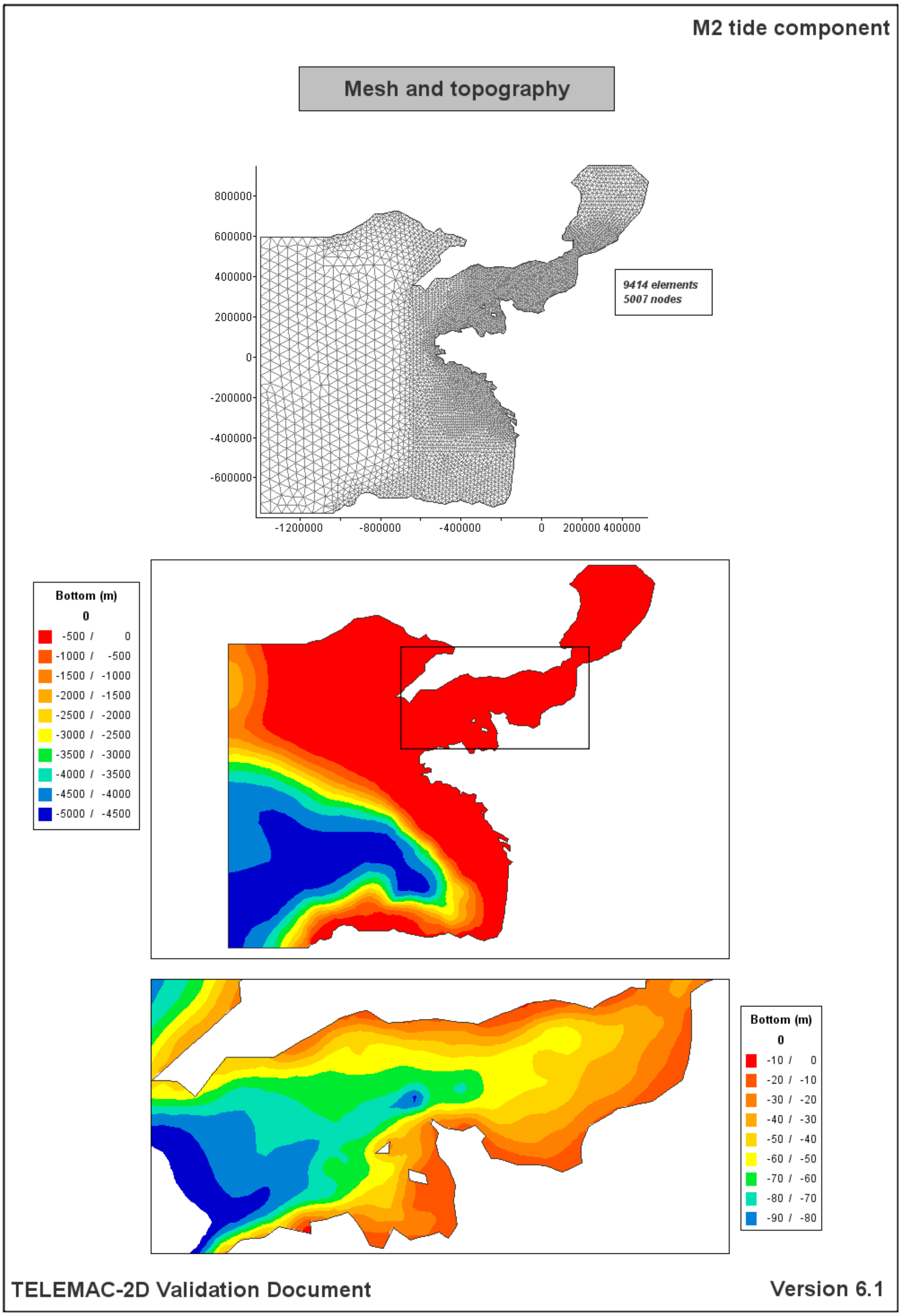


Figure 3.20.1 : Mesh and topography.

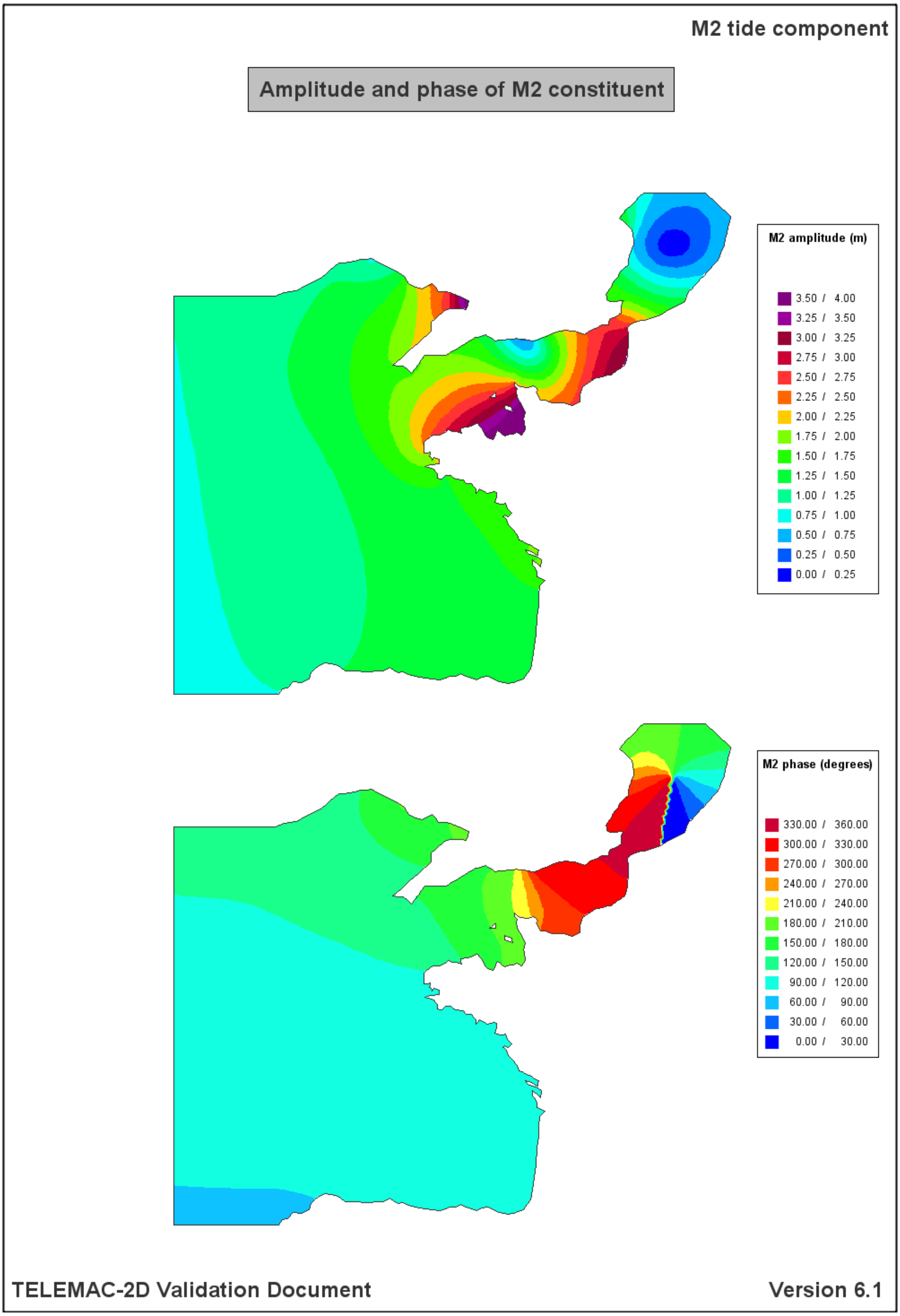


Figure 3.20.2 : Amplitude and phase of the M2 constituent.

3.21. POINT DISCHARGE WITHOUT DIFFUSION

Title	**Point discharge in a rectangular channel under steady state flow conditions. Calculation is carried out without diffusion.**
Initial study	**D. Lebris**
Last update	**May 2012**
Version	**TELEMAC-2D 6.1**

3.21.1. PURPOSE

This test demonstrates the ability of TELEMAC-2D to model a point discharge in a channel under a constant hydrodynamic field. Advection calculation is undertaken in order, on one hand to demonstrate that diffusion phenomena can be disconnected in TELEMAC and, on the other hand that the numerical schemes used to model the advective phenomena are little diffusive.

3.21.2. LINKED CLAIMS

- **Claim 2.2.1.11** TELEMAC-2D can be used to follow the behaviour of tracers (either conservative or decaying).
- **Claim 2.2.2.9** TELEMAC-2D is capable to represent areas of injection or extraction of fluid in the flow domain. This feature considers a source/sink of mass and of momentum in option. It is also possible to simulate a source/sink of tracer in the same location.
- **Claim 2.2.2.17** TELEMAC-2D is able to compute the dispersion by currents and diffusion of a tracer, with source or sink terms; this feature is employed only when water density is dependent upon the concentration in tracer.
- **Claim 2.3.2.2** The distribution of passive contaminant (tracer) can be determined by solution of the advection-diffusion equation.
- **Claim 2.4.2.2** The numerical methods used in TELEMAC-2D can provide an accurate solution of the tracer conservation equation.
- **Claim 2.4.2.13** TELEMAC-2D offers non diffusive solution methods of the advective terms of the flow equations.

3.21.3. APPROACH

The model is made of a rectangular channel. The grid, initialy built from square quadrangular elements, has been transformed into a triangular grid by division of the primary elements according to a diagonal line.

Geometry:

– Channel length = 50 m

– Channel width = 10 m

Mesh:

- 4000 triangular elements
- 2121 nodes

Hydrodynamics:

- Constant velocity equal to 1 m/s
- Water height 5 m

General parameters:

- Time step 0.1 sec
- Simulation period 500 sec

Initial conditions:

- Zero concentration in the whole domain

Boundary conditions:

- Channel banks : solid boundary without roughness
- Nil prescribed upstream concentration
- Free downstream concentration outflow
- Point source discharging 0.1 m^3/s with tracer concentration equal to 100 g/l

Numerical parameters:

- PSI advection scheme
- Solver : Conjugate gradient on normal equation
- Accuracy 10^{-9}

The model mesh and hydrodynamic field are presented at Figure 3.21.1.

3.21.4. RESULTS

Figure 3.21.2 shows the concentration changes in the domain at 30 an 60 seconds. Figure 3.21.3 shows the concentration longitudinal and transversal profiles at the same period.

The plume created by the spill is good rectilinear showing there is no lateral diffusion. This is particular showed by the transversal profile. However, a small numerical diffusion can be observed: it is amplified by the interpolations made by the postprocessor.

3.21.5. CONCLUSIONS

TELEMAC-2D is able to model advective transport phenomena.

3.21.6. STEERING FILE

```
/---------------------------------------------------------------------
/ TELEMAC2D Version v6p1 May 2012
/ Validation test case 21
/---------------------------------------------------------------------

/---------------------------------------------------------------------
/ INPUT-OUTPUT, FILES
/---------------------------------------------------------------------

TITLE                     ='Validation test case 21'
COMPUTATION CONTINUED     =YES

PREVIOUS COMPUTATION FILE ='hydro'
GEOMETRY FILE             ='geo'
REFERENCE FILE            ='ref'
NAMES OF TRACERS          ='TRAC1'
STEERING FILE             ='cas.txt'
BOUNDARY CONDITIONS FILE  ='cli.txt'
RESULTS FILE              ='res'

/---------------------------------------------------------------------
/ INPUT-OUTPUT, GRAPHICS AND LISTING
/---------------------------------------------------------------------

LISTING PRINTOUT PERIOD         =100
VARIABLES FOR GRAPHIC PRINTOUTS ='U,V,H,L,T1'
MASS-BALANCE                    =OUI
GRAPHIC PRINTOUT PERIOD         =100

/---------------------------------------------------------------------
/ EQUATIONS - TRACERS
/---------------------------------------------------------------------

NUMBER OF TRACERS         =1
FRICTION COEFFICIENT      =50.
INITIAL VALUES OF TRACERS =0.
LAW OF BOTTOM FRICTION    =2

/---------------------------------------------------------------------
/ EQUATIONS, BOUNDARY CONDITIONS
/---------------------------------------------------------------------

PRESCRIBED FLOWRATES      =0.;50.D0
PRESCRIBED ELEVATIONS     =5.D0;0.
PRESCRIBED TRACERS VALUES =0.D0;0.D0

/---------------------------------------------------------------------
/ EQUATIONS, SOURCE TERMS
/---------------------------------------------------------------------

ABSCISSAE OF SOURCES                =1.D0
VALUES OF THE TRACERS AT THE SOURCES =100.D0
WATER DISCHARGE OF SOURCES          =0.1D0
ORDINATES OF SOURCES                =5.D0

/---------------------------------------------------------------------
/ NUMERICAL PARAMETERS
/---------------------------------------------------------------------
```

```
TIME STEP                    =0.1
NUMBER OF TIME STEPS         =5000
INITIAL TIME SET TO ZERO =YES
TYPE OF ADVECTION            =2;5;1;1

/----------------------------------------------------------------------
/ NUMERICAL PARAMETERS, SOLVER
/----------------------------------------------------------------------

SOLVER ACCURACY =1.E-9
```

3.21.7. FIGURES

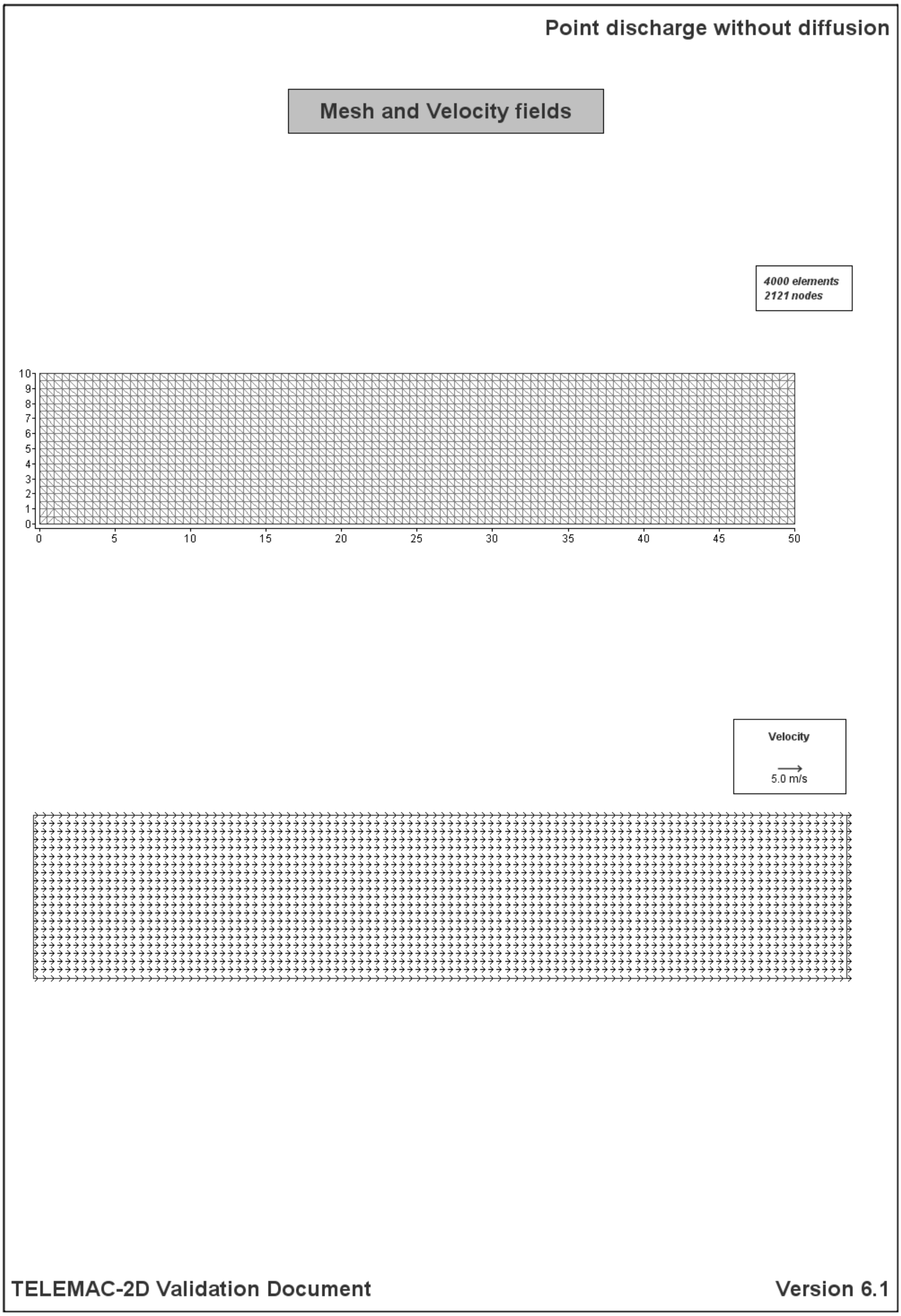


Figure 3.21.1 : Model mesh and hydrodynamic field.

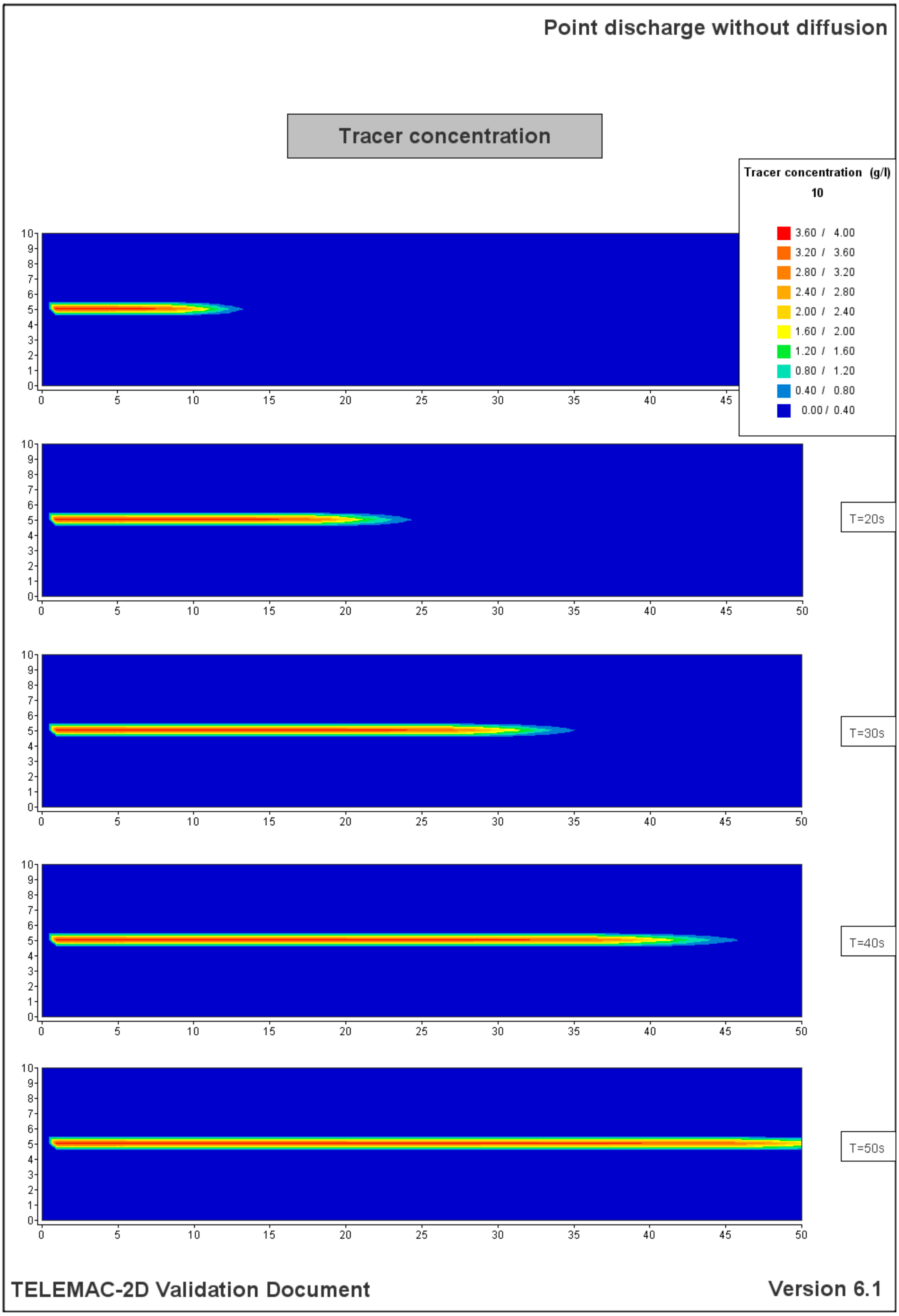


Figure 3.21.2 : Concentrations.

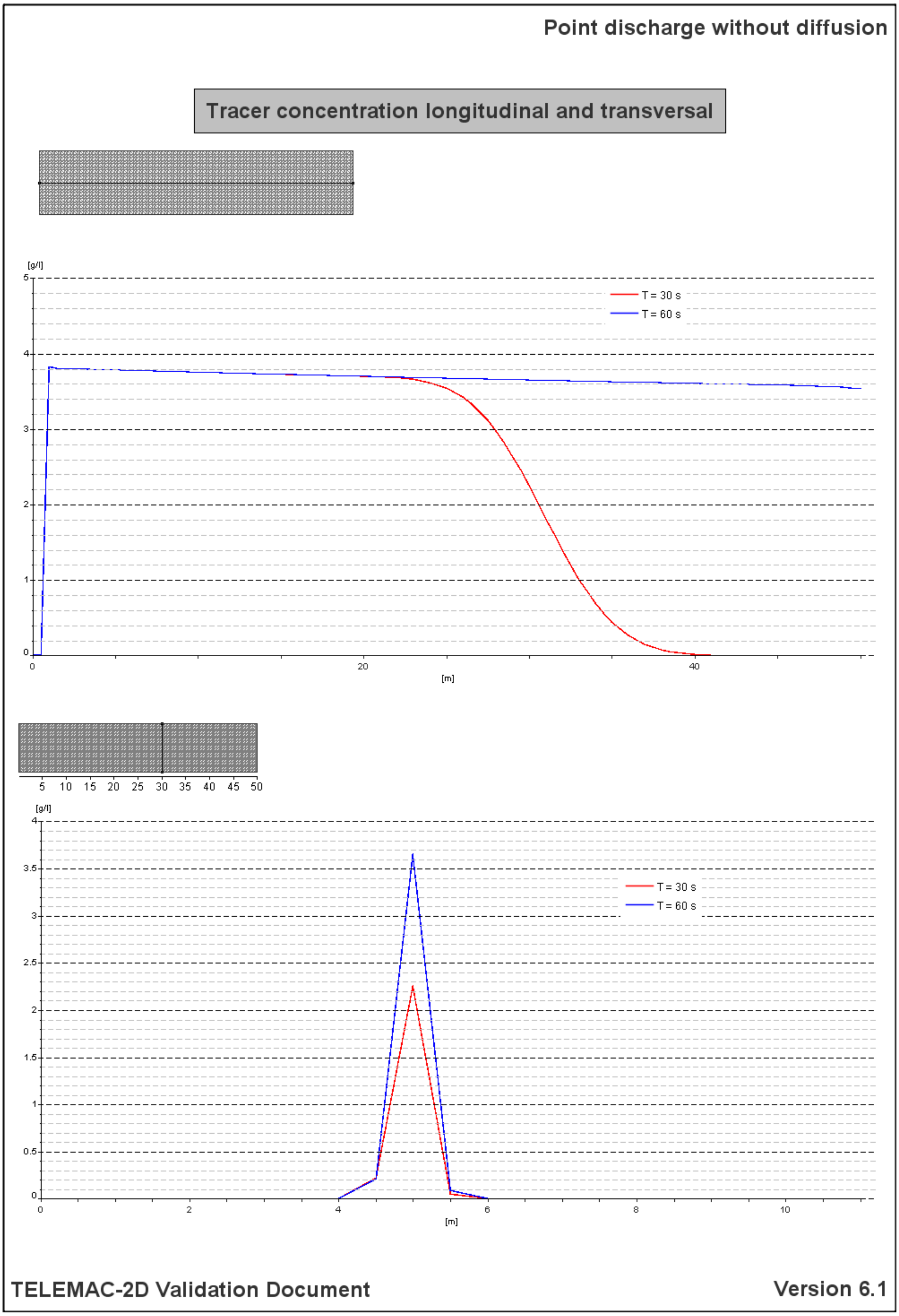


Figure 3.21.3 : Along and cross flow concentration Profiles.

3.22. POINT DISCHARGE WITH DIFFUSION

Title	**Point discharge in a rectangular channel under steady state flow conditions. Calculation is carried out with diffusion.**
Initial study	**D. Lebris**
Last update	**MAY 2012**
Version	**TELEMAC-2D 6.1**

3.22.1. PURPOSE

This test demonstrates the ability of TELEMAC-2D to model a point discharge in a channel under a constant hydrodynamic field. Advection and diffusion calculation is undertaken in order to demonstrate that TELEMAC-2D is able to model these two phenomena. The validation is made by comparison with analytical solution.

3.22.2. LINKED CLAIMS

- **Claim 2.2.1.11** TELEMAC-2D can be used to follow the behaviour of tracers (either conservative or decaying).
- **Claim 2.2.2.9** TELEMAC-2D is capable to represent areas of injection or extraction of fluid in the flow domain. This feature considers a source/sink of mass and of momentum in option. It is also possible to simulate a source/sink of tracer in the same location.
- **Claim 2.2.2.17** TELEMAC-2D is able to compute the dispersion by currents and diffusion of a tracer, with source or sink terms; this feature is employed only when water density is dependent upon the concentration in tracer.
- **Claim 2.3.2.2** The distribution of passive contaminant (tracer) can be determined by solution of the advection-diffusion equation.
- **Claim 2.4.2.2** The numerical methods used in TELEMAC-2D can provide an accurate solution of the tracer conservation equation.

3.22.3. APPROACH

The model is made of a rectangular channel. The grid, initialy build from square quadrangular elements, has been transformed into a triangular grid by division of the primary elements according to a diagonal line. Moreover, in order to have a more refined modelling, the grid has been refined compare to the one used in validation test 1 and 2.

<u>Geometry:</u>

- Channel length = 50 m
- Channel width = 10 m

<u>Mesh:</u>

- 4000 triangular elements
- 2121 nodes

Hydrodynamics:

- Constant velocity equal to 1 m/s
- Water height 5 m

General parameters:

- Time step 0.25 sec
- Simulation period 60 sec

Initial conditions:

- Zero concentration in the whole domain

Boundary conditions:

- Channel banks: solid boundary without roughness
- Nil prescribed upstream concentration
- Free downstream concentration outflow
- Point source discharging 0.1 m^3/s with tracer concentration equal to 100 g/l

Numerical parameters:

- PSI advection scheme
- Solver: Conjugate gradient on normal equation
- Solver accuracy 10^{-9}
- Diffusion coefficient 0.1 (m^2/s)

In this case, the TELEMAC-2D transport equation can be reduced in :

$$u \frac{\partial C}{\partial x} = K_x \left(\frac{\partial^2 C}{\partial x^2} + \frac{\partial^2 C}{\partial y^2} \right) + \frac{Q_e}{h}$$

This equation has a exact solution in steady flow given by:

$$C(x,y) = \frac{Q}{2\pi K_x h} \exp\left(\frac{u\,x}{2K_x}\right) K_0\left(\frac{u\,r}{2K_x}\right)$$

$$\text{With } r = \sqrt{x^2+y^2}$$

where :

x,y coordinates of the point from the source (m),

Q sediment discharge of the source (kg/s),

K_0 modified Bessel function, second sort and order zero.

The analytical solution is written in the water quality file. The formulation of the Bessel function was given in the book "Numerical Recipes in C [19].

The model mesh and hydrodynamic field are presented at Figure 3.22.1.

3.22.4. RESULTS

Figure 3.22.2 shows the concentration plume given by TELEMAC-2D and by the analytical solution after steady state reach. Figure 3.22.3 shows the concentration longitudinal and transversal profiles with comparison with analytical solution.

The plume created by the point discharge is typical of a diffusive tracer. The concentration longitudinal and transversal profiles computed by TELEMAC-2D are consistent with the analytical solution (except at the discharge point where the analytical solution gives an infinite concentration).

3.22.5. CONCLUSIONS

TELEMAC-2D is able to model advective and diffusive transport phenomena.

3.22.6. STEERING FILE

```
/----------------------------------------------------------------------
/ TELEMAC2D Version v6p1 May 2012
/ Validation test case 23
/----------------------------------------------------------------------

/----------------------------------------------------------------------
/ INPUT-OUTPUT, FILES
/----------------------------------------------------------------------
TITLE                     ='Validation test case 23'
COMPUTATION CONTINUED     =YES

FORTRAN FILE              ='main_tradif.f'
PREVIOUS COMPUTATION FILE ='hydro'
GEOMETRY FILE             ='geo'
REFERENCE FILE            ='ref'
NAMES OF TRACERS          ='TRAC1'
BOUNDARY CONDITIONS FILE  ='cli.txt'
RESULTS FILE              ='res'

/----------------------------------------------------------------------
/ INPUT-OUTPUT, GRAPHICS AND LISTING
/----------------------------------------------------------------------

LISTING PRINTOUT PERIOD         =100
VARIABLES FOR GRAPHIC PRINTOUTS ='U,V,H,L,T1,N'
MASS-BALANCE                    =OUI
GRAPHIC PRINTOUT PERIOD         =100

/----------------------------------------------------------------------
/ EQUATIONS
/----------------------------------------------------------------------

NUMBER OF TRACERS                    =1
FRICTION COEFFICIENT                 =50.
INITIAL VALUES OF TRACERS            =0.
LAW OF BOTTOM FRICTION               =2
COEFFICIENT FOR DIFFUSION OF TRACERS =0.1


/----------------------------------------------------------------------
/ EQUATIONS, BOUNDARY CONDITIONS
/----------------------------------------------------------------------
```

```
PRESCRIBED FLOWRATES       =0.;50.D0
PRESCRIBED ELEVATIONS      =5.D0;0.
PRESCRIBED TRACERS VALUES =0.D0;0.D0

/----------------------------------------------------------------------
/ EQUATIONS, SOURCE TERMS
/----------------------------------------------------------------------

ABSCISSAE OF SOURCES                  =1.D0
VALUES OF THE TRACERS AT THE SOURCES =100.D0
WATER DISCHARGE OF SOURCES            =0.1D0
ORDINATES OF SOURCES                  =5.D0

/----------------------------------------------------------------------
/ NUMERICAL PARAMETERS
/----------------------------------------------------------------------

TIME STEP                 =0.1
NUMBER OF TIME STEPS      =5000
INITIAL TIME SET TO ZERO =YES
TYPE OF ADVECTION         =2;5;1;1

/----------------------------------------------------------------------
/ NUMERICAL PARAMETERS, SOLVER
/----------------------------------------------------------------------

SOLVER ACCURACY =1.E-9
```

3.22.7. FIGURES

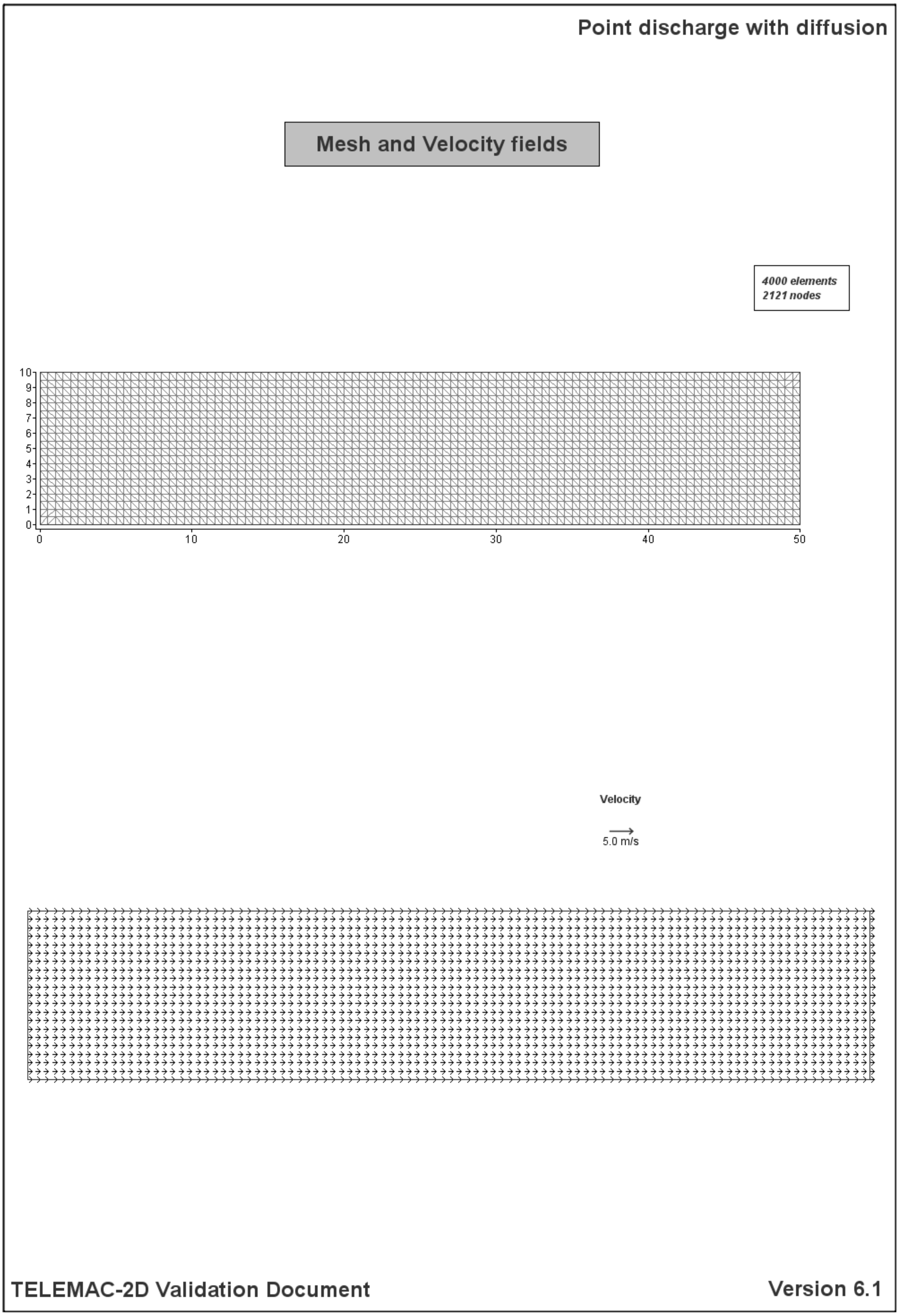


Figure 3.22.1 : Model mesh and hydrodynamic field.

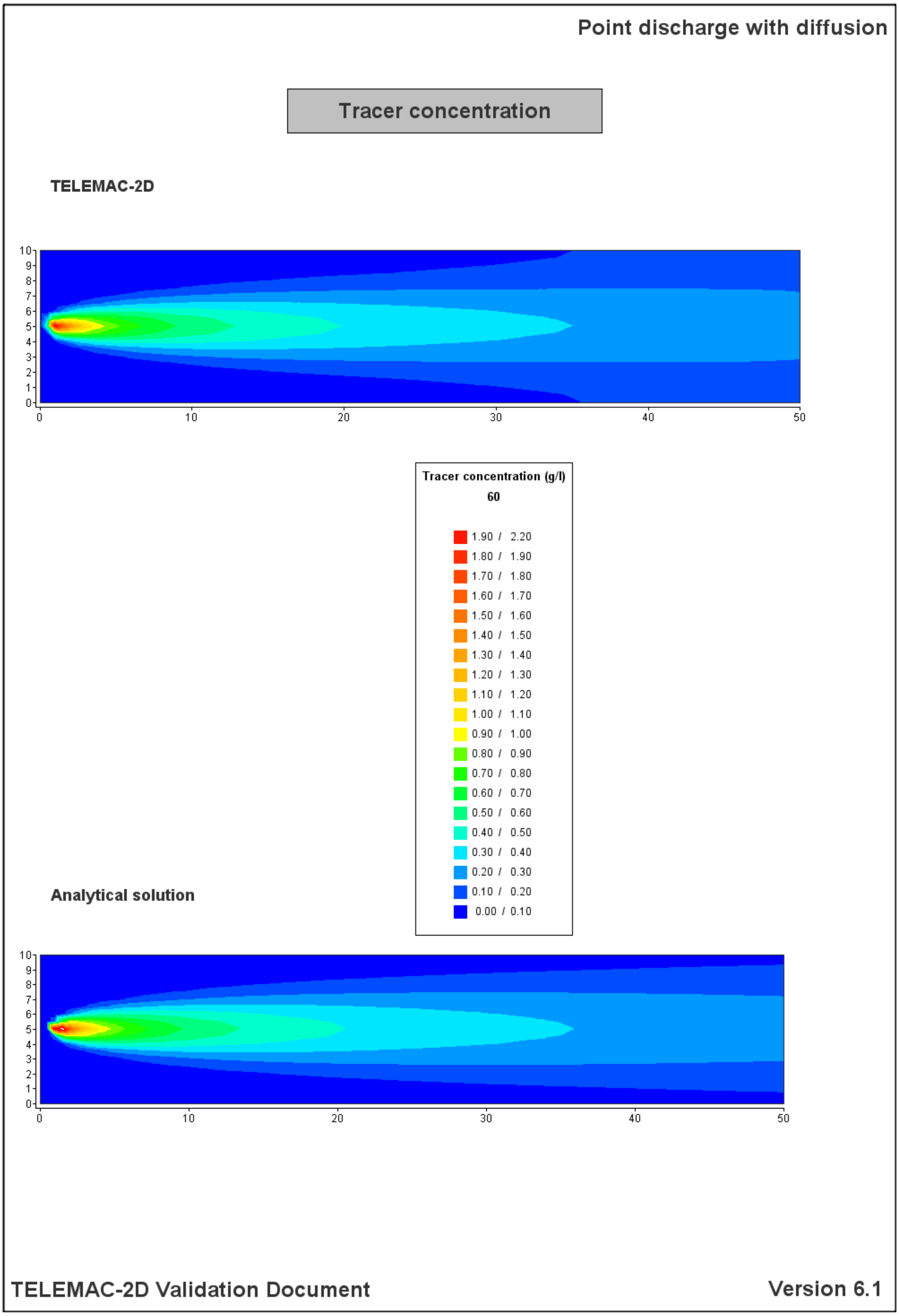


Figure 3.22.2 : Concentrations.

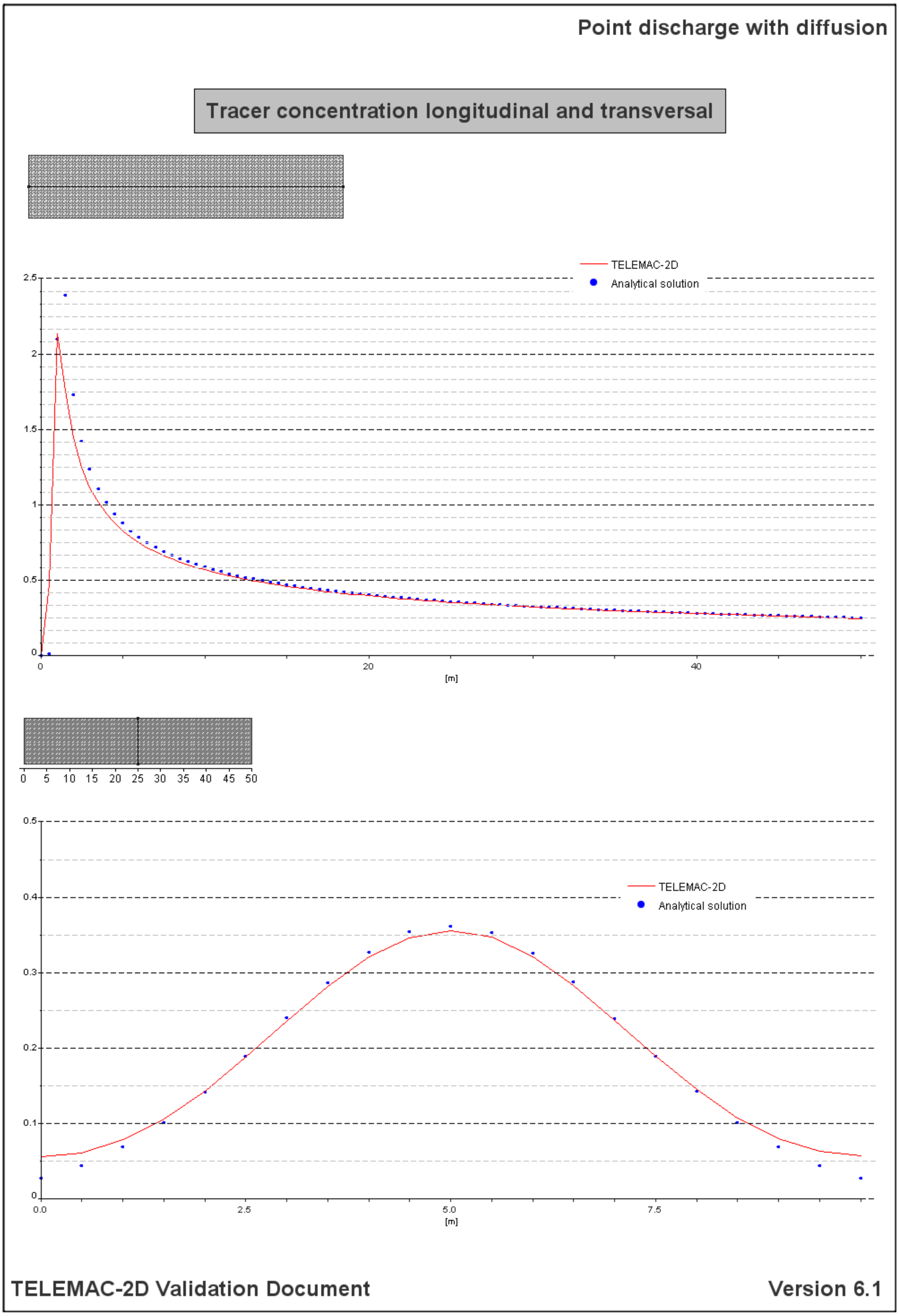


Figure 3.22.3 : Along and cross flow concentration Profiles.

Appendix A. GLOSSARY

Algorithmic implementation

The conversion of the conceptual model into a finite set of rules suitable for computation. This may involve spatial discretization schemes, time integration methods, solution procedures for algebraic equations, decision algorithms, etc.

Computational model

Software whose primary function is to model a certain class of physical systems. The computational model may include pre- and post-processing features, a user interface, and other ancillary programmes necessary in order to use the model in applications. However, this validation document primarily concerns the core of the computational model, consisting of the underlying conceptual model, its algorithmic implementation and software implementation.

Conceptual model

A mathematical/logical/verbal representation of a physical system or process. This representation may involve differential equations, discrete algebraic equations, decision graphs, or other types of conceptual descriptions.

Software implementation

The conversion of the algorithmic implementation into computer code. This includes coding of algorithms, use of standard mathematical software, design and implementation of data structures, etc. The term software implementation, for the purposes of this document, is limited to the computational core of the model. It does not include pre- and post-processing software, user interfaces, or other ancillary programmes associated with the computational model.

Appendix B. REFERENCES

1. *SINUSX - Version 2.0 - User Manual.* s.l. : EDF. HE-45/93.008.
2. *MATISSE - Version 1.0 - User Manual.* s.l. : EDF. HE-42/98/004/A.
3. **Hervouet, J-M.** *Hydrodynamics of Free Surface Flows modelling with the finite element method.* s.l. : Wiley, 2007. 978-0-470-03558-0.
4. **Kowalik, Z. et Murty, T. S.** *Numerical modelling of ocean dynamics.* Singapore : World Scientific Publishing Co., 1993.
5. **S.C., Martin J.L. and McCutcheon.** *Hydrodynamics and transport for water quality modelling.* s.l. : Lewis Publisher, 1999.
6. **Fischer, H. B., et al., et al.** *Mixing inland and coastal waters.* New-York : Academic Press, 1979.
7. **Abbott, M. B.** *Range of tidal flow modelling.* s.l. : Special Issue of the Journal of Hydraulic Engineering, ASCE, April 1997.
8. *TELEMAC 2D - Version 6.0 - User Manual.* 2010.
9. **Hervouet, J-M.** Vectorization and simplification of algorithms in finite element method. [éd.] EDF. Bulletin de la Direction des Etudes et Recherches, 1991, Vol. Série C Mathématiques, informatique, 1, pp. 1-37.
10. *Bibliothèque BIEF - Note de Principe et Descriptif Informatique.* s.l. : EDF. HE-43/92.16.
11. **Hervouet, J-M.** *Guide to programming in the Telemac system version 6.0.* s.l. : EDF R&D, 2010. H-P74-2009-00801-EN.
12. **Lauber, G. et Hager, W. H.** *Experiments to dambreak wave: Horizontal channel.* 1998. pp. 291-307. Vol. 36.
13. **Ritter, A.** *Die Fortplanzung der Wasserwellen (The propagation of water waves).* Zeitschrift Verein Deutscher Ingenieure. pp. 947-954. Vol. 36(33), in German.
14. **Stoker, J. J.** *Water waves.* s.l. : Wiley Interscience Publishers, 1957. pp. 333-341.
15. **Kumar Gurram, S., Karki, K. S. et Hager, W. H.** *Subcritical junction flow.* Journal of Hydraulic Engineering. 1997. pp. 447-455.
16. **Faure, T. D.** *Unsteady flow modelling on a beach using TELEMAC-2D.* [éd.] American Geophysical Union. Quantitative skill Assessment for Coastal Ocean Models, Coastal and Estuarine 1995 Annual conference of the Canadian Society for Civil Engineering. Ottawa, Ontario : s.n., June 1-3, 1995.
17. **Scott, D., Crookshank, N. et Zhang, J. I.** *Simulation of an idealised channel using Mike 11 and Mike 21.* [éd.] American Geophysical Union. Quantitative skill Assessment for Coastal Ocean Models, Coastal and Estuarine 1995 Annual conference of the Canadian Society for Civil Engineering. Ottawa, Ontario : s.n., June 1-3, 1995.
18. **Bijvelds, M., Kranenburg, C. et Stelling, G. S.** *3-D numerical simulation of turbulent shallow-water flow in a square harbour.* Journal of the Hydraulics Divisions, ASCE. 1997.
19. **Hervouet, J-M.** *A numerical simulation of the Malpasset dam-break with 2D Saint-Venant equations.* Colloque Barré De Saint Venant. Paris : s.n., August 1997.
20. **Bates, P. D., et al., et al.** *Internal and external validation of a two-dimensional finite-element code for river flood simulations.* Proceedings of the Institution of Civils Engineers, Water, maritime and Energy. Sept. 1998. pp. 127-141. Vol. 130.
21. **Bates, P. D., Anderson, M. G. et Hervouet, J-M.** *Computation of flood event using two-dimensional finite-element model and its comparison to fielsd data.* [éd.] P. Molinaro et L. Natale. In Modelling of Flood Porpagation Over Initially Dry Areas. New-York : s.n., 1994. pp. 238-248.
22. **Dee, Dick P.** A pragmatic approach to model validation. *Quantitative skill Assessment for Coastal Ocean Models. Coastal and Estuarine Studies.* 1995, Vol. 47, pp. 1-13.
23. *Guidelines for documenting the validity of computational modelling software.* s.l. : International Association for Hydraulic Research, June 1994.