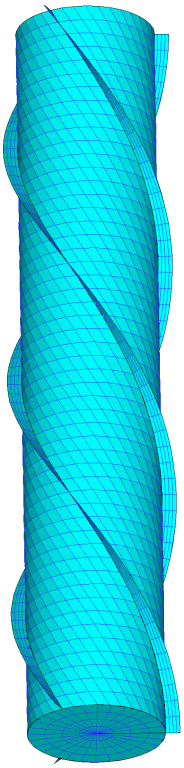
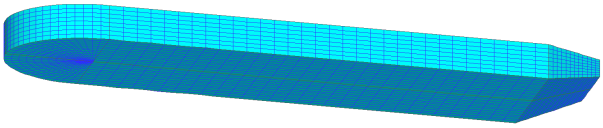
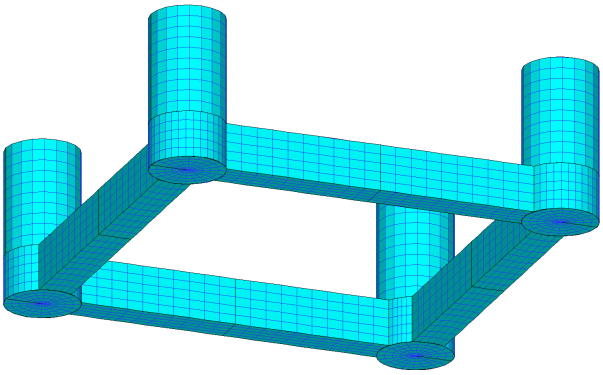
WAMIT®

USER MANUAL

Versions 6.3, 6.3PC, 6.3S, 6.3S-PC







WAMIT, Inc.

www.wamit.com

WAMIT®

USER MANUAL

Versions 6.3, 6.3PC, 6.3S, 6.3S-PC

This User Manual has been prepared for users of WAMIT Version 6.3, including the source-code Version 6.3, and the PC executable Version 6.3PC, and the second-order extension V6.3S.

WAMIT is distributed by WAMIT, Inc. under a special agreement with the Massachusetts Institute of Technology. Further information is available from the website shown below.

WAMIT, Inc.
822 Boylston St. – Suite 202
Chestnut Hill, MA 02467-2504
USA
phone 617-739-4488 *fax* 617-739-4499
www.wamit.com

TABLE OF CONTENTS

Chapter 1

INTRODUCTION

This manual describes the use of WAMIT (Version 6.3). This program exists in the standard source-code form, designated as 'V6.3', and in the PC-executable version 'V6.3PC' which is logically equivalent to 'V6.3'. A chapter concerning the use of the second-order extension 'V6.3S' is included for completeness. V6.3S includes the analysis for the second-order solution in bichromatic and bidirectional waves, including sum- and difference-frequency components.

V6.3 is a radiation/diffraction panel program developed for the linear analysis of the interaction of surface waves with offshore structures. A major feature of V6.0 and higher is the option to use a higher-order method based on B-splines to represent the velocity potential and pressure on the body surface, and to represent the geometry of the body surface in various manners. In addition to the conventional low-order panel discretization, these include new options to represent the body surface more accurately and/or with less work.

In V6.1, an option was introduced to link WAMIT with the Relational Geometry Kernel (RGKernel) of the CAD program MultiSurf With this option MultiSurf can be used to represent a wide variety of body geometries, and the same respresentation can be input directly in the higher-order solution of WAMIT. With this technique there is no need to approximate the geometry or to prepare extensive input files. (To take advantage of this approach the user must license MultiSurf and special .DLL files in addition to the standard WAMIT software.)

Starting in V6.2, a separate utility program, F2T (Frequency-to-Time domain), has been provided. F2T evaluates time-domain impulse-response functions (IRF's), from the outputs of WAMIT. IRF's can be evaluated for any combinations of linear outputs options 1-7 described in Chapter 4. IRFs may be useful for the time-domain simulation based on the linear hydrodynamic loads and other nonlinear constraints or excitations.

The execution of WAMIT has been simplified relative to earlier versions where two main programs ('POTEN' and 'FORCE') were invoked sequentially. In Version 6.0 and higher only one main program 'WAMIT' is used, with options to control the separate functions of the earlier two programs in the most effective manner for each application.

The remainder of this Chapter gives a general description of WAMIT Version 6.3 and changes made from earlier versions. Users of earlier versions should refer particularly to Sections 1.2, 1.3, 1.4, and 1.5 concerning changes and extensions introduced in Versions 6.0, 6.1, 6.2, and 6.3, respectively. A mark in the left margin, as on this line, is used to call attention to changes in Version 6.3.

The description and use of WAMIT for both the low-order and higher-order methods of solution has been unified as much as possible. Most but not all of the input and output files are 'generic', applicable to both methods in the same form. The principal exception is the Geometric Data File, which specifies the geometry of the body surface. To simplify the understanding and use of this User Manual, chapters are organized separately for generic information common to both methods, and for specific information which refers to either the low- or higher-order method separately.

In Chapter 2 a tutorial description is given to help the user get started using WAMIT in the PC environment. Source-code users may also benefit by reading this tutorial, or following the indicated examples on a PC. The examples described in Chapter 2 are for the simplest context of a single body.

Chapter 3 gives more detailed information regarding the generic input files, including the Potential Control File (POT) and Force Control File (FRC) which specify the principal non-geometric inputs for WAMIT. Also described in this Chapter are the optional files FNAMES.WAM and CONFIG.WAM, which are useful to specify input filenames and to specify various options.

Chapter 4 defines the various quantities which can be evaluated by WAMIT, and which are contained in the output files.

Chapter 5 describes special topics which are relevant only to the low-order method. This includes the Geometric Data File (GDF), which defines the coordinates of panel vertices, the use of the source formulation to evaluate the fluid velocity and second-order mean pressure on the body surface, the analysis of a body in the presence of vertical walls, and the analysis of bodies with thin elements such as damping plates or strakes.

Chapter 6 describes special topics which are relevant only to the higher-order method, including the subdivision used to represent the body surface and velocity potential on this surface, and the representation of the potential in terms of B-splines. Alternative methods for defining the body geometry are described including the use of low-order panels, the use of B-splines to provide a higher-order continuous definition, the use of explicit analytical formulae, and the use of MultiSurf geometry files. Starting with V6.2 WAMIT permits 'dipole patches' to represent thin submerged elements and their use is described in Section 6.10.

Chapters 7-9, which are generic, describe the use of extended features in WAMIT. These include the analysis of multiple interacting bodies (Chapter 7). the use of generalized modes of body motion which can be used to describe structural deformations, motions of hinged bodies, etc. (Chapter 8), and the use of a method to remove the effect of the irregular frequencies (Chapter 9).

Chapter 10 describes the relationships between memory requirements, numbers of un-

knowns, and input parameters, to provide a qualitative basis for estimating the requirements for RAM and hard disk storage, and for estimating run times. This Chapter also outlines the procedure for modification and use of .dll files, which can be used in Version 6 to describe the geometry and generalized modes.

Chapter 11 describes the second-order module V6.3S, including descriptions of the additional input files and definitions of the additional outputs.

A brief outline for the theoretical basis of WAMIT is presented in Chapter 12. Reference 26 contains a more complete review of the pertinent theory.

Chapter 13 describes the utility F2T (Frequency-to-Time domain) which can be used to transform the linear WAMIT outputs to the corresponding time-domain impulse-response functions.

A list of pertinent references and descriptions of the test runs are appended.

1.1 WAMIT Version 6

WAMIT is a radiation/diffraction program developed for the analysis of the interaction of surface waves with offshore structures. WAMIT is based on a three-dimensional panel method, following the theory which is outlined in Chapter 12. The main program consists of two top-level sub-programs POTEN and FORCE which evaluate the velocity potentials and desired hydrodynamic parameters, respectively. The water depth can be infinite or finite, and either one or multiple interacting bodies can be analyzed. The bodies may be located on the free surface, submerged, or mounted on the sea bottom. A variety of options permit the dynamic analysis of bodies which are freely floating, restrained, or fixed in position.

The flow is assumed to be ideal and time-harmonic. The free-surface condition is linearized (except in Version 6.1S where the second-order free-surface condition and body boundary conditions are imposed). We refer to this as the 'linear' or 'first-order' analysis. Mean second-order forces are included in this analysis, since they can be computed rigorously from the linear solution.

The radiation and diffraction velocity potentials on the body wetted surface are determined from the solution of an integral equation obtained by using Green's theorem with the free-surface source-potential as the Green function.

All earlier versions of WAMIT were based strictly on the low-order panel method, where the geometric form of the submerged body surface is defined by flat quadrilateral elements (low-order panels), and the solutions for the velocity potential and/or source strength are assumed constant on each panel. WAMIT Version 6 has been extended to include the complementary higher-order panel method based on a continuous B-spline representation for the velocity potential, and several alternative schemes for defining the body surface including explicit analytic formulae. The order of the B-splines is controlled by user-specified input parameters.

The two different uses of the word *order* should be noted to avoid confusion. Following

the usual conventions of marine hydrodynamics, *first-order* and *second-order* are always used here to refer to linearization of the boundary conditions and solution, whereas *low-order* and *higher-order* are used to refer to the method for representation of the body surface and solution.

The following quantities can be evaluated by WAMIT Version 6.3:

- Hydrostatic coefficients
- Added-mass and damping coefficients for all modes
- Added-mass coefficients for the limiting cases of zero or infinite wave periods
- Wave exciting forces and moments using the Haskind relations, or directly by pressure-integration from the solutions of the diffraction or scattering problems.
- Motion amplitudes and phases for a freely-floating body
- Forces restraining a body which is freely-floating in some but not all modes
- Hydrodynamic pressure and fluid velocity on the body surface
- Hydrodynamic pressure and fluid velocity in the fluid domain
- Free-surface elevation
- Horizontal drift forces and mean yaw moment by momentum integration
- All quantities listed above for user-specified generalized modes
- All components of the drift force and moment by pressure integration
- Drift force and moment in bidirectional waves

The following additional quantities can be evaluated by WAMIT Version 6.3S:

- Second-order forces on fixed or floating bodies
- Second-order pressure on the body surface
- Second-order pressure force on the waterline
- Second-order pressure in the fluid domain
- Second-order free-surface elevation
- Second-order motion amplitude on floating bodies

Two, one or no planes of geometric symmetry may be present. Part or all of the rigid-body modes can be analyzed. The program is designed to optimize the use of the available storage and minimize the computational effort for the specified planes of symmetry and modes. Matrix elements that share evaluations of the wave source potential are evaluated simultaneously.

Several techniques have been developed and implemented in WAMIT to improve the accuracy and efficiency of the solution and exploit the capabilities of a wide range of contemporary computing systems, ranging from personal computers to supercomputers. Important features of WAMIT include the use of special algorithms for the evaluation of the free-surface wave-source potential, the option to use direct, iterative, or block-iterative solution algorithms for the complex matrix equation, and the option to use either the low-order or higher-order panel methods. In combination these result in a fast, versatile, and robust code capable of analyzing offshore structures with complicated geometry.

WAMIT is designed to be flexible in its use with a variety of practical applications. It consists of two subprograms, POTEN and FORCE, which normally are run sequentially. POTEN solves for the radiation and diffraction velocity potentials (and source strengths) on the body surface for the specified modes, frequencies and wave headings. FORCE computes global quantities including the hydrodynamic coefficients, motions, and first- and second-order forces. Velocities and pressures on the body surface are evaluated by FORCE. Additional field data may also be evaluated by FORCE, including velocities and pressures at specified positions in the fluid domain and wave elevations on the free surface. Since the principal computational burden is in POTEN, the intermediate output data from this subprogram is saved in a binary 'P2F' file. Thus it is possible to make several runs with FORCE, varying the requested parameters to be output, without re-running POTEN in each instance. (V6.3S is an exception where the computational burden in FORCE can be greater than that in POTEN.)

In Version 6 the separate functions of POTEN and FORCE have been integrated into a single main program WAMIT, for ease of use. This differs from all prior versions where two separate main programs were employed. The use of Version 6 is initiated in all cases by executing the single command 'WAMIT'. This integration simplifies the use of WAMIT in typical applications where POTEN and FORCE are required sequentially, and options can be specified in the configuration file to perform separate runs of either subprogram, as explained in Chapter 2 and in Section 3.7.

Figure 1.1 shows the architecture of the two subprograms and the principal input/output files. (For simplification this figure does not include additional input files required for the case of multiple bodies, the optional spline control file which may be used to vary B-spline parameters in the higher-order analysis, additional input files which are required for the second-order analysis program V6.3S, and three output files which log errors, warnings, and other auxiliary data.)

The analysis for the generalized modes also requires an additional input file or special subroutine, to define the user-specified modes. Figure 9.1 in Chapter 9 shows the flow chart of POTEN for this case.

WAMIT Version 6 is written in FORTRAN-90 (ANSI/ISO Standard) and has been tested on a wide variety of contemporary computer systems. The subroutine DATTIM, which is used to set the date and time of the run, uses a generic F-90 intrinsic function and no longer requires modification of the source code for use on different systems. The only non-standard extension which has been retained from previous versions of WAMIT is the dollar-sign prompt '$', which facilitates interactive data input. Source-code users who

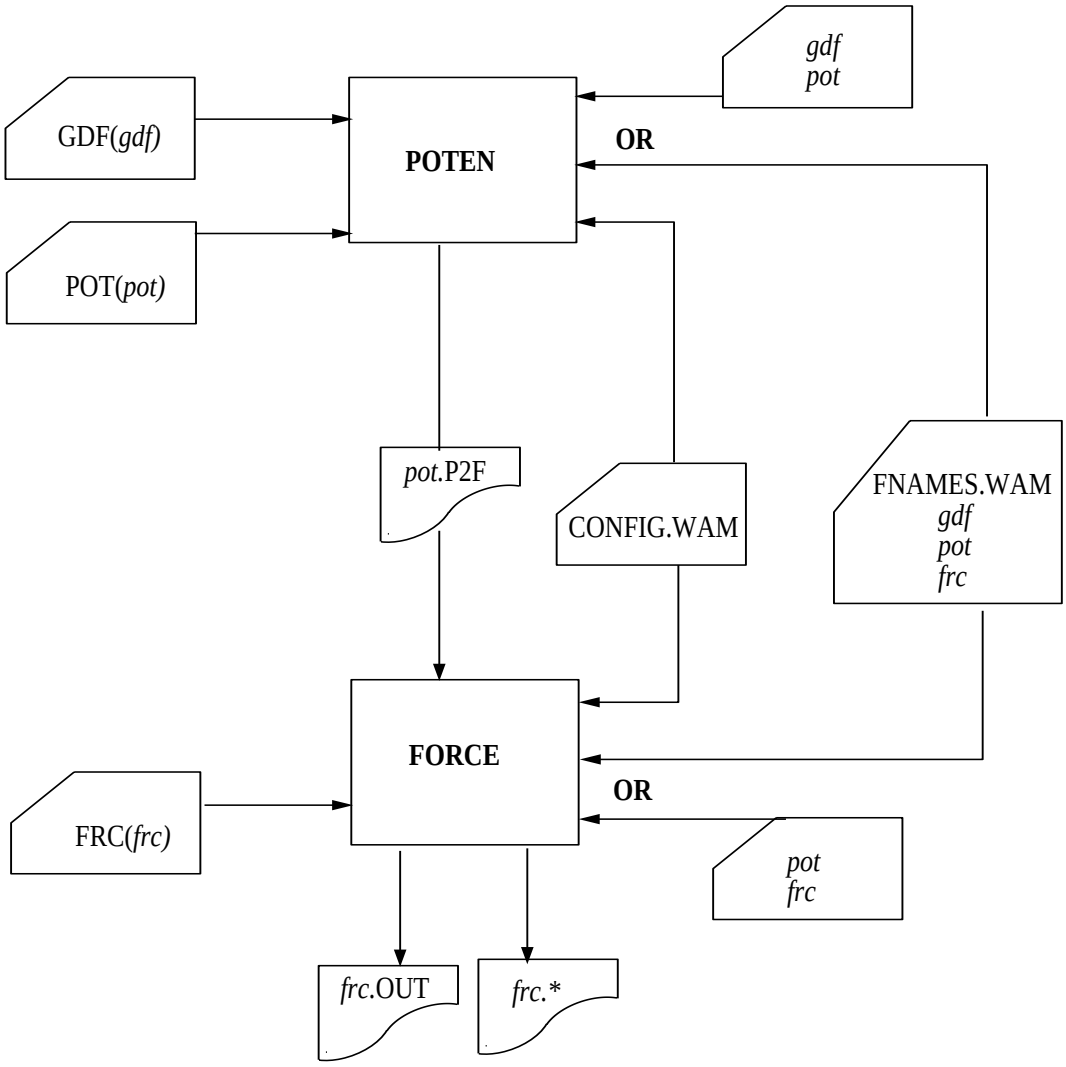


Figure 1.1: Flow chart of WAMIT showing the subprograms POTEN and FORCE with their associated input and output files. Filenames in italics are specified by the user. The three primary input files described in Chapters 3, 5, and 6 are indicated in the left-hand column. The names of these files are prescribed either by the optional file FNAMES.WAM, or by the interactive inputs represented by the top and bottom arrows in the right-hand column. Note that the P2F file output from POTEN is given the same filename as the input control file, with the extension P2F. The output file from FORCE is given the same filename as the force control file, with the extension OUT. The P2F file may be saved and reused for various applications of the FORCE module where the same velocity potentials apply. Asterisks ($*$) denote the extensions corresponding to each option in the numeric output files, as listed in the table in Section 3.3.

are not able to use this extension should remove the symbol $ by editing the source file RDFILESC.F.

The source code of WAMIT assumes the use of a PC-based Windows/DOS operating system. 'README.TXT' contains information regarding system requirements and configuration to run the PC-executable version of WAMIT. Appropriate modifications of the source code may be necessary for all other compilers.

1.2 CHANGES INTRODUCED IN Version 6.0

In relation to WAMIT Version 5.4, new features which affect the use of Version 6 are outlined below. For users familiar with versions of WAMIT prior to V5.4, and earlier versions of the higher-order code HIPAN, additional changes are also listed below. More detailed information is provided in the subsequent chapters as indicated.

- The main program WAMIT includes POTEN and FORCE as subprograms.
- The new input parameter ILOWHI has been added to control the use of the low- or higher-order panel methods.
- Alternative options have been introduced to specify the input filenames.
- Faster algorithms have been implemented to reduce redundant evaluations of the Green function.
- In the low-order method, some or all of the body panels can be defined to represent thin plates or appendages of small thickness, such as damper plates or strakes.
- For generalized modes the mode shapes can be specified in a separate .dll file.
- In the higher-order method the geometry can be specified by B-splines, explicit algorithms, or flat panels.
- Explicit algorithms to specify the geometry can be included in a .dll file, and related body dimensions can be specified in the .gdf input file. Several different generic types of bodies can be analyzed simply by inputting the relevant dimensions, without the effort required to prepare a conventional panelization.
- New parameters IALTPOT and IALTFRC have been included to provide a more unified and flexible range of input formats.
- Version 6.0PC has been compiled with the Compaq Digital Fortran compiler, optimized for use on PC systems with the Pentium CPU.

In relation to WAMIT Version 5.3, additional new features which were incorporated in Version 5.4 also apply to Version 6.0 as outlined below.

- All restrictions on the maximum number of inputs are removed. In V6.0 all of the relevant arrays are dimensioned at run time. In practice there may be practical limitations, e.g. regarding the number of panels, depending on the memory and disk capacity of the hardware and on the run time, but these limits will generally be much larger than in earlier versions.
- A new direct-solver subroutine has been utilized. This substantially reduces the run time when the direct solution method (ISOLVE=1) or block-iteration (ISOLVE>1) are used.
- The PC-executable code is compiled in a generic form for all licensed users, and specific licensees are identified by including a special binary input file **userid.wam**. This file must be available for input during the WAMIT runs. Provision has been made in the **config.wam** file to permit users to access this input file from a specified sub-directory, as explained in Section 3.9; this option avoids the need to copy **userid.wam** to all directories where WAMIT is used.
- Provision has been made in the configuration file (**config.wam** or ***.cfg**) to permit users to specify a different sub-directory or drive for scratch storage of large arrays, as explained in Section 3.9. This option effectively doubles the maximum limit on available disk memory for systems with multiple hard drives and/or partitions.
- The warning and error messages have been extended to provide more specific information concerning errors in input files.
- A new optional input array has been added to the .frc control file, as explained in Sections 3.3 and 3.4. This permits the user to specify that some or all of the modes selected in the inputs to POTEN are restrained with zero amplitude, and to re-run the FORCE subprogram alone. (Previously this required re-running POTEN without such modes.)
- Optional output files can be used to generate low-order GDF files and to generate data files for perspective plots of the panels and/or patches.

In relation to the higher-order code HIPAN Version 2, additional new features which are incorporated in WAMIT Version 6.0 are listed below. These features are described in Chapter 6.

- Options exist to define the B-spline parameters automatically, to achieve a specified maximum panel size.
- Multiple bodies with separate planes of symmetry can be input, with automatic reflection of the geometry.
- In the higher-order method the geometry can be specified by B-splines, explicit algorithms, or flat panels.
- Body geometry for barges and similar structures can be defined using a minimum number of flat quadrilateral elements, with the same input format as for the low-order analysis.

- Explicit algorithms to specify the geometry can be specified in a .dll file and related body dimensions can be specified in the .gdf input file.
- Filenames for the .gdf files and related data can be input either in the .pot file, as in HIPAN, or in the same format as used in earlier versions of WAMIT.

1.3 ADDITIONAL CHANGES IN Version 6.1

New features which are included in Version 6.1 are outlined below.

- Arrays of uniformly spaced wave periods and heading angles can be input more compactly in the Potential Control File, by specifying the number, first value, and increment (see Section 3.1).
- The configuration file CONFIG.WAM can be assigned a different filename, defined by the user, with the extension .CFG (see Section 3.7).
- In the higher-order method, the default points for hydrodynamic pressure on the body surface are modified, as described in Section 4.10.
- The hydrodynamic pressure on the body surface can be evaluated at user-specified points (x, y, z) which are listed in a special input file. This extension, which is available in both the low-order and higher-order solutions, facilitates the integration of WAMIT with structural-analysis codes where the hydrodynamic pressure is required at specified locations. This procedure is described in Section 4.11.
- The separate components of the hydrodynamic pressure on the body surface due to each of the radiation modes and due to the diffraction field can be output separately. This extension is useful for post-processing when the dynamics of the body are modified. This procedure is described in Section 4.12.
- In the higher-order solution method geometry models developed with MultiSurf can be used directly in WAMIT. This procedure is described in Section 6.7.
- One or two additional .dll files are required to run Version 6.1PC (see Sections 2.1 and 6.7)
- The second-order extension has been updated to include both the low-order and higher-order methods of solution, and to include other features of the V6 first-order code. Chapter 11 contains a description of these features.

1.4 ADDITIONAL CHANGES IN Version 6.2

New features which are included in Version 6.2 are outlined below.

- Internal tanks with free surfaces can be analysed either separately or coupled with the dynamics of vessels. (See Section 10.7)
- Impulse response functions (IRFs) in the time domain can be evaluated for the output options 1-7 using a utility program F2T. F2T is provided as a separate program along with WAMIT V6.2. F2T converts the outputs of WAMIT into IRFs using Fourier transforms. (See Chapter 13 and TEST 14A in Appendix A14.)
- In addition to the added mass and damping, the pressure and the fluid velocity on the body surface (Option 5) and at the field points (Options 6 and 7) can be evaluated for zero and infinite wave frequencies (Section 3.1).
- The domain of the parameters for B-Splines representing the body surface is no longer limited to $(-1, 1)$ and B-Splines defined over an arbitrary domain can be accepted. (Section 6.6)
- Zero and infinite frequencies can be specified with IRR > 0 (irregular frequency removal option), which was not permitted in the previous version. IRR > 0 is ignored for these two frequencies. This update enables users to run these frequencies with other frequencies in one execution of WAMIT. (Section 3.1)
- The subroutine FGR0I1, evaluating the Rankine source in two parallel horizontal walls, is extended to calcuate double derivatives and renamed as FGR0I2. This extension is necessary to evaluate the fluid velocity at zero and infinite frequencies when IOPTION(7)=1.
- The separate components of the hydrodynamic pressure and the fluid velocity at the field points due to each of the radiation modes and due to the diffraction field can be output separately. This extension is useful for post-processing when the dynamics of the body are modified. This procedure is described in Section 4.12.
- In the higher-order method, some or all of the body surface can be defined to represent thin structures (see Section 6.10). Similar extension was made in conjuction with the low-order method in V6.0.
- Mean drift forces by pressure integration on the rigid bodies can be evaluated when the generalized modes are used on other bodies in multi-body interaction. This option was not supported in previous versions.
- A new output file `wamitlog.txt` is created during the WAMIT run to provide an archival record of the run, including copies of the principal input files, and various auxiliary information. Further information is provided in Section 10.1.

1.5 ADDITIONAL CHANGES IN Version 6.3

New features which are included in Version 6.3 are outlined below.

- Uniform arrays of field points can be input in a more convenient manner. (See Section 3.10)
- A special option can be used to efficiently analyze the wave field generated by one or more wavemakers situated in planes of symmetry. (See Section 10.8)
- All six components of the mean drift forces and moments can be evaluated from the momentum flux through a control surface surrounding the body.
- A symmetry plane can be used when there are flat dipole elements on the plane of symmetry. (See Section 5.4 and Section 6.10)
- A supplementary output file *out*.hst is created to output the hydrostatic matrix of restoring coefficients, including generalized modes and/or tanks. (See Sections 4.9, 8.4, and 10.7.)
- The DLL library file GEOMXACT has been extended to include several new analytical geometries.
- The DLL library file NEWMODES has been extended to include generalized modes to represent a bank of paddle-type wavemakers, and to represent the vertical modes of a vessel with hinges.

Chapter 2

GETTING STARTED

In this Chapter instructions are provided for installing the WAMIT-PC software, and making simple test runs.

The two principal subprograms of WAMIT are POTEN and FORCE. POTEN solves for the velocity potential on the body surface, and optionally also for the source strength. FORCE evaluates physical parameters including the force and motion coefficients, and field data including the fluid pressure, velocity, and free-surface elevation. The basic sequence in a typical application of WAMIT is (1) prepare the input files; (2) run WAMIT. The principal results are then contained in output files which may be printed and post-processed. This architecture is illustrated in Figure 1.1.

The principal input files to the subprogram POTEN are the Potential Control File (POT) which specifies parameters including the fluid depth, wave periods, and wave heading angles, and the Geometric Data File (GDF), which describes the geometry of the structure to be analyzed. These files are discussed briefly below, and in more detail in subsequent Chapters. The principal input files to the subprogram FORCE are the Force Control File (FRC), which specifies inputs regarding the body dynamics, and the P2F file ('Poten to Force') which transfers data from POTEN to FORCE.

There are three additional input files which should also be considered: (1) Licensed users of WAMIT V6PC must utilize a unique input file **userid.wam**, which identifies the site license, (2) the optional input file **fnames.wam** is recommended to specify the input filenames, and (3) the optional configuration file (**config.wam** or ***.cfg**) may be used to specify certain options and other information used by the program. The file userid.wam is provided by WAMIT Inc. Samples of the fnames.wam and configuration files are included with the test runs; for example the files **fnames.1** and **config.1** are intended for use with TEST01. (Before running TEST01 the user must rename or copy these files, as explained below.)

2.1 INSTALLATION AND SETUP

The WAMIT-PC software is delivered in two forms, (1) on a CD-ROM disk, or (2) as a compressed zip file suitable for electronic transfer. This software includes the main executable program **wamit.exe** and a complete set of input files for the standard test runs TESTn (n=01,02, ...), and the labeled output files TESTn.OUT for these test runs. Also included is the text file **readme.txt** with additional information.

In the standard Version 6.3PC there are four additional dynamic link library files (DLL) which must be installed in the same directory (i.e. 'folder') as the executable file wamit.exe. If these DLL files are missing the program will not run, regardless of the inputs and options specified. The four DLL files are **geomxact.dll**, **newmodes.dll**, **rg2wamit.dll**, and **dforrt.dll**. (The extended Version including the capability to input MultiSurf models requires five DLL files, as explained in Section 6.7.)

Before installing the software a new main directory (folder) should be made. The recommended name is C:\WAMITv6, but the user may prefer to use a different drive.Copy all of the software to this directory, preserving the same subdirectories (folders) within the new directory as are on the CD ROM or in the zip file. If the software is received in a zip file by electronic transfer, it must be unzipped using a local utility, and all unzipped files should be saved in the directory C:\WAMITv6 and in subdirectories corresponding to those in the zip file. The text file **readme.txt** includes an outline of the directory tree and files. If any difficulties are encountered during installation and testing the user should confirm that the subdirectories and files correspond to the description in **readme.txt**.

2.2 DEMONSTRATION PROGRAMS

A special demonstration version of WAMIT can be downloaded in .zip format from the web site **www.wamit.com**. This program, **wamit_demo.exe**, can be used for demonstration or educational purposes, without obtaining an end-user license, subject to the conditions stated in the website. The installation and use of this software is the same as the licensed version, with the following exceptions:

- The program only accepts geometry inputs from the standard input files for the test runs described in the Appendix.
- The site license identification file **userid.wam** and DLL files are not required.
- The program runs are interrupted after display of the header, and the user is prompted to press the Enter key to continue the run.

The downloadable zip file includes all of the input files required to execute the standard test runs.

2.3 STANDARD TEST RUNS

Various standard test runs are included with the software, to illustrate different types of applications and features of the program. The results of these test runs can be used to confirm that the installation and setup of the program have been performed correctly by the user. The test runs also provide opportunities to use (and modify) existing input files, for tutorial purposes. The remainder of this Chapter is intended to guide new users through these procedures. Descriptions of each test run are included in the Appendix.

If the WAMIT software is installed in accordance with the instructions above, the required .EXE, .DLL, and USERID.WAM files will be installed in the directory C:\WAMITv6. All required input files for the standard test runs will be copied to the subdirectory C:\WAMITv6\TESTRUNS. Benchmark versions of the output files **test*.out** will be copied to the subdirectory C:\WAMITv6\TESTRUNS\OUT. These benchmark output files can be compared with results obtained by the user to ensure that the software is installed correctly.

Before running WAMIT with the standard test runs, the user should (1) open a DOS Command Prompt Window (in the Windows environment a Command Prompt Window is opened by clicking on 'Start', 'Programs', 'Command Prompt'), and (2) change the default subdirectory by entering the command `cd \WAMITv6\TESTRUNS`.

Since the executable file wamit.exe is resident in the directory C:\WAMITv6, the appropriate command to execute WAMIT is 'C:\WAMITv6\WAMIT'. There are three alternative shortcuts which may be used to simplify this command: (1) add C:\WAMITv6 to the system PATH, (2) copy the files wamit.exe, *.dll, and userid.wam from this directory to another directory which is included in the PATH, or (3) use the batch file **wamit.bat** which is supplied in the testruns subdirectory.

2.4 RUNNING TEST01

Test Run 01 evaluates the added-mass and damping coefficients, exciting forces, motions, wave elevations, field pressures, fluid velocities and drift forces for a freely-floating truncated vertical circular cylinder of radius 1.0m and draft 0.5m, in infinite water depth for three wave periods and one wave heading angle. Further details are contained in Appendix A.

The corresponding input files **test01.gdf**, **test01.pot**, and **test01.frc** are included in the subdirectory C:\WAMITv6\TESTRUNS. In order to specify the appropriate filenames during the run, first copy the file **fnames.01** to the file **fnames.wam**. (Copying is recommended, in preference to renaming the file, to preserve the original file.) The appropriate DOS command is

```
copy fnames.01 fnames.wam
```

Next enter **wamit** to start the run. During execution of the subprogram POTEN the monitor displays the starting time, and after the solutions for the velocity potentials are

obtained at each wave period the monitor will display a new line of information including the period (in seconds), time, and the maximum number of iterations required for the radiation and diffraction solutions. After the first solutions are displayed the results for subsequent periods run faster, since the panel integration of the Rankine components of the source potential are only evaluated initially and saved for reuse. After the third period the intermediate storage file **test01.p2f** is created on the disk, storing the velocity potentials and other inputs to the subprogram FORCE. Output from FORCE will appear relatively quickly on the screen, and the same output is stored in the file **test01.out**. The latter file includes useful identification information concerning the inputs, body parameters, run times and dates. This is followed for each period by tabulations of the hydrodynamic parameters requested in **test01.frc**. Assuming the standard version of **test01.out** has been saved in a subdirectory, as recommended in Section 2.1, the data in the new version of **test01.out** can be compared with the standard file with the same name. On a contemporary PC the total run time should be a few seconds.

2.5 RUNNING TEST11

Test Run 11 is intended to complement TEST01, but using the higher-order method of solution (ILOWHI=1). The body dimensions and other inputs are the same, but the surface of the cylinder and also the solution for the velocity potential are represented in a more accurate, continuous manner by B-splines, as explained in Chapter 6.

To run TEST11 follow the same procedure outlined above for TEST01, but replace '01' by '11' in all references to filenames and extensions. A comparison of the results from these complementary tests is summarized in Section A.11 of the Appendix.

2.6 OTHER TEST RUNS

The procedure for running the other test runs is the same as explained above, except for copying the appropriate files to **fnames.wam** from the original versions **fnames.n** where 'n' is the test number.

A special batch file **runtests.bat** is included in the licensed version and a similar file **rundemo.bat** in the demonstration version. Executing the corresponding batch file command will run all of the tests in succession.

2.7 MEMORY AND STORAGE RESTRICTIONS

WAMIT uses scratch files on the hard disk for temporary storage at runtime. Depending on the run parameters, the total number of scratch files and/or their size may become quite large. Normally these files are deleted by WAMIT after the files are no longer needed, and before the program stops. However some PC/Windows configurations save these deleted files in a 'recycled' directory, and this can cause the hard disk to become overloaded. Users

who experience this problem should delete the accumulated files in the recycled directory, or alternatively change their system setup to avoid saving a backup of all deleted files.

Contemporary Pentium PC systems generally have sufficient capacity for all of the Test Runs. Further information regarding memory requirements is given in Chapter 10.

2.8 MODIFYING THE INPUT FILES

A text editor can be used to edit the input files. Users should refer to Chapter 3 for a complete explanation of the data in these files, but a few simple modifications will be outlined here for tutorial purposes, in the context of Test Run 01. Before proceeding further check that the files FNAMES.WAM and CONFIG.WAM correspond to this test run, and re-copy these files if necessary following the instructions in Section 2.4.

As the first modification we might request FORCE to perform additional computations, for the same periods and wave headings analyzed by POTEN, but with modified values of the vertical center of gravity (VCG) and radii of gyration (XPRDCT) in the force control file TEST01.FRC. In the standard file VCG=0.0 (center of gravity in the waterplane) and the radii of gyration are set equal to 1.0 (the three diagonal elements of the XPRDCT matrix). A positive VCG will move the center of gravity above the waterplane, reducing the pitch/roll hydrostatic stability and affecting these RAO's in longer wave periods. Modifying the radii of gyration should change the same RAO's primarily at shorter wave periods. It is not necessary to re-run POTEN in this case, provided the file TEST01.P2F has been retained for the POTEN output. To avoid the extra run time of POTEN, add the line IPOTEN=0 to the configuration file config.wam (See Section 3.7, and also the file CONFIG.17b which includes the same line). After modifications are made to the file TEST01.FRC it is advisable to save the modified file a with different filename, and to edit the FNAMES.WAM file to show the corresponding new filename for the force control file. If this procedure is followed, the output file will carry the same filename (with the **.out** extension) to distinguish it from the original file TEST01.OUT. The user may then compare the RAO's in the different output files to discern the effect of these changes.

As the second modification WAMIT may be re-run with a more extensive list of wave periods. Edit the potential control file TEST01.POT with the following changes:

- on line 5 change the number of wave periods (NPER) from 3 to 10
- on line 6 replace the three original wave periods by ten new periods in decimal format
- depending on your preference, (1) save the modified file with the original name TEST01.POT, or (2) save the modified file with a new filename and correct the FNAMES.WAM file on line 2 with the new filename.

Since it is necessary to re-run POTEN in this case, either delete the line IPOTEN=0 if this was added to config.wam, or copy config.01 once again to config.wam. During the run, if the original filename TEST01.POT is retained, the user will be prompted whether or not to overwrite the old output file TEST01.P2F; overwriting is the simplest procedure

to follow, in this circumstance, otherwise the new .P2F filename must be specified before the FORCE run is executed.

The input files for other test runs can be used to illustrate various options and modifications.

Chapter 3

USING WAMIT

A typical application of the standard WAMIT program V6 or V6PC will consist of (a) preparing appropriate input files; (b) running WAMIT; and (c) using the resulting output files. Most of the required input files are 'generic', with the same format and data irrespective of whether the low-order or high-order method is used. These files are described in this Chapter. The principal exception is the geometric data file (GDF), which is described separately for the two methods in Chapters 5 and 6, respectively.

To simplify the presentation this Chapter will describe the required input files and resulting output files for a basic application involving the analysis of a single body. Further information is given in Chapters 7-9 for the appropriate modifications of the input files for specific purposes, including the analysis of multiple bodies, the analysis including generalized modes of body motion, and the option to remove irregular-frequency effects.

The execution of a WAMIT run is divided between two subprograms, POTEN and FORCE, as explained in Chapter 1. In special circumstances it is useful to run WAMIT and execute only one of the two subprograms, using the optional parameters IPOTEN=0 or IFORCE=0 to skip the corresponding subprogram execution. These parameters can be input in the configuration file, as explained below in Section 3.7. In the default case (IPOTEN=1, IFORCE=1) both subprograms are executed sequentially in the same run.

Some input files are used only by POTEN or FORCE, whereas other input files are used by both. The following table lists the input files which are prepared by the user, and indicates the relevant subprogram(s):

Filename	Usage	Description
runidp.pot	POTEN	Potential Control File (Sections 3.1-2)
body.gdf	POTEN	Geometric Data File (Chapters 5,6)
body.spl	POTEN	Spline Control File (Section 6.11)
FNAMES.WAM	POTEN/FORCE	Filenames list (Section 3.8)
∗.cfg	POTEN/FORCE	Configuration file (Section 3.7)
runidf.frc	FORCE	Force Control File (Sections 3.3-4)

Here *runidp* is a user-specified filename assigned to the POT file. The same filename is assigned to the P2F file, which is output by POTEN and input to FORCE. Similarly, *runidf* is a user-specified filename assigned to the FRC file and this filename is assigned to the principal output files from FORCE. *body* is the user-specified filename assigned to the GDF file. If the optional Spline Control File is used it must have the same filename.

The following table lists the output files which are produced by each subprogram:

Filename	Program	Description
runidp.p2f	POTEN	P2F File (binary data for transfer to FORCE)
errorp.log	POTEN	Error Log File (This section)
errorf.log	FORCE	Error Log File (This section)
runidf.out	FORCE	Formatted output file (Chapter 4)
runidf.num	FORCE	Numeric output files (Section 4.9)
body.pnl	FORCE	Panel data file (Section 4.9)
runidf.fpt	FORCE	List of field points (Section 4.9)

The structure of input and output files is illustrated in the flow chart shown in Figure 1.1. Further details are provided below.

The primary generic data files are the two 'control files' input to POTEN and FORCE. These are referred to as the Potential Control File (POT), with the extension '.pot', and the Force Control File (FRC), with the extension '.frc'.

All WAMIT input files are ASCII files. The first line of most files is reserved for a user-specified header, consisting of up to 72 characters which may be used to identify the file. If no header is specified a blank line must be inserted to avoid a run-time error reading the file. The remaining data in each file is read by a sequence of free-format READ statements. Thus the precise format of the input files is not important, provided at least one blank space is used to separate data on the same line of the file. Further details regarding the formats and names of files are contained in Section 3.10.

Additional input files may be used to assist in using WAMIT, to select various options, and to optimize its use for specific applications. The optional file FNAMES.WAM is used to specify the names of the GDF, POT, and FRC input files to avoid interactive input of these names (see Section 3.6). The optional input file CONFIG.WAM may be used to configure WAMIT and to specify various options, as described in Section 3.7. (In Version 6.2-3 this configuration file may be assigned a different filename by the user.) The third additional input file, the optional Spline Control File (SPL), is used only in the higher-order method, as described in Section 6.11.

In V6PC the input file 'userid.wam' is read by both POTEN and FORCE, to identify the licensee name and address for output to the headers at run time, and to write this information in the header of the .out output file. This file is prepared by WAMIT, Inc., and must be available for input to POTEN and FORCE at runtime. To be available for input, the file USERID.WAM must either be copied to the default directory with other input/output files, or else the pathname indicating the resident directory must be listed in CONFIG,WAM, as explained in Section 3.7.

Two alternative formats for the POT and FRC files are described separately in Sections

3.1-2 and 3.3-4.

Alternative Form 1 for the POT file is identical to the POT file used in earlier versions of WAMIT. Alternative Form 2 was developed initially for the higher-order program HIPAN, and is retained here as an option for convenience. Either Form 1 or Form 2 can be used irrespective of whether the low- or higher-order method is used. Form 2 is particularly convenient for the analysis of multiple bodies.

Alternative Forms 1 and 2 for the FRC file are identical to the corresponding alternative forms used in Version 5 of WAMIT. However a more consistent specification of these two alternatives is now available, via the parameter IALTFRC in the configuration file. For a rigid body which is freely floating, and not subject to external constraints, Alternative Form 1 (Section 3.3) may be used, with the inertia matrix of the body specified in terms of a 3×3 matrix of radii of gyration. Alternative Form 2 (Section 3.4) permits inputs of up to three 6×6 mass, damping, and stiffness matrices to allow for a more general body inertia matrix, and for any linear combination of external forces and moments. (A third alternative format may be used for multiple bodies, as described in Section 7.5.)

Several output files are created by WAMIT with assigned filenames. The output from POTEN for use by FORCE is stored in the P2F file (Poten to Force) and automatically assigned the extension P2F. The final output from FORCE is saved in a file with the extension OUT which includes extensive text, labels and summaries of the input data. FORCE also writes a separate numeric output file for the data corresponding to each requested option, in a more suitable form for post-processing; these files are distinguished by their extensions, which correspond to the option numbers listed in the table in Section 3.3.

Two additional numeric files are generated when the FRC file specifies either Option 5 (pressure and fluid velocity on the body surface) or Options 6-7 (pressure and fluid velocity at field points in the fluid), to assist in post-processing of these data. For Option 5 a 'panel geometry' file with the extension PNL is created with data to specify the area, normal vector, coordinates of the centroid, and moment cross-product for each panel on the body surface. For Options 6-7 a 'field point' file with the extension FPT specifies the coordinates of the field points in the fluid.

The filenames assigned to the various output files are intended to correspond logically with the pertinent inputs, and to simplify file maintenance. Thus the filename of the P2F file is derived in the program from the filename of the POT file, and similarly for the OUT output file from the FRC file. To make this convention more specific it is necessary to define the input filenames specified by the user. For the present discussion these are assumed to be, respectively *pot*.POT and *frc*.FRC where the italicized portion is user-specified. The resulting output filenames are then assigned as follows: *pot*.P2F is the output file from POTEN, input to FORCE. *frc*.OUT is the principal output file from FORCE. The various numeric output filenames are assigned the same name as the OUT file, but with extensions corresponding to each option number. (The alternative name OPTN may be specified by setting the optional switch NUMNAM=1 in the configuration file, as described in Section 3.7.) For Option 5, the panel data file is assigned the filename from the Geometric Data File, *gdf*.PNL, since this data is specific to the body identified

in the GDF file. If NUMNAM=1, the generic name 'gdf.pnl' is assigned to the panel data file. For Options 6-7, the field point file is assigned the filename *frc*.FPT (or OPTN.FPT, if NUMNAM=1), to correspond to the force control file and other numeric output files. These conventions are illustrated in Figure 1.1.

3.1 THE POTENTIAL CONTROL FILE (Alternative Form 1)

The Potential Control File is used to input various parameters to the POTEN subprogram. The name of the POT file can be any legal filename accepted by the operating system, with a maximum length of 16 ASCII characters, **followed by the extension '.pot'**.

Two alternative forms of the POT file can be used in Version 6, as specified by the optional parameter IALTPOT in the config.wam file. The default case (IALTPOT=1) is described in this Section. The alternative form (IALTPOT=2) is described separately in Section 3.2.

If IALTPOT is not specified, or if it is specified with the default value (IALTPOT=1), the data in the Potential Control File must be in the format listed below:

```
header
ISOR†
IRR†
HBOT {XBODY(1) XBODY(2) XBODY(3) XBODY(4)}†
IQUAD† ILOG† IDIAG†
IRAD IDIFF
MODE(1) MODE(2) MODE(3) MODE(4) MODE(5) MODE(6)
NPER
PER(1) PER(2) PER(3) ...  PER(NPER)
NBETA
BETA(1) BETA(2) BETA(3) ...  BETA(NBETA)
```

The symbol † denotes that the corresponding parameters may be input via the configuration file and not included in the Potential Control File (See Section 3.7). If the higher-order method is used (ILOWHI=1) the parameters ISOR, IQUAD, IDIAG are ignored, but they must be included either in the configuration file or in the Potential Control File to correctly read the remainder of the data.

The data shown on each line above are read consecutively by corresponding read statements. Thus it is recommended to preserve the line breaks indicated above, except that if a large number of periods (PER) and/or wave heading angles (BETA) are input, these may be placed on an arbitrary number of consecutive lines.

The definition of each variable in the Potential Control File is as follows:

'header' denotes a one-line ASCII header dimensioned CHARACTER∗72. This line is available for the user to insert a brief description of the file.

ISOR is the integer used to specify whether the source strength is evaluated:

ISOR= 0: Do not evaluate the source strength.

ISOR= 1: Evaluate the source strength.

The source strength is required in the low-order method (ILOWHI=0) if FORCE evaluates the fluid velocity on the body (IOPTN(5)=2 or 3), the pressure/free-surface elevation

or velocity in the fluid domain by the source formulation (IOPTN(6)=2 or IOPTN(7)=2) and the mean drift force and moment from pressure integration (IOPTN(9)=1 or 2). Further information on these options is given in Section 5.2. Running POTEN with ISOR=1 requires substantially longer run time and larger scratch storage. In the higher-order method (ILOWHI=1) all of the FORCE evaluations are made directly from the solution for the velocity potential, and ISOR=0 is required.

IRR is the integer used to specify whether the irregular frequencies are removed or not. A detailed explanation for this parameter is provided in Chapter 9.

IRR= 0: Do not remove the effect of the irregular frequencies.

IRR= 1: Do remove the effect of the irregular frequencies. (User needs to represent the interior free surface by panels or patches.)

IRR= 2: Do remove the effect of the irregular frequencies. (Program projects the body panels on the interior free surface.)

IRR= 3: Do remove the effect of the irregular frequencies. (Program automatically discretizes the interior free surface.)

IRR>1 is not permitted in the higher-order method (ILOWHI=1).

The parameters ISOR and IRR can be input either in the POT or the configuration file. If ISOR or IRR is specified in the configuration file the corresponding line of the POT file should be deleted, as described in Section 3.7.

HBOT is the dimensional water depth. By convention in WAMIT, a value of HBOT less than or equal to zero is interpreted to mean that the water depth is infinite. It is recommended to set HBOT=-1. in this case. If HBOT is positive it must be within the range of values

$$10^{-5} < \mathrm{HBOT} \times \omega^2/\mathrm{GRAV} < 10^5$$

where $\omega = 2\pi/\mathrm{PER}$ is the radian frequency of the incident waves and GRAV is the gravitational acceleration constant. For each run the value of GRAV is input in the GDF file, as described in Chapters 5 and 6. All dimensional inputs with the units of length, including HBOT, must be consistent with the input GRAV.

XBODY(1), XBODY(2), XBODY(3) are the dimensional (X, Y, Z) coordinates of the origin of the **body-fixed coordinate system** relative to the **global coordinate system**, input in the same unit of length as GRAV. The global coordinate system is required when walls are present (Section 5.3) and when multiple bodies are analyzed (Chapter 7). The global coordinate system is also used in place of the body coordinate system to define field-point data (fluid pressures, velocities, and free-surface elevation). Normally, in the absence of walls or multiple bodies, the coordinates XBODY(1) and YBODY(1) should be set equal to zero unless it is desired to refer the field-point data to a different coordinate system from that of the body. (The origin of the global coordinate system is on the free surface.) The incident-wave velocity potential is defined relative to the global coordinate system. Consequently, the phases of the exciting forces, motions, hydrodynamic pressure and field velocity induced by the incident wave are understood relative to the incident-wave elevation at $X = Y = 0$.

XBODY(4) is the angle in degrees of the $x-$axis of the body coordinate system relative to the $X-$axis of the global system in counterclockwise sense (see Figure 5.2).

The array XBODY may be moved from the POT file to the configuration file (See Section 3.7).

IQUAD, ILOG, IDIAG are control indices which may be used to increase the precision of the panel integration of the Green function and its derivatives, at the expense of computation time; in each case the default setting zero will minimize the computation time. IQUAD and IDIAG are only used in the low-order method (ILOWHI=0). However if the Alternative Form 1 POT file is used IQUAD and IDIAG must be input, even when ILOWHI=1, to avoid an error reading the input data.

IQUAD= 0: The integration of the regular wavelike part of the Green function and its derivatives is carried out by using a single node at the centroid of each panel.

IQUAD= 1: The integration is carried out by using a four-node Gauss quadrature. (This option can be used to verify the accuracy of computations carried out with the faster single-node quadrature. However this practice is generally not as useful as the alternatives of increasing the number of panels or using the higher-order method.)

ILOG= 0: The logarithmic singularity is included with the wavelike component of the Green function and is integrated by quadrature over each panel.

ILOG= 1: The logarithmic singularity in the Green function is subtracted and integrated analytically for pairs of panels for which the Rankine image singularity $1/r'$ is also integrated analytically. (This option produces more accurate results.) ILOG=1 is required when panels are defined in the plane of the free surface, including the following two cases: (1) bodies with horizontal physical surfaces lying in the plane of the free surface, and (2) use of the irregular frequency option, where panels are located on the interior free surface inside the body waterline. In these two cases, execution of the program is interrupted with an error message if ILOG=0.

IDIAG= 0: In determining those pairs of panels where the above analytic integration is required, the distance between their centroids is compared with the characteristic length based on the square root of their area.

IDIAG= 1: The characteristic length of each panel is based on its maximum diagonal. [This option is more accurate for panels with very large aspect ratios.]

Generally it is recommended to use the default values IQUAD= 0, ILOG= 0, and IDIAG= 0. ILOG= 1 may be useful when studying local characteristics such as run up near the waterline of the body. IDIAG= 1 may be useful when some of the panels used to describe the body have a very large aspect ratio.

Any or all of these control indices may be moved from the POT file to the configuration file (See Section 3.7).

IRAD, IDIFF are indices used to specify the components of the radiation and diffraction problems to be solved. The following options are available depending on the values of IRAD and IDIFF:

IRAD= 1: Solve for the radiation velocity potentials due to all six rigid-body modes of motion.

IRAD= 0: Solve the radiation problem only for those modes of motion specified by setting the elements of the array MODE(I)=1 (see below).

IRAD= −1: Do not solve any component of the radiation problem.

IDIFF= 1: Solve for all diffraction components, i.e. the complete diffraction problem.

IDIFF= 0: Solve only for the diffraction problem component(s) required to evaluate the exciting forces in the modes specified by MODE(I)=1.

IDIFF= −1: Do not solve the diffraction problem.

MODE is a six-element array of indices, where I=1,2,3 correspond to the surge, sway and heave translational modes along the body-fixed (x, y, z) axes, and I=4,5,6 to the roll, pitch and yaw rotational modes around the same axes, respectively. Each of these six indices should be set equal to 0 or 1, depending on whether the corresponding radiation mode(s) and diffraction component(s) are required. (See the options IRAD=0 and IDIFF=0 above.)

The MODE array in the radiation solution specifies which modes of the forced motion problem will be solved. To understand the significance of the MODE array in the diffraction solution, it should be noted that, when symmetry planes are defined, the complete diffraction problem is decomposed into symmetric/antisymmetric components in a manner which permits the most efficient solution, and when IDIFF=0, only those components of the diffraction potential required to evaluate the exciting force for the specified modes are evaluated. For example, if ISX=1, IDIFF=0, MODE(1)=1, and the remaining elements of MODE are set equal to zero, then the only component of the diffraction potential which is solved is that part which is antisymmetric in x. If the complete diffraction potential is required, for example to evaluate the drift forces or field data, IDIFF should be set equal to one. (For further discussion of these options see Section 3.3.)

NPER is the number of wave periods to be analyzed. NPER must be an integer. The following alternatives can be used in Versions 6.2-3:

$NPER = 0$: Read inputs and evaluate hydrostatic coefficients only.

$NPER > 0$: Execute the hydrodynamic analysis for NPER wave periods PER

$NPER < 0$: Execute the hydrodynamic analysis for $|NPER|$ wave periods as explained below

If NPER= 0, POTEN and FORCE will run but not execute any hydrodynamic analysis. This option can be used to test for errors in input files, and to evaluate the hydrostatic coefficients in the OUT file. If NPER=0 the array PER must be removed from the Potential Control File.

PER is the array of wave periods T in seconds, or of optional inputs as specified by the parameter IPERIO. Normally the values of PER must be positive. By using the optional parameter IPERIO in the configuration file, it is possible to replace the input array of wave periods by a corresponding array with values of the radian frequencies $\omega = 2\pi/T$, infinite-depth wavenumbers KL, or finite-depth wavenumbers νL. The wavenumbers

are nondimensionalized by the length L=ULEN, and defined relative to the gravitational acceleration g=GRAV. Both ULEN and GRAV are input in the GDF file. The following table gives the definitions of each input and the corresponding value of IPERIO:

IPERIO	Input	Definition
1	Period	T
2	Frequency	$\omega = 2\pi/T$
3	Infinite-depth wavenumber	$KL = \omega^2 L/g$
4	Finite-depth wavenumber	$\nu L \quad (\nu L \tanh \nu H = \omega^2 L/g)$

If the fluid depth is infinite (HBOT ≤ 0), $K = \nu$ and there is no distinction between the inputs for the last two cases. The default case IPERIO=1 is assumed if IPERIO is not specified in the configuration file. Regardless of the form of these inputs, the output data is unchanged, with wave periods in seconds and the nondimensional finite-depth wavenumbers specified in the OUT file.

The limiting values of the added mass coefficients may be evaluated for zero or infinite period by specifying the values PER= 0.0 and PER< 0.0, respectively. In V6.2-3, the limiting values of the body pressure and velocity and the field pressure and velocity due to the radiation solution may be evaluated(see Section 4.13). These special values can be placed arbitrarily within the array of positive wave periods. These special values are always associated with the wave period, irrespective of the value of IPERIO and the corresponding interpretation of the positive elements of the array PER. For example, the effect of the parameter IPERIO=2 and the array PER with the four inputs 0., 1., 2., -1. is identical to the default case IPERIO=1 with the array PER equal to 0., 2π, π, -1. These limiting values can be included in runs where the irregular-frequency removal option is used (IRR$\geq$1) (In V6.1 and lower, a separate run must be made for these limiting values after setting IRR=0.)

If $NPER < 0$ a special convention is followed to assign a total of $|NPER|$ wave periods, frequencies, or wavenumbers with uniform increments, starting with the value equal to PER(1) and using the increment equal to the value shown for PER(2). In this case only two values PER(1),PER(2) should be included in the POT file. This option is convenient when a large number of uniformly spaced inputs are required. The two following examples show equivalent sets of input data in lines 5 and 6 of the POT file:

```
8    (NPER)
1.0  2.0  3.0  4.0  5.0  6.0  7.0  8.0     (PER array)
```

and

```
-8    (NPER)
1.0  1.0                                   (PER(1), increment)
```

This convention is applied in the same manner for all IPERIO values, irrespective of whether PER represents the wave period, frequency, or wavenumber. Special attention is required when zero-period or zero-frequency inputs are required, following the definitions as specified above. For the example described in the preceding paragraph with IPERIO=2,

the inputs NPER=-4 and (PER = -1. 1.) will result in the sequence of wave frequencies equal to (zero, infinity, 1., 2.).

NBETA is the number of incident wave headings to be analyzed in POTEN. (Additional heading angles may be specified subsequently in FORCE, as described in Section 3.3.) NBETA must be an integer. The following alternatives can be used in Version 6.:

$NBETA = 0$: There are no incident wave heading angles. (IDIFF=-1)

$NBETA > 0$: Execute the hydrodynamic analysis for NBETA wave angles BETA

$NBETA < 0$: Execute the hydrodynamic analysis for $|NBETA|$ wave angles as explained below

BETA is the array of wave heading angles in degrees. The wave heading is defined as the angle between the positive $x-$axis of the global coordinate system and the direction in which the waves propagate, as shown in Figure 12.1. The sign of the wave heading is defined by applying the right-hand rule to the body fixed system. In POTEN the wave headings specified in the Potential Control File pertain to the solution of the diffraction problem only. NBETA may be set equal to 0 if IDIFF$= -1$; in this case wave heading angles specified in the Potential Control File are ignored in Alternative Form 1. In Alternative Form 2, described in Section 3.2, if NBETA=0 no data BETA should be included in the POT file.

If $NBETA < 0$ a special convention is followed to assign a total of $|NBETA|$ wave angles, with uniform increments, starting with the value equal to BETA(1) and using the increment equal to the value shown for BETA(2). In this case only two values BETA(1),BETA(2) should be included in the POT file. This option is convenient when a large number of uniformly spaced inputs are required. The two following examples show equivalent sets of input data in lines 7 and 8 of the POT file:

```
8    (NBETA)
0.0 45.0 90.0 135.0 180.0 225.0 270.0 315.0     (BETA array)
```

and

```
-8    (NBETA)
0.0 45.0                                    (BETA(1), increment)
```

3.2 THE POTENTIAL CONTROL FILE (Alternative Form 2)

If IALTPOT=2 is specified in the configuration file, the data in the Potential Control File must be in the format listed below:

```
header
IRR†
HBOT
IRAD IDIFF
NPER
PER(1) PER(2) PER(3)...PER(NPER)
NBETA
BETA(1) BETA(2) BETA(3) ...  BETA(NBETA)
NBODY
GDF(1)
XBODY(1,1) XBODY(2,1) XBODY(3,1) XBODY(4,1)
MODE(1,1) MODE(2,1) MODE(3,1) MODE(4,1) MODE(5,1) MODE(6,1)
NEWMDS(1)
GDF(2)
XBODY(1,2) XBODY(2,2) XBODY(3,2) XBODY(4,2)
MODE(1,2) MODE(2,2) MODE(3,2) MODE(4,2) MODE(5,2) MODE(6,2)
NEWMDS(2) ·
.
.
GDF(NBODY)
XBODY(1,NBODY) XBODY(2,NBODY) XBODY(3,NBODY) XBODY(4,NBODY)
MODE(1,NBODY) MODE(2,NBODY) MODE(3,NBODY) ...  MODE(6,NBODY)
NEWMDS(NBODY)
```

The data shown on each line above is read consecutively by corresponding read statements. Thus it is necessary to preserve the line breaks indicated above, but if a large number of periods (PER) and/or wave heading angles (BETA) are input, these may be placed on an arbitrary number of consecutive lines.

The definitions of the data on lines 1-8 (header, IRR, HBOT, IRAD, IDIFF, NPER, PER, NBETA, BETA) are the same as given in Section 3.1 for the Alternative Form 1 POT File. As in Alternative Form 1, the dagger shown after IRR denotes that this parameter must be included either here, or in the configuration file, but not in both.

Definitions of the other data above are as follows:

NBODY is the number of bodies. Except for the analysis of multiple bodies (Chapter 7), NBODY=1.

GDF(K) is the name of the Geometric Data File of the Kth body.

XBODY(1,K), XBODY(2,K), XBODY(3,K) are the dimensional (X, Y, Z) coordinates of the origin of the **body-fixed coordinate system** of the kth body relative to the **global coordinate system**, input in the units of the length ULEN. The incident-wave

velocity potential is defined relative to the global coordinate system. Consequently, the phases of the exciting forces, motions, hydrodynamic pressure and field velocity induced by the incident wave are understood relative to the incident-wave elevation at $X = Y = 0$.

XBODY(4,K) is the angle in degrees of the $x-$axis of the body coordinate system of the Kth body relative to the $X-$axis of the global system in counterclockwise sense.

NEWMDS(K) is the number of generalized modes for Kth body. Except for the analysis of generalized modes (Chapter 8), NEWMDS=0.

When the Alternative Form 2 POT file is used, the parameters NBODY and NEWMDS must be specified in the POT file, even if they have the default values NBODY=1 and/or NEWMDS=0. The name(s) specified for the GDF file(s) take precedence over the GDF filename, if any, that is specified in the FNAMES.WAM file.

3.3 THE FORCE CONTROL FILE (Alternative form 1)

The Force Control File (FRC) is used to input various parameters to the FORCE subprogram. The name of the FRC file can be any legal filename accepted by the operating system, with a maximum length of 16 ASCII characters, **followed by the extension '.frc'**.

In this Section the first form of the FRC file is described, in which the input of the body inertia matrix is simplified, and it is assumed that the body is freely floating. (For the more general alternative form see Section 3.4.)

The data in the Alternative 1 FRC file is listed below:

```
header
IOPTN(1) IOPTN(2) IOPTN(3) IOPTN(4) IOPTN(5) IOPTN(6) IOPTN(7) IOPTN(8) IOPTN(9)
VCG
XPRDCT(1,1) XPRDCT(1,2) XPRDCT(1,3)
XPRDCT(2,1) XPRDCT(2,2) XPRDCT(2,3)
XPRDCT(3,1) XPRDCT(3,2) XPRDCT(3,3)
NBETAH
BETAH(1) BETAH(2) ...  BETAH(NBETAH)
NFIELD
XFIELD(1,1) XFIELD(2,1) XFIELD(3,1)
XFIELD(1,2) XFIELD(2,2) XFIELD(3,2)
XFIELD(1,3) XFIELD(2,3) XFIELD(3,3)
.
.
XFIELD(1,NFIELD) XFIELD(2,NFIELD) XFIELD(3,NFIELD)
```

The index IOPTN(9) was added in Version 5, and must be included in the FRC file. Users of earlier versions of WAMIT must take special care to add this extra index to old files before using Version 6 and higher. Since FORTRAN free-format read statements are used to read this file, if IOPTN(9) is missing its value will be read from the line containing VCG, VCG will be read as XPRDCT(1,1), etc. If there are no values of BETAH or XFIELD specified in the FRC file, the final result will be that there are not sufficient lines of data and an error message will be displayed that there is erroneous data in the FRC file. However if values of BETAH and/or XFIELD are specified it is possible for all of the data to be read without detecting an error, with unpredictable results. This possibility can be prevented by adding a non-numeric comment at the end of the line containing the IOPTN array, as illustrated in Test Run 01.

The definition of each variable in the Force Control File is as follows:

'header' denotes a one-line ASCII header dimensioned CHARACTER∗72. This line is available for the user to insert a brief description of the file.

IOPTN is an array of option indices. These indicate which hydrodynamic parameters are to be evaluated and output from the program. The available options, descriptions and numeric file names are as follows:

Option	Description	Filename
1	Added-mass and damping coefficients	*frc*.1
2	Exciting forces from Haskind relations	*frc*.2
3	Exciting forces from diffraction potential	*frc*.3
4	Motions of body (response amplitude operator)	*frc*.4
5p	Hydrodynamic pressure on body surface	*frc*.5p
5v	Fluid velocity vector on body surface	*frc*.(5vx,5vy,5vz)
6	Pressure/ free-surface elevation at field points	*frc*.6
7	Fluid velocity vector at field points	*frc* .(7x,7y,7z)
8	Mean drift force and moment from momentum	*frc*.8
9	Mean drift force and moment from pressure	*frc*.9

The evaluation and output of the above parameters is accordance with the following choice of the corresponding index:

IOPTN(I) = 0: do not output parameters associated with option I.

IOPTN(I) = 1: do output parameters associated with option I.

Options 4, 5, 6, 7, 8 and 9 may have additional values as listed below:

IOPTN(4)

IOPTN(4) = 0: do not output response amplitude operator, RAO

IOPTN(4) = $\pm$1: do output RAO by Haskind exciting force

IOPTN(4) = $\pm$2: do output RAO by diffraction exciting force

IOPTN(4) = $-$3: output field data only for specified radiation modes

The use of IOPTN(4)=-1, -2, or -3 is explained in Sections 3.5.

IOPTN(5)

IOPTN(5) = 0: do not output pressure and fluid velocity on the body

IOPTN(5) = 1: do output pressure on the body

IOPTN(5) = 2: do output fluid velocity on the body

IOPTN(5) = 3: do output both pressure and fluid velocity on the body

IOPTN(6)

IOPTN(6) = 0: do not output pressure in the fluid and/or free-surface elevation

IOPTN(6) = 1: do output pressure in the fluid and/or free-surface elevation by the potential formulation

IOPTN(6) = 2: do output pressure in the fluid and/or free-surface elevation by the source formulation

IOPTN(7)

IOPTN(7) = 0: do not output fluid velocity in the fluid

IOPTN(7) = 1: do output fluid velocity in the fluid by the potential formulation

IOPTN(7) = 2: do output fluid velocity in the fluid by the source formulation

IOPTN(8)

IOPTN(8) = 0: do not output mean force and moment from momentum integration

IOPTN(8) = 1: do output mean force and moment only for unidirectional waves

IOPTN(8) = 2: do output mean force and moment for all combinations of wave headings

IOPTN(9)

IOPTN(9) = 0: do not output mean force and moment from pressure integration

IOPTN(9) = 1: do output mean force and moment only for unidirectional waves

IOPTN(9) = 2: do output mean force and moment for all combinations of wave headings

The options IOPTN(5)=2 and 3, IOPTN(6)=2, and IOPTN(7)=2 apply only for the low-order method (ILOWHI=0), and require the source formulation (ISOR=1). IOPTN(9)=1 and 2 require the source formulation (ISOR=1) if and only if the low-order method (ILOWHI=0) is used. If the higher-order method (ILOWHI=1) is used, IOPTN(9) can take any of the values 0,1,2 with ISOR=0.

The settings of the indices IOPTN(I) must be consistent with themselves and with the indices IRAD, IDIFF, and NBETA set in the Potential Control File. Error messages are generated if inconsistent indices are input. Otherwise, the indices IRAD, IDIFF and IOPTN(I), I=1,...,9 can be selected in any way the particular application may suggest. Three principal applications are as follows:

Forced motions in calm water (the radiation problem). In this case the modes of possible motion are specified by the MODE(I) indices in the Potential Control File. The diffraction index IDIFF should be set equal to -1. The corresponding linear force coefficients are obtained with Option 1. Field pressures, velocities, free-surface elevations and drift forces follow from the corresponding options 5-9.

Note that in the default case the pressures, velocities, and drift forces are not separated according to each mode, and the consequence of setting more than one mode to be nonzero is to superpose all such modes with unit amplitude. For example if IRAD=1 and IDIFF=-1, the default outputs from options 5-9 correspond to unit amplitudes of motion in all six degrees of freedom, with the same phase. Generally this is a nonphysical problem and care should be taken to avoid it. The simplest procedure to evaluate these outputs for each mode is to set MODE(I)=1 for only one mode at a time. However this requires a separate run of WAMIT for each mode. In V6.2-3 it is possible to output separate results for options 5-7 for each mode of forced motion in a single run, using the configuration parameters INUMOPT5, INUMOPT6, INUMOPT7, as explained in Section 4.12.)

Diffraction of incident waves by a stationary structure (the diffraction problem).

In this case the radiation index IRAD should be set equal to -1. To solve the complete diffraction problem set IDIFF=1, with corresponding outputs from the options 3,5,6,7,8,9 in FORCE. (If IOPTN(4)=0 and IDIFF=1 it is assumed that the body is stationary, irrespective of IRAD. Thus, after a complete WAMIT run where the P2F file is output from POTEN and saved, it is possible to execute another run using only FORCE with the body motions both free and fixed, without re-running POTEN.)

In V6.2-3 it is possible to output separate results for options 5-7 for the diffraction problem, using the configuration parameters INUMOPT5, INUMOPT6, INUMOPT7, as explained in Section 4.12.

Body motions in incident waves. In this case the index IRAD and IDIFF are set equal to 0 (body free only in specified modes) or 1 (body free in all modes). Body motions are obtained from the solution of the equations of motion using Option 4. The resulting field data and drift forces are evaluated for this particular combination of the radiation and diffraction solutions.

VCG Dimensional z-coordinate of the center of gravity of the body relative to the origin of the body-system, input in the same units as the length ULEN.

XPRDCT is the 3×3 matrix of the body radii of gyration about the body-fixed axes, where I,J=1,2,3 correspond to (x, y, z) respectively, input in the same units as the length ULEN. More precisely, the elements of the body inertia matrix m_{ij} are evaluated for $i, j = 4, 5, 6$ according to the algorithm $m_{ij} = m \times \mathrm{XPRDCT(i-3, j-3)} \times |\mathrm{XPRDCT}(i-3, j-3)|$. Here the body mass m is evaluated from the displaced mass of fluid, and the absolute value is used in the last factor so that negative products of inertia can be specified. The remaining elements of m_{ij} are evaluated assuming the body is freely floating in equilibrium, based on the calculated values of the displaced volume and center of buoyancy and on the specified value of VCG. In practical cases the matrix XPRDCT is symmetric. Zeroes may be specified if the body motions are not evaluated.

NBETAH is the number of Haskind wave headings, defined below. NBETAH must be an integer, greater than or equal to zero.

BETAH is an array of length NBETAH defined as the Haskind wave headings in degrees. The Haskind wave headings may be introduced in the Force Control File as an option, to enable evaluations to be made of the Haskind exciting forces (Option 2) and body motions in waves (Option 4) at heading angles not included in the Potential Control File. This option is feasible since the evaluation of Haskind exciting forces requires only the radiation potentials already determined by POTEN (see Section 4.3). This is a useful feature since a relatively small number of wave headings for the diffraction problem may be specified in the Potential Control File and the time required to solve many diffraction problems in POTEN greatly exceeds the time required to evaluate the Haskind exciting forces in FORCE. Since the number of Haskind wave headings will affect the subsequent READ statements for data in the Force Control File, it is important to ensure that this number corresponds with the prescribed integer NBETAH. In particular, if NBETAH= 0 no values of BETAH should be included and NFIELD should appear on the next line of the Force Control File. If

NBETAH> 0 is specified, the settings of the IOPTN switches are automatically set equal to 0 for options 3,5,6,7,8.

NFIELD is the number of points in the fluid domain(free surface) where the hydrodynamic pressure(wave elevation) and/or velocity are to be evaluated. NFIELD must be an integer, greater than or equal to zero.

XFIELD is a three-dimensional array with dimensions $3 \times$ NFIELD, defining the dimensional global coordinates of field points where the pressure/wave elevation and/or fluid velocity vector will be evaluated. Here I=1,2,3 correspond to the (X, Y, Z) coordinates. If $Z = 0$ the resulting output should be interpreted as the nondimensional wave elevation, otherwise as the nondimensional pressure. If NFIELD= 0 no input should be made for the array XFIELD.

Additional data is required to specify the field point array XFIELD when field points are placed inside internal tanks. This departure from the usual default procedure is described in Sections 3.7 and 10.7.

3.4 THE FORCE CONTROL FILE (Alternative form 2)

In this Section the second alternative form of the FRC file is described, where it is possible to specify separately three independent external force matrices including the mass matrix of the body, an external damping matrix, and an external stiffness matrix. This permits the analysis of bodies which are not freely floating in waves, with arbitrary linear external forces and moments, and also permits the specification of the complete body mass matrix instead of the simpler radii of gyration (cf. Section 3.3).

The format of the Alternative 2 FRC file is shown below:

```
header
IOPTN(1) IOPTN(2) IOPTN(3) IOPTN(4) IOPTN(5) IOPTN(6) IOPTN(7) IOPTN(8) IOPTN(9)
RHO
XCG YCG ZCG
IMASS
EXMASS(1,1) EXMASS(1,2) ...  EXMASS(1,6+NEWMDS)
EXMASS(2,1) EXMASS(2,2) ...  EXMASS(2,6+NEWMDS)
.
.
EXMASS(6+NEWMDS,1) EXMASS(6+NEWMDS,2) ...  EXMASS(6,6+NEWMDS)
IDAMP
EXDAMP(1,1) EXDAMP(1,2) ...  EXDAMP(1,6+NEWMDS)
EXDAMP(2,1) EXDAMP(2,2) ...  EXDAMP(2,6+NEWMDS)
.
.
EXDAMP(6+NEWMDS,1) EXDAMP(6+NEWMDS,2) ...  EXDAMP(6,6+NEWMDS)
ISTIF
EXSTIF(1,1) EXSTIF(1,2) ...  EXSTIF(1,6+NEWMDS)
EXSTIF(2,1) EXSTIF(2,2) ...  EXSTIF(2,6+NEWMDS)
.
.
EXSTIF(6+NEWMDS,1) EXSTIF(6+NEWMDS,2) ...  EXSTIF(6+NEWMDS,6+NEWMDS)
NBETAH
BETAH(1) BETAH(2) ...  BETAH(NBETAH)
NFIELD
XFIELD(1,1) XFIELD(2,1) XFIELD(3,1)
XFIELD(1,2) XFIELD(2,2) XFIELD(3,2)
XFIELD(1,3) XFIELD(2,3) XFIELD(3,3)
.
.
XFIELD(1,NFIELD) XFIELD(2,NFIELD) XFIELD(3,NFIELD)
```

The recommended procedure to designate Alternative Form 2 is to assign the parameter IALTFRC=2 in the configuration file, as described in Section 3.7. In earlier versions of

WAMIT this parameter was assigned differently, by inserting the integer 2 on a separate line after the header line of the FRC file, as shown below:

```
header
2
IOPTN(1) IOPTN(2) ...  IOPTN(9)
RHO
.
.
```

This variant of the FRC file format is still supported in Version 6, to enable old files to be read correctly. However it is deprecated, and may not be supported for later versions of WAMIT.

The header, IOPTN array, and all lines beginning with the variable NBETAH, are identical to the data in the Alternative form 1 FRC file, as defined in Section 3.3. In the remainder of this Section the data which differ in Form 2 are described.

RHO Dimensional density of the fluid, in the same units as used for the external force matrices and for GRAV.

XCG YCG ZCG Dimensional coordinates of the body center of gravity in terms of the body coordinate system and in the same units as ULEN.

IMASS This index is either 0 or 1, to signify if the external mass matrix EXMASS is read. If the value of the index is zero, the matrix EXMASS is not included in the FRC file and the program assumes that all values in this matrix are zero. If the value of the index is one, the matrix EXMASS is included in the FRC file.

EXMASS is the (6+NEWMDS) $\times$ (6+NEWMDS) dimensional inertia matrix of the body about the body-fixed axes. (For a conventional rigid body this is a 6×6 dimensional matrix as defined in Reference [3], page 149, equation 141.) Each element in this matrix is added to the corresponding added mass of the body, in setting up the equations of body motions.

IDAMP This index is either 0 or 1, to signify if the external damping matrix EXDAMP is read. If the value of the index is zero, the matrix EXDAMP is not included in the FRC file and the program assumes that all values in this matrix are zero. If the value of the index is one, the matrix EXDAMP is included in the FRC file.

EXDAMP is the (6+NEWMDS) $\times$ (6+NEWMDS) dimensional damping matrix of an arbitrary external force or moment acting on the body, e.g. from a mooring cable subject to viscous damping. The value of each element in this matrix is added to the corresponding linear wave damping coefficient of the body, in setting up the equations of body motions.

ISTIF This index is either 0 or 1, to signify if the external mass matrix EXSTIF is read. If the value of the index is zero, the matrix EXSTIF is not included in the FRC file and the program assumes that all values in this matrix are zero. If the value of the index is one, the matrix EXSTIF is included in the FRC file.

EXSTIF is the (6+NEWMDS) × (6+NEWMDS) dimensional stiffness matrix of an arbitrary external force or moment acting on the body, e.g. from an elastic mooring cable. In setting up the equations of body motions, the value of each element in this matrix is added to the corresponding restoring coefficient of the body, including both hydrostatic pressure and the gravitational moment due to the body's mass, as defined in Reference [3], page 293, equation 145. (The vertical inertia force due to heave, EXMASS(3,3), is assumed equal to the body mass and is used to derive the gravitational restoring moment of the body. In any situation where this assumption is not satisfied, due to the presence of an external vertical inertia force, the gravitational restoring moment should be corrected for this difference via the stiffness matrix EXSTIF.)

The units of EXMASS, EXDAMP, EXSTIF must correspond to those used to specify the fluid density RHO and the length ULEN, with time measured in seconds. These matrices must be defined with respect to the body-fixed coordinate system.

It is also possible to use Alternative Form 2 with the external force matrices in separate files. In this case the index IMASS, IDAMP, and/or ISTIF is set equal to 2, and followed by the corresponding file name:

```
header
IOPTN(1) IOPTN(2) IOPTN(3) IOPTN(4) IOPTN(5) IOPTN(6) IOPTN(7) IOPTN(8) IOPTN(9)
RHO
XCG YCG ZCG
2
MASS (file name containing inertia matrix)
2
DAMP (file name containing damping matrix)
2
STIF (file name containing stiffness matrix)
NBETAH
BETAH(1) BETAH(2) ...  BETAH(NBETAH)
NFIELD
XFIELD(1,1) XFIELD(2,1) XFIELD(3,1)
XFIELD(1,2) XFIELD(2,2) XFIELD(3,2)
XFIELD(1,3) XFIELD(2,3) XFIELD(3,3)
.
.
XFIELD(1,NFIELD) XFIELD(2,NFIELD) XFIELD(3,NFIELD)
```

MASS is a file name which contains external inertial forces to the bodies and interactive inertial forces between bodies. For example, the data in MASS for a body are listed below:

```
header
EXMASS(1,1) EXMASS(1,2) ...  EXMASS(1,6+NEWMDS)
EXMASS(2,1) EXMASS(2,2) ...  EXMASS(2,6+NEWMDS)
.
```

```
.
EXMASS(6+NEWMDS,1) EXMASS(6+NEWMDS,2) ...  EXMASS(6+NEWMDS,6+NEWMDS)
```
For N interacting bodies, the number of data in MASS are NDFR × NDFR, where NDFR=$\sum_{n=1}^{N}(6 + \text{NEWMDS}(n))$ is the total number of rigid body modes and generalized modes. Similarly the external damping and stiffness can be specified to the files DAMP and STIF.

In summary, there are three legal values for IMASS, IDAMP, ISTIFF:

I=0: no matrix or file name is input, matrix is assumed equal to zero

I=1: matrix follows on subsequent lines

I=2: file name follows on next line

The recommended procedure to designate Alternative Form 2 is to assign the parameter IALTFRC=2 in the configuration file, as described in Section 3.7, and use the formats above. In earlier versions of WAMIT the parameter IALTFRC was assigned differently, by inserting the integer 2 on a separate line after the header line of the FRC file, as shown below:

```
header
2
IOPTN(1) IOPTN(2) ...  IOPTN(9)
RHO
.
.
```

This variant of the FRC file format is still supported in Version 6, to enable old files to be read correctly. However it is deprecated, and may not be supported for later versions of WAMIT. If a file with this variant is input, Version 6 issues a warning message.

3.5 DEFINITION OF FIXED OR FREE MODES

It is possible to specify that a sub-set of the modes of body motion analyzed in POTEN can be fixed in FORCE. As a simple example, consider a single body with six degrees of rigid-body motions, all of which have been analyzed in POTEN (either by setting IRAD=1 or by setting IRAD=0 and setting all six elements of MODE=1). Normally, in the FORCE analysis (IOPTN(4)) the body motions in all six degrees of freedom are computed. Now suppose that the body is restrained in the vertical modes (heave, roll, pitch) as would be the case for the first-order motions of a tension-leg platform. This condition can be analyzed in FORCE by modifying the Force Control File in the following manner:

(1) assign a negative value to IOPTN(4) (-1 to use the Haskind exciting force or -2 to use the diffraction exciting force).

(2) insert two new lines of data after IOPTN (before VCG or RHO):

NDFR
MODE(1),MODE(2),MODE(3), ... MODE(NDFR)

Here NDFR is the total number of possible radiation modes and MODE is an array with the value of each element 0 if the mode is fixed or 1 if the mode is free. For the example described above, NDFR=6 and MODE = (1,1,0,0,0,1). Thus surge, sway, and yaw are free while heave, roll, and pitch are fixed.

When this option is employed the RAO's output for the free modes are defined in the conventional manner, as the amplitudes of body motions in the corresponding degrees of freedom (cf. Section 4.4). For the fixed modes the RAO's are replaced by the loads acting on the body in the corresponding directions. In this case the corresponding modal index in the output file is shown with a negative value, to signify the change. For the example described above, the output RAO for heave is equal to the vertical load acting on the body (equal and opposite to the load on the restraining structure), and preceded by the index -3. The TLP Test Runs (06, 07, 14) described in the Appendix illustrate this application.

For a single body with no generalized modes NDFR=6 in all cases. For the analysis of multiple bodies (cf. Chapter 7), with no generalized modes, NDFR=6*NBODY. If generalized modes are analyzed (cf. Chapter 8) NDFR is the total number of modes for all bodies, including both rigid-body modes and generalized modes. Thus, in the most general case, NDFR=6*NBODY+$\sum_{n=1}^{\text{NBODY}}$ NEWMDS(n).

It is also possible to perform the analysis in FORCE for a problem where no incident waves exist, after running POTEN with incident waves present and with the diffraction solution obtained. To suppress the incident waves in FORCE, IOPTN(4)=-3 is assigned in the FRC file. In that case the body pressure/velocity, field point pressure/velocity, and mean drift forces are evaluated assuming that the body is oscillating in the specified free mode(s), without incident waves.

3.6 FILENAMES LIST 'FNAMES.WAM'

An optional input file may be used to specify the filenames of the primary input files CFG, POT, FRC, and GDF. Use of this optional file is recommended, particularly to facilitate batch processing. The optional file must be named FNAMES.WAM. (The name 'fnames.wam' must be used for this file if the system is case sensitive. Source-code users can redefine this name if it is desired to do so.) The optional file is simply a list of the other input file names, including their respective extensions '.cfg', '.pot', '.frc', and '.gdf'. If this file does not exist, or if it is incomplete, the user is prompted to supply the missing filenames interactively. In Version 6 the number and order of these file names is arbitrary, but they must include the specified extensions. If the alternative (IALTPOT=2) format is used, as explained in Section 3.2, the GDF filename(s) are specified in the POT file and it is not necessary to specify any GDF filename in the FNAMES.WAM file. (In this case a GDF filename that exists in the FNAMES.WAM file is ignored.) In Version 6.2-3 the filename of the configuration file may be input in the FNAMES.WAM file, with the extension '.cfg', as explained below.

3.7 CONFIGURATION FILE 'CONFIG.WAM' OR '*.CFG'

The configuartion file may be used to specify various parameters and options in WAMIT. The configuration file can be assigned the generic name 'config.wam', as in previous versions of WAMIT. Alternatively, the filename of this file can be specified by the user, with the extension '.cfg'. The latter option, which is illustrated in the Test Runs in the Appendix, permits configuration files to be associated with specific names of runs or other input files. If a configuration file is included in the FNAMES.WAM list with the extension '.cfg' this file is used. If no configuration file is included in the FNAMES.WAM list, the generic name CONFIG.WAM is assigned.

The complete list of inputs which may be specified in the configuration file are as follows (in alphatetic order):

Parameter	Alternate File	Default value
IALTFRC		1
IALTFRCN		1,1,1,...
IALTPOT		1
ICTRSURF		0
IDIAG†	POT	
IFIELD_ARRAYS		0
IFORCE		1
IGENMDS		0
ILOWGDF		0
ILOWHI		0
ILOG†	POT	
INUMOPT5		0
INUMOPT6		0
INUMOPT7		0
IPERIO		1
IPLTDAT		0
IPNLBPT		0
IPOTEN		1
IQUAD†	POT	0
IQUADI†	SPL	4
IQUADO†	SPL	3
IRR†	POT	
ISCATT		0
ISOLVE		0
ISOR†	POT	
ITANKFPT		0
KSPLIN	SPL	3
MAXITT		35
MAXMIT		8
MAXSCR		2

MODLST		0
MONITR		0
NEWMDS(n)†	POT_2	0
NOOUT		1
NPTANK(n)		
NUMHDR		0
NUMNAM		0
PANEL_SIZE		-1.0
RHOTANK		
SCRATCH_PATH		current directory
USERID_PATH		current directory
XBODY(n)†	POT or GGDF	

Parameters marked † must be input once and only once, either in the configuration file, or in the other files specified following the symbol †. All other parameters in the list above are optional, with the specified default value assigned when the parameter is not included explicitly.

The parameters marked † are defined in the Sections pertaining to the respective input files (Sections 3.1, 6.9, 7.1 respectively for the POT, SPL, GGDF files). POT_2 denotes the Alternative Form 2 POT file format as explained in Section 3.2.

NEWMDS(n), NPTANK(n) and XBODY(n) denote the NEWMDS, NPTANK and XBODY arrays for body number n. When NBODY=1 these can be written more simply as NEWMDS, NPTANK and XBODY. If NBODY>1 the number of each body should be indicated in parenthesis, as illustrated in the example in Section 7.6. If IALTPOT= 1 nonzero values of NEWMDS must be input for each body in the configuration file. If IALTPOT= 2 the parameters NEWMDS and XBODY must be input for each body in the POT file as shown in Section 3.2.

Explanations of the other parameters which may be specified in the configuration file are as follows:

IALTFRC is an integer specifying the alternative forms of the FRC file as explained in Sections 3.3, 3.4, and 7.3-5.

IALTFRC= 1: use the Alternative Form 1 FRC format shown in Section 3.3,

IALTFRC= 2: use the Alternative Form 2 FRC format shown in Section 3.4,

IALTFRC= 3: use the Alternative Form 3 FRC format shown in Section 7.5.

See Sections 3.4 and 7.5 concerning the specification of Alternative Forms 2 and 3 FRC files by special modification of the data on line 2, as in earlier versions of WAMIT. This option is deprecated and may not be supported beyond Version 6.

The default value is IALTFRC= 1.

IALTFRCN is an integer array of length NBODY. The value of each array element, 1 or 2, designates the corresponding Alternative Form of the separate FRC files for each body, as explained in Section 7.5. This array is only used if IALTFRC=3 and NBODY>1.

IALTPOT is an integer specifying the alternative forms of the POT file and GGDF file.

IALTPOT= 1: use the Alternative Form 1 POT format shown in Section 3.1, and define the GDF filename in FNAMES.WAM. For multiple bodies the separate GDF filename for each body is assigned in the Global GDF as explained in Section 7.1.

IALTPOT= 2: use the POT format shown in Section 3.2, with the GDF filename(s) assigned in the POT file.

IALTPOT= −1: use the POT format shown in Section 3.1. This is a special alternative intended to permit using V6 for multiple bodies without changing the Global GDF format required in earlier versions of WAMIT. Further details are given in Section 7.1. Use of this option is deprecated, and may not be supported beyond Version 6.

The default value is IALTPOT= 1.

ICTRSURF is an integer specifying the alternative forms to evaluate the drift forces based on integration of the momentum flux over a user-defined control surface (see Chapter 14).

ICTRSURF= 0: Do not evaluate the control-surface drift forces.

ICTRSURF= 1: Evaluate the control-surface drift forces using Alternative 1 integration of the momentum flux.

ICTRSURF= 2: evaluate the control-surface drift forces using Alternative 2 integration of the momentum flux.

The default value is ICTRSURF= 0.

IFIELD_ARRAYS is an integer parameter specifying if uniform arrays of field points are assigned in the FRC control file, using a compressed format as explained in Section 3.10.

IFIELD_ARRAYS= 0: Field point data is assigned only as shown in the conventional FRC files, as explained in Sections 3.3 and 3.4.

IFIELD_ARRAYS= 1: Additional field point data is assigned in the conventional FRC files, as explained in Section 3.10.

The default value is IFIELD_ARRAYS= 0.

IFORCE is an integer parameter specifying if the FORCE subprogram is executed during the WAMIT run.

IFORCE= 0: Do not execute FORCE.

IFORCE= 1: Do execute FORCE.

The default value is IFORCE= 1.

IGENMDS is an integer parameter specifying the option to input the geomeric data associated with the mode shapes of generalized modes, when NEWMDS > 0. The details on the use of the generalized mode option are described in Chapter 8.

IGENMDS= 0: use a seperate program DEFMOD to input the geometric data for generalized modes. This option can be used only with the low order method (ILOWHI=0).

In this case the user prepares a subroutine in DEFINE to describe the generalized mode shapes.

IGENMDS= 1: use a DLL file containing the subroutine NEWMODES. The user modifies NEWMODES to specify the mode shape. This option can be used with both the low- and higher-order options.

The default value is IGENMDS= 0.

ILOWGDF is an integer parameter specifying the option to generate a low-order GDF output file based on the input geometry (see Section 10.6). If ILOWHI=0, the original input panels are used. If ILOWHI=1 the low-order output panels are generated from the panels of the higher-order geometry as defined in Section 6.1. If ILOWGDF> 1, the panels are subdivided into ILOWGDF×ILOWGDF sub-panels.

ILOWGDF= 0: Do not generate the output file *gdf*_LOW.GDF.

ILOWGDF≥ 1: Generate the output file *gdf*_LOW.GDF.

The default value is ILOWGDF= 0.

ILOWHI is an integer parameter specifying the use of the low-order or higher-order panel method.

ILOWHI= 0: Low-order panel method (Chapter 5)

ILOWHI= 1: Higher-order panel method (Chapter 6)

INUMOPT5 is an integer parameter specifying the option to evaluate the separate radiation and diffraction components of the body pressure and velocity. (see Section 4.12).

INUMOPT5= 0: output the total body pressure and velocity, as in Version 6.0.

INUMOPT5= 1: output the total body pressure and velocity in the formatted output file (.out). Output the separate components in the numeric output files (.5p, .5vx, .5vy, .5vz).

INUMOPT6 is an integer parameter specifying the option to evaluate the separate radiation and diffraction components of the pressure at field points.(see Section 4.12).

INUMOPT6= 0: output the total field pressure, as in Version 6.1.

INUMOPT6= 1: output the total field pressure in the formatted output file (.out). Output the separate components in the numeric output file (.6).

INUMOPT7 is an integer parameter specifying the option to evaluate the separate radiation and diffraction components of the fluid velocity at field points. (see Section 4.12).

INUMOPT7= 0: output the total field velocity, as in Version 6.1.

INUMOPT7= 1: output the total field velocity in the formatted output file (.out). Output the separate components in the numeric output files (.7x, .7y, .7z).

IPERIO is an integer parameter specifying the input data for PER in the POT file.

IPERIO= 1: Input periods in seconds for PER

IPERIO= 2: Input radian frequencies for PER

IPERIO= 3: Input infinite-depth wavenumbers for PER

IPERIO= 4: Input finite-depth wavenumbers for PER

The default value is IPERIO= 1.

IPLTDAT is an integer parameter specifying the option to generate the output files *gdf*_PAN.DAT and *gdf*_PAT.DAT, used for plotting the panel and patch discretizations (see Section 10.6).

IPLTDAT= 0: Do not generate the output files *gdf*_PAT.DAT and *gdf*_PAN.DAT

IPLTDAT$\geq$ 1: Generate the output files *gdf*_PAT.DAT and *gdf*_PAN.DAT for plotting the panel and patch discretizations with IPLTDAT$\times$IPLTDAT subdivisions.

The default value is IPLTDAT= 0.

IPNLBPT is an integer parameter specifying the option to evaluate the body pressure at specified points (x, y, z) listed in the input file *gdf*.BPI. (see Section 4.11).

IPNLBPT= 0: output the body pressure at the panel centroids, or on a uniform parametric mesh in the higher-order method, as in Version 6.0.

IPNLBPT$\geq$ 1: output the body pressure at the points listed in the input file *gdf*.BPI. These points are defined with reference to the body-fixed dimensional Cartesian coordinates for each body.

IPNLBPT≤ -1: output the body pressure at the points listed in the input file *gdf*.BPI. These points are defined with reference to the global dimensional Cartesian coordinate system.

The default value is IPNLBPT= 0. When IPNLBPT$\neq$0 and ILOWHI=0, the absolute value of IPNLBPT specifies the number of panels nearest to each point specified in gdf.BPI. See Section 4.11 for further details.

IPOTEN is an integer parameter specifying if the POTEN subprogram is executed during the WAMIT run.

IPOTEN= 0: Do not execute POTEN.

IPOTEN= 1: Do execute POTEN.

The default value is IPOTEN= 1.

ISCATT is an integer parameter specifying whether the diffraction or the scattering problem is solved to obtain the diffraction potential. The diffraction potential may be solved by the equation (12.12) which we define as the diffraction problem. Alternatively in the scattering problem, the scattered potential is solved by (12.13) and the diffraction potential is obtained from equation (12.8): This option is only available in the low-order method (ILOWHI=0). In the higher-order method (ILOWHI=1) an error message is generated if ISCATT=1.

ISCATT= 0: Solve the diffraction problem

ISCATT= 1: Solve the scattering problem

The default value is ISCATT= 0.

ISOLVE is an integer parameter specifying the method of solution for the linear systems in POTEN.

ISOLVE= 0: Use the iterative solver

ISOLVE= 1: Use the direct solver

ISOLVE= N For $N \geq 2$: Use the block iterative solver with N blocks

The default value is ISOLVE= 0.

The direct solver is based on a partial-pivoting LU decomposition algorithm known as Gauss elimination. In general, the iterative solver is faster than the direct solver as NEQN, the size of the linear system, increases. However, there are cases where the direct solver may have advantage. For applications where NEQN is relatively small the direct solver may be somewhat faster than the iterative solver. This is particularly the case where the number of right-hand sides is large, as in cases where the diffraction solution is required for a large number of wave headings, or where a large number of radiation solutions are required for the analysis of multiple bodies or generalized modes. The direct solver is also useful in applications where the iterative solver fails to converge (See Section 12.6). If the direct solver is used it is necessary for MAXSCR to be at least as large as NEQN.

The block iterative solver is based on the combination of the local LU decomposition for each diagonal block and the algorithm of the iterative solver. This option may be used in cases where the the iterative solver does not converge sufficiently fast, but where the available RAM does not permit assigning MAXSCR=NEQN, or where NEQN is so large that the direct solver is too time-consuming (the cpu required for the LU decomposition is proportional to NEQN^3). In general, the number of iterations is reduced as the block size increases.

ISOLVE=0 is recommended when ILOWHI=0. In some cases the iterative method does not converge after MAXITT iterations (see below). In this case, the user can i) set ISOLVE=1 to invoke the direct solver ii) set ISOLVE=N, (N$\geq$2) to invoke the block iterative method, iii) keep ISOLVE=0 and increase MAXITT or iv) a combination of ii) and iii).

ISOLVE=1 is recommended when ILOWHI=1 because the iterative method is not effective in general. For a large number of unknowns, the block-iterative method can be efficient. But the number of blocks should be chosen carefully so that iteration converges sufficiently fast.

In the standard test runs described in Appendicies A and B, the iterative or block-iterative solvers converge within the default number of iterations MAXITT=35, for all cases where these solvers are used, except for TEST02.

ITANKFPT is an integer parameter specifying the option to input field point (fpt) coordinates in the .frc control file, either in the conventional format (default) or in the format required when some or all of the field points are located within the internal tanks.

ITANKFPT= 0: All field points are in the exterior domain. Field points are input as specified in Sections 3.3 and 3.4, with the coordinates of each field point on one line. The line containing the input data for the n'th field point is

`XFIELD(1,n),XFIELD(1,n),XFIELD(1,n)`

ITANKFPT= 1: Field points are listed in the .frc control file with the tank number preceding the coordinates. Thus, in place of the line shown above, the correponding data are input as follows:

```
ITANK(n) XFIELD(1,n) XFIELD(1,n) XFIELD(1,n)
```

ITANK(n) is an integer which specifies the tank number containing the n'th field point. If ITANK(n)=0, the field point is in the exterior domain. The numbering of the tanks corresponds to the order of the NPTANK indices, as explained below.

The default value is ITANKFPT= 0.

MAXITT is the integer parameter used to specify the maximum number of iterations in the iterative solver of POTEN. (Further information is given above regarding the parameter ISOLVE.)

The default value MAXITT=35 is recommended for general use.

MAXMIT is the maximum number of iterations in the adaptive integration used to evaluate the momentum integral for the drift force and moment (Option 8). (The maximum number of integration ordinates in the azimuthal direction is equal to 2**MAXMIT.)

The default value MAXMIT=8 is recommended for general use. A warning message is issued if the integration does not coverge to 4 significant digits within the specified number of iterations.

Often the warning message is issued because of the smaller component of drift force or moment. This force or moment can be either converged for most practical purposes or too small to be important in practice. It is recommended to check the practical importance of this quantity. Further check on the convergence of the result can be made by increasing MAXMIT gradually. Since the computational time increases exponentially, it is not recommended to use significantly large MAXMIT than the default value.

MAXSCR is the integer parameter used to specify the available RAM for scratch storage in POTEN. The default value MAXSCR=2 is specified in V6PC to minimize the required RAM. Users should assign MAXSCR in to have the largest practical value as described in Section 10.4.

MODLST is the integer parameter used to control the order in which the added-mass and damping coefficients, exciting forces, and RAO's for different modes of motion are written to the output files. (In V5.3 the added-mass and damping coefficients were not included in this option.)

MODLST= 0: Outputs are in ascending order of the modal indices.

MODLST= 1: Outputs are in the order evaluated for each of the corresponding left-hand-sides.

These two alternatives differ only if NLHS is greater than one. The default value is MODLST= 0.

MONITR is the integer parameter used to control the display of output to the monitor during the execution of FORCE.

MONITR= 0: Outputs to the monitor are abbreviated, consisting of the header text and displays of each wave period as it is executed. This option is convenient in long runs of FORCE with extensive lines of output, to permit monitoring the progress of the execution.

MONITR= 1: Outputs of all data evaluated by FORCE are displayed on the monitor during execution, in the same format as in the OUT file.

The default value is MONITR= 0.

NOOUT is an integer array with length 9, used to control the output to the OUT file. Each element of the array corresponds to one of the 9 options in FRC. If the element equals 0, the corresponding output is omitted from the OUT file.

NOOUT(I)= 0 Omit the output corresponding to IOPTN(I) in the OUT file

NOOUT(I)= 1 Include the output corresponding to IOPTN(I) in the OUT file

If NOOUT is included in the configuration file all 9 integers must be specified. An example is shown on the next line, which specifies that all outputs are included in the OUT file except the pressures and/or fluid velocities on the body panels.

NOOUT= 1 1 1 1 0 1 1 1 1

This option can be useful to avoid very long OUT files, since the data for option 5 is generally much more extensive than for the other options. The default value NOOUT(I)= 1 for $I = 1, ..., 9$ is assigned if NOOUT is not included in the configuration file. (The data for each specified option is always included in the corresponding numeric output file, regardless of the array NOOUT.)

NPTANK is an integer array used to specify the panel or patch indices of internal tanks. The data in this array are in pairs, denoting the first and last index for each tank. An even number of indices must be included on each line. More than one line can be used for multiple tanks, and/or multiple tanks can be defined on the same line. If NBODY>1, the body number(s) for each body containing tanks must be appended to the parameter name. Only integer data are read for the array NPTANK, with spaces separating each index. Other ASCII characters may be used to delimit the data in pairs. Further details and examples are given in Section 10.7.

NUMHDR is the integer parameter used to control writing of a one-line header in the numeric output files.

NUMHDR= 0: No headers are included.

NUMHDR= 1: A one-line header is included in the numeric output files specifying the file name, date, and time.

The default value is NUMHDR= 0.

NUMNAM is the integer parameter used to control the assignment of filenames to the numeric output files.

NUMNAM= 0: Numeric filenames are assigned based on the filename of the FRC input control file. (The same filename is used for the OUT output file.)

NUMNAM= 1: Numeric filenames are assigned as 'optn'.

The default value is NUMNAM= 0.

PANEL_SIZE is a parameter used for automatic subdivision of patches in the higher-order panel method. Further information is given in Chapter 6.

PANEL_SIZE $\leq$ 0.0: subdivide patches into panels as specified by the parameters NU,NV in the SPL file.

PANEL_SIZE $>$ 0.0: subdivide patches into panels so that the maximum length of each panel is approximately equal to the value of this parameter in dimensional units.

RHOTANK is a real array used to specify the density of fluid in internal tanks. The density specified is relative to the density ρ of the fluid in the external domain outside the bodies, as defined in Chapter 4. The data in the array RHOTANK must be input in the same order as the data in the array NPTANK. Multiple lines of this parameter may be used, with an arbitrary number of data on each line, but each line must begin with 'RHOTANK='. The total number of tanks NTANKS is derived from the inputs NPTANK in the POTEN run. If fewer than NTANKS values of RHOTANK are specified, the remainder of the array is assigned the last non-negative value. Thus if the density is the same for all tanks, only the first value is required. Zero may be assigned, but negative values of the density must not be assigned. RHOTANK is only used in the FORCE run, and may be changed if separate FORCE runs are made using the same POTEN outputs. Further details and examples are given in Section 10.7.

The following two parameters are applicable to the PC version only.

SCRATCH_PATH is the path designating a directory (folder) for storage of some scratch arrays. If this input is not used all scratch storage is in the default directory where the program is run. If a different directory is specified, about half of the scratch arrays will be stored in the default directory, and the remaining arrays will be stored in the designated alternative directory. This option permits users with two or more disk drives to distribute the scratch storage, thereby increasing the usable disk storage. The example below illustrates this option. (The user must make the specified directory, if it does not already exist, prior to running the program.) Spaces cannot be used in the pathname, as explained below.

USERID_PATH designates the directory (folder) where the input file USERID.WAM is stored. (This input file is required for users of V6PC.) It is convenient to store USERID.WAM, the executable WAMIT.EXE, and the dynamic link libraries DFORRT.DLL, GEOMX-ACT.DLL, NEWMODES.DLL together in one directory (e.g. C:\WAMIT). In this case USERID_PATH should be specified as in the example below. (Note that the executables can be run from another directory provided their resident directory is included in the system path specified in the AUTOEXEC.BAT file.) Spaces cannot be used in the pathname, as explained below.

The syntax of the configuration file is illustrated in the following example. To specify each of the desired inputs, the corresponding parameter is displayed, followed by an '=' sign, followed by the value of the parameter. These lines may be in any order. Lines which do not contain an '=' sign are ignored. Comments may be inserted following the value of a parameter on the same line, separated by at least one blank space. (Since a blank space is used to designate a comment, the names of directories or folders in SCRATCH_PATH and USER_PATH cannot include blank spaces.) All elements of the arrays IALTFRCN(NBODY), NOOUT(9) and XBODY(4) should be displayed in order on one line. All other parameters must be input singly on separate lines.

The following example of a configuration file illustrates all of the possible input parameters for NBODY=1. Section 7.6 shows additional inputs for NBODY>1. For clarity the parameters are arranged in alphabetic order, but their actual order is arbitrary.

IALTFRC=1 (use FRC format in Section 3.1)
IALTPOT=1 (use POT format in Section 3.3)
ICTRSURF=1 (use Alternative 1 evaluation of control surface drift forces)
IDIAG=0 (omit IDIAG from POT file)
IFIELD_ARRAYS=1 (arrays of field point data is assigned, as in Section 3.10)
IFORCE=1 (run the FORCE subprogram)
IGENMDS=1 (run DEFINE subroutine in .dll file for generalized modes)
ILOG=0 (omit ILOG from POT file)
ILOWGDF=0 (do not output low-order GDF from higher-order geometry)
ILOWHI=0 (use low-order method)
INUMOPT5=1 (output the separate components of the body pressure/velocity)
INUMOPT6=1 (output the separate components of the field pressure)
INUMOPT7=1 (output the separate components of the field velocity)
IPERIO=1 (POT file contains wave periods in seconds)
IPLTDAT=0 (do not output data for plotting geometry)
IPNLBPT=1 (evaluate body pressure at points specified in body coordinates)
IPOTEN=1 (run the POTEN subprogram)
QUAD=0 (omit IQUAD from POT file)
IQUADI=4 (inner Gauss quadrature order in higher-order method)
IQUADO=3 (outer Gauss quadrature order in higher-order method)
IRR=0 (omit IRR from POT file)
ISCATT=1 (Solve for scattered potential)
ISOLVE=2 (Use two blocks in iterative solver)
ISOR = 1 (solve for source strength – omit ISOR from POT file)
ITANKFPT=1 (input field points in internal tanks with special format)
KSPLIN = 3 (Use 3rd-order B-splines for potential in higher-order method)
MAXITT = 35 (Maximum number of iterations in solver)
MAXMIT = 8 (Maximum number of iterations in momentum integral)
MAXSCR = 1024 (store $\leq 1024 \times 1024$ LHS matrix elements in RAM)
MODLST =1 (Outputs in same order as left-hand-sides)
MONITR=1 (display all FORCE output to monitor)
NEWMDS=0 (No generalized modes)
NOOUT=1 1 1 1 0 1 1 1 1 (output all but body pressure/velocity data)
NPTANK= 8-11 12-15 (range of panels/patches on two internal tanks)
NUMHDR=1 (write headers to numeric output files)
NUMNAM=1 (Numeric filenames are assigned as 'OPTN')
PANEL_SIZE = 0.1 (automatic subdivision of patches in higher-order method)
RHOTANK = 0.6 1.0 (relative fluid densities in the two tanks)
SCRATCH_PATH=D:\TEMP
USERID_PATH=C:\WAMIT
XBODY = 0.0 0.0 0.0 0.0 (omit XBODY from POT file)

3.8 FILE NAMES

The names of the input data files should not exceed **20 characters** in length including extensions and should have **one period** separating the filename and extension. Provisions are made in WAMIT to guard against unintended loss of old output files. If the names specified for the P2F and OUT files are identical to existing files, the user is prompted interactively to choose between (1) changing the new output filename, or (2) overwriting the old file. If a new filename is specified interactively it must include the desired extension. For example, if the name CYL.FRC is retained in Session 2 as described in Section 2.2, instead of a modified name such as CYL2.FRC, the user will be prompted at the start of the FORCE subprogram with the choice of either overwriting the old file CYL.OUT or specifying another name for the new OUT file. If the default setting NUMNAM=0 is used, the same safeguard will apply to the numeric output files, minimizing the possibility that these are lost during a subsequent run. Otherwise, if NUMNAM=1, the OPTN output files are assigned the same generic names for all runs, and old OPTN files are overwritten without warning when a new run is made; this option avoids the proliferation of old output files, but requires the user to rename or otherwise preserve the contents of OPTN files which are to be saved.

For batch processing it is important to avoid interactive interrogation from the programs. Thus the user should delete or rename P2F and/or OUT files from previous runs, if the same names will be assigned (from the POT and/or FRC control filenames respectively in a new run.)

In the User Manual capital letters are generally used for file names to provide emphasis and visual correspondence with systems where file names are displayed in capital letters. A distinction must be made for systems such as UNIX, which are case sensitive. Most filenames and extensions which are assigned by WAMIT are specified in lowercase letters (via assignments of type CHARACTER in the source file modulesc.f, module MAINC_ARRAYS). This applies in particular to the input files 'fnames.wam' and 'config.wam', and to the extensions '.p2f', '.out', etc. The only exceptions to this convention are the scratch files opened temporarily by WAMIT with the explicit names 'SCRATCHA', SCRATCHB', etc.

In previous versions of WAMIT the extensions '.gdf', '.pot', and '.frc' were recommended but not required for the GDF, POT, and FRC input files. In Version 6 these extensions are required, unless the corresponding assignments are changed in the source code. (This new requirement permits a more flexible arrangement of the optional input file fnames.wam.)

In V6.2-3, a separate utility program F2T.EXE is provided. 'INPUTS.F2T' is reserved as a generic input filename to this program.

3.9 FILE FORMAT

The free-format READ statements read only the specified data on a line, or on subsequent lines if there is insufficient data on the first line. Comments inserted after the specified data are ignored. Thus it is possible for the user to include comments at the ends of selected lines in the input files, to identify the data on these lines. Such comments should be separated from the data by at least one blank space. This format is illustrated in the input files of Test Run 01. Generally, comments at the ends of appropriate lines, which contain non-numeric ASCII characters, will ensure that execution is interrupted with an error message if insufficient data is contained on the line.

When blocks of data are written on multiple lines, and read by a single READ statement, comments are only permitted after all of the data is read. In the POT file, for example, comments could be placed after the last elements of the arrays PER and BETA, but not on intermediate lines which contain these arrays, and similarly for field point coordinates in the FRC file. A special restriction applies to the format of the GDF file when the higher-order method is used (ILOWHI=1) and the geometry is defined by MultiSurf (IGDEF=2), or explicitly (IGDEF <0 or >2). In this case it is possible to input relevant body dimensions in the GDF file, as illustrated in Section 6.8 and in most of the higher-order test runs. In this case the data included in the GDF file (starting on line 5) must be restricted to columns 1-80. Note that an extra line of data has been added to the GDF file in Versions 6.2-3, in all cases where (IGDEF <0 or >1), to specify the number of lines to be read beyond line 5. Users of Version 6.0 must insert this extra line in GDF files where the higher-order method is used, except in the cases IGDEF=0 and IGDEF=1 where no changes are required. Also, IGDEF=2 is reserved in V6.2-3 for MultiSurf inputs and should not be used for explicit geometry definition as permitted in V6.0.

3.10 UNIFORM ARRAYS OF FIELD POINTS

In some applications large numbers of field points are required, with a spacing on a rectangular grid. In Version 6.3 a new option has been introduced to facilitate inputting this data in the FRC file. This option is specified by assigning the parameter IFIELD_ARRAYS= 1 in the CFG file, as explained in Section 3.7. When IFIELD_ARRAYS= 1, additional data is appended to the FRC file immediately after the line(s) containing NFIELD and XFIELD. The following shows the complete Alternative 1 FRC file, replacing the format shown in Section 3.3:

```
header
IOPTN(1) IOPTN(2) IOPTN(3) IOPTN(4) IOPTN(5) IOPTN(6) IOPTN(7) IOPTN(8) IOPTN(9)
VCG
XPRDCT(1,1) XPRDCT(1,2) XPRDCT(1,3)
XPRDCT(2,1) XPRDCT(2,2) XPRDCT(2,3)
XPRDCT(3,1) XPRDCT(3,2) XPRDCT(3,3)
NBETAH
BETAH(1) BETAH(2) ...  BETAH(NBETAH)
NFIELD
XFIELD(1,1) XFIELD(2,1) XFIELD(3,1)
XFIELD(1,2) XFIELD(2,2) XFIELD(3,2)
XFIELD(1,3) XFIELD(2,3) XFIELD(3,3)
.
.
XFIELD(1,NFIELD) XFIELD(2,NFIELD) XFIELD(3,NFIELD)
NFIELD_ARRAYS
ITANKFLD(1)
NFX(1) X1(1) DELX(1)
NFY(1) Y1(1) DELY(1)
NFZ(1) Z1(1) DELZ(1)
ITANKFLD(2)
NFX(2) X1(2) DELX(2)
NFY(2) Y1(2) DELY(2)
NFZ(2) Z1(2) DELZ(2)
.
.
ITANKFLD(NFIELD_ARRAYS)
NFX(NFIELD_ARRAYS) X1(NFIELD_ARRAYS) DELX(NFIELD_ARRAYS)
NFY(NFIELD_ARRAYS) Y1(NFIELD_ARRAYS) DELY(NFIELD_ARRAYS)
NFZ(NFIELD_ARRAYS) Z1(NFIELD_ARRAYS) DELZ(NFIELD_ARRAYS)
```

The additional data is defined as follows:

NFIELD_ARRAYS is the number of separate arrays. NFIELD_ARRAYS must be an

integer, greater than or equal to zero.

ITANKFLD(n) (n=1,2,...,NFIELD_ARRAYS) is an integer which specifies if the array n is in the exterior fluid domain, or in an internal tank. ITANKFLD=0 denotes the exterior fluid domain. In cases where the field point array is in an internal tank ITANKFLD is assigned with the same integer as the number of the tank, as explained in Section 10.7. In all cases where no internal tanks are included, ITANKFLD=0 must be specified.

NFX, NFY, NFZ are positive integers, specifying the number of elements in the array parallel to the X,Y,Z axes. (The total number of elements in each array is the product of these three integers.)

X1, Y1, Z1 are the coordinates of the first point in the array.

DELX, DELY, DELZ are the distances between adjacent points in the array, in each direction.

If NFX=1, indicating that there is only one point in the direction parallel to the x-axis, the value of DELX is irrelevant, but must be assigned to prevent a read error. Similarly, for NFY=1 and/or NFZ=1 the values of DELY and/or DELZ are irrelevant.

The field points assigned using this procedure are augmented to the list of field points (if any) assigned in the conventional manner as explained in Section 3.3. If the array option is used for all field points, then NFIELD=0 must be assigned. At runtime NFIELD is increased to include all field points. The complete list of all field points is output in the .FPT file.

Test23 illustrates the use of this option.

Chapter 4

DEFINITION OF QUANTITIES EVALUATED BY WAMIT

The forces and other quantities evaluated by WAMIT are output in a standard nondimensional form, in terms of the appropriate combinations of the water density ρ, the acceleration of gravity g, the incident-wave amplitude A, frequency ω,and the length scale L defined by the input parameter ULEN in the GDF file. (The volume and coordinates of the center of buoyancy are exceptions. They are output in a dimensional form.)

Body motions and forces are defined in relation to the origin of the same Cartesian coordinate system relative to which the panel offsets are defined. Note that this origin may be located on, above or below the free surface. *If planes of symmetry are defined for the body, the origin must always lie on these planes of symmetry.* The x- and y-axes must be parallel to the mean position of the free surface.

The notation and definitions of physical quantities here correspond with those in Reference [3], except that in the latter reference the y axis is vertical.

All of the evaluated quantities are included with appropriate identification in the OUT output file. To facilitate post-processing the same quantities are also saved in the numeric output files, following the format defined in Section 4.9.

In the cases of field data and mean drift forces (Options 5-9) the definitions given below in Sections 4.5-4.8 apply to the complete solution for the combined radiation and diffraction problems. The same quantities can be evaluated separately for either the radiation or diffraction problems, as explained in Section 4.10.

For the sake of simplicity, the definitions which follow in this Section assume that the origin of the coordinate system is located on the free surface. Special definitions apply to some quantities if vertical walls are defined, as explained in Section 5.3.

4.1 HYDROSTATIC DATA

All hydrostatic data can be expressed in the form of surface integrals over the mean body wetted surface S_b, by virtue of Gauss' divergence theorem.

a) Volume

$$\forall = -\iint_{S_b} n_1 x dS = -\iint_{S_b} n_2 y dS = -\iint_{S_b} n_3 z dS$$

All three forms of the volume are evaluated in WAMIT, as independent checks of the panel coordinates, and printed in the summary header of the output file. The median volume of the three is used for the internal computations. If it is less than 10^{-30}, a warning is displayed and the coordinates of the center of buoyancy are set equal to zero. For bottom-mounted structures, where panels are not defined on the bottom, the last integral defined above differs from the correct submerged volume as noted in Section 3.1.

b) Coordinates of center of buoyancy

$$x_b = \frac{-1}{2\forall} \iint_{S_b} n_1 \, x^2 dS$$

$$y_b = \frac{-1}{2\forall} \iint_{S_b} n_2 \, y^2 dS$$

$$z_b = \frac{-1}{2\forall} \iint_{S_b} n_3 z^2 dS$$

c) Matrix of hydrostatic and gravitational restoring coefficients

$C(3,3) = \rho g \iint_{S_b} n_3 dS$	$\bar{C}(3,3) = C(3,3)/\rho g L^2$
$C(3,4) = \rho g \iint_{S_b} y n_3 dS$	$\bar{C}(3,4) = C(3,4)/\rho g L^3$
$C(3,5) = -\rho g \iint_{S_b} x n_3 dS$	$\bar{C}(3,5) = C(3,5)/\rho g L^3$
$C(4,4) = \rho g \iint_{S_b} y^2 n_3 dS + \rho g \forall z_b - m g z_g$	$\bar{C}(4,4) = C(4,4)/\rho g L^4$
$C(4,5) = -\rho g \iint_{S_b} x y n_3 dS$	$\bar{C}(4,5) = C(4,5)/\rho g L^4$
$C(4,6) = -\rho g \forall x_b + m g x_g$	$\bar{C}(4,6) = C(4,6)/\rho g L^4$
$C(5,5) = \rho g \iint_{S_b} x^2 n_3 dS + \rho g \forall z_b - m g z_g$	$\bar{C}(5,5) = C(5,5)/\rho g L^4$
$C(5,6) = -\rho g \forall y_b + m g y_g$	$\bar{C}(5,6) = C(5,6)/\rho g L^4$

where $C(i,j) = C(j,i)$ for all i, j, except for $C(4,6)$ and $C(5,6)$. For all other values of the indices i, j, $C(i,j) = 0$. In particular, $C(6,4) = C(6,5) = 0$.

In C(4,4), C(4,6), C(5,5) and C(5,6), m denotes the body mass. When Alternative form 1 is used for the FRC file (Section 3.3) the body mass is computed from the relation $m = \rho\forall$. When Alternative form 2 is used for the FRC file (Section 3.4) the body mass is defined by EXMASS(3,3).

4.2 ADDED-MASS AND DAMPING COEFFICIENTS

$$A_{ij} - \frac{i}{\omega} B_{ij} = \rho \iint_{S_b} n_i \varphi_j dS$$

$$\bar{A}_{ij} = \frac{A_{ij}}{\rho L^k} \qquad \bar{B}_{ij} = \frac{B_{ij}}{\rho L^k \omega}.$$

where $k = 3$ for $i, j = 1, 2, 3$, $k = 4$ for $i = 1, 2, 3$, $j = 4, 5, 6$ or $i = 4, 5, 6$, $j = 1, 2, 3$ and $k = 5$ for $i, j = 4, 5, 6$.

4.3 EXCITING FORCES

a) Exciting forces from the Haskind relations

$$X_i = -i\omega\rho \iint_{S_b} \left(n_i \varphi_0 - \varphi_i \frac{\partial \varphi_0}{\partial n} \right) dS$$

b) Exciting forces from direct integration of hydrodynamic pressure

$$X_i = -i\omega\rho \iint_{S_b} n_i \varphi_D dS$$

$$\bar{X}_i = \frac{X_i}{\rho g A L^m},$$

where $m = 2$ for $i = 1, 2, 3$ and $m = 3$ for $i = 4, 5, 6$.

4.4 BODY MOTIONS IN WAVES

Two alternative procedures are followed to evaluate the body motions in waves, corresponding respectively to the Alternative 1 (Section 3.3) and Alternative 2 (Section 3.4) FRC control files.

In Alternative 1, which is restricted to a body in free stable flotation without external constraints, the following relations hold

$$m = \rho \forall$$

$$x_b = x_g, \qquad y_b = y_g$$

where m is the body mass and (x_g, y_g, z_g) are the coordinates of the center of gravity.

The inertia matrix is defined as follows.

$$M = \begin{pmatrix} m & 0 & 0 & 0 & mz_g & -my_g \\ 0 & m & 0 & -mz_g & 0 & mx_g \\ 0 & 0 & m & my_g & -mx_g & 0 \\ 0 & -mz_g & my_g & I_{11} & I_{12} & I_{13} \\ mz_g & 0 & -mx_g & I_{21} & I_{22} & I_{23} \\ -my_g & mx_g & 0 & I_{31} & I_{32} & I_{33} \end{pmatrix}. \tag{4.1}$$

WAMIT equates the body mass to the mass of the displaced water in free flotation. The moments of inertia I_{ij} are defined in terms of the corresponding radii of gyration r_{ij}, defined by the relation

$$I_{ij} = \rho \forall r_{ij} |r_{ij}|.$$

The array XPRDCT(I,J) input to WAMIT contains the radii of gyration input with the same units of length as the length scale ULEN defined in the panel data file.

In the Alternative 2 format of the FRC file the matrices $M_{ij} + M^E_{ij}$, B^E_{ij} and C^E_{ij} are input by the user to include the possibility of external force/moment constraints acting on the body.

The complex amplitudes of the body's motions ξ_j are obtained from the solution of the 6×6 linear system, obtained by applying Newton's law

$$\sum_{j=1}^{6} \left[-\omega^2 (M_{ij} + M^E_{ij} + A_{ij}) + i\omega (B_{ij} + B^E_{i,j}) + (C_{ij} + C^E_{ij}) \right] \xi_j = X_i.$$

where the matrices M^E_{ij}, B^E_{ij} and C^E_{ij} are included only in the Alternative 2 case. Note that in the Alternative 2 case the user must specify the body inertia matrix M_{ij} and include it in the total inertia matrix $M_{ij} + M^E_{ij}$ specified in the FRC file.

The non-dimensional definitions of the body motions are

$$\bar{\xi_i} = \frac{\xi_i}{A/L^n},$$

where $n = 0$ for $i = 1, 2, 3$ and $n = 1$ for $i = 4, 5, 6$.

4.5 HYDRODYNAMIC PRESSURE

The complex unsteady hydrodynamic pressure on the body boundary or in the fluid domain is related to the velocity potential by the linearized Bernoulli equation

$$p = -\rho \frac{\partial \varphi}{\partial t}.$$

The total velocity potential is defined by

$$\varphi = \varphi_D + i\omega \sum_{j=1}^{6} \xi_j \varphi_j,$$

where the radiation and diffraction velocity potentials are defined in Section 12.1. In order to render the velocity potential and the hydrodynamic pressure non-dimensional, we define

$$\bar{p} = \frac{p}{\rho g A} = \bar{\varphi}_D + KL \sum_{j=1}^{6} \bar{\xi}_j \bar{\varphi}_j.$$

where $K = \omega^2/g$ and

$$\bar{\varphi}_D = \frac{\varphi_D}{igA/\omega}, \qquad \bar{\varphi}_j = \frac{\varphi_j}{L^{n+1}}$$

with $n = 0$ for $j = 1, 2, 3$ and $n = 1$ for $i = 4, 5, 6$.

The body pressure can be evaluated separately for the diffraction or radiation problems by following the procedure described in Section 4.12. When the radiation components are output separately, the nondimensional pressure due to jth mode is defined by

$$\bar{p} = \frac{p}{\rho g \xi_j L^n} = KL\bar{\phi}_j \quad \text{or} \quad \bar{p} = \frac{p}{\rho g a_j L^{n+1}} = -\bar{\phi}_j$$

where a_j is the acceleration in the same unit as the gravitional constant specified in the GDF file for $j = 1, 2, 3$ and the angular acceleration in rad/s^2 for $j = 4, 5, 6$. $n = 0$ for $j = 1, 2, 3$ and $n = 1$ for $j = 4, 5, 6$.

4.6 FREE-SURFACE ELEVATION

The free surface elevation is obtained from the dynamic free-surface condition

$$\eta = -\frac{1}{g} \left(\frac{\partial \varphi}{\partial t} \right)_{z=0},$$

and in non-dimensional form

$$\bar{\eta} = \frac{\eta}{A} = \left(\bar{\varphi}_D + KL \sum_{j=1}^{6} \bar{\xi}_j \bar{\varphi}_j \right)_{z=0} ,$$

where $\bar{\varphi}$ is defined as in Section 4.5. [Note that the non-dimensional field hydrodynamic pressure and wave elevation are equal to the non-dimensional velocity potential at the respective positions.]

These parameters can be evaluated separately for the diffraction or radiation problems by following the procedure described in Section 4.10. When the radiation components are output separately, the nondimensional free-surface elevation due to jth mode is defined by

$$\bar{\eta} = \frac{\eta}{\xi_j L^n} = KL\bar{\phi}_j$$

where $n = 0$ for $j = 1, 2, 3$ and $n = 1$ for $j = 4, 5, 6$.

The evaluation of the pressure or free-surface elevation requires special caution close to the body surface. Within a distance on the order of the dimensions of the adjacent panel(s), field-point quantities cannot be computed reliably. More specific limits can be ascertained by performing a sequence of computations and studying the continuity of the result. Approaching the body along a line normal to the centroid of a panel will minimize this problem. See Reference [12] regarding the computation of run-up at the intersection of the body and free surface.

4.7 VELOCITY VECTOR ON THE BODY AND IN THE FLUID DOMAIN

The non-dimensional velocities evaluated by WAMIT are defined in vector form by

$$\bar{\mathbf{V}} = \frac{\mathbf{V}}{igA/(\omega L)} = \bar{\nabla}\bar{\varphi}_D + KL \sum_{j=1}^{6} \bar{\xi}_j \bar{\nabla}\bar{\varphi}_j$$

where

$$\bar{\nabla} = L\nabla$$

is the non-dimensional gradient operator.

These parameters can be evaluated separately for the diffraction or radiation problems by following the procedure described in Section 4.10.

The evaluation of the velocity requires special caution close to the body surface, in the same manner as the pressure and free-surface elevation. (See Section 4.6 above.) When the radiation components are output separately, the nondimensional velocity due to jth mode is defined by

$$\bar{V} = \frac{V}{v_j L^n} = \bar{\nabla}\bar{\varphi}_j$$

where v_j denotes the velocity of the body for $j = 1, 2, 3$ and the angular velocity for $j = 4, 5, 6$. $n = 0$ for $j = 1, 2, 3$ and $n = 1$ for $j = 4, 5, 6$.

4.8 MEAN DRIFT FORCE AND MOMENT

The definition of the non-dimensional mean drift force and moment in unidirectional waves is

$$\bar{F}_i = \frac{F_i}{\rho g A^2 L^k}$$

where $k = 1$ for the forces ($i = 1, 2, 3$), and $k = 2$ for the moments ($i = 4, 5, 6$).

For bi-directional waves of the same period, with complex amplitudes (A_1, A_2) and corresponding angles of incidence (β_1, β_2), the nondimensional outputs $\bar{F}_i(\beta_1, \beta_2)$ are the coefficients such that the total dimensional mean drift force or moment exerted on the body is given by the equation

$$F_i(\beta_1, \beta_2) = \rho g L^k (|A_1|^2 \bar{F}_i(\beta_1, \beta_1) + |A_2|^2 \bar{F}_i(\beta_2, \beta_2) + 2Re[A_1 A_2^* \bar{F}_i(\beta_1, \beta_2)])$$

Note that $\bar{F}_i^*(\beta_2, \beta_1) = \bar{F}_i(\beta_1, \beta_2)$, where the asterisk ($*$) denotes the complex conjugate.

In Option 8, the evaluation of the horizontal drift force and vertical moment is based on the momentum conservation principle in its general form (see References [4] and [26]), without the assumption of energy conservation. This permits the analysis of cases where the body motions are affected by non-conservative effects, such as external damping. The azimuthal integration required to evaluate the momentum flux is performed by an adaptive quadrature formula in subroutine MDRFTM. [The integration is performed iteratively, with convergence specified by the criterion of absolute or relative errors in each drift force less than TOL=10^{-4}. The maximum number of iterations is controlled by the parameter MAXMIT=8, which is specified in the main program. A warning message is displayed in the event that this convergence criterion is not satisfied.]

In Option 9, the evaluation of the drift force and moment is based on integration of the pressure over the body surface, using the relations in [10] and [17], as summarized in Section 12.7. When ICTRSURF=1 the drift force and moment are also evaluated based on the momentum flux across the control surface, using equations (12.57-60).

These parameters can be evaluated separately for the diffraction or radiation problems by following the procedure described in Section 4.10.

4.9 FORMAT OF NUMERIC OUTPUT FILES

For each of the nine options in the FORCE subprogram, separate output files of numeric data are generated as listed in Section 3.3. The hydrodynamic parameters in these files are output in the same order as in the OUT file, and listed in the following format:

OPTN.1:	PER	I	J		$\bar{A}_{ij}$	$\bar{B}_{ij}$		
OPTN.2:	PER	BETA	I		Mod($\bar{X}_i$)	Pha($\bar{X}_i$)	Re($\bar{X}_i$)	Im($\bar{X}_i$)
OPTN.3:	PER	BETA	I		Mod($\bar{X}_i$)	Pha($\bar{X}_i$)	Re($\bar{X}_i$)	Im($\bar{X}_i$)
OPTN.4:	PER	BETA	I		Mod($\bar{\xi}_i$)	Pha($\bar{\xi}_i$)	Re($\bar{\xi}_i$)	Im($\bar{\xi}_i$)
OPTN.5P:	PER	BETA	M	K	Mod($\bar{p}$)	Pha($\bar{p}$)	Re($\bar{p}$)	Im($\bar{p}$)
OPTN.5VX:	PER	BETA	M	K	Mod($\bar{V}_x$)	Pha($\bar{V}_x$)	Re($\bar{V}_x$)	Im($\bar{V}_x$)
OPTN.5VY:	PER	BETA	M	K	Mod($\bar{V}_y$)	Pha($\bar{V}_y$)	Re($\bar{V}_y$)	Im($\bar{V}_y$)
OPTN.5VZ:	PER	BETA	M	K	Mod($\bar{V}_z$)	Pha($\bar{V}_z$)	Re($\bar{V}_z$)	Im($\bar{V}_z$)
OPTN.6:	PER	BETA	L		Mod($\bar{p}$)	Pha($\bar{p}$)	Re($\bar{p}$)	Im($\bar{p}$)
OPTN.7X:	PER	BETA	L		Mod($\bar{V}_x$)	Pha($\bar{V}_x$)	Re($\bar{V}_x$)	Im($\bar{V}_x$)
OPTN.7Y:	PER	BETA	L		Mod($\bar{V}_y$)	Pha($\bar{V}_y$)	Re($\bar{V}_y$)	Im($\bar{V}_y$)
OPTN.7Z:	PER	BETA	L		Mod($\bar{V}_z$)	Pha($\bar{V}_z$)	Re($\bar{V}_z$)	Im($\bar{V}_z$)

OPTN.8:	PER	BETA_1	BETA_2	I	Mod($\bar{F}_i$)	Pha($\bar{F}_i$)	Re($\bar{F}_i$)	Im($\bar{F}_i$)
OPTN.9:	PER	BETA_1	BETA_2	I	Mod($\bar{F}_i$)	Pha($\bar{F}_i$)	Re($\bar{F}_i$)	Im($\bar{F}_i$)
	[PER	BETA_1	BETA_2	-I	Mod($\bar{F}_{io}$)	Pha($\bar{F}_{io}$)	Re($\bar{F}_{io}$)	Im($\bar{F}_{io}$)]

(Depending on the value of NUMNAM, the filenames OPTN will be replaced by *frc*.)

If option 5 is specified and INUMOPT5=1, as explained in Section 4.12, the numeric output files .5P, .5VX, .5VY, .5VZ contain the separate components of the radiation and diffraction pressure and velocity in the following modified format:

OPTN.5P:	PER	M	K	Re($\bar{p}_1$)	Im($\bar{p}_1$)	Re($\bar{p}_2$)	Im($\bar{p}_2$)	...
	PER	BETA	M	K	Re($\bar{p}_D$)	Im($\bar{p}_D$)		

Here ... denotes the remaining components for modes 3,4,5,6 if the six rigid-body modes are specified for a single body. More generally when different sets of modes are evaluated for one or multiple bodies, these are output in sequence. For each wave period the radiation pressures are listed for all values of M and K before the diffraction pressures. Corresponding formats apply for the fluid velocity components in the files OPTN.5VX, OPTN.5VY, OPTN.5VZ.

Starting in Version 6.3, the supplementary output file *out*.hst is created in the following format, to output values of the hydrostatic matrix C_{ij}:

out.hst:	I	J	C(I,J)

If Option 5 is specified and IPNLBPT=0, the supplementary output file *gdf*.PNL is created in the following format:

gdf.PNL:	M	K	XCT	YCT	ZCT	AREA	n_x	n_y	n_z	$(\mathbf{r}\times\mathbf{n})_x$	$(\mathbf{r}\times\mathbf{n})_y$	$(\mathbf{r}\times\mathbf{n})_z$

If option 6 is specified and INUMOPT6=1 or option 7 is specified and INUMOPT7=1, the numeric output files .6, or .7X, .7Y and 7Z contain the separate components of the

radiation and diffraction pressure and velocity in the following modified format:

OPTN.6:	PER	L	Re($\bar{p}_1$)	Im($\bar{p}_1$)	Re($\bar{p}_2$)	Im($\bar{p}_2$)	...
	PER	BETA	L	Re($\bar{p}_D$)	Im($\bar{p}_D$)		

Here ... denotes the remaining components for modes 3,4,5,6 if the six rigid-body modes are specified for a single body. More generally when different sets of modes are evaluated for one or multiple bodies, these are output in sequence. For each wave period the radiation pressures are listed for all values of L before the diffraction pressures. Corresponding formats apply for the fluid velocity components in the files OPTN.7X, OPTN.7Y, OPTN.7Z. (See section 4.12.)

If Option 6 or 7 is specified, the supplementary output file *frc*.FPT will be created in the following format:

OPTN.FPT: L XFIELD(L) YFIELD(L) ZFIELD(L)

Except as noted below, the definitions of parameters in these files are as follows:

I, J: Mode indices

M: Index for quadrant (2 planes of symmetry) or half (1 plane of symmetry).

(If no planes of symmetry are specified, or if IPNLBPT>0, then M=1.)

K: Index for panels on the body surface

L: Index for field points

PER: Period

BETA: Wave heading

BETA_1, BETA_2: Two wave headings for the mean drift forces and moments

XCT, YCT, ZCT: Dimensional global coordinates of panel centroid.

AREA: Dimensional value of the area of a panel

n_x, n_y, n_z: Components of the unit vector normal to K-th panel in local coordinate system

$(\mathbf{r}\times\mathbf{n})_x$, $(\mathbf{r}\times\mathbf{n})_y$, $(\mathbf{r}\times\mathbf{n})_z$: Components of the cross product of the position vector to the centroid of the K-th panel and it's normal vector, in the local coordinate system. Here $\mathbf{r}$ is given in dimensional units.

XFIELD, YFIELD, ZFIELD: Dimensional global coordinates of the field point

All output quantities are nondimensionalized as defined in Sections 4.2-8. Complex quantities are defined by the magnitude (Mod), phase in degrees (Pha), and also in terms of the real (Re) and imaginary (Im) components. The phase is relative to the phase of an incident wave at the origin of the global coordinates system.

In Option 5, when IPNLBPT$\neq$ 0, the index M refers to the body index and K refers to the body point in the order listed in the .bpi input file and .bpo output file. The file *gdf*.pnl is only output when IOPTN(5)>0 and IPNLBPT=0. If ILOWHI=1 the data output in this file differ from those shown above as follows:

K: Index for points on the body surface (See section 4.10)

XCT, YCT, ZCT: Dimensional global coordinates of points

AREA: Product $J\delta U\delta V$ where J is the Jacobian at the point, and δU, δV denote the differential increments between points in parametric coordinates.

n_x, n_y, n_z: Components of the unit vector normal to the body surface at each point

$(\mathbf{r} \times \mathbf{n})_x$, $(\mathbf{r} \times \mathbf{n})_y$, $(\mathbf{r} \times \mathbf{n})_z$: Components of the cross product of the position vector at each point.

In Option 8, the mean force and moment are output only for modes I=1, 2 and 6, corresponding to the two horizontal forces and yaw moment, respectively.

In Option 9, the six components of the mean forces and moments, $\bar{F}_i$, are output on the first six lines, with positive indices (i=1,2,...,6). These are the components of the force and moment vectors, defined with respect to the inertial reference frame corresponding to the mean position of the body coordinate system. When **IRAD**$\neq -1$, three additional components of the moment $\bar{F}_{io}$, are output and identified by negative indices (i=-4,-5,-6). These are the components of the moment about the moving origin, denoted by 'o' in Figure 12-2. In all cases the components of the vector force and moment are defined with respect to the inertial (mean) coordinate system.

In Option 9, if ICTRSUFT=1, the drift forces evaluated from momentum flux on the control surface are output in the numeric output file OPTN.9c, in the same format as shown above for OPTN.9.

If NBODY>1, the panels of all bodies are merged with a common index K, following the same order as the body numbers in the global GDF file (See Chapter 7).

4.10 BODY PRESSURE OUTPUT FOR THE HIGHER-ORDER METHOD

If the higher-order method is used (ILOWHI=1), Option 5 is selected in the FRC file, and IPNLBPT=0, the pressure and the fluid velocity on the body surface are output at the points corresponding to equally spaced points in parametric space. These points are defined in parametric space as the midpoints of the set of $(KU+1)*(KV+1)$ panel subdivisions on each patch (see Chapter 6). The coordinates, the extended normal vector corresponding to 6 rigid body modes and the Jacobian are output in the .pnl file. The value of the Jacobian at the prescribed point replaces the panel area in the format shown in Section 4.9. The pressure and the fluid velocity vector at these points are output in the files 5p, 5vx, 5vy and 5vz in the same format as shown in Section 4.9. (If IPNLBPT$\neq$0 is assigned in the configuration file an alternative option is utilized with the points on the body surface specified by the user, as described in Section 4.11.)

When the above options are specified a second output file .5pb is also generated. This file contains the B-spline coefficients and other relevant parameters for the evaluation of the pressure and its derivatives on the body surface. The total pressure coefficient (ϕ), the diffraction pressure (ϕ_D) and the radiation pressure (ϕ_R) are output separately. The radiation pressure has as many components as the number of modes specified in the POT file, including generalized modes. Following the definition of the nondimensional pressure (Section 4.5) these three components are related by the equation

$$\phi = \phi_D + KL \sum_j \xi_j \phi_j$$

where KL is the nondimensional infinite depth wavenumber, ξ_j is the nondimensional motion amplitude and j is the mode index.

The total pressure coefficient is output always. The diffraction pressure coefficient is output when IRAD> -1 and IDIFF>-1. Since the total pressure is the same as the diffraction pressure if IRAD=-1, the diffraction pressure is not output in this case. The radiation pressure coefficient is output when IRAD> -1.

The data in the .5pb file is useful for special post-processing purposes, such as for interfacing with structural loads analyses. The content of the .5pb numeric output file is listed below:

```
HEADER
ISX,ISY
ULEN
NPATCH
IRAD,IDIFF
NPER,NBETA
NEQN
NLHS
NDFR
NBODY
```

```
((XBODY(L,J),L=1,4),J=1,NBODY)
((XBCS(L,J),L=1,2),J=1,NBODY)
(IBPTH(L),L=1,NPATCH)
(IBMOD(L),L=1,NBODY)
(IGEO(J),J=1,8)
(ILHS(J),J=1,4)
(IFLAT(L),L=1,NPATCH)
(KU(L),KV(L),NU(L),NV(L) ,L=1,NPATCH)
(NMDS(J),J=1,4)
(ICOL(J),J=1,NDFR)
((MDS(L,J),L=1,NDFR),J=1,4)
(BETA(NB),NB=1,NBETA) (omit if IDIFF=-1)
Loop over number of periods (repeat NPER times)
  PER(IP),WVNFIN(IP),WVNUM,IFREQ
  IF block starts (if IFREQ=0)
    IF block starts (if IRAD>-1 and IDIFF>-1)
      Loop over wave-headings starts (repeat NBETA times)
        (WRAO(IM,NB),IM=1,NDFR)
      Loop over wave-headings ends (repeat NBETA times)
      Loop over wave-headings starts(repeat NBETA times)
        Loop over number of symmetric images (repeat MXNLHS times)
          Loop over number of patches (repeat NPATCH times)
            (WPRS(I,M,NB),I=NP+1,NQ) (omit if IFLAT(L)=-1)
          End of the loop over number of patches
        End of the loop over symmetric images
      Loop over wave-headings ends (repeat NBETA times)
    IF block ends (if IRAD>-1 and IDIFF>-1)
    IF block starts (if IDIFF >-1)
      Loop over wave-headings starts (NBETA times)
        Loop over number of symmetric images (repeat MXNLHS times)
          Loop over number of patches (repeat NPATCH times)
            (WBD(I,M,NB),I=NP+1,NQ) (omit if IFLAT(L)=-1)
          End of the loop over number of patches
        End of the loop over symmetric images
      Loop over wave-headings ends (NBETA times)
    IF block ends (if IDIFF >-1)
  IF block ends (if IFREQ=0)
  IF block starts ( if IRAD>-1 )
    Loop over left hand side starts (repeat NLHS times)
      Loop over number of modes for each left-hand-side
       MDI (mode index)
        Loop over number of symmetric images (repeat MXNLHS times)
          Loop over number of patches (repeat NPATCH times)
            (WBR(I,ICOL(MDI)),I=NP+1,NQ) (omit if IFLAT(L)=-1)
```

```
          End of the loop over number of patches
        End of the loop over symmetric images
      End of the loop over number of modes for each left-hand-side
    Loop over left hand side ends (repeat NLHS times)
  IF block ends ( if IRAD>-1 )
End of the loop over number of periods
```

NP+1 and NQ are the pointers of the first and the last B-spline coefficients of the unknown velocity potential on patch L.

HLINE: header line

ISX,ISY: Symmetry index (1/0 = symmetric/asymmetric).

ULEN: Characteristic length specified in GDF.

NPATCH: Number of patches.

IRAD, IDIFF: Radiation/diffraction problem indices.

NPER, NBETA: Number of periods and wave headings.

NEQN : The total number of unknown B-spline coefficients.

NLHS : Number of components to be solved when the total solution is decomposed into symmetry and antisymmetry components for the body having geometric symmetry

NDFR : The total number of degrees of freedom. It equal to the sum of the degrees of freedom of each body.

NBODY : Total number of bodies.

XBODY : Normalized coordinates of the origin of body coordinate system and its orientation relative to the global coordinates system.

XBCS : XBCS(1,I) and XBCS(2,I) are cosine and sine of XBODY(4,I)

IBPTH(L) : Body index for patch index L.

IBMOD(N) : Global modes counter. Number of modes prior to the present body N.

IGEO : Parameter used to determine the sign of the pressure/velocity on the reflected patches (see MODE.F)

ILHS : Pointer of the given LHS among NLHS components

IFLAT : Index for patches on the free surface.(IFLAT=-1, patches on interior free surface. IFLAT=1, patches for flat physical surface on the free surface. IFLAT=0, patches not on the free surface.)

KU,KV,NU,NV: Orders and panels

NMDS : For given LHS, total number of modes of radiation problem.

MDS : For given LHS, MDS stores NMDS modes indices.

ICOL : The solution such as motion amplitude is stored in the order which is not ascending from mode 1 (surge). ICOL stores the pointer in that sequence for all modes.

BETA : Wave headings

PER WVNFIN WVNINF IFREQ: Period, finite depth wave number, infinite depth

wave number, period index (IFREQ=0: normal period, IFREQ=1: infinite or zero period). When IFREQ=1, the total and diffraction pressure coefficient are not output in .5pb.

WRAO (I,J): Complex motion amplitude (I: modes, J:wave heading)

WPRS: Total pressure coefficient (I: unknown coefficient, M:reflection J:wave heading)

WBD: Diffraction pressure coefficient (I: unknown coefficient, M:reflection J:wave heading)

MDI: Mode index.

WBR: Radiation pressure coefficient (I: unknown coefficient, ICOL(MDI):pointer of mode MDI)

4.11 BODY PRESSURE AND FLUID VELOCITY AT SPECIFIED POINTS

If IOPTN(5)> 0, the hydrodynamic pressure and fluid velocity on the body surface can be evaluated. The points where the pressure is evaluated depend on the parameter IPNLBPT in the configuration file. In the default case IPNLBPT=0, the pressure is evaluated at the panel centroids in the low-order method (ILOWHI=0) or at a set of uniformly spaced parametric points on each patch in the higher-order method (ILOWHI=1).

If IPNLBPT$\neq$ 0 the body pressure is evaluated at points on the body which are specified by the user in a special input file *gdf*.bpi (Body Point Input). The format of this file is as follows:

```
header
NBPT
X(1) Y(1) Z(1)
X(2) Y(2) Z(2)
.
.
X(NBPT) Y(NBPT) Z(NBPT)
```

Note that the filename of this file must be the same as the filename of the .GDF file.

If $IPNLBPT > 0$, the data in the .bpi file is read and interpreted to be in dimensional body-fixed coordinates. If $IPNLBPT < 0$, the data in the .bpi file is read and interpreted to be in dimensional global coordinates. (The relationship between these two coordinate systems is defined by the array XBODY, as defined in Section 3.1.)

The procedure followed to evaluate the body pressure at these specified points is different in the low-order (ILOWHI=0) and higher-order (ILOWHI=1) solutions. These are described separately below.

If ILOWHI=0, the solution is based on piecewise constant values of the potential on each panel based on colocation at the panel centroids. In order to evaluate the pressure at other points an interpolation procedure is adopted. This interpolation is based on a user-specified number NNEAR of nearest panel centroids. The parameter NNEAR is determined from the absolute value of the input parameter IPNLBPT. IPNLBPT=4 is recommended, when the input points are in body-fixed coordinates. In this case the program searches and identifies the four nearest panel centroids to each specified input point, and assigns weights to each of these panels based on the (inverse) distance to each centroid. The pressure is output in the .5p numeric output file with the following format:

OPTN.5P: PER BETA IBODY IPOINT Mod($\bar{p}$) Pha($\bar{p}$) Re($\bar{p}$) Im($\bar{p}$)

This format and the definitions of the data are the same as in Section 4.9, except that the index IBODY is used to specify the body index and IPOINT is used to specify the index of the input point in the .bpi file (J=1,2,...,NBPT) for each body. Similar output files .5vx,

.5vy, .5vz contain the components of the fluid velocity on the body surface in the same format, when IOPTN(5)≥2. In addition to these hydrodynamic outputs, a supplementary output file *gdf*.bpo (Body Point Output) is created with the following format:

gdf.BPO: M N1 R1 N2 R2 N3 R3 N4 R4

Here M is the quadrant index, N1 is the panel index of the nearest panel and R1 is the radial distance from the specified point (x, y, z) to the centroid of the panel. Successive pairs (Ni, Ri) are the index and radial distance to the other panel centroids, where (i=1,2,...,NNEAR). (In the example shown above NNEAR=4.)

In the higher-order method (ILOWHI=1) the solution for the velocity potential and pressure is represented by continuous B-splines on each patch. For each specified input point (x, y, z) the program searches for the patch index and (U, V) coordinates of the point on this patch which is closest to the input point. The pressure is evaluated at the corresponding point (U, V) and output in the .5p and .5v∗ files with the same format as shown above. In this caes the supplementary output file *gdf*.bpo contains the following data for each point:

gdf.BPO: K M NP U V R I XI XN

Here K is the body point index, M is the quadrant index, NP is the patch index, and (U,V) are the parametric coordinates on the patch. R is the radial distance from the point (U,V) on the patch to the specified (x, y, z) point. An iterative procedure is used to find (U,V), with a specified convergence tolerance of 1.0E-4 for the radial distance in nondimensional Cartesian coordinates. (When the length scale of the patch is larger than 1.0 the tolerance is increased by a factor equal to this length scale, estimated from the Jacobian of the parametric transformation at the center of the patch.) I is the number of iteration. A maximum of 16 iterations are used in this search, and if I=17 this indicates noncovergence of the search. A warning message is generated if nonconvergence occurs for one or more input points, showing the total number of unconverged points. XI is the position vector of the output point on the body surface and XN is the normal vector at XI, both in body coordinates system.

In both the low-order and higher-order implementations, the input data in the .bpi file should correspond to points which lie as close as possible to the body surface.

If points in the BPI files are very close to intersections of adjacent patches in the higher-order method, the index NP in the BPO file should be checked to verify that the correct patch is used, especially in cases where there is ambiguity between the pressure on a conventional patch and the pressure jump on a dipole patch. Similarly, in the low-order method, the panel indices, N1, N2, ... in the BPO file can be checked to verify the corresponding output at the points in BPI files is the pressure obtained from those on the conventional body panels or the pressure jump on the dipole panels.

4.12 RADIATION AND DIFFRACTION COMPONENTS OF THE PRESSURE AND VELOCITY

If IOPTN(5)> 0, the parameter INUMOPT5 in the configuration file can be used to control the outputs of the body pressure in the numeric output files. In the default case INUMOPT5=0 the body pressure output in the numeric output files is the same total pressure as in the formatted .out output file, as defined in Section 4.5. Alternatively, if INUMOPT5=1, the separate components $KL\bar{\varphi}_j$ and $\bar{\varphi}_D$ are output in the numeric output file .5p and the corresponding components of the fluid velocity on the body surface are output in the files .5vx, .5vy, .5vz. Here $\bar{\varphi}_j$ is the nondimensional potential in mode j, and $\bar{\varphi}_D$ is the nondimensional potential of the diffraction problem with the body fixed, as defined in Section 4.5. In this case the format of the numeric output files is modified, as shown in Section 4.9.

If IOPTN(6) $>$ 0, the parameter INUMOPT6 in the configuration file can be used to control the outputs of the field pressure in the numeric output file. In the default case INUMOPT6=0, as in previous versions of WAMIT, the field pressure output in the numeric output files is the same total pressure as in the formatted .out output file, as defined in Section 4.5. Alternatively, if INUMOPT6=1, the separate components $KL\bar{\varphi}_j$ and $\bar{\varphi}_D$ are output in the numeric output file .6. Similarly, if IOPTN(7) $>$ 0, the parameter INUMOPT7 in the configuration file can be used to control the outputs of the field velocity in the numeric output files, .7x, .7y and .7z. The format of the numeric output files is shown in Section 4.9.

4.13 RADIATION PRESSURE AND VELOCITY FOR ZERO AND INFINITE PERIODS

In the two limiting cases of zero and infinite period (or equivalently, infinite and zero frequency), it is possible to evaluate the pressure and velocity on the body (Option 5) and in the fluid (Options 6 and 7). This extension is particularly important in the context of evaluating the corresponding time-domain impulse response functions, as explained in Chapter 13. These extended outputs are only included in the numeric output files if the corresponding parameters INUMOPT5, INUMOPT6, INUMOPT7 are assigned with value 1 in the .cfg file. The formats of the corresponding numeric output files are explained in Section 4.9.

Special definitions are applied to the radiation pressure and velocity in the case of zero frequency, which is identified in the output files by a negative value of the wave period. In general, for nonzero finite values of the frequency, the nondimensional outputs for the radiation pressure and velocity are as defined in Sections 4.5 and 4.7. Thus the output pressure for each radiation mode is $KL\bar{\varphi}_j$ and the output velocity for each mode is $KL\bar{\nabla}\bar{\varphi}_j$. However for the two limiting cases, where $KL = 0$ or $KL = \infty$, the factor KL is omitted from the outputs for options 5,6,7. The following table summarizes these definitions:

frequency	period	pressure	velocity
$\omega = 0$	PER< 0	$\bar{\varphi}_j$	$\bar{\nabla}\bar{\varphi}_j$
$0 < \omega < \infty$	PER$=(2\pi/\omega)$	$KL\bar{\varphi}_j$	$KL\bar{\nabla}\bar{\varphi}_j$
$\omega = \infty$	PER$=0$	$\bar{\varphi}_j$	$\bar{\nabla}\bar{\varphi}_j$

Chapter 5

THE LOW-ORDER METHOD (ILOWHI=0)

This Chapter includes specific topics which are applicable when the low-order method is used, as in earlier versions of WAMIT. The essential features of this method are (a) the geometry of the body is represented by an ensemble of flat quadrilateral panels, or facets, and (b) the solutions for the velocity potential, and optionally for the source strength, are approximated by piecewise constant values on each panel.

The geometry of the body is specified in this case by a Geometric Data File (GDF) which includes the Cartesian coordinates of each vertex of each panel, listed sequentially. In addition the GDF file specifies the characteristic length ULEN used for nondimensionalization of outputs, the value of the gravitational acceleration constant GRAV in the same units of measurement, the number of panels NPAN, and two symmetry indices ISX, ISY, as described in Section 5.1. The syntax for data in this file follows the same requirements outlined for the generic input files in Chapter 3.

When the low-order method is used there are three options which can be used for special purposes. These include the source formulation, analysis of a body near one or two orthogonal vertical walls, and the analysis of bodies with thin elements such as damper plates or strakes.

When the source formulation is used, as described in Section 5.2, the solution is computed both for the velocity potential and for the equivalent source distribution which generates this potential. The source formulation must be used, in the low-order method, if the mean drift force and moment are evaluated by pressure integration, and more generally if the fluid velocity is required on the body surface.

The option to include one or two vertical walls, described in Section 5.3, is based on the method of images. The body geometry is specified in the usual manner by a GDF file, and the program automatically represents the image body (or bodies, if two walls are present).

If the body has thin elements, there are two possible approaches. The first is to panel both sides of these elements, with a finite thickness to separate the two sides. The disadvantage of this approach is that, as a general rule, the size of the panels must be comparable

to the thickness, and thus a very large number of small panels may be required to achieve accurate results. The second approach is to reduce the thickness to zero, and represent the corresponding elements of the body by special 'dipole panels'. This approach is analogous to the thin-wing approximation in lifting-surface theory [21]. Version 6 permits the user to specify a set of dipole panels, as described in Section 5.4. This option facilitates the analysis of bodies with damper plates, strakes, and similar thin elements, without the need to use very large numbers panels or to artificially increase the thickness.

Starting in V6.2, the same extension is made for ILOWHI=1 and the users may specify dipole patches to represent thin structures as described in Section 6.10.

5.1 THE GEOMETRIC DATA FILE

In the low-order method the wetted surface of a body is represented by an ensemble of connected four-sided facets, or panels. The Geometric Data File contains a description of this discretized surface, including the body length scale, gravity, symmetry indices, the total number of panels specified, and for each panel the Cartesian coordinates x, y, z of its four vertices. A panel degenerates to a triangle when the coordinates of two vertices coincide. The order in which the panels are defined in the file is unimportant, but each panel must be described completely by a set of 12 real numbers (three Cartesian coordinates for each vertex) which are listed consecutively, with a line break between the last vertex of each panel and the first vertex of the next. The value of gravity serves to define the units of length, which apply to the body length scale, panel offsets, and to all related parameters in the other input files. The coordinate system x, y, z in which the panels are defined is referred to as the *body coordinate system.* The only restrictions on the body coordinate system are that it is a right-handed Cartesian system and that the $z-$axis is vertical and positive upward.

The name of the GDF file can be any legal filename accepted by the operating system, with a maximum length of 16 ASCII characters, **followed by the extension '.gdf'**.

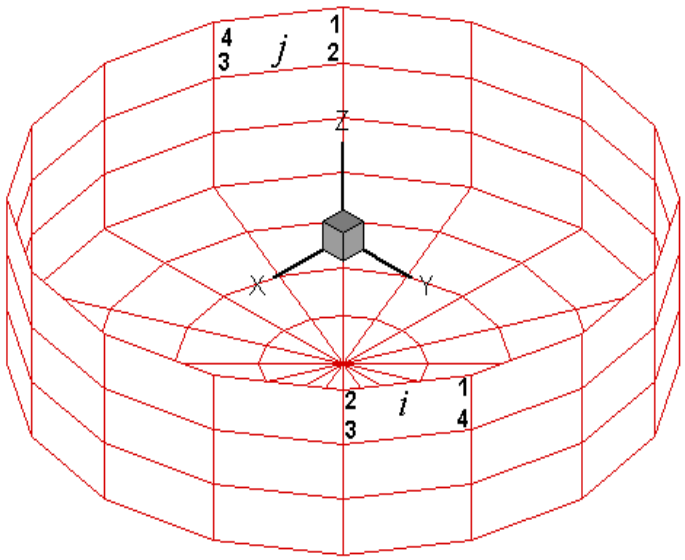


Figure 5.1: Discretization of a circular cylinder showing the convention for panel vertex numbering. The perspective view is from above the free surface, showing portions of the exterior and interior of the cylinder (lower and upper portions of the figure, respectively). The view of panel i is from the 'wet side', *inside* the fluid domain, so the vertex ordering appears *anti-clockwise.* The view of panel j is from the 'dry side' *outside* the fluid domain, so the vertex ordering appears *clockwise.*

The data in the GDF file can be input in the following form:

```
header
ULEN GRAV
ISX ISY
NPAN
X1(1) Y1(1) Z1(1) X2(1) Y2(1) Z2(1) X3(1) Y3(1) Z3(1) X4(1) Y4(1) Z4(1)
X1(2) Y1(2) Z1(2) X2(2) Y2(2) Z2(2) X3(2) Y3(2) Z3(2) X4(2) Y4(2) Z4(2)
.
.
.
.   .   . . . . .   .   . . . . X4(NPAN) Y4(NPAN) Z4(NPAN)
```

Each line of data indicated above is input by a separate FORTRAN READ statement, hence line breaks between data must exist as shown. Additional line breaks between data shown above have no effect on the READ statement, so that for example the user may elect to place the twelve successive coordinates for each panel on four separate lines. (However the format used above is more efficient regarding storage and access time.)

Input data must be in the order shown above, with at least one blank space separating data on the same line.

The definitions of each entry in this file are as follows:

'header' denotes a one-line ASCII header dimensioned CHARACTER∗72. This line is available for the user to insert a brief description of the file, with maximum length 72 characters.

ULEN is the dimensional length characterizing the body dimension. This parameter corresponds to the quantity L used in Chapter 4 to nondimensionalize the quantities output from WAMIT. ULEN can be input in any units of length, meters or feet for example, as long as the length scale of all other inputs is in the same units. ULEN must be a positive number, greater than 10^{-5}. An error return and warning statement are generated if the last restriction is not satisfied.

GRAV is the acceleration of gravity, using the same units of length as in ULEN. *The units of time are always seconds.* If lengths are input in meters or feet, input 9.80665 or 32.174, respectively, for GRAV.

ISX, **ISY** are the geometry symmetry indices which have integer values $0, +1$. (A negative value is assigned when a vertical wall is present. This case is discussed in Section 5.3.) If ISX and/or ISY =1, $x = 0$ and/or $y = 0$ is a geometric plane of symmetry, and the input data (panel vertex coordinates X,Y,Z and their total number NPAN) are restricted to one quadrant or one half of the body, namely the portion $x > 0$ and/or $y > 0$. Conversely, if ISX=0 and ISY=0, the complete submerged surface of the body must be represented by panels.

ISX = 1: The $x = 0$ plane is a geometric plane of symmetry.

ISX = 0: The $x = 0$ plane is not a geometric plane of symmetry.

ISY = 1: The $y = 0$ plane is a geometric plane of symmetry.

ISY = 0: The $y = 0$ plane is not a geometric plane of symmetry.

For all values of ISX and ISY, the (x, y) axes are understood to belong to the body system. The panel data are always referenced with respect to this system, even if walls or other bodies are present.

NPAN is equal to the number of panels with coordinates defined in this file, i.e. the number required to discretize a quarter, half or the whole of the body surface if there exist two, one or no planes of symmetry respectively.

X1(1), Y1(1), Z1(1) are the (x, y, z) coordinates of vertex 1 of the first panel, X2(1), Y2(1), Z2(1) the (x, y, z) coordinates of the vertex 2 of the first panel, and so on. These are expressed in the same units as the length ULEN. The vertices must be numbered in the *counter-clockwise* direction when the panel is viewed from the fluid domain, as shown in Figure 3.1. The precise format of each coordinate is unimportant, as long as there is at least one blank space between coordinates, and the coordinates of the four vertices representing a panel are listed sequentially.

There are two situations when panels lie on the free surface, and thus all four vertices are on the free surface: (1) the discretization of a structure which has zero draft over part or all of its submerged surface, and (2) the discretization of the interior free surface for the irregular frequency removal as described in Chapter 9. For the first case, where the panels are part of the physical surface, the panel vertices must be numbered in the counter-clockwise direction when the panel is viewed from the fluid domain as in the case of submerged panels. For the second case, where the panel is interior to the body and non physical, the vertices must be numbered in the *clockwise* direction when the panel is viewed from inside the structure (or in the *counter-clockwise* direction when the panel is viewed from above the free surface). Details of the discretization of the interior free surface are provided in Chapeter 9.

Although the panels on the free surface are legitimate in these two special cases, a warning message is displayed by WAMIT when it detects panels which have four vertices on the free surface. This is to provide a warning to users for a possible error in the discretization other than the above two exceptional cases. The run continues in this case, without interruption. An error message is displayed with an interruption of the run when the panels have only three vertices on the free surface, unless two adjacent vertices are coincident. (The latter provision permits the analysis of a triangular panel with one side in the free surface.)

The three Cartesian coordinates of four vertices must always be input for each panel, in a sequence of twelve real numbers. Triangles are represented by allowing the coordinates of two adjacent vertices to coincide, as in the center bottom panels shown in Figure 3.1.

Two adjacent vertices are defined to be coincident if their included side has a length less than ULEN $\times 10^{-6}$. An error return results if the computed area of any panel is less than ULEN$^2 \times 10^{-10}$.

The input vertices of a panel do not need to be co-planar. WAMIT internally defines planar panels that are a best fit to four vertices not lying on a plane. However it is advisable to discretize the body so that the input vertices defining each panel lie close to a plane, in order to achieve good accuracy in the computed velocity potentials. An error message is printed if a panel has two intersecting sides. A warning message is printed if a panel is 'convex' (the included angle between two adjacent sides exceeds 180 degrees).

The origin of the body coordinate system may be on, above or below the free surface. The vertical distance of the origin from the free surface is specified in the Potential Control File. The same body-system is also used to define the forces, moments, and body motions. (See Chapter 5 regarding the change in reference of phase relations when walls are present.)

Only the wetted surface of the body should be paneled, and then only half or a quarter of it if there exist one or two planes of symmetry respectively. This also applies to bodies mounted on the sea bottom or on one or two vertical walls. The number of panels NPAN refers to the number used to discretize a quarter, half or the whole body wetted surface if two, one or no planes of symmetry are present respectively.

The displaced volume of the structure deserves particular discussion. Three separate algorithms are used to evaluate this quantity, as explained in Section 4.1. Except for the special case where the structure is bottom-mounted, the three evaluations (VOLX, VOLY, VOLZ) should be identical, but they will generally differ by small amounts due to inaccuracies in machine computation and, more significantly, to approximations in the discretization of the body surface.

For a bottom-mounted structure VOLZ is less than the true volume due to the missing panels in the bottom. In this case a substantial reduction should be observed for VOLZ, but VOLX and VOLY should be nearly equal. For the same reason, substantial differences may occur if a body is mounted in a vertical wall unless the origin of the body coordinates is in the plane of the wall. With these exceptions, substantial differences between the three volumes may indicate errors in the GDF input data.

A unique value of the displaced volume is required in computing the hydrostatic parameters, and in evaluating the body inertia for the motions of the freely-floating body. In these cases the displaced volume of fluid is based on the median (middle value of the three when ranked according to value) of VOLX, VOLY, and VOLZ. A warning message is displayed by WAMIT if the median volume is less than 10^{-30}.

A general-purpose pre-processor has been developed for preparation of GDF files, using the MultiSurf geometric modelling program.[1]

[1] AeroHydro, Inc., 54 Herrick Rd., Southwest Harbor, Maine 04679 USA 207-244-4100 (www.aerohydro.com)

5.2 USE OF THE SOURCE FORMULATION (ISOR=1)

This section describes the evaluation and use of the source strength, in the context of calculating the fluid velocity components on the body and the mean drift force and moment based on pressure integration in uni- and bi-directional waves.

In order to evaluate effectively the tangential components of the fluid velocity on the body (and hence the second-order mean pressure), the solution for the velocity potential based on Green's theorem is augmented if ISOR=1 by the corresponding solution for the source distribution on the body surface.

Setting the parameter ISOR=1 in the POT file or in the CONFIG.WAM file specifies that the source-distribution integral equation is solved in addition to the velocity-potential integral equation. Extended values are assigned to the option switches IOPTN(5), IOPTN(6), IOPTN(7) and IOPTN(9) in the FRC file, to control the evaluation of the fluid velocity on the body and the pressure-integrated drift force/moment, respectively. Explanations of the new output quantities and changes in input are explained in the following section. A brief description of the theory is provided in Section 12.5. (Further details are given in [10] and [26].)

Values of the horizontal drift force and vertical drift moment can be compared with the corresponding outputs evaluated using momentum conservation. In general the results obtained from integration of the second-order pressure will require a finer discretization on the body surface, particularly in the vicinity of sharp corners.

Body symmetries can be exploited to minimize computing time. Special attention must be given to the evaluation of the drift forces, since these are dependent on quadratic products of the first-order solution. For example, if the body has two planes of symmetry the vertical first-order exciting force and heave response can be evaluated simply by setting IRAD=-1, IDIFF=0, MODE(3)=1, and the remaining MODE indices equal to zero. This will not give the correct vertical drift force on the body, however, since the components of the diffraction potential and body motions which are odd functions of x and y have not been evaluated. In general the drift forces should be evaluated only after evaluating all components of the first-order potential, i.e. by setting IDIFF=1 for the stationary body and IRAD=1 and IDIFF=1 for the freely floating body in the POT file. [An example of a valid short cut exists if both the body geometry and the hydrodynamic flow field are symmetrical about a plane of symmetry; then it is not necessary to evaluate first-order potentials which are odd about that plane since these would vanish. For example, if the body is symmetrical about $y = 0$ and the incident-wave heading angle is either zero or 180°, the drift force and moment can be obtained by setting MODE(n)=0 for $n = 2, 4, 6$.]

To calculate the mean drift forces it is necessary to evaluate the runup or, equivalently, the velocity potential at the waterline. Since the program utilizes the velocity potentials at the centroids of the panels adjacent to the waterline for the runup, it is advisable to use panels with small vertical dimensions near the waterline.

In the Force Control File (FRC) extended options are assigned to IOPTN(5), IOPTN(6), IOPTN(7) and IOPTN(9), as follows (See also Section 3.3):

IOPTN(5) is the index used to specify whether the pressure and/or velocity are evaluated on the body boundary:

IOPTN(5) = 0: Do not output either the pressure or velocity.

IOPTN(5) = 1: Output the pressure only.

IOPTN(5) = 2: Output the velocity only (requires ISOR= 1).

IOPTN(5) = 3: Output both the pressure and the velocity (requires ISOR= 1).

IOPTN(6) is the index used to specify whether the pressure in the fluid and/or free surface elevation are evaluated.

IOPTN(6) = 0: do not output pressure and/or free-surface elevation

IOPTN(6) = 1: do output pressure and/or free-surface elevation by the potential formulation

IOPTN(6) = 2: do output pressure and/or free-surface elevation by the source formulation

IOPTN(7) is the index used to specify whether the velocity in the fluid is evaluated.

IOPTN(7) = 0: do not output fluid velocity in the fluid

IOPTN(7) = 1: do output fluid velocity in the fluid by the potential formulation

IOPTN(7) = 2: do output fluid velocity in the fluid by the source formulation

IOPTN(9) is the index used to specify whether the mean drift forces are evaluated from pressure integration:

IOPTN(9) = 0: do not evaluate the mean force and moment

IOPTN(9) = 1: do evaluate the mean force and moment for each wave heading (requires ISOR= 1).

IOPTN(9) = 2: do evaluate the mean force and moment for all combinations of the wave headings (requires ISOR= 1).

The non-dimensional fluid velocity is evaluated in vector form with respect to the global coordinate system at the centroid of each panel. The non-dimensional definition of the fluid velocity is given in Section 4.7.

Three coordinate systems are used in the analysis of the second order mean forces and moments. The first is the global coordinate system (an inertial coordinate system) whose origin should be on the free-surface with the vertical axis positive upward. The phases of time-harmonic quantities in WAMIT are defined with respect to the phase of the incident wave at the origin of the global coordinate system. The second coordinate system is the body-fixed coordinate system. The position and orientation of this coordinate system at rest are defined relative to the global coordinates by the input array XBODY (see Chapter 3). The third coordinate system, introduced to provide additional flexibility in the mean-force analysis, is the inertial coordinate system which coincides with the body-fixed coordinate system at rest. This third system is illustrated in Section 12.5, Figure 12.2.

The non-dimensional definitions of mean drift forces and moments are defined as in Section 4.8. Typical outputs are provided in Test03 described in the Appendix (Section A.3). The following features should be noted in the output:

i) On the line showing the number of iterations required for the solution, the second number inside the parenthesis indicates the maximum number of iterations required for the solution of the linear system for the evaluation of the source strength.

ii) The components of the fluid velocity on the body surface are defined in reference to the global coordinate system.

iii) The three drift force and moment components evaluated by momentum conservation are defined with reference to the axes of the **global coordinate system**.

iv) The six components of the drift force and moment evaluated by pressure integration are defined with reference to the axes of the **body coordinate system at rest** (O of Figure 12.2). Also listed in the OUT file, in the two right columns, are the components of the moment vector defined with respect to the **body-fixed coordinate system** (o of Figure 12.2).

The format of the numerical output files is given in Section 4.9.

5.3 BODY NEAR VERTICAL WALLS

WAMIT includes the option to account for images of the body in the presence of one vertical wall, or two vertical walls which intersect at a right angle.

Figure 5-2 defines two coordinate systems, one fixed on the body and the other on the wall. The axes of the former are denoted by (x, y, z) and those of the latter by (X, Y, Z). In the presence of wall(s), the global coordinate system defined in Section 3.1 must coincide with the wall coordinate system as defined in Figure 5-2. Only one body may be analyzed in the presence of walls (NBODY=1). The drift force and moment cannot be evaluated by momentum integration.

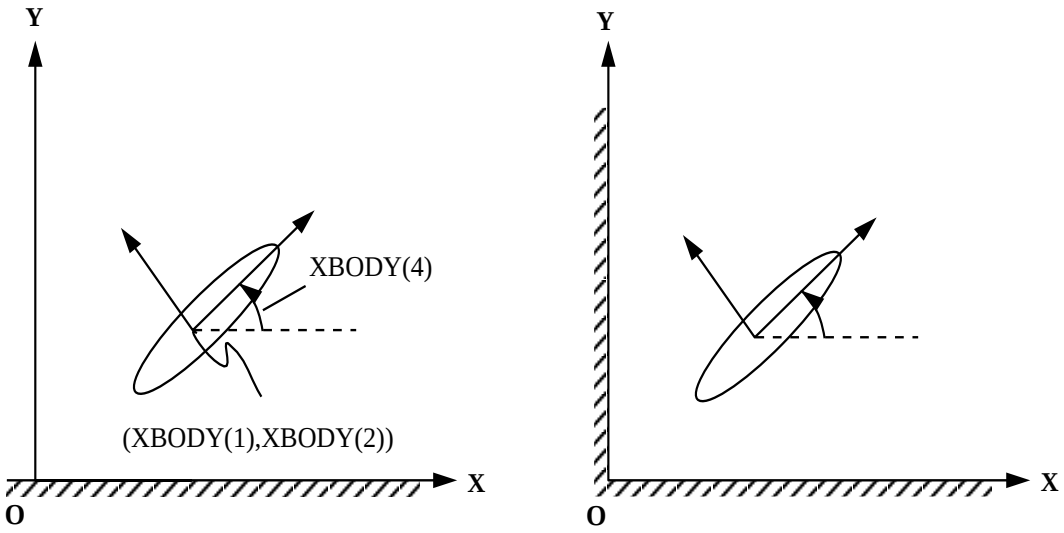


Figure 5.2: Definition sketch of coordinates

In the GDF file the panels are defined as in Section 5.1, in terms of the body coordinate system. Negative values (-1, -2) are assigned to the symmetry indices **ISX, ISY** in place of the conventional corresponding values (0, 1) to indicate the presence of either one or two walls, as follows:

ISX $= -1$ or -2: The $X = 0$ plane is a plane wall.
ISY $= -1$ or -2: The $Y = 0$ plane is a plane wall.

If ISX or ISY is assigned the value -1, no local plane of symmetry is assumed and the body must be represented completely (or half of it must be represented in the case where the complementary plane of symmetry is specified). Conversely, if the body has a local plane of symmetry $x = 0$ and/or $y = 0$, ISX and/or ISY can be assigned the value -2, with panels defined in the GDF file only for the portion of the body where $x \geq 0$ and/or

$y \geq 0$. For example, ISX $= -2$, ISY $= 0$ signify that a wall is present on the $X = 0$ plane and that the body is symmetric about the local $x = 0$ plane. In this case only the half of the body needs to be discretized by the input panels. When ISX $= 1$, ISY $= -2$, a wall is present on the $Y = 0$ plane and the body has two local planes of symmetry.

In the Potential Control File the vector **XBODY(1),XBODY(2),XBODY(3)** specifies the dimensional (X, Y, Z) coordinates of the origin of the body-fixed coordinate system relative to wall system, in the units of the length ULEN. **XBODY(4)** is the angle in degrees formed by the body x-axis and the X-axis of the wall system, as defined in Figure 5-2. The values of the incident wave heading angles **BETA** are defined with respect to the positive X-axis.

An important detail to note is the definition of the incident-wave amplitude and its physical interpretation. The 'incident-wave' is defined as the incoming wave component prior to reflection from the wall(s), and A is the corresponding amplitude. After reflection the resulting wave field, in the absence of the body, is an oblique standing wave with maximum free-surface elevation $2A$. In the special case $\beta = 0$ the incident wave propagates parallel to the wall, without a distinct reflected component, but the physical amplitude of this wave is $2A$. Some consequences of this definition are noted in the comparison of Test Runs 04 and 19 (see Sections A.4 and A.19).

In the Force Control File the array IOPTN is unchanged from the definitions in Section 3.3. Since momentum integration cannot be used to determine the mean drift force and moment, IOPTN(8)=1 is ignored. IOPTN(9) is used in the normal manner to evaluate the drift force and moment from pressure integration. The Haskind wave heading angles **BETAH** are defined with respect to the walls in the same manner as the array BETA above. The coordinates of field points **XFIELD** where the pressure, wave elevation, and velocity are evaluated, are defined as in Section 3.3 relative to the wall-mounted system.

The incident-wave velocity potential is defined relative to the wall-mounted coordinate system. Consequently, the phases of the exciting forces, motions, hydrodynamic pressure and field velocity induced by the incident wave are understood relative to the incident-wave elevation at $X = Y = 0$. In addition the fluid velocity vector components are given with respect to the wall-mounted coordinate system.

The other definitions of output quantities in Chapter 4 are unchanged.

5.4 BODIES WITH THIN SUBMERGED ELEMENTS

In Version 6 an extension has been developed to analyze bodies which consist partially (or completely) of elements with zero thickness. These elements are represented by panels in the same manner as conventional body panels (Section 5.1). Figure 5-3 shows a typical example of a floating spar with thin helical strakes. This structure is analyzed in Test Run 09, described in the Appendix (Section A.9).

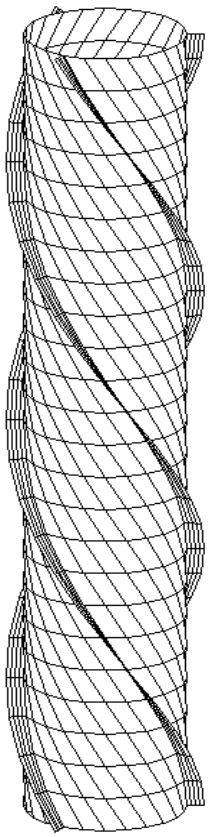


Figure 5.3: Perspective view of a floating spar with three helical strakes, showing the conventional panels on the spar and dipole panels on the strakes.

The velocity potential on the zero-thickness elements is represented by dipoles alone, with no corresponding sources. The panels are referred to as 'dipole panels'. Since both sides of the dipole panels adjoin the fluid, the order of the vertex coordinates shown in Figure 5-1 (clockwise or counter-clockwise) is irrelevant.

In the GDF file the dipole panels must be included separately, after the conventional panels which represent the remainder of the body surface (and also the interior free surface, if IRR=1). The format of the GDF file is as follows:

```
header
ULEN GRAV
ISX ISY
NPANC
X1(1) Y1(1) Z1(1) X2(1) Y2(1) Z2(1) X3(1) Y3(1) Z3(1) X4(1) Y4(1) Z4(1)
X1(2) Y1(2) Z1(2) X2(2) Y2(2) Z2(2) X3(2) Y3(2) Z3(2) X4(2) Y4(2) Z4(2)
.
.
.   .   . . . . .   .   . . . . X4(NPANC) Y4(NPANC) Z4(NPANC)
NPAND
X1(1) Y1(1) Z1(1) X2(1) Y2(1) Z2(1) X3(1) Y3(1) Z3(1) X4(1) Y4(1) Z4(1)
X1(2) Y1(2) Z1(2) X2(2) Y2(2) Z2(2) X3(2) Y3(2) Z3(2) X4(2) Y4(2) Z4(2)
.
.
.   .   . . . . .   .   . . . . X4(NPAND) Y4(NPAND) Z4(NPAND)
```

Here NPANC is the number of conventional panels, NPAND is the number of dipole panels, and NPAN=NPANC+NPAND is the total number of panels in the GDF file. The first part of the GDF file, ending with the line which includes X4(NPANC), is identical to a conventional GDF file for the same body without the dipole panels. The remainder of the file, starting with NPAND, defines the dipole panels which represent the zero-thickness elements.

The same format can be extended to multiple bodies, with dipole panels specified for some or all of the bodies, following the procedure described in Chapter 7.

The source formulation cannot be used if dipole panels are included. Thus the fluid velocity on the body cannot be evaluated, and the mean drift force/moment can only be evaluated by the momentum method. An error message is output and the run is terminated if ISOR=1 with dipole panels specified in the GDF file.

It is possible to analyze bodies which consist entirely of zero-thickness elements, by setting NPANC=0.

A symmetry plane can be used when there are flat thin elements represented by dipole panels on the plane of symmetry. As an example, when a keel on the centerplane $y = 0$ is represented by dipole panels, either the port or starboard side of the vessel can be defined in the GDF file with ISY=1

Chapter 6

THE HIGHER-ORDER METHOD (ILOWHI=1)

The higher-order method which is introduced as an option in Version 6 is fundamentally different from the low-order panel method described in Chapter 5. The body geometry can be represented by different techniques including flat panels, B-spline approximations, geometry models developed in MultiSurf, and explicit analytical formulae. The velocity potential on the body is represented by B-splines in a continuous manner, and the fluid velocity on the body can be evaluated by analytical differentiation. In most applications this provides a more accurate solution, with a smaller number of unknowns, compared to the low-order method.

This higher-order method is developed from the earlier research code HIPAN, which has been described in References [18-20].

A brief outline of the method is provided in Sections 6.1-6.3, to give the necessary background for several input parameters which must be specified. This includes the subdivision of the body surface into patches, the further subdivision of the patches into panels, and the use of B-splines to develop approximations on these surfaces. It is important to note in this context that a panel is not restricted to be a flat quadrilateral in physical space, but can be a general surface in space with continuous curvature to fit the corresponding portion of the body as precisely as is appropriate.

The number of patches NPATCH is specified in the GDF file. Various options exist to specify the other input parameters which determine the number or size of the panels, order of the B-splines, and order of the Gauss quadratures used to integrate over each panel. Section 6.4 describes the data in the Geometric Data File (GDF) which is common to all applications of the higher-order method. Sections 6.5-6.8 describe the four different options for describing the body geometry, and the corresponding inputs. Section 6.9 describes the procedure for modifying the GEOMXACT subroutine to represent the geometry of user-specified bodies.

If the body has thin elements, there are two possible approaches as in the low-order method (see Chapter 5). The first is to represent both sides of these elements with patches.

As a general rule, this approach requires a large number of panel subdivisions of the patches as the thickness of the elements decreases and thus becomes inefficient. The second is to reduce the thickness to zero and represent the elements by special 'dipole patches', analogous to the thin-wing approximation in lifting-surface theory [21]. Version 6.2 permits the user to specify a set of dipole patches, as described in Section 6.10.

Section 6.11 describes the optional Spline Control File (SPL) which can be used to define the orders of the B-Splines, Gauss quadratures, and the numbers of panel subdivisions on each patch.

The maximum size of the panels (measured in dimensional units) can be specified in the configuration file, instead of specifying the number of panels on each patch in the SPL file. This is particularly convenient to achieve a panel size that is commensurate with the body dimensions and wavelength. Default values of the remaining parameters in the SPL file (B-spline and Gauss quadrature orders) are assigned automatically, if not input by the user. Section 6.12 describes this procedure, which permits users to exploit the flexibility and efficiency of the higher-order method with a minimum of inputs.

6.1 SUBDIVISION OF THE BODY SURFACE IN PATCHES AND PANELS

The body surface is first defined by one or more 'patches', each of which is a smooth continuous surface in space. Contiguous patches meet at a common edge, where the coordinates are continuous but the slope may be discontinuous. A simple illustrative example is provided by the circular cylinder of finite draft shown in Figure 6.1. (The same cylinder is shown in Figure 5.1 as it would be represented by low-order panels.) Since there are two planes of geometric symmetry we consider only one quadrant, represented by the shaded portion of Figure 6.1. Two patches are used, one for the flat horizontal bottom and the other for the curved cylindrical side. The important properties of the patches are that (a) the surface is smooth, with continuous coordinates and slope, on each patch, and (b) the ensemble of all patches represents the complete body surface (or one half or quarter of that surface, if one or two planes of symmetry exist).

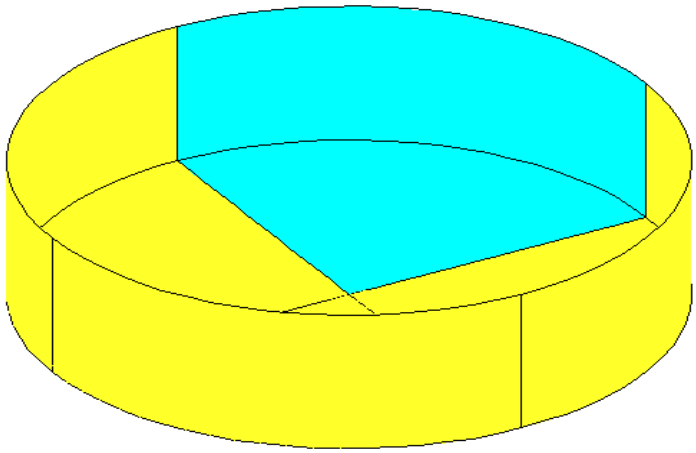


Figure 6.1: Representation of the circular cylinder by two patches on one quadrant, shown by the shaded portion, with reflections about the two planes of symmetry.

On each patch a pair of *parametric coordinates* (u, v) are used to define the position.

The parametric coordinates are normalized so that they vary between ± 1 on the patch. Continuing with the example in Figure 6.1, denoting the cylinder radius R and the draft D and defining conventional circular cylindrical coordinates (r, θ, z), appropriate choices for the parametric coordinates are

$$u = \frac{4\theta}{\pi} - 1, \qquad v = 1 - 2\frac{r}{R} \tag{6.1}$$

on the bottom, where $0 \le \theta \le \pi/2$ and $0 \le r \le R$, and

$$u = \frac{4\theta}{\pi} - 1, \qquad v = -2\frac{z}{D} - 1 \tag{6.2}$$

on the side, where $0 \le \theta \le \pi/2$ and $-D \le z \le 0$. In order to give a consistent definition for the normal vector we impose the right-hand convention: if the fingers of the right hand are directed from $+u$ toward $+v$ the thumb should point *out* of the fluid domain and *into* the interior domain of the body. With these definitions the Cartesian coordinates (x, y, z) of any point on either patch can be expressed in terms of the parametric coordinates (u, v).

More generally, any physically relevant body surface can be represented by an ensemble of appropriate patches, where the Cartesian coordinates of the points on each patch are defined by the *mapping functions*

$$x = X(u, v), \quad y = Y(u, v), \quad z = Z(u, v) \tag{6.3}$$

This is the fundamental manner in which the body surface is represented for the higher-order option of WAMIT. Alternative methods for prescribing these mapping functions are described separately in Sections 6.4-6.7.

In order to provide a systematic procedure for refining the accuracy of approximations on each patch, a set of smaller surface elements are defined, as described in Section 6.2. For this purpose each patch is sub-divided in a rectangular mesh, in parametric space. These elements are referred to as *panels*. Note that while these panels are flat and rectangular in parametric space, they are unrestricted in physical space except for the requirement that they represent a subdivided element of the corresponding patch. Thus, in general, these panels are curved surfaces in physical space. (In some references, such as [22], panels are called 'sub-patches', or simply 'patches'). Figure 6.2 shows the example where the side and bottom of the shaded quadrant in Figure 6.1 are each subdivided into four panels.

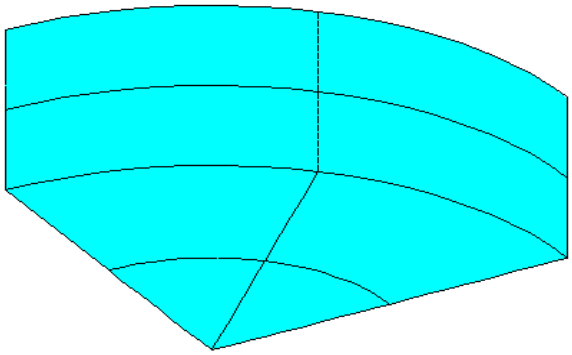


Figure 6.2: Subdivision of one quadrant of the cylinder shown in Figure 6,1 into panels. In this case $N_u = N_v = 2$ on both patches.

6.2 B-SPLINE REPRESENTATION OF THE SOLUTION

The other important subject to consider is the manner in which the velocity potential is represented on each patch. Desirable properties of this representation are that it should be smooth and continuous, corresponding to the physical solution for the fluid flow over the surface, with control over the accuracy. B-splines are used for this purpose. More specifically, the velocity potential is represented by a tensor product of B-spline basis functions

$$\phi(u,v) = \sum_{j=1}^{M_v} \sum_{i=1}^{M_u} \phi_{ij} U_i(u) V_j(v) \tag{6.4}$$

Here $U_i(u)$ and $V_i(v)$ are the B-spline basis functions of u and v, and M_u and M_v are the number of basis functions in u and v, respectively. The unknown coefficients ϕ_{ij} are determined ultimately by substituting this representation in the integral equation for the

potential, as described in Chapter 12. The total number of unknowns on a patch is $M_u \times M_v$.

In the low order panel method, the accuracy of the numerical solution depends on the number of panels. (To a lesser extent, the panel arrangement, such as cosine spacing, may also affect the accuracy of the solution.) In the higher-order method the accuracy depends on two parameters: the *order* of the basis functions and their number M_u and M_v. Order is defined as the degree of the polynomial plus one. For example, a quadratic polynomial $u^2 + au + b$ is of order three. We denote the order of $U(u)$ and $V(v)$ by K_u and K_v, respectively. Further information regarding the B-spline basis functions can be found in Reference [22].

While K_u and K_v are input parameters specified by users, M_u and M_v are not direct input parameters to WAMIT. Instead, users may specify the number of panel subdivisions on each patch, N_u and N_v. (In standard B-spline terminology, these correspond to knots.) Alternatively, users can specify the desired size of each panel in physical space, and the program will automatically assign the corresponding inputs N_u and N_v on each patch to achieve this objective.

The relations between the number of basis functions and the number of panels are as follows:

$$M_u = N_u + K_u - 1 \quad M_v = N_v + K_v - 1 \tag{6.5}$$

Since $K_u = K_v = 1$ in the low-order panel method, the number of unknowns is the same as the number of panels. Chapter 6 of [18] contains examples showing how the accuracy of the solution depends on K and N for various geometries.

6.3 ORDER OF GAUSS QUADRATURES

Another topic which must be considered is the integration over patch surfaces. Since the Galerkin method is used to solve the boundary integral equation, as described in Chapter 12, this integration is carried out first with respect to the source point, and then with respect to the field point. These are referred to respectively as the *inner* and *outer* integrations which are carried out in parametric space. For this purpose, each patch is sub-divided into $N_u \times N_v$ panels, and Gauss-Legendre quadrature is applied on each panel. The orders of the Gauss quadratures are specified by input parameters. Experience with a variety of applications has shown that it is sufficient to set the order of the outer integrals with respect to (u, v) equal to $(K_u,\, K_v)$ and the order of the inner integrals equal to $(K_u + 1,\, K_v + 1)$.

6.4 THE GEOMETRIC DATA FILE

In the higher-order method the first part of the GDF file is as follows:

```
header
ULEN  GRAV
ISX  ISY
NPATCH  IGDEF
```

Subsequent data may be included in the GDF file after these four lines, depending on the manner in which the geometry of the body is represented. (See Sections 6.5-6.8.)

The data on the first three lines are identical to the low-order method as described in Section 5.1. Thus:

'header' denotes a one-line ASCII header dimensioned CHARACTER∗72.

ULEN is the dimensional length characterizing the body dimensions, used to nondimensionalize the quantities output from WAMIT.

GRAV is the acceleration of gravity, using the same units of length as in ULEN.

ISX, **ISY** are the geometry symmetry indices which have integer values $0, +1$ to denote no symmetry, or symmetry about the plane $x = 0$ or $y = 0$ respectively.

The data on line 4 of the GDF file are defined as follows:

NPATCH is equal to the number of patches used to describe the body surface, as explained in Section 6.1. If one or two planes of symmetry are specified, NPATCH is the number of patches required to discretize a half or one quadrant of the whole of the body surface, respectively.

IGDEF is an integer parameter which is used to specify the manner in which the geometry of the body is defined. Four specific cases are relevant, corresponding respectively to the representations explained in Sections 6.5, 6.6, 6.7 and 6.8:

IGDEF = 0: The geometry of each patch is a flat quadrilateral, with vertices listed in the GDF file.

IGDEF = 1: The geometry of each patch is represented by B-splines, with the corresponding data in the GDF file.

IGDEF = 2: The geometry is defined by inputs from a MultiSurf .ms2 file.

IGDEF < 0 or > 2: The geometry of each patch is represented explicitly by a special subroutine, with optional data in the GDF file

6.5 GEOMETRY REPRESENTED BY LOW-ORDER PANELS (IGDEF=0)

The simplest option to define the body geometry is appropriate if each patch of the body surface is a flat quadrilateral in physical space. In this case the vertices of each patch are input via the GDF file in the same format as described in Section 5.1 for the low-order method:

```
header
ULEN  GRAV
ISX  ISY
NPATCH  0
X1(1) Y1(1) Z1(1) X2(1) Y2(1) Z2(1) X3(1) Y3(1) Z3(1) X4(1) Y4(1) Z4(1)
X1(2) Y1(2) Z1(2) X2(2) Y2(2) Z2(2) X3(2) Y3(2) Z3(2) X4(2) Y4(2) Z4(2)
.
.
.
.   .   . . . . .   .   . . . . X4(NPATCH) Y4(NPATCH) Z4(NPATCH)
```

The data in the first four lines are defined above, in Section 6.4. Note that IGDEF=0 is assigned on line 4. The patch vertices ($X1, Y1, Z1, \ldots, X4, Y4, Z4$) are defined in precisely the same manner as the panel vertices in Section 5.1. The convention defined in Figure 5-1 must also be applied here, with the vertices numbered in the anti-clockwise direction when the patch is viewed from the fluid domain.

This option is particularly useful in the case of structures which consist of a small number of flat surfaces. Examples include rectangular barges, similar vessels with rectangular moonpools, the Hibernia platform (a star-shaped bottom-mounted cylinder), etc. In such cases it is not necessary or desirable to use a large number of small patches on each flat surface, as would be necessary to achieve accurate results with the low-order method. The most efficient procedure is to use the smallest number of patches which permits a complete representation of the structure. For a simple rectangular barge, one quadrant can be represented with three patches (bottom, side, end). If a rectangular moonpool is centered amidships, 6 patches are required with two on the bottom and two on the walls of the moonpool.

This option also might be useful to check the accuracy of a low-order application, using the same GDF file for both (except that IGDEF=0 must be assigned for the higher-order input). This provides a more general scheme for improving the accuracy compared to the low-order option IQUAD=1. Two caveats should be noted in this context. First, since each low-order panel is replaced by a patch, the number of patches may be quite large; this will result in substantially longer run times and memory requirements as compared with the low-order method. Secondly, if the flat low-order panels do not correspond exactly to the body surface, this part of the low-order approximation is not refined by such a check.

6.6 GEOMETRY REPRESENTED BY B-SPLINES (IGDEF=1)

The most general approach to represent the geometry in the higher-order method is the same as that which was first developed in [18,19]. In this approach each patch of the body is represented by B-splines, in an analogous manner to the representation of the velocity potential (Section 6.2). The panel subdivision (knot vector) and the order of the B-splines can be assigned independently between the geometry and the potential. If the subdivisions and orders are the same, this is analogous to the isoparametric approach in finite-element analysis. In V6.2, the domain of the parameters of the B-SPLINES representing the geometry is no longer limited to $(-1, 1)$. Arbitrary limits can be used and they are normalized to $(-1, 1)$ in the program.

More specifically, the mapping function $\mathbf{X} = (X, Y, Z)$ defined by Equation (6.3) is represented on each patch in the tensor-product form

$$\mathbf{X}(u, v) = \sum_{j=1}^{M_v^{(g)}} \sum_{i=1}^{M_u^{(g)}} \mathbf{X}_{ij} U_i(u) V_j(v) \tag{6.6}$$

Here $U_i(u)$ and $V_i(v)$ are the B-spline basis functions of u and v, and $M_u^{(g)}$ and $M_v^{(g)}$ are the number of basis functions in u and v, respectively. (The superscripts are used to distinguish these geometric parameters from the corresponding parameters used to represent the potential in Section 6.2.) As in (6.5),

$$M_u^{(g)} = N_u^{(g)} + K_u^{(g)} - 1 \quad M_v^{(g)} = N_v^{(g)} + K_v^{(g)} - 1 \tag{6.7}$$

where $K_u^{(g)}$ and $K_v^{(g)}$ are the orders of the respective B-splines. These parameters, and the values of the unknown coefficients $\mathbf{X}_{ij}$, are assigned for each patch in the GDF file.

The format of the GDF file is as follows:

```
header
ULEN  GRAV
ISX  ISY
NPATCH  1
NUG(1) NVG(1)
KUG(1) KVG(1)
VKNTUG(1,1) ...  VKNTUG(NUA(1),1)
VKNTVG(1,1) ...  VKNTVG(NVA(1),1)
XCOEF(1,1) XCOEF(2,1) XCOEF(3,1)
XCOEF(1,2) XCOEF(2,2) XCOEF(3,2)
.
.
.
XCOEF(1,NB(1)) XCOEF(2,NB(1)) XCOEF(3,NB(1))
.
```

```
.
.
NUG(NPATCH) NVG(NPATCH)
KUG(NPATCH) KVG(NPATCH)
VKNTUG(1,NPATCH) ...  VKNTUG(NUA(NPATCH),NPATCH)
VKNTVG(1,NPATCH) ...  VKNTVG(NVA(NPATCH),NPATCH)
XCOEF(1,1) XCOEF(2,1) XCOEF(3,1)
XCOEF(1,2) XCOEF(2,2) XCOEF(3,2)
.
.
.
XCOEF(1,NB(NPATCH)) XCOEF(2,NB(NPATCH)) XCOEF(3,NB(NPATCH))
```

Here **IGDEF**=1 is assigned on line 4 to specify the B-spline representation of the geometry.

NUG(I) and **NVG(I)** are the numbers of panel subdivisions of the u and v coordinates on I-th patch.

KUG(I) and **KVG(I)** are the orders of B-splines

VKNTUG(J,I) is the B-spline knot vector in u on patch I. J=1,2,...NUA(I)
NUA(I)=NUG(I)+2*KUG(I)-1.

VKNTVG(J,I) is the B-spline knot vector in v on patch I. J=1,2,...NVA(I)
NVA(I)=NVG(I)+2*KVG(I)-1.

XCOEF(1,K)) XCOEF(2,K) XCOEF(3,K) are the components of the vector coefficient $\mathbf{X}_{ij}$ in (6.6). These are defined in terms of the single array index K, where K=1,2,...,NB(I). Here NB(I) is the total number of coefficients on patch I, given by the relation NB(I)=(NUG(I)+KUG(I)-1)×(NVG(I)+KVG(I)-1).

TEST11 (Appendix, Section A.11) is an example of this type of GDF input file.

6.7 GEOMETRY REPRESENTED BY MULTISURF (IGDEF=2)

Version 6.2 includes the option to import .ms2 geometry database files from the CAD program MultiSurf directly into WAMIT, and to represent the geometry during execution of WAMIT by linking to the MultiSurf kernel. A detailed description of this option is contained in Reference 24. The principal advantages of this option are (a) the representation of the geometry can be developed using the CAD environment of MultiSurf, and (b) this representation can be transferred to WAMIT without significant effort or approximations.

Two special .dll files are required: RGKERNEL.DLL and RG2WAMIT.DLL. The 'real' versions of these files are not included in the standard WAMIT license. Users who intend to use this option may license RGKERNEL and RG2WAMIT as part of an extended version of WAMIT, or separately. The standard distribution of WAMIT Version 6.2PC includes a 'dummy' file with the name "rg2wamit.dll". This enables WAMIT to be executed without the 'real' files. As explained in Section 2.1, the PC-executable version of WAMIT (wamit.exe) must be accompanied by four .dll files (geomxact.dll, newmodes.dll, dforrt.dll, rg2wamit.dll). The dummy version of rg2wamit.dll can be distinguished from the real version in two ways: (a) the dummy filename uses lower-case letters (rg2wamit.dll), and (b) the size of this file is smaller, as indicated in the following table:

version	name	size (approximate)
dummy	rg2wamit.dll	24Kb
real	RG2WAMIT.DLL	78Kb

Note that the size of these files is approximate, and may change with updates and subsequent versions, but the disparity in size will serve to distinguish the dummy and real files.

To proceed with this option a user should first prepare the MultiSurf model for the body following the procedure in the MultiSurf documentation. A special appendix 'Using the WAMIT-RGKernel Interface' is included in this User Manual (Appendix C). The output file from MultiSurf will include a filename specified by the user and the extension '.ms2'. This file will be referred to below as 'body.ms2'. If the .ms2 file is missing or cannot be found, a WAMIT runtime error message 'Error return from subroutine RGKINIT' is generated, and the log file 'RGKLOG.TXT' will contain a statement that the designated .ms2 file could not be opened.

In its simplest form, the GDF input file required to run WAMIT should be in the following format:

```
header
ULEN  GRAV
ISX  ISY
NPATCH   2
3
```

```
(path)body.ms2
*
0 0 0
```

The first four lines are explained in Section 6.4. IGDEF=2 is assigned by the second integer on line 4. Line 5 contains an integer specifying the number of subsequent lines to be read from the .gdf file. Line 6 contains the name of the .ms2 file, and may include the optional path if this file is in a different directory (folder). The asterisk (∗) on line 7 is a default specifier to indicate that all visible surfaces in the .ms2 file are to be included; alternatively if only a subset of these surfaces are submerged these may be designated by following the instructions in Appendix C. Line 8 includes three integer parameters with default values zero, which may be used to control the accuracy of the geometry evaluation in RGKernel, and also to modify the convention regarding the direction of the unit normal. Further information is contained in Appendix C. TEST11C and TEST20 in Appendix A are examples showing typical WAMIT runs for a circular cylinder and for a barge. Additional examples are included in Reference 24.

A pre-processor utility GDF2MS2.EXE has been developed by AeroHydro Inc.[1] to convert low-order WAMIT .gdf input files to .ms2 geometry database files for MultiSurf. Its results depend on the organization and content of the .gdf file. In general this utility will create correctly dimensioned points for building a surface model in MultiSurf; and if the .gdf file is suitably structured it is possible to create appropriate surface patches for higher-order analysis with the IGDEF=2 option.

[1] AeroHydro, Inc., 54 Herrick Rd., Southwest Harbor, Maine 04679 USA 207-244-4100 (www.aerohydro.com)

6.8 ANALYTIC REPRESENTATION OF THE GEOMETRY

This option can be used in cases where the geometry of the body can be defined explicitly, with the fundamental advantage that the definition of the body geometry is exact and that the only numerical approximation which remains is in the representation of the velocity potential. Further details and examples based on this method are contained in Reference 25. The domain of parameters must be (-1.,1) in analytic representation.

The formulae required to define the geometry must be coded in FORTRAN, in the file GEOMXACT.F. This file can be compiled separately as a .dll file and linked with WAMIT at runtime. This special arrangement makes it possible for users of the PC executable code to modify GEOMXACT for their own particular applications. Another feature of this option is the possibility to input relevant body dimensions in the GDF file. Thus the body dimensions can be changed without modification of the code.

In the version of GEOMXACT.F and GEOMXACT.DLL as supplied with the WAMIT software, there are several subroutines to produce various generic body shapes as listed in the table below. Most of these subroutines are illustrated in the higher-order test runs described in Appendix A. The dimensions of these generic bodies can be modified by introducing appropriate data in the GDF file. Thus there is a variety of possibilities for exploiting this option with or without special programming efforts. Several different subroutines can be collected in a library, and identified with specific reserved values of the index IGDEF which is input in the GDF file. The WAMIT software includes the FORTRAN library file GEOMXACT.F, where several examples of these subroutines are included. Note that IGDEF=0 or 1 are reserved for the options described in Sections 6.5 and 6.6, and thus IGDEF$\geq$ 2 or IGDEF≤ -1 are appropriate values to select for the analytic representation option. In the WAMIT software package as distributed, several negative values IGDEF≤ -1 have been used for the test runs, and for other pertinent examples which may be useful. Thus it is recommended that any new additions to this library developed by users should be identified with positive values IGDEF$\geq$ 2.

Continuing with the example of the circular cylinder shown in Figures 6.1 and 6.2, the subroutine CIRCYL can be used without modification. CIRCYL is included in the source file GEOMXACT.F and selected by specifying IGDEF=-1. The relevant dimensions are the radius and draft (and also ULEN and GRAV), which are specified in the GDF file in the following format:

```
header
ULEN  GRAV
1  1
2   -1
2
RADIUS  DRAFT
INONUNIF
```

Here the symmetry indices ISX=1 and ISY=1 have been assigned, as well as the parameters NPATCH=2 and IGDEF=-1. The number 2 on line 5 indicates that two lines follow in the file to be read as input data. In addition to the dimensions of the cylinder, the parameter INONUNIF is used in subroutine CIRCYL to specify either uniform (INONUNIF=0) or nonuniform (INONUNIF=1) mapping between the parametric coordinates (U,V) and the Cartesian coordinates (X,Y,Z).

Uniform mapping uses linear functions to transform V to the vertical coordinate on the side, and to the radial coordinate on the bottom (and interior free surface). When the nonuniform mapping option is selected the vertical coordinate on the side is a cubic polynomial in V, and the radial coordinate on the other patches is a quadratic polynomial in V, such that the first derivatives vanish at the corner and at the intersection of the side and free surface. This nonuniform mapping is analogous to the use of 'cosine spacing' in the low-order panel method, to achieve a finer discretization of the solution near these boundaries. The motivation for using the nonuniform mapping option is discussed in Appendix A.11, where both options are compared, and in more detail in Reference 25. The code in the subroutine CIRCCYL may be used as a guide for other geometries where nonuniform mapping is desirable.

Before using this GDF file the user should assign appropriate values for the parameters ULEN, GRAV, RADIUS, DRAFT, INONUNIF, and an appropriate header. As noted in Section 3.7, this data must be contained within columns 1-80 of the GDF file.

In the normal case described above, NPATCH=2, corresponding to the side and bottom of the cylinder. Two other situations exist where the same subroutine can be used: (1) for a bottom-mounted cylinder NPATCH=1 and DRAFT is assigned with the same value as the fluid depth HBOT, and (2) if NPATCH=3 the interior free surface is included to permit the removal of irregular-frequency effects (IRR=1) as described in Chapter 9. The restriction DRAFT<HBOT must be imposed if NPATCH>1. Figure 6.3 illustrates the patch numbering to achieve this flexibility.

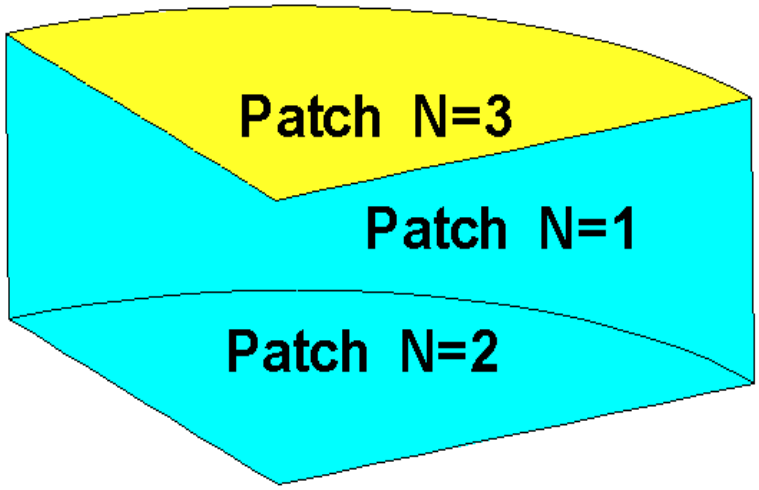


Figure 6.3: One quadrant of the cylinder shown in Figure 6.1 showing the patch numbering system which permits using the subroutine CIRCYL with NPATCH=1 (bottom-mounted caisson), NPATCH=2 (floating cylinder of finite draft), or NPATCH=3 (floating cylinder with a patch on the interior free surface to remove irregular-frequency effects). The view is from above the free surface, looking toward the interior of the cylinder.

Other subroutines are also included in GEOMXACT.F to define a variety of bodies, in all cases with IGDEF<0 so that positive values of IGDEF>2 will be reserved for users. Twelve of the subroutines are listed in the table below and explained in detail here. Several of these are used for the higher-order Test Runs described in Appendix A. Several additional subroutines are also included in GEOMXACT for possible use by users. The details of these other subroutines are explained in comments which are inserted in the source code of each subroutine in the file GEOMXACT.F.

The following table lists the twelve subroutines which are described in more detail below:

IGDEF	SUBROUTINE	NPATCH	GDF INPUTS
-1	CIRCCYL	1,2,3	RADIUS,DRAFT
			INONUMAP
-2	ELLIPCYL	1,2,3	A,B,DRAFT
-3	SPHERE	1,2	RADIUS
-4	ELLIPSOID	1,2	A,B,C
-5	BARGE	2,3,4	HALFLEN,HALFBEAM,DRAFT
-6	BARGEMP	6,7	HALFLEN,HALFBEAM,DRAFT
			XMP,YMP
-7	CYLMP	3,4	RADIUS,DRAFT,RADMP
-8	TORUS	1,2	RCIRC,RAXIS,ZAXIS
-9	TLP	11,12	RADIUS,DRAFT,HSPACE
			WIDTH,HEIGHT
-10	SEMISUB	-	XL,Y1,Y2,Z1,Z2
			DCOL,RCOL,NCOL
-11	FPSO	4,6	XBOW,XMID,XAFT
			HBEAM,HTRANSOM
			DRAFT,DTRANSOM
-12	SPAR		RADIUS DRAFT
			WIDTH THICKNESS TWIST NSTRAKE
			IRRFRQ
			IMOONPOOL, RADIUSMP
			IMPGEN

The last column indicates the dimensions to be included in the GDF file. Where a second line of dimensions is shown in the table (barge with moonpool, TLP, and semisub) the GDF file should follow the same format as illustrated in the test runs TEST14 and TEST15.

Brief descriptions of each subroutine are given below. More specific information is included in the comments of each subroutine. In all cases the subroutine defines one quadrant of a body with two planes of symmetry (ISX=ISY=1). These bodies can be combined for multiple-body analysis, as described in Chapter 7, without modifications of the subroutines.

CIRCCYL defines a circular cylinder as explained above.

ELLIPCYL defines an elliptical cylinder with semi-axes A,B. If A=B=RADIUS the results are identical to using CIRCCYL. The options NPATCH=1 (bottom mounted) and NPATCH=3 (IRR=1) are the same as for CIRCCYL. The semi-axes A and B coincide with the $x-$ and $y-$axis of the body coordinate system, respectively.

SPHERE defines a floating hemisphere, with one patch on the body surface. If NPATCH=2 the interior free surface is included for use with the irregular-frequency option (IRR=1).

ELLIPSOID defines an ellipsoid with semi-axes A,B,C, floating with its center in the plane of the free surface. (C is equal to the draft.) If A=B=C=RADIUS the results are

identical to using SPHERE. The semi-axes A, B and C coincide with the $x-$, $y-$ and $z-$axis of the body coordinate system, respectively.

BARGE defines a rectangular barge with length equal to 2×HALFLEN and beam equal to 2×HALFBEAM. If NPATCH=2 and DRAFT=HBOT the barge is a bottom-mounted rectangular caisson. If NPATCH=4 the interior free surface is included for use with the irregular-frequency option (IRR=1). The longitudinal and transverse directions coincide with the $x-$ and $y-$axis of the body coordinate system, respectively.

BARGEMP defines a rectangular barge with a rectangular moonpool at its center. The moonpool is bounded by vertical walls $x = \pm$XMP and $y = \pm$YMP. Other dimensions are the same as for BARGE. In the normal case, NPATCH=6, separate patches are on the end and side, two patches on the bottom, and two patches for the moonpool walls. Optionally, if NPATCH=7, the moonpool free surface is represented by an additional patch; this is an alternate scheme for the analysis of moonpools, using generalized modes to describe the free surface so that resonant modes can be damped. TEST17B illustrates this scheme. The longitudinal and transverse directions coincide with the $x-$ and $y-$axis of the body coordinate system, respectively.

CYLMP defines a spar-type structure consisting of a circular cylinder with a concentric moonpool of constant radius RADMP. In the normal case, NPATCH=3, separate patches are on the outer side of the cylinder, on the bottom, and on the interior wall of the moonpool. Optionally, if NPATCH=4, the moonpool free surface is represented by an additional patch; this is an alternate scheme for the analysis of moonpools, using generalized modes to describe the free surface so that resonant modes can be damped. TEST17B gives a description of this scheme.

TORUS defines a floating or submerged torus, as illustrated in Figure 6.4. The sections of the torus are circles of radius RCIRC, with their axes on a circle of radius RAXIS in the horizontal plane $z =$ZAXIS. If -RCIRC<ZAZIS<RCIRC the torus is floating, and if ZAXIS<-RAXIS the torus is submerged. One quadrant of the surface is represented by one patch. If the torus is floating, and NPATCH=2, the free surface inside the "moonpool" is represented by an additional patch, as in CYLMP.

TLP defines a generic tension-leg platform (TLP) with four circular columns connected by rectangular pontoons. The bottom surfaces of the columns and pontoons are at the same draft and the columns are equally spaced in a square array. The quadrant is defined to include one column and half of the adjoining pontoons. The column radius RADIUS and draft DRAFT and a half of the horizontal spacing between the axes of adjacent columns HSPACE are specified on one line of the GDF file. The pontoon width WIDTH and height HEIGHT are specified on a separate line. The width of the pontoons is restricted so that they do not intersect off the columns. In the special case WIDTH=RADIUS×$\sqrt{2}$ the pontoon corners coincide on the column and NPATCH=11. This includes eight patches on the top, sides, and bottom of the pontoons, one patch on the column above the pontoons, one patch on the column outside the pontoons, and one patch on the column bottom. In the general case WIDTH<RADIUS×$\sqrt{2}$, NPATCH=12 with the 12th patch on the column between the inside corners of adjacent pontoons. This case is illustrated in the test run TEST14.

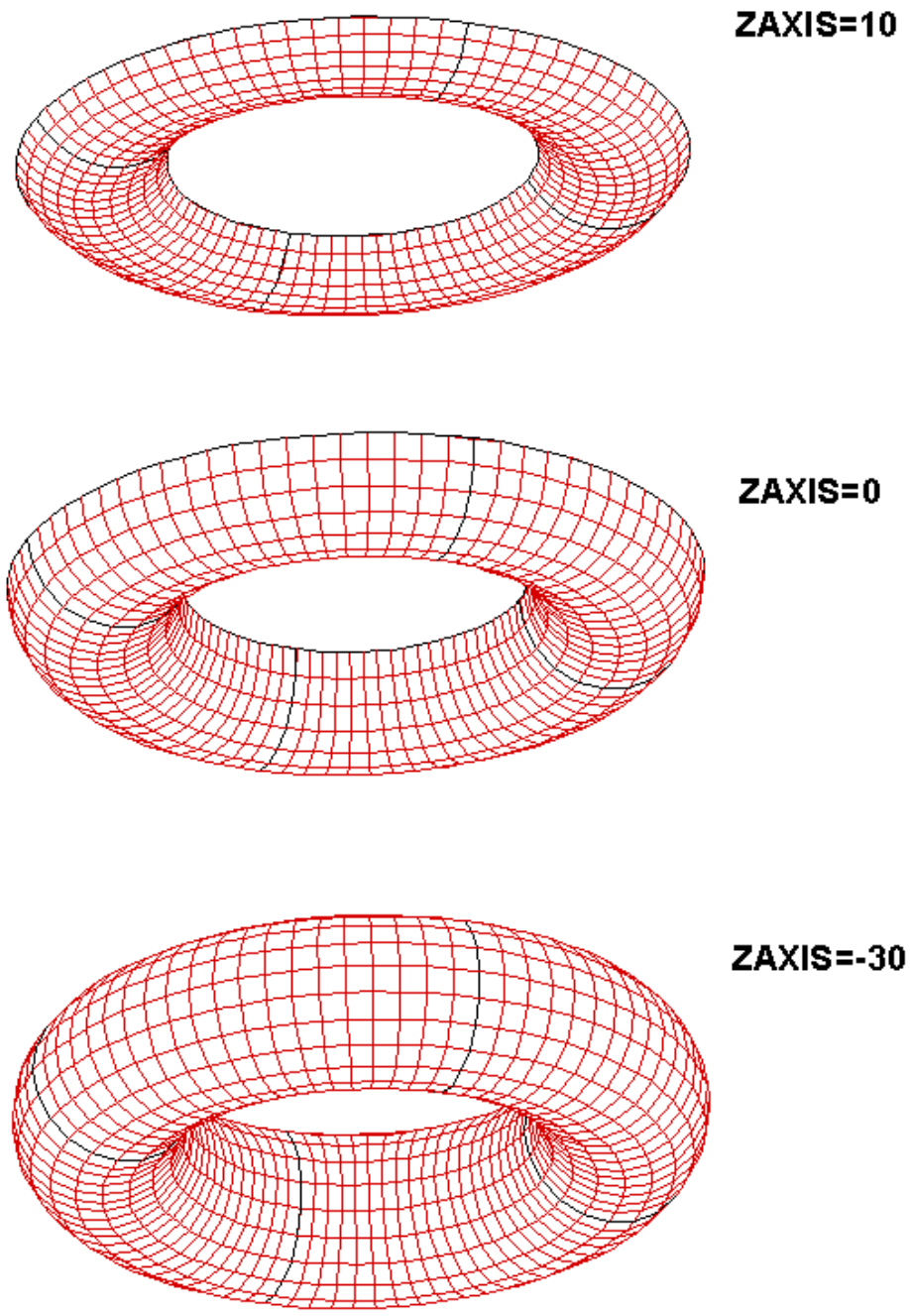


Figure 6.4: Perspective views of the torus, with RCIRC=20, RAXIS=60, and three different values of ZAXIS as shown. ZAXIS>0 in the top figure corresponds to the axis above the free surface. In the middle figure the axis is in the plane of the free surface, and the sections are semi-circles. In these two figures the torus is floating, with the upper edges of the body in the plane of the free surface. The bottom figure shows a complete submerged torus. The dark lines indicate the boundaries between adjacent quadrants, with one patch on each quadrant.

SEMISUB defines a generic semi-submersible with two rectangular pontoons and NCOL equally-spaced circular columns on each pontoon. The pontoon dimensions include the total length XL, transverse coordinates of the inner/outer pontoon sides Y1, Y2, and vertical coordinates of the bottom and top horizontal surfaces Z1, Z2. Note that $0<Y1<Y2$ and $Z1<Z2\leq 0$. (The overall beam is equal to $2\times$Y2 and the draft is equal to -Z1.) The pontoon ends are semi-circular. NPATCH depends on the number of columns, and their spacing, as explained in the subroutine header. If Z2=0 and NPATCH=2 the pontoons intersect the free surface. The test run TEST15 illustrates the use of this subroutine for a semi-sub with submerged pontoons and five columns on each pontoon.

FPSO defines a monohull ship with a form representative of the 'Floating Production Ship Offloading' type. (A perspective view of this vessel is shown on the cover page.) The hull consists of three portions: (1) an elliptical bow with a flat horizontal bottom, vertical sides, and semi-elliptical waterlines, (2) a rectangular mid-body with a flat horizontal bottom, vertical sides, and constant beam, and (3) a prismatic stern with rectangular sections. The dimensions XBOW, XMID, XAFT define the longitudinal extent of these three portions. The total length of the vessel is equal to (XBOW+XMID+XAFT), and the origin of the coordinate system is defined at the midship section, half-way between the bow and stern. The dimensions include the half-beam HBEAM, half-width of the transom HTRANSOM, maximum draft DRAFT, and transom draft DTRANSOM. In the general case NPATCH=6, with the patch indices 1-6 corresponding respectively to (1) the horizontal bottom, (2) the vertical portion of the bow, (3) sides of the mid-body, (4) transom, (5) sloping bottom on the prismatic stern, and (6) sloping side on the prismatic stern. The prismatic stern portion can be omitted by setting NPATCH=4, XAFT=0.0, HTRANSOM=HBEAM, and DTRANSOM=DRAFT.

SPAR defines a spar with strakes and with an optional moonpool. The number of patches varys depending on the optional configuration. RADIUS is the radius of the spar. DRAFT is the vertical length. WIDTH and THICKNESS are the width and thickness of the strakes. Helical form of strakes can be generated by specifying nonzero TWIST which represents the number of revolutions from top to bottom in the counter-clockwise direction viewed from the top. IRRFRQ=1 includes the interior free surface and, in this case, IRR=1 should be specified in the configuration file. IRRFRQ=0 indicates no interior free suface patch. The spar may have a uniform circular moonpool at the center. IMOONPOOL=1 includes a moonpool and IMOONPOOL=0 does not. RADIUSMP is the radius of the moonpool. IMPGEN=1 includes the moonpool free surface to specify the generalized modes on that surface. Otherwise set IMPGEN=0.

6.9 MODIFYING THE DLL SUBROUTINE GEOMXACT

If a body which is not included in the examples above can be described explicitly by analytic formulae (either exactly or to a suitable degree of approximation) a corresponding subroutine can be added to the GEOMXACT.F file. Reference can be made to the source file GEOMXACT.F and to the subroutines already provided to understand the appropriate procedures for developing new subroutines.

Users of WAMIT Version 6PC cannot modify the source code in general. However GEOMXACT has been separated from the rest of the source code, and compiled separately as a dll (dynamic link library) to be linked to the rest of the executable code at run time. Thus users of the PC-executable code can modify or extend GEOMXACT for their own applications.

Source code users can modify GEOMXACT directly, and compile it with any suitable compiler to link with the rest of the program. Special DEC directives in the source code, which stipulate the dll status, should not affect other compilers. (Since the DEC directives are lines of code beginning with `!DEC$`, these should be interpreted as comment lines by other compilers.)

The following points are intended to provide further background information, and should be consulted in conjunction with the code and comments in GEOMXACT.

- The principal inputs are the parametric coordinates u, v, represented in the code by scalars U and V.
- The principal outputs are the Cartesian coordinates $\mathbf{X}$, represented by the array X of dimension 3, and the corresponding derivatives with respect to (U,V) which are represented by the arrays XU, XV with the same dimension.
- These arguments, and all associated dimensions, are of type REAL*4 (single precision).
- In a typical run, GEOMXACT is called a very large number of times. Users modifying this code should ensure that the new code is efficient from the standpoint of CPU time.
- The arrays X,XU,XV are initialized to zero before calls to GEOMXACT. Thus it is only necessary to evaluate nonzero elements in the subroutine.
- Other inputs in the argument list include the body index IBI (to distinguish multiple bodies), the patch index IPI, and the parameter IGDEF, all of type INTEGER.
- To facilitate reading user-specified data in the GDF file, an initial call is made to GEOMXACT with IPI=0 to designate this purpose. If the user intends to read data from the GDF file, appropriate code must be included in the subroutine following the examples which are contained in the original version of GEOMXACT.F as delivered to the user. It is important to use the attribute SAVE for any input data or intermediate data which must be preserved in the subroutine after the initial call.

- Users may place all of their own code in a new subroutine and name it GEOMXACT, or in a subsidiary subroutine called by GEOMXACT. The latter arrangement, which is followed in the GEOMXACT.F file distributed with WAMIT, effectively produces a library of subroutines which can all be accessed by the corresponding values of the parameter IGDEF.

- Some or all of the geometric data may be input in a user-defined file, separate from the .GDF file. In this case standard FORTRAN coding conventions should be followed, with the user's file(s) opened, read, and closed in the initial call to the GEOMXACT subroutine. Unit numbers should be assigned above 300 to avoid conflicts with other open files in WAMIT. The sample DLL file CADDLL.F contains an example of this procedure. In all cases the GDF file must contain at least 6 lines, including the last line 0 (NLINES) if there is no additional data to input from the GDF file.

In order to use GEOMXACT for any of the purposes described in this Chapter, the file GEOMXACT.DLL must be in the same directory as WAMIT.EXE.

Instructions for making new DLL files are included in Section 10.5.

6.10 BODIES WITH THIN SUBMERGED ELEMENTS

In Version 6.2 an extension has been made to analyze bodies which consist partially (or completely) of elements with zero thickness. The patches representing these elements are referred to as 'dipole patches'. Dipole patches are represented in the same manner as the conventional body surface (Section 6.5-6.8). Since both sides of the dipole patches adjoin the fluid, the direction of the normal vector is irrelevant. On the dipole patches, the unknowns are the difference of the velocity potential. The positive difference acts in the direction of the normal vector.

To analyze bodies with zero-thickness elements, the corresponding dipole patches should be identified in the GDF file. Specifically, the number of dipole patches and their indices must be inserted between the 4th and 5th lines of the GDF described in Sections 6.5-6.8.

The format of the GDF file is as follows:

```
header
ULEN GRAV
ISX ISY
NPATCH IGDEF
NPATCH_DIPOLE=NPATCHD
IPATCH_DIPOLE=IPATCHD(1),IPATCHD(2),...,IPATCHD(NPATCHD)
...
```

Here ... on the last line indicates that the parameters are the same as the those in the GDF without dipole patches. NPATCHD is the number of dipole patches and IPATCHD is an array containing the integer indices of the dipole patches. These must be preceded by character strings, 'NPATCH_DIPOLE=' and 'IPATCH_DIPOLE=', respectively.

As an example, the floating spar shown in Figure 5-3 is analysed by the higher-order method in Test Run 21. The total number of patches is seven: three on the side of the cylinder, three on the strakes and one on the bottom of the cylinder. The corresponding patch indices of the patches on the side are 1, 3, and 5, those on the strakes are 2,4, and 6, and that on the bottom is 7.

The mean drift force/moment can only be evaluated by the momentum method. A warning message is output when the option for the pressure integration is specified.

A symmetry plane can be used when there are flat thin elements represented by dipole patches on the plane of symmetry. As an example, when a keel on the centerplane $y = 0$ is represented by dipole patches, either the port or starboard side of the vessel can be defined in the GDF file with ISY=1

6.11 THE OPTIONAL SPLINE CONTROL FILE

The optional Spline Control File (SPL) may be used to control various parameters in the higher-order method. These include the panel subdivision on each patch, the orders of the B-splines used to represent the potential, and the orders of Gauss quadrature used for the inner and outer integrations over each panel. If the SPL file is used it must have the same filename as the corresponding GDF file for the same body, with the extension '.spl'.

The format of the SPL file is as follows:

```
header
NU(1) NV(1)†
KU(1)† KV(1)†
IQUO(1)† IQVO(1)†
IQUI(1)† IQVI(1)†
NU(2) NV(2)†
KU(2)† KV(2)†
IQUO(2)† IQVO(2)†
IQUI(2)† IQVI(2)†
.
.
.
NU(NPATCH) NV(NPATCH)†
KU(NPATCH)† KV(NPATCH)†
IQUO(NPATCH)† IQVO(NPATCH)†
IQUI(NPATCH)† IQVI(NPATCH)†
```

NU and **NV** are the numbers of panels along the u and v coordinates.

KU and **KV** are the orders of B-splines along the u and v coordinates. These parameters should be greater than or equal to 2. Recommended values are given below.

IQUO and **IQVO** are the orders of Gauss quadrature for the outer integration. These parameters should be greater than 1 and ≤ 16. Recommended values are given below.

IQUI and **IQVI** are the orders of Gauss quadrature for the inner integration. These parameters should be greater than 1 and ≤ 16. Recommended values are given below.

NU/NV (marked by †) should not be specified in the SPL file when PANEL_SIZE>0 is assigned in the **CONFIG.WAM** file (See Section 3.9). In that case the program automatically assigns appropriate values to NU and NV on each patch with the objective that the maximum physical length of each panel is equal to PANEL_SIZE. This parameter is specified in the same dimensional units of length as the data in the GDF file. This option is especially convenient for convergence tests, where the size of all panels can be reduced simultaneously.

Similarly, KU/KV, IQUO/IQVO and IQUI/IQVI (marked by †) should not be specified in the SPL file when nonzero values are assigned to **KSPLIN**, **IQUADO** and **IQUADI**,

respectively, in the **CONFIG.WAM** file. (See Section 3.9.) In this case, the program sets KU and KV equal to **KSPLIN**, IQUO and IQVO to **IQUADO**, and IQUI and IQVI to **IQUADI**.

If these parameters are assigned in the SPL file, separate assignments must be made for each patch as indicated in the above format. Conversely, parameters which are assigned in CONFIG.WAM are global, with the same value assigned to all patches and all bodies. Similarly, if KU/KV, IQUO/IQVO or IQUI/IQVI are included in the SPL file, separate values are assigned to the u and v coordinates whereas if these parameters are assigned via global parameters KSPLIN, IQUADO, IQUADI the same values are used for both coordinates.

Experience using the higher-order method indicates that quadratic (KSPLIN=3) or cubic (KSPLIN=4) B-splines are generally appropriate to represent both the geometry and velocity potential, with the former (KSPLIN=3) preferred when the body shape is relatively complex and the latter (KSPLIN=4) when the body is smooth and continuous (e.g. a sphere). Most of the test runs described in the Appendix use KSPLIN=3. Experience also suggests that efficient choices for the inner and outer Gauss integrations are equal to KSPLIN+1 and KSPLIN, respectively. Tests for accuracy and convergence can be achieved most easily and effectively by increasing the numbers of panels, either by increasing NU and NV or by decreasing the parameter PANEL_SIZE. This procedure permits systematic convergence tests to be made easily and efficiently, without simultaneously changing the other parameters or inputs.

6.12 THE USE OF DEFAULT VALUES TO SIMPLIFY INPUTS

Experience with the higher-order method indicates that for typical applications the global parameters defined above may be assigned the values KSPLIN=3, IQUADO=3, IQUADI=4. These default values are assigned by the program automatically, if they are not assigned in the CONFIG.WAM file and if there is no SPL input file available to open and read with the same filename as the GDF file. In the latter case, however, the parameter PANEL_SIZE must be specified with a nonzero positive value in CONFIG.WAM. This is the simplest way to use the higher-order method since it does not require the user to input the B-spline and Gauss quadrature orders either locally in the SPL file or globally in the CONFIG.WAM file.

The following table summarizes the options for inputting these parameters:

gdf.spl	config.wam	NONE
NU,NV	PANEL_SIZE	error
KU,KV	KSPLIN	3
IQUO,IQVO	IQUADO	3
IQUI,IQVI	IQUADI	4

Here the first column indicates inputs in the optional SPL file and the second column indicates the corresponding inputs in the CONFIG.WAM file. The third column indicates the default values which are set if there is no SPL file and if the parameters are not included in the CONFIG.WAM file. **It is important not to specify the same parameters in both the SPL and CONFIG.WAM files, since this will cause errors reading the data in the SPL file.**

In summary, the simplest way to use the higher-order method is to specify PANEL_SIZE only, in the CONFIG.WAM file, and ignore all of the other parameters shown in this table.

The values of these parameters are displayed for each patch in the header of the .out file. When the parameter PANEL_SIZE is used, its value is also displayed on the line indicating that the higher-order method is used.

6.13 ADVANTAGES AND DISADVANTAGES OF THE HIGHER-ORDER METHOD

Some advantages and disadvantages of the higher-order method in comparison of the low-order method are listed below.

Advantages:

1. The higher-order method is more efficient and accurate in most cases. More precisely, the higher-order method converges faster than the low-order method, when the number of panels is increased in both. (Comparisons for various geometries can be found in [18,19]). Thus accurate solutions can be obtained more efficiently with the higher-order method.
2. Various forms of geometric input are possible, including the explicit representation. When it is possible to use this approach it is relatively simple to input the geometry and modify its dimensions for each run.
3. The pressure and velocity on the body surface are continuous. Continuity of the hydrodynamic pressure distribution is particularly useful for the analysis of structural loads.

Disadvantages:

1. The linear system which must be solved for the velocity potentials is not as well conditioned in the low-order method. Thus the iterative method for the solution of the linear system fails to converge in many cases. The direct or block-iterative solution options are recommended in these cases. Since the size of the linear system (number of unknowns) is significantly smaller than for the low-order method, this generally does not impose a substantial computational burden.
2. The second-order pressure due to the square of the fluid velocity is unbounded at sharp corners. The approximation of this pressure by higher-order basis functions is more difficult than in the low-order method. The result may be less accurate unless the mapping accounts for the flow singularity near the corner.

Chapter 7

ANALYSIS OF MULTIPLE BODIES (NBODY>1)

WAMIT includes the capability to analyze multiple bodies which interact hydrodynamically and mechanically. Each of the separate bodies may oscillate independently with up to six degrees of freedom. (Additional generalized modes can be defined for each body, as described in Chapter 8.) The bodies may be freely floating, fixed, or constrained by external forces. The basic theory for multibody interactions with waves is similar to that of a single body as described in Chapter 12. The principal extension is to increase the maximum number of degrees of freedom from 6 for a single body, to $6N$ for N bodies (N is hereafter used to denote the number of bodies and the index $K = 1, 2, ..., N$ is used to denote each of the N bodies.). For example, when two bodies are present the maximum possible number of degrees of freedom becomes 12, 6 for each body. In this example, modes $7, 8, 9$ represent translation of body 2 in the direction of the x,y and z-axes of the coordinate system fixed on that body. These modes correspond to surge, sway and heave for the second body, respectively. Modes $10, 11, 12$ represent rotation about the same axes. The extension of this convention to more bodies is evident. Thus, some output quantities are given in vector or matrix form with dimensions $6N$ or $(6N) \times (6N)$, respectively.

Separate GDF files must be input for each body. The specification of the GDF filenames and coordinates of each body is different in the two alternative POT file formats (Sections 3.1 and 3.2). Further details are included in the corresponding sections below. The individual GDF files for each body are unchanged from the case where $N = 1$, thus GDF files can be combined without modification to analyze multiple-body configurations. However the units of measurement, identified by the parameter GRAV, must be the same for all bodies. The program assigns GRAV based on the value in the GDF file for the first body (K=1). If the value of GRAV for another body differs from this by more than a small tolerance (GTOL=0.1) the run is terminated with an appropriate error message.

WAMIT assumes that there are no planes of hydrodynamic symmetry when $N > 1$. If geometric symmetry is specified for individual bodies, via their respective GDF input files, the program reflects about the corresponding planes and increases the number of panels accordingly. The total number of unknowns is the sum of the number required to

describe each body, including reflections. Thus the run times and memory requirements are substantially increased.

Walls are not allowed when $N > 1$.

The multiple-body extensions are essentially the same for analyses based on the low-order and higher-order methods. However it is impossible to use both methods simultaneously for different bodies. If NBODY>1 and ILOWHI=1 the option IALTPOT=2 must be used.

The GEOMXACT subroutines described in Section 6.8 can be used for multiple bodies, but special attention is required unless all of the bodies using the same subroutine have identical dimensions. Most of the existing GEOMXACT subroutines read in the appropriate dimensions from the GDF file in scalar form, to initialize parameters within the subroutine for subsequent calls. If the same subroutine is initialized again for another body, the dimensions are overwritten. As a result the dimensions in the last GDF file are applied to all of the bodies using the same subroutine. In cases where bodies with different dimensions are represented by the same subroutine, the dimensions used in subsequent calls should be saved within the subroutine as arrays with dimension NBODY. The subroutine FPSO12 in GEOMXACT illustrates this procedure for two FPSO hulls with different dimensions. To call attention to this problem, a warning message is issued if two or more bodies use the same value of IGDEF in GEOMXACT.

There are three alternative ways to input parameters to the FORCE subprogram. Alternative Form 1 can be used for freely floating bodies, and the more general Alternative 2 Form can be used for bodies subject to external forces. Alternative 3 includes a Global Force Control file (GFRC) and a separate FRC file for each body, each being in either Form 1 or 2. These three alternative are described respectively in Sections 7.3, 7.4, and 7.5.

7.1 INPUT TO POTEN (IALTPOT=±1)

If IALTPOT=1 is specified in the configuration file, or if this parameter is not specified and is set by default, a Global Geometric Data File (GGDF) must be input. The name of this GGDF file is specified in the FNAMES.WAM file, or interactively if no GDF filename is included in FNAMES.WAM. The filenames of each individual body are input in the GGDF file. This option can only be used for the low-order method (ILOWHI=0).

If IALTPOT=1 the data in the Global Geometric Data File (GGDF) are listed below:

```
header
-1.0 GRAV
NBODY
GDF(1)
XBODY(1,1) XBODY(2,1) XBODY(3,1) XBODY(4,1)
MODE(1,1) MODE(2,1) MODE(3,1) MODE(4,1) MODE(5,1) MODE(6,1)
.
.
.
GDF(N)
XBODY(1,N) XBODY(2,N) XBODY(3,N) XBODY(4,N)
MODE(1,N) MODE(2,N) MODE(3,N) MODE(4,N) MODE(5,N) MODE(6,N)
```

If IALTPOT=-1, corresponding to earlier versions of WAMIT, the data in the Global Geometric Data File (GGDF) are listed below:

```
header
-1.0 GRAV
NBODY
GDF(1)
XBODY(1,1) XBODY(2,1) XBODY(3,1) XBODY(4,1)
IRAD(1) IDIFF(1)
MODE(1,1) MODE(2,1) MODE(3,1) MODE(4,1) MODE(5,1) MODE(6,1)
.
.
.
GDF(N)
XBODY(1,N) XBODY(2,N) XBODY(3,N) XBODY(4,N)
IRAD(N) IDIFF(N)
MODE(1,N) MODE(2,N) MODE(3,N) MODE(4,N) MODE(5,N) MODE(6,N)
```

The only difference between these two formats is the inclusion of separate values IRAD,IDIFF for each body when IALTPOT=-1. The latter is generally not necessary, and is considered redundant with the assignment of IRAD and IDIFF in the POT file. However users of earlier versions who wish to do so may continue to use old GGDF files by assigning

IALTPOT=-1. This option is deprecated and may not be supported in later versions of WAMIT.

'header' denotes a one-line ASCII header dimensioned CHARACTER∗72.

-1.0 is the flag which indicates that the file is a Global GDF file. (Any real number less than or equal to zero is acceptable.) (Note that in an ordinary GDF file this parameter is ULEN, which must be positive.)

GRAV is the acceleration of gravity, using the same units of length as in ULEN(K) of the GDF files. If lengths are input in meters or feet, input 9.80665 or 32.174, respectively for GRAV. (The same units of length must be used in each GDF file if $N > 1$.)

NBODY is the total number of bodies.

GDF(K) is the name of the K-th Geometric Data File.

XBODY(1,K),XBODY(2,K),XBODY(3,K) are the (X, Y, Z) coordinates of the origin of the body-fixed coordinate system of the K-th body, relative to the *global coordinate system*, input in the dimensional units of the length ULEN(K). The global coordinate system is used in place of the body coordinate system to define field-point data (fluid pressures, velocities, and free-surface elevation). The origin of the global coordinate system must be located on the free surface.

XBODY(4,K) is the angle in degrees between the $x-$axis of the body coordinate system of the K-th body and the $X-$axis of the global system (see Figure 5.2). The direction of the global Z-axis, and of the z-coordinate in each body coordinate system, must be positive upward and perpendicular to the undisturbed free surface.

IRAD(K), IDIFF(K) are indices used to specify the components of the radiation and diffraction problems to be solved for the K-th body. The following options are available, depending on the values of IRAD(K) and IDIFF(K):

IRAD(K)= 1: Solve for the radiation velocity potentials due to all six rigid-body modes of motion of the K-th body.

IRAD(K)= 0: Solve the radiation problem only for those modes of motion specified by setting the elements of the array MODE(I,K)=1 (see below).

IRAD(K)= -1: Do not solve any component of the radiation problem.

IDIFF(K)= 1: Solve for all diffraction components, i.e. the complete diffraction problem.

IDIFF(K)= 0: Output only the exciting forces in the modes specified by MODE(I,K)=1.

IDIFF(K)= -1: Do not solve the diffraction problem.

MODE(I,K) is a six-element array of indices for the K-th body, where I=1,2,3 correspond to the surge, sway and heave translational modes along the body-fixed (x, y, z) axes, and I=4,5,6 to the roll, pitch and yaw rotational modes about the same axes, respectively. Each of these six indices should be set equal to 0 or 1, depending on whether the corresponding radiation mode(s) and diffraction component(s) are required. (See the options IRAD(K)=0 and IDIFF(K)=0 above.)

The data in the GDF file for each individual body is as described in Section 3.1. Panel coordinates and symmetry indices are defined in terms of the local body coordinates of

the respective body, as for a single-body application. For each body there is a respective dimensional length ULEN(K), and a corresponding value of gravity GRAV(K). ULEN(K) must be input in the same dimensional units for every body. ULEN(K) must be a positive number greater than 10^{-5}. An error return and warning statements are generated if the last restriction is not satisfied. ULEN(1), the characteristic length of the first body, is used to nondimensionalize the outputs as described in Chapter 4.

The values GRAV(K) in all GDF files are read, and used to check for consistency of units among the different bodies. An error return occurs if any GRAV(K) differs from GRAV(1) by more than 0.1 units in absolute value (0.1 m/s^2 or 0.1 ft/s^2 if these units are used.) (The value of GRAV(1) is used for all calculations and outputs.) The Potential Control File (POT) is described in Section 3.1, and is unchanged when $N > 1$. However the values of the array XBODY in the POT file (or in the optional configuration file), and array MODE which are specified in the POT file are all ignored, since these are superseded by the individual values for each body in the GGDF file. (If IALTPOT=-1, the same statement applies to IRAD, IDIFF.)

7.2 INPUT TO POTEN (IALTPOT=2)

If IALTPOT=2 is assigned in the configuration file, the POT file is as described in Section 3.2. This Alternative Form of the POT file includes all of the relevant data for each body as is described above for the GGDF file when IALTPOT=1. (This is considered to be a more efficient scheme to adopt for multiple-body analyses.)

To summarize, when NBODY>1 and IALTPOT=2, the POT file should include the applicable value of NBODY, followed by appropriate data for each body, as shown in Section 3.2. A separate GDF file must be input for each body. No other changes are required in the input files.

7.3 INPUT TO FORCE (IALTFRC=1)

If IALTFRC=1 the format of the FRC file is as shown below:

```
header
IOPTN(1) IOPTN(2) IOPTN(3) IOPTN(4) IOPTN(5) IOPTN(6) IOPTN(7) IOPTN(8) IOPTN(9)
VCG(1)
XPRDCT(1,1,1) XPRDCT(1,2,1) XPRDCT(1,3,1)
XPRDCT(2,1,1) XPRDCT(2,2,1) XPRDCT(2,3,1)
XPRDCT(3,1,1) XPRDCT(3,2,1) XPRDCT(3,3,1)
VCG(2)
XPRDCT(1,1,2) XPRDCT(1,2,2) XPRDCT(1,3,2)
XPRDCT(2,1,2) XPRDCT(2,2,2) XPRDCT(2,3,2)
XPRDCT(3,1,2) XPRDCT(3,2,2) XPRDCT(3,3,2)
.
.
VCG(N)
XPRDCT(1,1,N) XPRDCT(1,2,N) XPRDCT(1,3,N)
XPRDCT(2,1,N) XPRDCT(2,2,N) XPRDCT(2,3,N)
XPRDCT(3,1,N) XPRDCT(3,2,N) XPRDCT(3,3,N)
NBETAH
BETAH(1) BETAH(2) ...  BETAH(NBETAH)
NFIELD
XFIELD(1,1) XFIELD(2,1) XFIELD(3,1)
XFIELD(1,2) XFIELD(2,2) XFIELD(3,2)
XFIELD(1,3) XFIELD(2,3) XFIELD(3,3)
.
.
XFIELD(1,NFIELD) XFIELD(2,NFIELD) XFIELD(3,NFIELD)
```

The only difference relative to the case of a single body (Section 3.3), is that the VCG and 3×3 matrix of each body's radii of gyration are entered in succession.

7.4 INPUT TO FORCE (IALTFRC=2)

If IALTFRC=2 the format of the FRC file is the same as described in Section 3.4 for a single body, except that the array specifying (XCG,YCG,ZCG) is extended to include all bodies, and the external force matrices have dimensions NDFR × NDFR. NDFR= $\sum_{n=1}^{N}(6+\text{NEWMDS}(n))$ is the total number of degrees of freedom including all rigid body modes and generalized modes. The normal format is as follows:

```
header
IOPTN(1) IOPTN(2) IOPTN(3) IOPTN(4) IOPTN(5) IOPTN(6) IOPTN(7) IOPTN(8) IOPTN(9)
RHO
XCG(1) YCG(1) ZCG(1) XCG(2) YCG(2) ZCG(2) ...
...  XCG(N) YCG(N) ZCG(N)
IMASS
EXMASS(1,1) EXMASS(1,2) ...  EXMASS(1,NDFR)
EXMASS(2,1) EXMASS(2,2) ...  EXMASS(2,NDFR)
.
.
EXMASS(NDFR,1) EXMASS(NDFR,2) ...  EXMASS(NDFR,NDFR)
IDAMP
EXDAMP(1,1) EXDAMP(1,2) ...  EXDAMP(1,NDFR)
EXDAMP(2,1) EXDAMP(2,2) ...  EXDAMP(2,NDFR)
.
.
EXDAMP(NDFR,1) EXDAMP(NDFR,2) ...  EXDAMP(NDFR,NDFR)
ISTIF
EXSTIF(1,1) EXSTIF(1,2) ...  EXSTIF(1,NDFR)
EXSTIF(2,1) EXSTIF(2,2) ...  EXSTIF(2,NDFR)
.
.
EXSTIF(NDFR,1) EXSTIF(NDFR,2) ...  EXSTIF(NDFR,NDFR)
NBETAH
BETAH(1) BETAH(2) ...  BETAH(NBETAH)
NFIELD
XFIELD(1,1) XFIELD(2,1) XFIELD(3,1)
XFIELD(1,2) XFIELD(2,2) XFIELD(3,2)
XFIELD(1,3) XFIELD(2,3) XFIELD(3,3)
.
.
XFIELD(1,NFIELD) XFIELD(2,NFIELD) XFIELD(3,NFIELD)
```

As in Section 3.4, the integers (IMASS,IDAMP,ISTIF) are set equal to one if the matrix follows, and equal to zero if no corresponding external matrix is included in the file. Omitting the matrix is equivalent to including the matrix with zero values for all elements.

The same format can be used with the external force matrices in separate files and with the corresponding filenames replacing the matrices in the FRC file. This option is specified by the values (IMASS,IDAMP,ISTIF)=2:

```
header
IOPTN(1) IOPTN(2) IOPTN(3) IOPTN(4) IOPTN(5) IOPTN(6) IOPTN(7) IOPTN(8) IOPTN(9)
RHO
XCG(1) YCG(1) ZCG(1) XCG(2) YCG(2) ZCG(2) ...
...  XCG(N) YCG(N) ZCG(N)
2
MASS (filename containing inertia matrix)
2
DAMP (filename containing damping matrix)
2
STIF (filename containing stiffness matrix)
NBETAH
BETAH(1) BETAH(2) ...  BETAH(NBETAH)
NFIELD
XFIELD(1,1) XFIELD(2,1) XFIELD(3,1)
XFIELD(1,2) XFIELD(2,2) XFIELD(3,2)
XFIELD(1,3) XFIELD(2,3) XFIELD(3,3)
.
.
XFIELD(1,NFIELD) XFIELD(2,NFIELD) XFIELD(3,NFIELD)
```

The separate external force data files MASS, DAMP, STIF contain a one-line header plus the three corresponding matrices shown in the first format.

As in Section 3.4, it is also possible in Version 6 to specify the Alternative Form 2 by inserting the integer 2 in the second line, but that option is deprecated.

The first line of this file, and all lines beginning with the variable NBETAH, are identical to the data in Alternative 1 FRC file. The data which differ in Alternative 2 are described in Section 3.4.

7.5 INPUT TO FORCE (IALTFRC=3)

Alternative Form 3 includes one Global FRC file (GFRC) and N FRC files. The FRC file for each body can take either the form of Alternative 1 or Alternative 2. With this option existing FRC files for single bodies can be used without modification. Note however that this precludes the consideration of external mass, damping and stiffness forces which produce coupling interactions between the bodies.

If IALTFRC=3 the input parameters in the GFRC file are listed below:

```
header
IOPTN(1) IOPTN(2) IOPTN(3) IOPTN(4) IOPTN(5) IOPTN(6) IOPTN(7) IOPTN(8) IOPTN(9)
RHO
FRC(1)
FRC(2)
.
.
FRC(N)
NBETAH
BETAH(1) BETAH(2) ...  BETAH(NBETAH)
NFIELD
XFIELD(1,1) XFIELD(2,1) XFIELD(3,1)
XFIELD(1,2) XFIELD(2,2) XFIELD(3,2)
XFIELD(1,3) XFIELD(2,3) XFIELD(3,3)
.
.
XFIELD(1,NFIELD) XFIELD(2,NFIELD) XFIELD(3,NFIELD)
```

The first three lines of this file, and all lines beginning with the variable NBETAH, are identical to the data in the Alternative Form 2 FRC file.

FRC(K) is the name of the FRC file for body K. The Form of each separate file must be 1 or 2, and this is specified by the optional array IALTFRCN in the configuration file, as described in the following section.

Some of the data given in N FRC files are read but neglected, if the same data is given in the GFRC file. For example, the data IOPTN and RHO in the FRC files for each body are neglected and the corresponding parameters provided by the GFRC file are used.

As with IALTFRC=2, it is possible to specify IALTFRC=3 by inserting the integer 3 on line 2 of this file, but this practice is deprecated in Version 6 and may not be supported in later versions.

7.6 PARAMETERS IN THE CONFIGURATION FILE

The configuration file described in Section 3.9 includes a few inputs that are relevant to the analysis of multiple bodies. These include NBODY, XBODY, NEWMDS, and IALTFRCN. If IALTFRC=3 the array IALTFRCN must be specified in configuration, unless the default values 1 are applicable for all bodies. The parameters NBODY, XBODY, and NEWMDS can be specified either in configuration or in other input files, as indicated in Section 3.9. A typical configuration file used for all of these parameters is as follows, with explanatory comments on each line:

```
IALTFRC=3    (use FRC format in Section 7.5)
IALTFRCN=2 1 2  (Form of FRC file for each body)
IALTPOT=1    (use POT format in Section 3.1)
IDEFINE=1   (run DEFINE subroutine in .dll file for generalized modes)
ILOWHI=0    (use low-order method)
IPERIO=1    (POT file contains wave periods in seconds)
NEWMDS(1)=0  (Body 1: No generalized modes)
NEWMDS(2)=4  (Body 2: 4 generalized modes)
NEWMDS(3)=0  (Body 3: No generalized modes)
XBODY(1) = -5.0 0.0 0.0 0.0  (Body 1 coordinates)
XBODY(2) = 0.0 0.0 0.0 0.0  (Body 2 coordinates)
XBODY(3) = 5.0 0.0 0.0 90.0  (Body 3 coordinates)
```

7.7 OUTPUT

The nondimensionalizations given in Chapter 4 hold for all output quantities. L=ULEN(1), the dimensional length for Body 1, is the characteristic length and is used for the nondimensionalization of the output quantities.

The added mass (A_{ij}), damping coefficients (B_{ij}) and hydrostatic coefficients($C(i,j)$) are matrices of dimension up to $6N \times 6N$. These quantities are defined in the direction of the axes of the corresponding body coordinate systems. For example $A_{1,9}$ is the added mass in the direction of the x-axis of the coordinate system of Body 1 (surge added mass) due to the motion of the Body 2 in the direction of the z-axis of Body 2 (heave motion).

The forces (X_i) and the motion amplitudes (ξ_i) are vectors of dimension up to $6N$. These quantities are also defined in the direction of the axes of the coordinates system of the corresponding body. For example, X_{17} is the pitch exciting moment about the y-axis of Body 3. The **phases** of the forces and motion amplitudes and of the field quantities such as the field pressure and field velocity, are defined relative to the phase of the incident wave at the origin of the global coordinate system.

The pressure drift force and moment (Option 9) returns values for each body in its respective body coordinate system.

When Option 8 is specified (momentum drift force and moment) the quantities calculated are the global horizontal drift forces and mean yaw moment acting on the entire ensemble of bodies. It is possible to compare these outputs with the total drift force and moment from pressure integration, by summing the latter outputs for the forces and moments on each body. This provides a useful check on consistency. Special attention is required if the body coordinates are not parallel to the global coordinates system.

Chapter 8

GENERALIZED BODY MODES (NEWMDS>0)

WAMIT includes the capability to analyze generalized modes of body motion, which extend beyond the normal six degrees of rigid-body translation and rotation. These generalized modes can be defined by the user to describe structural deformations, motions of hinged bodies, and a variety of other modes of motion which can be represented by specified distributions of normal velocity on the body surface. To simplify the discussion it will be assumed that only one body is analyzed, i.e. NBODY=1. The analysis of multiple bodies with generalized modes is discussed below in Section 8.5.

Each generalized mode is defined by specifying the normal velocity in the form

$$\varphi_{jn} = n_j = u_j n_x + v_j n_y + w_j n_z \tag{8.1}$$

where $j > 6$ is the index of the mode. (The first six indices $j = 1, 2, ..., 6$ are reserved for the conventional rigid-body modes.) The displacement vector (u_j, v_j, w_j) is defined by the user in a special subroutine which can be accessed and modified by the user. The displacement vector can be any physically relevant real function of the body coordinates (x, y, z) which can be defined by FORTRAN code.

Corresponding to these modes are the generalized hydrodynamic force components, which are defined as in Sections 4.2 and 4.3 with the extended normal-vector n_i and the corresponding radiation solutions φ_i. Further discussion of the pertinent theory may be found in References [13] and [26].

The following four examples, which are intended to illustrate applications, are simplified from the computational examples in [13].

1. A ship with simplified transverse and vertical bending modes described by the Legendre polynomial of order 2:

$$u_7 = 0, \quad v_7 = P_2(q), \quad w_7 = 0 \tag{8.2}$$

$$u_8 = 0, \quad v_8 = 0, \quad w_8 = P_2(q) \tag{8.3}$$

Here NEWMDS=2, $P_2(q) = \frac{3}{2}q^2 - \frac{1}{2}$ is the Legendre polynomial and $q = 2x/L$ is the normalized horizontal coordinate varying from -1 to +1 over the length L of the ship.

2. A vertical column, bottom mounted, with three orthogonal cantilever modes described by shifted Jacobi polynomials:

$$u_7 = q^2, \quad v_7 = 0, \quad w_7 = 0 \tag{8.4}$$

$$u_8 = 6q^3 - 5q^2, \quad v_8 = 0, \quad w_8 = 0 \tag{8.5}$$

$$u_9 = 28q^4 - 42q^3 + 15q^2, \quad v_9 = 0, \quad w_9 = 0 \tag{8.6}$$

Here NEWMDS=3, and $(q = z/HBOT + 1)$ is the normalized vertical coordinate varying from 0 at the bottom to 1 at the free surface.

3. Two bodies connected by a simple hinge joint at the origin, permitting each body to pitch independently about the y-axis. In this case the six rigid-body modes are defined as if the hinge is rigid, and the new mode $(j = 7)$ is specified by the vector

$$u_7 = zn_x\text{sgn}(x), \quad v_7 = 0, \quad w_7 = |x|n_z \tag{8.7}$$

with NEWMDS=1. Here $\text{sgn}(x)$ is equal to ± 1 according as $x > 0$ or $x < 0$.

4. An array of five identical bodies, all described by panels in the same GDF file as if for a single body with five separate elements; the bodies are centered at transverse positions $y = -4w, -2w, 0, 2w, 4w$ to simulate images of the central body in the presence of tank walls at $y = \pm w$. The surge, heave, and pitch exciting forces on the central body are then specified by the exciting force coefficients X_7, X_8, X_9. The corresponding new modes are defined which have the same normal velocities on the central body and zero on the images. Here NEWMDS=3, and the vectors (u_j, v_j, w_j) are all zero, except in the range $-w < y < w$ where

$$u_7 = 1 \tag{8.8}$$

$$w_8 = 1 \tag{8.9}$$

$$u_9 = z \quad w_9 = -x \tag{8.10}$$

(With these definitions the exciting force coefficients can be evaluated either from the Haskind relations or direct pressure integrals as defined in Section 4.3, with the surface integrals extended over the ensemble of the central body and its images.)

In the generalized mode analysis one or two planes of geometric symmetry can be exploited to reduce the computational burden, when the body geometry permits. In such cases it is necessary to define the generalized modes to be either symmetrical or antisymmetrical. These symmetries must be specified in the subroutine by assigning one of the integers (1-6) to the array ISYM for each of the generalized modes. The value of this integer signifies that the symmetries of the generalized mode are the same as the corresponding rigid-body mode. Since the symmetries of surge (j=1) and pitch (j=5) are always the same, and likewise for sway (j=2) and roll (j=4), specifying ISYM=1 or 5 is equivalent,

and similarly ISYM=2 or 4 is equivalent. If there are no planes of symmetry, any integer in the range (1-6) may be assigned to ISYM.

Any problem which can be analyzed with the NBODY option can also be analyzed with the generalized mode option. If geometric symmetry planes exist for the ensemble of all bodies, the use of the generalized mode method is more efficient computationally. On the other hand, the preparation of input files generally is simpler in the NBODY approach.

Complex generalized modes can be analyzed by superposition of the separate real and imaginary parts, each of which is treated as a separate real mode. For example, the specifications $w_7 = \cos(kx)$, $w_8 = \sin(kx)$ define two vertical deflections which can be superposed with a phase difference of 90° to represent a snake-like traveling wave along the body.

Two alternative program units have been provided in the WAMIT software package to facilitate the use of generalized modes. The first method, also used with previous versions of WAMIT, uses a separate program DEFMOD to evaluate the geometric data associated with generalized modes. DEFMOD contains a subroutine DEFINE, which can be modified by the user to compute the displacement vector for different generalized modes. This method is retained in Version 6 and can be used with the low-order method (ILOWHI=0). The second method, which is applicable for both the low-order and higher-order methods, uses a DLL file containing a special subroutine NEWMODES with a library of lower-level routines where different types of generalized modes can be defined. Since NEWMODES is contained in a DLL file it can be modified by the user, in a similar manner to the GEOMXACT file for defining bodies analytically (cf. Section 6.9).

The first method, using the program DEFMOD, can be used with any suitable FORTRAN compiler. Three separate runs must be made, first with WAMIT to set up the input file for DEFMOD, then with DEFMOD, and finally with WAMIT to solve for the potentials in the usual manner. In the second method only one run of WAMIT is required, however users of the PC executable code must compile the DLL file following the instructions below.

In Section 8.1 the input files are described for performing the generalized mode analysis for a single body. Section 8.2 describes the use of DEFMOD, and Section 8.3 describes the alternative use of the DLL file NEWMODES. The definitions of hydrostatic restoring coefficients are described in Section 8.4 and the analysis of multiple bodies (NBODY>1) is described in Section 8.5.

Several test runs are used to illustrate the use of generalized modes, including the use of both DEFMOD and NEWMODES and the appropriate input files. TEST08 (ILOWHI=0) and TEST18 (ILOWHI=1) analyze a bottom-mounted column with bending modes. TEST16 analyzes a rectangular barge with bending modes. TEST17(a,b) illustrates the use of generalized modes to analyze damped motions of a moonpool. TEST23 uses generalized modes to analyse a bank of 'paddle' wavemakers. TEST 24 analyses the motions of a vessel with five separate segments connected by hinges. Further information is contained in the Appendix.

8.1 INPUT FILES

Two input parameters NEWMDS and IGENMDS control the implementation of the generalized mode option. NEWMDS specifies the number of generalized modes, with the default value zero. Instructions for inputting NEWMDS in the POT file (if IALTPOT=2) or in the configuration file (if IALTPOT=1) are given in Sections 3.2 and 3.7 respectively. IGENMDS is an integer used to specify the definition of the generalized modes, using either the DEFMOD program or the NEWMODES subroutine library. IGENMDS is input in the configuration file. The default value IGENMDS=0 is used if the program DEFMOD is used, as explained in Section 8.2. If IGENMDS is nonzero, the DLL file NEWMODES is used. In the latter case, the value of IGENMDS can be used to identify appropriate subroutines within the NEWMODES library, as explained in Section 8.3.

The definition of nondimensional outputs corresponding to each mode of motion cannot be specified in general, without prescribing the dimensions of each mode. To avoid this complication, the parameter ULEN should be set equal to 1.0 in the GDF file whenever generalized modes are analyzed. Except for this restriction, the GDF and POT input files are not changed.

In the POT file the six rigid-body modes can be specified as free or fixed in the usual manner, by appropriate choices of the index IRAD and array MODE (see Section 3.2). The program assumes that all generalized modes ($j > 6$) are free, and sets the array elements MODE(j) equal to one for these modes during the computations.

The options in FORCE have the same effect for generalized modes as for the rigid-body modes, except for restrictions on the mean drift forces evaluated by direct pressure integration (Option 9) and control surfaces (Option 9c). The two horizontal drift forces and the vertical drift moment can be evaluated, including all pertinent motions of the body in the rigid-body and generalized modes, using the momentum analysis (Option 8). Options 9 and 9c cannot be used for bodies with generalized modes. In the analysis of multiple bodies (NBODY>1), where some but not all of the bodies have generalized modes, Options 9 and 9c can be used for the bodies with no generalized modes.

The Alternative Form 2 of the FRC file (IALTFRC=2) should be used to specify the appropriate mass, damping, and stiffness matrices for the body including its extended modes. For example, in case 1 above, to account for the mass and stiffness of the ship hull it is necessary to include corresponding 8×8 matrices which correctly specify these coefficients for the distribution of internal mass within the ship and for its bending motion. The FRC files used for Test Runs 08, 16, 17, and 18 illustrate appropriate forms.

8.2 USING DEFMOD WITH THE LOW-ORDER METHOD

To facilitate the definition of the vectors (u_j, v_j, w_j) by users, a pre-processor program DEFMOD is provided in Fortran source code to input the values of the normal velocity for each generalized mode, at the centroid of each body panel. DEFMOD includes a short

subroutine DEFINE, which should be modified by the user for each application. In the DEFMOD subroutine as provided, DEFINE evaluates the bending modes of the vertical column used in the Test Run TEST08. (The same subroutine is included in NEWMODES and used for TEST18.) The four examples itemized in the introduction of this Chapter are included in separate files DEFINE.1, DEFINE.2, DEFINE.3, and DEFINE.4 to illustrate the preparation of appropriate subroutines. (Additional modes are included in these files.)

The evaluation of the normal velocity (8.1) for generalized modes requires a specification of the vectors u, v, w, and normal components n_x, n_y, n_z at the centroid of each panel. The first WAMIT run is aborted after writing these data to a file, and also the panel areas which are required to evaluate the hydrostatic coefficients for the generalized modes. The user is then requested to run DEFMOD. After DEFMOD is run, WAMIT may then be re-run to complete the analysis in the normal manner. This procedure is described in greater detail below, and illustrated by the flow chart in Figure 8.1. There are two input/output files associated with DEFMOD, both denoted by the filename (*gdf*) of the GDF file. The file *gdf*.PRE (for PRE-processing) contains for each panel the centroid coordinates $\mathbf{x}$, the area, and the six components of $\mathbf{n}$ and $\mathbf{x} \times \mathbf{n}$. The file *gdf*.MOD (for MODes) contains the generalized normal velocity on each panel, and the hydrostatic coefficients. These two files are prepared automatically by WAMIT and DEFMOD, and do not require special attention by the user.

The first run of WAMIT is made with NEWMDS > 0 specified in the configuration file. The pre-processor file *gdf*.PRE is output to the hard disk, and execution of WAMIT is interrupted with the following message:

'PREMOD run completed – now run DEFMOD'

This first run of WAMIT is interrupted if NEWMDS > 0 and if there is not already in the default directory an input file with the name *gdf*.MOD. For this reason the user must rename or delete old MOD files prepared with the same *gdf* filename.

The output file *gdf*.PRE includes a one-line header for identification, followed by one line containing the symmetry indices (ISX, ISY), number of input panels (NEQN), and the maximum number of degrees of freedom which can be assigned in the WAMIT run (MAXDFR). Each of the remaining lines of this file corresponds to a panel, in the same order as the GDF file, and includes the coordinates $\mathbf{x}=(x, y, z)$ of the panel centroid, its area, and the six components of the normal vector $\mathbf{n}$ and cross-product $\mathbf{x}\times\mathbf{n}$.

Either before or after the first run of WAMIT the user should modify the subroutine DEFINE in the program DEFMOD, specifying the number of generalized modes, their symmetries with respect to the geometric planes of symmetry of the body, and appropriate code for computing the vectors (u_j, v_j, w_j). DEFMOD should then be compiled and linked with an appropriate FORTRAN compiler. (Since DEFMOD is a self-contained FORTRAN file, linking is a trivial operation.)

DEFMOD is executed after the first run of WAMIT, using the *gdf*.PRE file as input to generate the output file *gdf*.MOD. This MOD file generated by DEFMOD includes the normal velocities of each generalized mode at each panel centroid. Also included at the end of this file are the generalized hydrostatic coefficients computed from equation (8.12).

After creating the *gdf*.MOD input file, execute WAMIT again to continue the run. From this on to the completion of the run the procedure is identical to that where NEWMDS=0.

The first and second runs are distinguished by the absence or presence, respectively, of the file *gdf*.MOD. Thus the existence of an old file with the same name is important. If changes are made only in the POT file (e.g. changing the wave periods), an existing MOD file can be reused without repeating the first run of WAMIT. On the other hand, if changes are made in the GDF file (e.g. changing the number of panels), and if the same *gdf* filename is used for the new GDF file, the old MOD file must be renamed or deleted before running WAMIT. If changes are made in DEFMOD (e.g. changing the definitions and/or number of new modes), the old PRE file can be used as input to create the new MOD file. A warning message is issued by DEFMOD before overwriting an old MOD file with the same name. No warning message is issued by WAMIT before overwriting an old PRE file with the same name.

8.3 USING THE DLL SUBROUTINE NEWMODES

The source code for the DLL file NEWMODES.F is provided with the WAMIT software to facilitate the specification of generalized modes by users, including users of the WAMIT PC executable version. Source-code users may prefer to compile and link NEWMODES together with WAMIT, or alternatively to keep it as a separate DLL file. PC executable users must compile their modified versions of NEWMODES, following the instructions below. This procedure is analogous to the modification of the DLL file GEOMXACT, as described in Section 6.9.

The file NEWMODES.F includes the main subroutine NEWMODES, and a library of specific subroutines used for different applications. The library can be modified or extended by users to describe generalized modes for other applications. In all cases the calls to these specific subroutines are made from NEWMODES. Thus the user has the capability to make appropriate modifications or extensions and to implement these with the executable version of WAMIT, without the necessity to access to the remaining source code for WAMIT.

The principal inputs to NEWMODES are the Cartesian coordinates (X,Y,Z) of a point on the body surface, specified in the vector form X(1),X(2),X(3), and the corresponding components of the unit normal vector XN at the same point. These inputs are provided by the calling unit of WAMIT, and the user does not need to be concerned with providing these inputs. The principal outputs, which the user must specify in an appropriate subroutine for each generalized mode, are (1) the symmetry index of the mode, (2) the normal component of the displacement, and (3) the vertical component of the displacement. The symmetry indices, which are defined above in the Introduction, identify the symmetry of each mode (assuming the body has one or two planes of symmetry) by assigning the values (1,2,3,...,6) to indicate the same symmetries as the corresponding rigid-body modes. The normal component of the displacement, denoted VELH in NEWMODES, is computed from the product of the displacement vector (U,V,W) and the normal vector XN. The

vertical component of the displacement, denoted ZDISP in NEWMODES, is identical to the component W of the displacement vector (U,V,W).

Several other inputs are included in the argument list of NEWMODES to simplify its use and increase its computational efficiency. These include the body index IBI, the vector IBMOD which specifies the starting index for each body in the global array of mode indices, the patch/panel index IPP, the vector NEWMDS which specifies the number of generalized modes for each body, the integer IGENMDS specified in the configuration file, and an integer IFLAG which specifies the required outputs from each call. These inputs are explained in more detail below. Unit numbers for three files are also included in the argument list to facilitate input of user-defined data, and the output of error messages.

The body index IBI is useful for multiple-body computations, where different generalized modes may be specified on each body. If NBODY=1, the index IBI=1.

For the analysis of a single body, the mode index j is assigned consecutively as explained after equation (8.1), with the first generalized mode $j = 7$ and the last generalized mode $j = 6+NEWMDS(1)$. For multiple bodies the arrays for each body are concatenated in succession. The vector IBMOD(1:NBODY) points to the last mode index j used for the preceding body. Thus, in general, for body IBI, the first generalized mode is $j = IBMOD(IBI)+7$ and the last generalized mode is $j = IBMOD(IBI)+6+NEWMDS(IBI)$. Thus the pointer IBMOD is useful to assign generalized modes correctly when multiple bodies are analyzed.

The patch/panel index IPP can be used to identify specific portions of the body. In the low-order method (ILOWHI=0), IPP identifies the local panel number where the input point X,Y,Z is located (usually at the centroid of this panel). In this case, if NBODY>1, IPP is the *local* panel number of the body IBI, as listed in the corresponding GDF file. In the higher-order method (ILOWHI=1) IPP identifies the *global* patch number where the input point is located. (If NBODY=1 the local and global patch numbers are identical. If NBODY>1 the global patch numbers are assigned consecutively for all bodies.) When NBODY > 1 and a part of the bodies are input, WAMIT automatically reflects the surface. The following convention should be followed to find the appropriate patch numbers after the reflections. For each body where one reflection occurs, the original patches IPP=1 to NP are reflected to NP+1 to 2NP. When two reflections occurs, the first reflection is about $x = 0$. Then the second reflection occurs for the patches from IPP=1 to IPP=2NP+1, 4NP about $y = 0$. For example, if the first quadrant of a body is defined by one patch, the next three patches will be in the order of the second, fourth and the third quadrant.

The subroutine MOONPOOL_FS used for the test runs TEST17a/b is an example where IPP is useful. In this case the generalized modes are defined as nonzero only on the free surface inside the moonpool, identified as patch 4 for this geometry (see Section 6.8). Thus the use of the patch index is simpler and more efficient than performing a test based on the values of the input coordinates (X,Y,Z).

The input IGENMDS should be specified in the configuration file with a nonzero value, when NEWMODES is used to define generalized modes. Different values of IGENMDS can be useful in NEWMODES to identify different subroutines of the library. This is illustrated in the version supplied with WAMIT, where the integers 16, 17, 18 are used to identify the

corresponding test runs and associated subroutines.

The input IFLAG is generated internally by the calling routine, with three possible values. Initially NEWMODES is called once with IFLAG=-1, to assign the array IMODE specifying the symmetry index for each mode. In subsequent calls where only the normal component of the displacement vector VELH is required, IFLAG=1. If both VELH and the vertical component ZDISP are required (for computations of hydrostatic coefficients) IFLAG=2. In the higher-order method a large number of calls are made to NEWMODES with IFLAG=1. If computational efficiency is important this should be considered in modifying or extending the subroutines in NEWMODES.

The comments inserted in the NEWMODES file should be consulted for further details.

In order to use NEWMODES for any of the purposes described in this Chapter, the file NEWMODES.DLL must be in the same directory as WAMIT.EXE.

Instructions for making new DLL files are included in Section 10.5.

In some cases it is useful to input data from a special input file, so that the subroutines in NEWMODES can be used with different values of relevant parameters. For example, the subroutine WAVEMAKER inputs the depth of the pivot axis from a special file WAVEMAKER_DEPTH.DAT, as illustrated in TEST23, and the subroutine HINGE_MODES inputs data from XHINGE.DAT, as illustrated in TEST24. The NEWMODES argument IFILEDLL specifies a file number (also known in FORTRAN as a unit number) which is reserved for this purpose. Users who modify the DLL subroutines or make new DLL subroutines should follow the code in these two subroutines for guidance in inputting data from an external file. In order to make it possible to copy this data file to the log file WAMITLOG.TXT, as is done for other input files, the file number of the log file IWAMLOG is included in the argument list. Error messages which are generated in DLL subroutines can be added to the error files ERRORP.LOG and ERRORF.LOG by using the file number IERROR.

8.4 HYDROSTATICS

To evaluate the motions of a body including generalized modes, it is necessary to evaluate the corresponding hydrostatic coefficients. In general these are defined by the matrix (Reference 13, equation 2.17)

$$c_{ij} = \rho g \iint_{S_b} n_j (w_i + z D_i) dS \tag{8.11}$$

Here D_i denotes the divergence of the vector (u_i,v_i,w_i), assumed to be continuous in the vicinity of the body surface.

In cases where $D_i = 0$ the hydrostatic matrix can be evaluated uniquely from the vertical component w_i. For these cases the simplified hydrostatic matrix

$$c_{ij} = \rho g \iint_{S_b} n_j w_i dS \tag{8.12}$$

is computed and no further steps are required by the user. This computation is performed in DEFMOD if that program is used, or internally in WAMIT if the DLL subroutine NEWMODES is used.

In special applications where $D_i \neq 0$, the hydrostatic coefficients can be programmed specially by modifying the code in the main program of DEFMOD. Alternatively, the extra contribution from the last term in (8.11) can be included as an 'external' force, in the stiffness matrix of the FRC file.

The hydrostatic coefficients c_{ij} are output, as part of the complete hydrostatic matrix, in the file *out*.hst, with the format indicated in Section 4.9. (Here *out* denotes the filename of the .out file for the run.)

8.5 NBODY ANALYSIS

When it is desirable to do so, the NBODY and generalized mode analyzes can be combined. An example where this might be effective is if two separate bodies are in close proximity, one or both of them are undergoing structural deflections, and there are no planes of geometric symmetry for the ensemble of two bodies.

If NBODY>1, separate values of NEWMDS must be specified for each body. If IALTPOT=1 this is done in the configuration file using a separate line for each body with the syntax 'NEWMDS(m) = n'. Here m is the body index and n is the number of new modes for that body. For example if there are three bodies, and body 1 has 3 generalized modes, body 2 has none, and body 3 has 2, two lines should be added to the configuration file as follows:

NEWMDS(1)=3
NEWMDS(3)=2

(The additional line NEWMDS(2)=0 may be added for clarity but is not required.)

If IALTPOT=2 the value of NEWMDS for each body is specified in the POT file, as shown in Section 3.2.

In the approach described in Section 8.2, prior to the NBODY run of WAMIT, the subroutine DEFMOD must be executed for each body, to prepare the corresponding MOD file. This procedure is carried out separately for each body for which generalized modes are specified, with an appropriate subroutine DEFINE corresponding to the generalized modes of that body. The procedure for doing this is identical to that described in Section 8.2 for a single body.

In the approach described in Section 8.3, the subroutines in NEWMODES should be organized in a logical manner so that the generalized modes for each body are defined. Usually this can be done most effectively by using separate subroutines for each body, but that is not necessary. The body index IBI is used to identify the body for each call.

The numbering sequence for these modes in the output files is with the new modes of each body following the six rigid-body modes of the same body. Thus, in the example above, the nine exciting-force coefficients of body 1 are denoted by X_j ($j = 1, 2, ..., 9$, the

six conventional coefficients of body 2 by ($j = 10, 11, ..., 15$, and the eight coefficients of body 3 by ($j = 16, 17, ..., 23$).

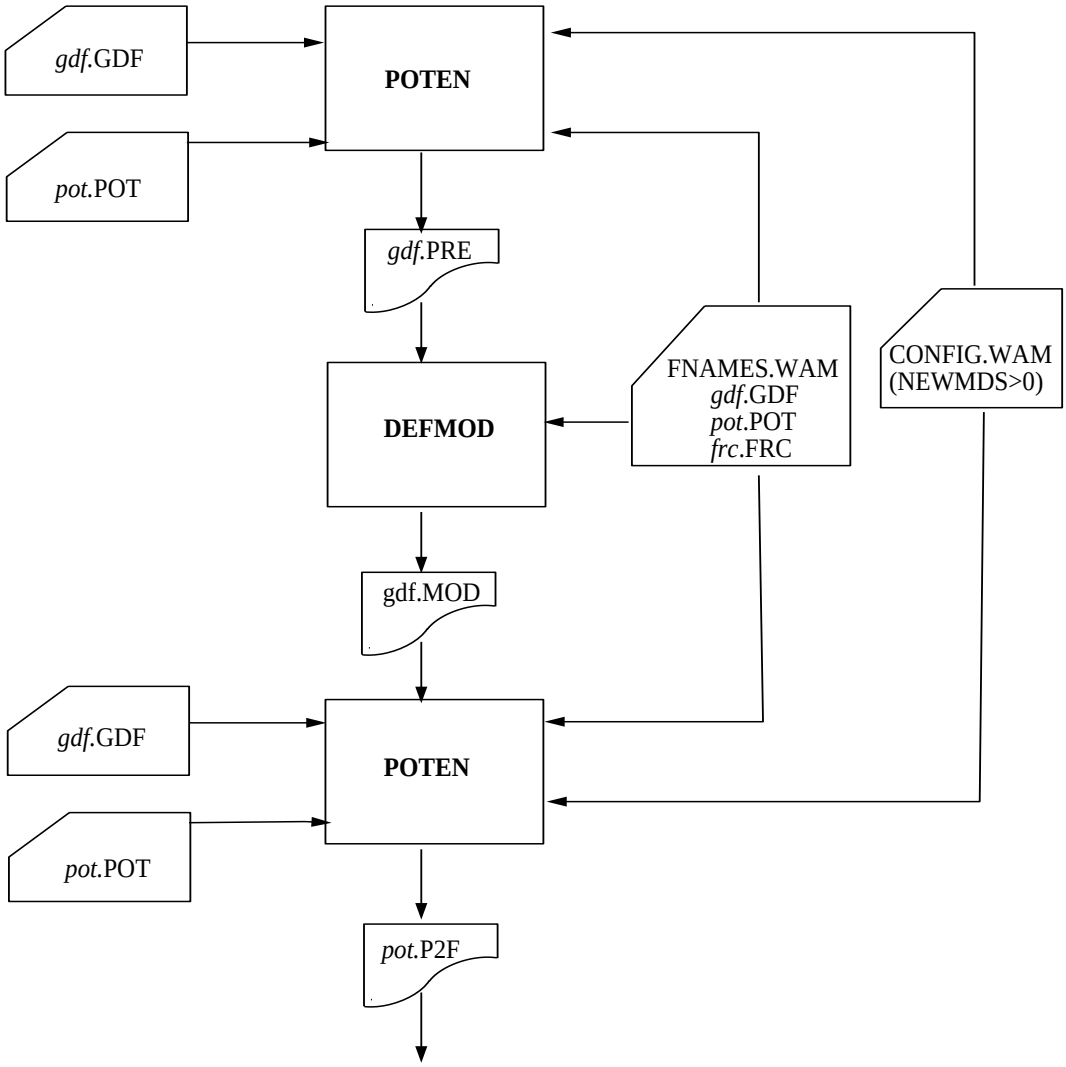


Figure 8.1: Flow chart showing the use of WAMIT and DEFMOD to define and analyze generalized modes in the low-order method (ILOWHI=0). In this example IALTPOT=1, and the number of generalized modes NEWMDS is specified in the configuration file. On the first run of WAMIT the centroid coordinates, area, normal vector $\mathbf{n}$, and cross-product $\mathbf{x}\times\mathbf{n}$ are output for each panel to the file *gdf*.PRE, for pre-processing by the program DEFMOD. After modification by the user to specify the desired modes, DEFMOD is run with the input file *gdf*.PRE, to produce the output file *gdf*.MOD containing the normal velocity of each new mode at the panel centroids, and also the hydrostatic coefficients. After this pre-processing is completed, WAMIT is run again in the normal manner, as in the flow chart of Figure 1-1. Note that WAMIT reads the GDF and POT input files on both runs, although the data in the POT file is only used on the second run.

Chapter 9

USE OF IRREGULAR FREQUENCY OPTION

WAMIT includes a method for removing the effects of irregular frequencies on both the velocity potential and the source strength. An outline of the method is given in Section 12.8, with more detailed information in References [8], [16] and [26]. In this method, the computational domain includes the interior free surface of the body and it is necessary to discretize it as well as the body surface. The integer switch IRR controls the removal of the effect of irregular frequencies. This parameter is required either in the POT file or in the configuration file, as explained in Sections 3.1, 3.2 and 3.9. Depending on the value of IRR, the discretization of the interior free surface may be provided by the user or it may be done automatically by the program. Explanation of the parameter IRR and the discretization of the interior free surface are given in the following Sections.

9.1 INPUT PARAMETERS

The parameter IRR may be specified in the POT file as shown in Section 3.1 and 3.2, or it may be assigned in the configuration file as shown in Section 3.9. The definition of IRR is as follows.

IRR is the integer used to specify whether the effect of the irregular frequency is removed or not. The values IRR=(0,1,2,3) are legal in the low-order method (ILOWHI=0). Only the values IRR=(0,1) are legal in the higher-order method (ILOWHI=1).

IRR= 0: Do not remove the effect of the irregular frequencies.

IRR= 1: Do remove the effect of the irregular frequencies.

In the low-order method (ILOWHI=0) the user must discretize the free surface. The coordinates of these panel vertices must be included in GDF file. (These must precede dipole panels, as explained in Section 5.4. Otherwise, the locations of the free-surface

panels are arbitrary, within the GDF file.) The vertices of the free surface panels must be numbered in the *clockwise* direction when the panel is viewed from inside the body or in the *counter-clockwise* direction when the panel is viewed from above the free surface. The whole, half or a quadrant of the interior free surface must be discretized in accordance with the discretization of the body surface when there is geometric symmetry. The number of equations, the parameter NEQN in the GDF file, should be the sum of the number of panels on the body and the free surface.

In the higher-order method (ILOWHI=1) the user must represent the interior free surface with one or more patches, in the same manner as for the body surface. This must be done with the normal pointing downward, in the negative $z-$direction. In the context of the right-hand-rule stated in Section 6.1, this is equivalent to requiring that the parametric coordinates (u, v) for the free-surface patch must be defined such that the positive normal vector in the (u, v) plane points downward from the free surface into the body. The subroutine CIRCCYL in GEOMXACT.F is an example where the interior free surface is represented analytically as one extra patch. The locations of the free-surface patches are arbitrary, within the GDF file.

IRR= 2: Do remove the effect of the irregular frequencies.

The program projects the body panels onto the free surface to generate panels on the free surface. IRR=2 should not be specified for a certain body shape such as a Tension Leg Platform with pontoons. For these bodies, a vertical line intersects the body surface more than once and thus the projection of two body panels overlap on the free surface. The GDF file contains only the body panels (it is the same as the GDF file used with IRR= 0). This option cannot be used with the higher-order method.

IRR= 3: Do remove the effect of the irregular frequencies.

With IRR= 3, the program automatically generates a discretized interior free surface, and stores the panel vertices in a special output file *gdf*.IDF. In this file, the vertices of the free surface panels are appended to the data in the GDF file. An outline of this procedure is described below in Section 9.2. Since this automatic discretization cannot accommodate abnormal waterline shapes, the user should visualize the paneling on the interior free surface using the IDF file, to check the quality of the interior free surface discretization. This option cannot be used with the higher-order method.

When IRR$\geq$ 1, the convergence rate of the iterative solver (ISOLVE=0) is reduced. It is recommended to use either the block iterative solver or the direct solver.

The parameter ILOG in the POT file or the configuration file should be set equal to 1 when IRR$\geq$ 1. The special limits of zero or negative (infinite) wave period should not be included in the POT file when IRR$\geq$1 .

The generation of free-surface panels in the low-order method, or free-surface patches in the higher-order method, can generally be facilitated by using MultiSurf. Various techniques may be considered for this purpose, including the higher-order integration option described in Section 6.7 and Appendix C.

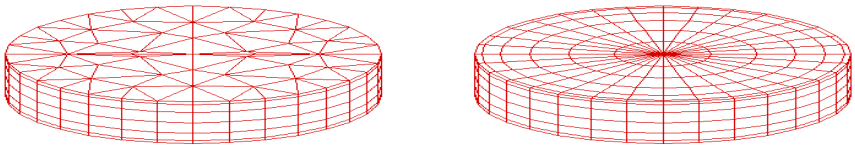


Figure 9.1: Automatic discretizations on the interior free surface of a circular cylinder. The figure on the left shows the result of the discretization algorithm used when ISOR=0. The figure on the right shows the result of the special algorithm used when ISOR=1 to ensure continuity of the panels adjacent to the waterline.

9.2 AUTOMATIC FREE-SURFACE DISCRETIZATION (IRR=3)

The program discretizes the interior free surface based on i) the algorithm described in [8] when ISOR=0 and ii) in a regular pattern when ISOR=1 as described below. Figure 9.1 shows these two discretizations on the interior free surface of a truncated cylinder. In i) (ISOR=0) the interior free surface is discretized with triangular panels. In general, the side of the triangular panels is similar to the average length of the waterline segments. due to the $O(1)$ aspect ratio of the panels, the number of panels on the interior free surface is not so large as in i).

In ii) (ISOR=1) the connectivity between the free surface panels and the waterline segments is enforced. The necessity of this connectivity in the source formulation is explained in [16]. The program first finds the centroid of the waterplane area enclosed by a waterline contour formed by a set of waterline segments. Then the two nodes of each waterline segment are connected to the centroid forming triangles with vertices on the centroid and on the two nodes. The triangles are discretized by quadrilateral panels (two sides of each panel are on the lines from the centroid to the nodes) from the waterline segments to the centroid with denser discretization toward the waterline. The last panel is a small triangle which is used in the subsequent part of the program as a flag indicating the end of the connectivity. In this way, the connectivity is guaranteed between certain groups of panels starting from quadrilateral panels contiguous to waterline segment. **This option should not be used for any body where a ray from the centroid of the waterplane area intersects the waterline more than once. In that case the IRR=1 option should be used.**

When IRR=1 and ISOR=1, the interior free surface panels should be discretized in a following manner. First, define a set of interior panels which are contiguous with the waterline segments, one panel for each segment, and having aspect ratios similar to the adjoining body panels. Continue this process recursively toward the centroid of the waterplane area. As the process continues moving inward toward the centroid, the panel size can be increased and the continuity between adjacent panels is less important.

When IRR=1 and ISOR=0 it is recommended to discretize the interior free surface with the panels having the following properties: i) Panels have O(1) aspect ratio. ii) The length of the sides of the panels is similar to the waterline segments.

When the option IRR=3 is used, special care is required in preparing the GDF file to ensure that there are no significant gaps between the vertices of adjacent panels at the waterline. In addition it is desirable to avoid panels at the waterline which are of extremely small dimensions. These requirements are best understood in the context of the procedure used by the program to define the waterline contour from the data in the GDF file.

As the first step for the automatic discretization of the free surface, the program searches for and identifies all of the waterline segments between the panel vertices which are in the plane of the free surface or within a specified small distance of this plane. If the nondimensional absolute value of the vertical coordinate is less than TOL=1.E-5 the vertex is defined to be in the plane of the free surface. (This parameter TOL can be modified in the source code, in the subroutine WTLPN of the file 'getip.f'.)

To avoid including very small segments, which would produce a nonuniform discretization, waterline segments are neglected if their lengths are smaller than a specified minimum length. The minimum length for this purpose is the product of another prescribed tolerance (TOL=10E-3, defined in subroutine WLSORT of the same file) times the average length of the waterline segments. (In computing this average length, segments which are very small are neglected.) The program then rearranges the waterline segments in consecutive order, based on testing the distance between the end points of the segments. If this distance is less than the product of TOL times the average length, the program assumes the two segments are contiguous. When there is a gap larger than this product, the program outputs the error message 'Waterline panels do not form a closed contour'.

Since waterline segments with a nondimensional length less than TOL are neglected, it is possible for this error message to be output even when the actual gap is less than TOL, due to the simultaneous occurrence of one or more gaps and/or small panels. More specifically, the error message is output when the cumulative length[1] of successive small waterline segments is larger than the prescribed tolerance for the gap.

When this error message is encountered, the user is advised to re-discretize the body surface, to avoid panels with very small waterline segments and/or large gaps between adjacent panels, or alternatively to use the IRR=1 option which requires the user to discretize the free surface. It is also possible to overcome this problem on an *ad hoc* basis by modifying the parameter TOL in the subroutine WLSORT of the file 'genfi.f'. Modification of TOL is potentially dangerous, if there are relatively short waterline segments.

[1] The cumulative length is defined here as the length of the vector sum of the waterline segments

The parameter SCALEH is defined in WAMIT with the value 1.4. This parameter is used when IRR=3 and ISOR=0, to set the typical length ratio between the sides of the triangular panels on the free surface and the side of body panels.

Chapter 10

SPECIAL TOPICS

10.1 ERROR MESSAGES AND LOG FILES

Numerous checks are made in WAMIT for consistency of the data in the three input files. Appropriate error messages are displayed on the monitor to assist in correcting erroneous inputs. Output files containing warning and error messages are created after each execution of the subprograms POTEN and FORCE. **errorp.log** contains messages from POTEN and **errorf.log** from FORCE. The existing .LOG file, in the directory where the program runs, is overwritten with every new run. When the program runs successfully without any warning or error, the .LOG file contains two lines: a header line including the date and time when the program starts to run and a line indicating the completion of the run.

Error messages are associated with problems where the program execution is halted. Warning messages indicate that a possible error may occur, but under certain circumstances the results may be correct. (Examples include failure of the convergence tests for various numerical integrations, which sometimes result from inappropriate choices of characteristic length scales or of overly conservative convergence tolerances. Another example is in the case of diffraction by a body with one or two planes of symmetry, where it is possible to compute the exciting forces at certain heading angles without solving for all components of the diffraction potential; in this case the warning message states that the solution is non-physical, whereas at some heading angles the exciting forces will be correctly evaluated.)

Error and warning messages generated during execution of the POTEN sub-program are output to the file **errorp.log**, and messages generated during execution of the FORCE sub-program are output to the file **errorf.log**. The same outputs are displayed on the monitor. Since some of these messages may be lost on the monitor due to scrolling of other outputs, a special warning message is generated at the end of the run to alert users when significant messages are contained in these two files.

Starting in V6.2, a new file **wamitlog.txt** is output during the run. This file is intended to provide an archival record of the run. The file includes the starting and ending time and date for each sub-program, copies of the principal input files, and copies of the outputs in the files **errorp.log** and **errorf.log**. (Since the GDF input files are relatively long in the

low-order method, and also in the higher-order method when the geometry is defined by low-order panels or B-splines (ILOWHI=0 or 1, respectively), in these cases only the first 10 lines of the GDF file are copied to the file **wamitlog.txt**. Another point to note is that the maximum width of lines of data is truncated to 80 characters in **wamitlog.txt**. The existing **wamitlog.txt** file, in the directory where the program runs, is overwritten with every new run.

Additional data is included in the file **wamitlog.txt** as explained below.

When a warning message occurs indicating that the 'Number of subdivisions exceeds MAXSQR' for the Rankine integration over a higher-order panel, the Cartesian coordinates of the field point and source point are output to `wamitlog.txt` so that the user can more easily check if there is a singularity or inconsistency in the geometry definition in the vicinity of these points.

When the input data in the Force Control Files for one or more bodies are in the Alternative form 1, as defined in Section 3.3, the nondimensional inertia matrix for each body is included in the file **wamitlog.txt**. This is particularly useful when the analysis of a body is first performed using Alternative form 1, and then changed for subsequent extensions to Alternative form 2, for example when external damping is imposed on an otherwise freely floating body. The normalizing factors for the nondimensional inertia matrix are the products of the fluid density, and appropriate powers of the characteristic length parameter ULEN. In preparing a force control file for Alternative form 2, as defined in Section 3.4, these normalizing factors must be included in the inputs EXMASS when these are derived from the nondimensional inertia matrix.

10.2 RESERVED FILE NAMES

To avoid conflicting filenames, users are advised to reserve the extensions GDF, POT, FRC, SPL, P2F, OUT, PNL, FPT, PRE, MOD, HST, CSF, CSP, 1, 2, 3, 4, 5p, 5vx, 5vy, 5vz, 6, 7x, 7y, 7z, 8, 9, and 9c for WAMIT input and output. Other reserved filenames include CONFIG.WAM, FNAMES.WAM, ERRORP.LOG, ERRORF.LOG, WAMITLOG.TXT, SCRATCH* (here *=A,B,C,...,O), as well as WAMIT.EXE, DEFMOD.FOR, DEFMOD.EXE, and for V6PC, the DLL files GEOMXACT.F and NEWMODES.F. Source-code users can modify the extensions by editing the appropriate assignments (in the source file modulesc.f, module MAINC_ARRAYS).

10.3 SCRATCH FILES

Two types of temporary scratch files are opened during execution of the subprogram POTEN. One group are opened formally as scratch files using the FORTRAN convention, with filenames which are assigned by the compiler. The second group are opened with the temporary filenames SCRATCHA, SCRATCHB, ..., SCRATCHO. All of these files are deleted prior to the end of the run, but if execution is interrupted by the user (or by power interruption to the system) some or all of the above scratch files may remain on the hard disk. In the latter case the user is advised to delete these files manually.

10.4 MEMORY REQUIREMENTS AND NUMBER OF UNKNOWNS

The system memory requirements of WAMIT depend on the number of number of separate solutions for the velocity potential (and source strength, if ISOR=1), and the number of unknowns in each solution. This section is intended to provide the user with an understanding of these relationships, and of the manner in which WAMIT optimizes the solution of a given problem.

The number of simultaneous equations NEQN, equal to the number of unknowns, is used here to denote the dimension of the linear system solved for the determination of the radiation and diffraction velocity potentials on the body surface, and the corresponding source strengths when the option ISOR=1 is selected. The value of NEQN for each run is listed in the header of the OUT output file.

In the low-order method (ILOWHI=0) the number of equations NEQN is equal to the number of panels NPAN specified in the GDF input file(s), except when reflections about body planes of symmetry are required as explained below. If dipole panels are input, as explained in Section 5.4, NPAN is the sum of the conventional panels and dipole panels. If the irregular-frequency option is used, additional panels are required on the interior free surface inside of the body waterline, as described in Chapter 9, and the number of equations is equal to the total number of panels including the interior free surface, regardless of whether this is included in the GDF file and parameter NPAN explicitly (IRR=1), or the interior free surface panels are added by the program (IRR=2 or IRR=3).

In the higher-order method (ILOWHI=1) NEQN depends on the number of patches, panels, and the order of the B-splines:

$$\mathrm{NEQN} = \sum_{i=1}^{N_P} (\mathrm{NU}(i) + \mathrm{KU}(i) - 1) \times (\mathrm{NV}(i) + \mathrm{KV}(i) - 1) \tag{10.1}$$

Here N_P=NPATCH is the total number of patches, NU and NV are the numbers of panels on each patch, and KU and KV are the orders of the B-splines used to represent the solution. NU and NV are specified by the user in the CONFIG.WAM or SPL files, as explained in Sections 6.9-10. KU and KV are specified in the CONFIG.WAM or SPL files, or assigned the default value KU=KV=3. When reflections about planes of symmetry are

required, NPATCH is increased in the same manner as for NPAN.

If the parameter PANEL_SIZE in the CONFIG.WAM file is positive, NU and NV are determined internally in the program, as explained in Sections 3.7 and 6.9. In this case it is not generally possible to determine the exact values assigned to NU and NV on each patch. NU and NV are determined such that the estimated maximum physical dimension of each panel is less than PANEL_SIZE. The value of NEQN printed in the header of the OUT file can be used to verify estimates of NU and NV in this case.

When planes of geometric symmetry ($x = 0$ and/or $y = 0$ of the body coordinate system) do not coincide with the $X = 0$ and/or $Y = 0$ planes of the global coordinate system, WAMIT assumes no hydrodynamic symmetry with respect to those planes. In these cases the program reflects about the corresponding planes and increases the number of panels accordingly. The other case where planes of hydrodynamic symmetry are not utilized is the analysis of multiple interacting bodies (Chapter 7). In this case the number of unknowns NEQN is the total number of panels required to describe the entire bodies, and NLHS=1. The following list summaries the cases where the number of unknowns is increased due to reflection:

1. When planes of geometric symmetry ($x = 0$ and/or $y = 0$) of the body coordinate system do not coincide with the $X = 0$ and/or $Y = 0$ planes of the global coordinate system, due to nonzero values of the input parameters XBODY(1), XBODY(2), XBODY(4) in the POT file or config.wam file. In this case the program assumes that there are no planes of hydrodynamic symmetry, and the body geometry is reflected about its specified planes of geometric symmetry.

2. In the multiple-body analysis described in Chapter 7 (NBODY Option), the same procedure applies as in (1) above. (In some special cases the configuration of multiple bodies may be symmetrical about one or two global planes, but the program does not check for that possibility. In such cases more economical computations can be performed using generalized modes.)

3. In the analysis of a body near vertical walls described in Chapter 5.3, the same procedure applies as in (1) above.

'Total number of panels' refers to the number of panels used to represent the entire body surface. WAMIT takes into account flow symmetries in setting up the linear systems, therefore the number of unknowns and total number of panels are different if body geometry symmetry planes are present. If 0, 1, or 2 planes of symmetry are specified, the total number of panels is equal to NPAN, 2×NPAN, or 4×NPAN, respectively. Since the computational burden of solving the linear system of equations is at least proportional to NEQN^2, a substantial reduction in computational effort is achieved by imposing the planes of symmetry when this is physically appropriate.

Provision is made in WAMIT to specify a subset of modes to be analyzed separately, thus reducing the memory required and the run time. If the user anticipates the analysis of more than one mode it is more efficient to only once, for all modes of interest.

Since the issue of hydrodynamic symmetry is so important, it should be emphasized that the separate analysis of symmetric and antisymmetric modes of motion applies not only to the obvious cases of radiation modes, such as surge, sway, and heave, but also to the more complex solution of the diffraction problem, even in oblique waves. This is achieved in WAMIT by decomposing the complete diffraction (or scattering) solution as the sum of four separate components that are respectively even or odd functions of the horizontal coordinates. (Physically these can be interpreted as the solutions of problems where standing waves are incident upon the body.)

In considering memory requirements a distinction must be made between storage in RAM and on the hard disk. RAM requirements are affected by NEQN, and are different in the low-order and higher-order methods. Usually NEQN can be significantly smaller when the higher-order method is used, compared to the analysis of the same problem with the low-order method. But when ILOWHI=0, RAM can be used only for arrays which are linear in NEQN and thus it is possible to analyze structures with a large number of panels and unknowns, and to analyze simultaneously the different radiation and diffraction solutions of interest. When ILOWHI=1, one complex matrix of dimension NEQN, requiring 8*NEQN^2 bytes, is stored in RAM. All other arrays stored in RAM are linear in NEQN.

The most important parameter which affects the space required on the hard disk is the number of equations, and unknowns, NEQN. For large values of NEQN the required amount of scratch storage on the hard disk is proportional to NEQN^2. If the low-order method is used (ILOWHI=0), the number of bytes required for these arrays on the hard disk can be estimated from the equation

$$[(4 \times \mathrm{NLHS}) \times (1 + 4 \times \mathrm{ISOR}) \times (3 + \mathrm{ILOG}) + 2]\mathrm{NEQN}^2 - 8[min(\mathrm{MAXSCR}, \mathrm{NEQN})]^2$$

where min(MAXSCR,NEQN) denotes the minimum of these two parameters. If the higher-order method is used (ILOWHI=1), the number of bytes can be estimated from the equation

$$[(4 \times \mathrm{NLHS}) \times (3 + \mathrm{ILOG})]\mathrm{NEQN}^2$$

In these equations the parameter NLHS is the number of left-hand-sides appropriate to the analysis. If all modes of motion are studied simultaneously, for a body with 0, 1, or 2 planes of symmetry specified, NLHS is equal to 1, 2, or 4, respectively. As an example, the truncated vertical cylinder described in the Chapter 5, which has 2 planes of symmetry and a total of 1024 panels, requires about 3.4 megabytes of scratch storage on the hard disk to analyze all modes of motion simultaneously, i.e. for the run described in the Chapter 5.

If the storage requirements of a run exceed the available disk space a system error will be encountered; in this event the user should either increase the available disk space or reduce the number of panels or solutions.

Subroutine ITRCC (the iterative solver) reads matrix elements in each iteration step from the hard-disk. The run time in this subroutine can be reduced substantially by storing some or all of elements in available RAM. The parameter MAXSCR defines the dimension of a square sub-array which can be stored temporarily in available RAM. Since the coefficients

of this array are complex, the corresponding storage requirement in RAM is 8*MAXSCR^2 bytes, for systems which use 8 bytes for a single-precision complex number. Thus the parameter MAXSCR should be determined initially by estimating the size of excess RAM, after the program is loaded, and setting the largest integer value of MAXSCR such that 8*MAXSCR^2 does not exceed the excess RAM which is available. If a value of MAXSCR is specified which is larger than NEQN, MAXSCR is reduced by the program at run time, and set equal to NEQN.

If the option ISOLVE= 1 is selected, to utilize the direct solver for the linear system of equations, the entire left-hand-side matrix must be stored in RAM. This can only be achieved if MAXSCR≥NEQN. Otherwise, if ISOLVE= 1, an appropriate error message is generated at run time.

If the option ISOLVE> 1 is selected, to utilize the block iterative solver, the diagonal block matrices must be stored in RAM, each block at a time for the local LU decomposition. This can only be achieved if MAXSCR is equal to or greater than the dimension of the diagonal blocks. Otherwise the size of the diagonal blocks is reduced to MAXSCR internally.

10.5 MODIFYING DLL FILES

The files GEOMXACT.F and NEWMODES.F can be modified by users following the instructions in Sections 6.8 and 8.3. This makes it possible for users of the PC-executable code to develop special subroutines for the definitions of the body geometry and generalized modes, respectively, and to link these subroutines with WAMIT at runtime. Users of the source code have two alternative options: (1) maintain these two files separately from the remainder of the code, and use the DLL option if it is available with their compiler, or (2) compile and link the two source files directly with the remainder of the code.

The following procedure should be followed by PC-executable users to make a modified version of GEOMXACT.DLL, using Compaq Visual Fortran. For NEWMODES.DLL the procedure is identical.

- Open a new project 'geomxact' as a Fortran Dynamic Link Library
- Add geomxact.f to the project
- Build a release version of geomxact.dll
- Copy the new version of geomxact.dll to the working directory for WAMIT

It may be possible to use other FORTRAN compilers to make the DLL file, but certain conventions in calling subroutines must be consistent with those of Compaq Visual Fortran. Further information is provided in [23], Chapters 8 and 18.

10.6 AUXILIARY OUTPUT FILES FOR THE GEOMETRY

Version 6.2 includes options to generate three auxiliary geometry output files, with the names *gdf*_PAN.DAT, *gdf*_PAT.DAT, and *gdf*_LOW.GDF. (Here *gdf* is the filename of the GDF input file. For a run with NBODY>1 the GDF filename for the first body (N=1) is used.) These files contain the Cartesian coordinates of panels or patches in formats suitable for perspective plotting, and the coordinates of new low-order panels which are derived from the input geometry. In all cases the coordinates are dimensional, and defined in the same units as specified in the input GDF file(s).

The data in *gdf*_LOW.GDF has the same definitions and format as a conventional low-order GDF file (Section 5.1). The coordinates of the panel vertices are defined with respect to the body coordinate system, corresponding to the original GDF inputs. If NBODY>1 the file *gdf*_LOW.GDF represents the body identified as $N = 1$. If the original body panels are reflected by the program, the file *gdf*_LOW.GDF will include subdivided panels for the reflected body. (This will occur if NBODY>1, if walls are present, or if the body is not symmetric with respect to the global coordinate system.) If ILOWGDF=1 and the body is reflected by the program, *gdf*_LOW.GDF will contain the original body panels (without subdivision) plus their images about the reflected planes of symmetry.

The data in *gdf*_PAN.DAT and *gdf*_PAT.DAT are defined with respect to the global coordinate system. In a WAMIT run with NBODY>1 the data for all of the bodies are included. The figures in Sections A5 and A13 illustrate this feature.

These auxiliary files are described separately below for the low-order and higher-order methods.

In the low-order method (ILOWHI=0) the vertex coordinates of the body panels are stored in the output file *gdf*_PAN.DAT in a format suitable for input to plotting programs such as Tecplot. This facilitates the use of perspective plots to illustrate and check the GDF inputs. If (ILOWHI=0) the file *gdf*_PAN.DAT is output in the finite-element format FEPOINT, as specified for use with the Tecplot program. Examples of these plots are included in the Appendix for each low-order test run (Sections A1-A9). The integer parameter IPLTDAT in the file CONFIG.WAM is used to specify whether or not to generate this output file. In the default case (IPLTDAT=0) no file is generated. If (IPLTDAT>0) the file is generated.

In the low-order method (ILOWHI=0) the optional output file *gdf*_LOW.GDF is controlled by the integer parameter ILOWGDF in the file CONFIG.WAM. If ILOWGDF> 0 the output file *gdf*_LOW.GDF is generated, with all of the original panels subdivided into ILOWGDF×ILOWGDF sub-divisions. The first three lines are copied from the GDF input file. The total number of sub-divided panels is included on line 4. This option can be used to increase the number of panels, and hence to increase the accuracy of the solution for the potential or source strength. However this subdivision scheme does not increase the accuracy of the geometric representation of the body, since the subdivided panels are coplanar with the original panels. Only one body can be subdivided in this manner.

In the higher-order method (ILOWHI=1) the optional output files *gdf*_PAT.DAT and *gdf*_PAN.DAT specify the vertex coordinates of both the patches and panels, as defined in

Chapter 6. If (ILOWHI=1) these files are in the ordered-list format POINT, as specified for use with the Tecplot program. Examples of these plots are included in the Appendix for each higher-order test run (Sections A11-A19). The integer parameter IPLTDAT in the file CONFIG.WAM is used to specify whether or not to generate these output files. In the default case (IPLTDAT=0) no files are generated. If (IPLTDAT>0) the files are generated. The number of panel subdivisions on each patch is determined by the parameters NU and NV in the SPL file, as explained in Section 6.11. If IPLTDAT=1 the data file *gdf*_PAT.DAT contains only the four vertices of each patch, and the file *gdf*_PAN.DAT contains only the four vertices of each panel. If IPLTDAT>1, each element is subdivided into IPLTDAT×IPLTDAT sub-elements. Subdivision of the elements is useful when perspective plots are constructed for bodies with curved boundaries of the patches and panels. When the curvature is large, IPLTDAT should be increased to give a more accurate plot. (IPLTDAT=5 is used for the plots shown in Sections A11-A19.)

In the higher-order method (ILOWHI=1) the optional output file *gdf*_LOW.GDF is controlled by the integer parameter ILOWGDF in the file CONFIG.WAM. If ILOWGDF> 0 a low-order GDF file is generated, using the panel vertices of the higher-order geometry with ILOWGDF×ILOWGDF sub-divisions. The first three lines are copied from the higher-order GDF input file. The total number of sub-divided panels is included on line 4. This option can be used to generate low-order GDF files for any of the geometries which can be input to the higher-order method, including geometries represented by a small number of flat patches (Section 6.5), B-splines (Section 6.6), and geometries which are defined in the subroutine GEOMXACT (Section 6.8). In each case the number of low-order panels can be increased by increasing the value of ILOWGDF. The coordinates of the panels are in the same body-fixed dimensional system as the original input data.

These optional files are generated in the POTEN subprogram, after reading the geometry input files and before looping over the wave periods. If NPER=0 these files can be generated quickly, without the extra time required to solve for the potential and hydrodynamic parameters.

10.7 INTERNAL TANKS

Version 6.2 includes the options to analyze the linear hydrodynamic parameters for a fluid inside an oscillatory tank, or to analyze the coupled problem where one or more tanks are placed within the interior of one or more bodies, including their dynamic coupling. Usually the fluid in each tank will be bounded above by a free surface, but in special cases a rigid boundary surface can be placed above the fluid to represent a tank entirely filled by fluid. The following discussion pertains to the situation where a free surface is present in each tank. The free surface boundary condition in each tank is linearized in the same manner as for the exterior free surface. Special attention is required near the eigenfrequencies of the tanks, where nonlinear effects are significant. A two-dimensional study of this problem, including nonlinear motions in the tank, is presented in Reference [26].

The tank geometry is defined in the same manner as for the exterior surface of each body, using either the low-order method with conventional panels, or the higher-order method. In both cases it is essential that the normal vector points away from the 'wet' side of the tank surface, as explained in Section 5.1 (ILOWHI=0) and Section 6.1 (ILOWHI=1). In the context of a tank, this means that the normal vector points out of the tank and into the interior space of the body. In the low-order method the tank panels are included with the conventional body panels in the .GDF file. In the higher-order method the tank patches are defined in the same manner as the body surface, using one of the options listed in Sections 6.5 to 6.9. In both cases the tank panels or patches are identified by their starting and ending indices, which must be listed in the .CFG file using the parameter name NPTANK, as explained in Section 3.7.

Test 22 is an example, where the body is an FPSO containing two rectangular internal tanks. Patches 1-7 represent the exterior surface of the FPSO, patches 8-11 represent tank 1, and patches 12-15 represent tank 2. (When IRR=1, this convention is modified, with patches 8-10 used for the interior free surface of the FPSO and patches 11-18 used for the tanks, as explained in the header of the GEOMXACT.DLL subroutine FPSOINT.) One side of each tank is represented by four rectangular patches, with their vertex coordinates included in the GDF file TEST22.GDF.

The panels or patches which represent each tank must be contiguous. Separate tanks can be grouped together, or interspersed arbitrarily within the description of the exterior surface. It is recommended to group all of the tanks together, either at the beginning or at the end of the panels/patches which define the exterior surface. When IRR=1 and ILOWHI=0, the interior free surface panels must be placed at the end of the GDF file, as explained in Section 9.1, and thus if tanks are included they must be located before the panels which define the interior free surface.

The vertical positions of tanks can be specified arbitrarily, in other words the free surface of each tank can be defined independently of the other tanks and the exterior free surface. For each tank the free surface is defined to coincide with the highest point of the panel vertices or patch corners defining that tank. In TEST 22, the free surface of one tank coincides with the exterior free surface, i.e. the plane $z = 0$, and the free surface of the other tank is elevated.

The solutions for the velocity potential (or source strength) are performed independently for the exterior fluid domain, and for each interior tank. Thus the mutual locations of these surfaces are irrelevant. Small gaps between them do not cause problems, and they may even coincide without introducing numerical difficulties. When irregular-frequency removal is used (IRR>0), the entire interior free surface inside the body should be described, ignoring any possible intersections with tanks. Thus the same interior free surface should be used with or without the presence of tanks.

Two types of tank parameters must be included in the CFG file, as explained in Section 3.7:

NPTANK is an integer array used to specify the panel or patch indices of internal tanks. The data in this array are in pairs, denoting the first and last index for each tank. An even number of indices must be included on each line. More than one line can be used for multiple tanks, and/or multiple tanks can be defined on the same line. If NBODY>1, the body number(s) for each body containing tanks must be appended to the parameter name. Only integer data are read for the array NPTANK, with spaces separating each index. Other ASCII characters may be used to delimit the data in pairs.

RHOTANK is a real array used to specify the density of fluid in internal tanks. The density specified is relative to the density ρ of the fluid in the external domain outside the bodies, as defined in Chapter 4. The data in the array RHOTANK must be input in the same order as the data in the array NPTANK. Multiple lines of this parameter may be used, with an arbitrary number of data on each line, but each line must begin with 'RHOTANK='. The total number of tanks NTANKS is derived from the inputs NPTANK in the POTEN run. If fewer than NTANKS values of RHOTANK are specified, the remainder of the array is assigned the last non-negative value. Thus if the density is the same for all tanks, only the first value is required. Zero may be assigned, but negative values of the density must not be assigned. RHOTANK is only used in the FORCE run, and may be changed if separate FORCE runs are made using the same POTEN outputs. Further details and examples are given in Section 10.8.

The following are equivalent formats for the required lines in the file TEST22.CFG:

```
NPTANK= (8-11) (12-15) (patch indices for two tanks)
RHOTANK= 0.6 0.6 (fluid densities for tanks one and two)


NPTANK= 8 11 (patch indices for tank one)
NPTANK= 12 15 (patch indices for tank two)
RHOTANK= 0.6
```

These examples illustrate the following rules:

1. Only integer data are recognized in NPTANK. Arbitrary ASCII characters other than these can be used both as comments and to delimit the pairs of indices for each tank, according to the user's preferences. At least one blank space must be used to

separate pairs of indices. Comments appended to these lines must not include integer characters.

2. The total number NTANKS of tanks to be included is determined by the number of pairs of indices in NPTANK inputs, in this case NTANKS=2. The same number of densities RHOTANK is required for the analysis, but it is not necessary to input repeated values if all of the densities are the same (or if all of the densities after a certain point are the same). The order of the tank densities must correspond to the order of the index pairs in NPTANK.

3. The data in RHOTANK are real numbers. After specifying all NTANKS values, arbitrary comments can be appended as in the first example above, but if fewer than RHOTANK real numbers are assigned the remainder of the line should be left blank, as in the second example above.

When field pressures and/or velocities are required for field points inside the tanks (Options 6-7) special inputs are required. In this case, when the field point coordinates XFIELD are input as explained in Section 3.3, the parameter ITANKFPT=1 must be specified in the .cfg file, as explained in Section 3.7, and the format of the XFIELD inputs in the .frc file must include the corresponding number of each tank, or zero for the exterior domain. When all of the field points are defined by arrays, using the option described in Section 3.10, the parameter ITANKFPT is not used and may be deleted from the .cfg file.

Inputs which relate to the body's mass including VCG and the radii of gyration XPRDCT (IALTFRC=1), or XCG, YCG, ZCG and the inertia matrix EXMASS (IALTFRC=2) refer to the mass of the body alone, without the tanks (or with the tanks empty). The same definitions apply to the outputs of these quantities in the header of the .out file and in the log file wamitlog.txt. When IALTFRC=1 is used, the body mass is derived from the displaced fluid mass corresponding to the body volume, minus the fluid mass in the tanks.[1]

The body volumes, center of buoyancy, and hydrostatic restoring coefficients displayed in the header of the .out file are calculated from the exterior wetted surface of the body, and are not affected by the tanks. The volumes, densities, values of ZTANK, and hydrostatic restoring coefficients for the tanks are listed separately after the corresponding data for the hull. When the higher-order option (ILOWHI=1) is used, the header of the .out file includes a list of all the patches, and the corresponding tank numbers ITANK for patches which are defined as interior tanks.

Hydrodynamic parameters which are physically relevant for the tanks alone (options 1,5,6,7,9) can be computed by inputting only the patches or panels for the tanks. In this case IDIFF=-1 should be used, and the outputs for options 5,6,7,9 correspond to the combination of radiation modes specified by the IMODE array, with unit amplitude of each mode. The damping coefficients for the tanks should be practically zero. From momentum

[1] In WAMIT Versions prior to 6.312 the inertia matrix output in the log file wamitlog.txt when IALTRFRC=1 is not reduced by the fluid mass in the tanks. The correct inertia matrix is used in the equations of motion and in all hydrodynamic outputs which depend on the RAO's. Users of earlier versions can correct the output in the log file by multipyling each element of the inertia matrix by the factor $1 - (\mathrm{RHOTANK} \times \mathrm{VOL}_T)/\mathrm{VOL}_B)$. Here VOL_T is the sum of the volumes of all tanks in the body and VOL_B is the volume of the body. These quantities are all specified in the header of the .out file.

conservation the outputs from option 8 are zero, and the horizontal drift forces and vertical drift moment from option 9 should be practically zero.

Internal tanks affect the hydrostatic restoring coefficients C_{ij}, as described in Section 12.10. The complete hydrostatic matrix, which is output in the *out*.hst file, includes the effects of the tanks. Note that these outputs are not the same as the hydrostatic matrix for the hull alone as described above, and displayed in the .out file. For example the hydrostatic coefficient C_{33} in the .out file is equal to the total area of the waterplane inside the waterline of the body, whereas the coefficient C_{33} in the .hst file is reduced by the product of the free-surface area and density for each tank.

When both tanks and the external hull surfaces are included, hydrodynamic and hydrostatic parameters which are relevant for the hull alone, with no internal tanks, can be computed by setting RHOTANK=0.0

10.8 RADIATED WAVES FROM WAVEMAKERS IN TANK WALLS

Version 6.3 includes the option to analyze the waves generated by one or more wavemakers in the plane(s) of symmetry $x = 0$ and/or $y = 0$. This configuration corresponds to a wave tank with rectangular walls in the planes of symmetry. The opposite walls of the wave tank are assumed to have absorbing beaches, represented here by open domains extending to infinity. If the symmetry indices ISX=1, ISY=1 are input, with wavemakers in one or both planes of symmetry, the walls extend to infinity in both the x and y directions. Alternatively, with ISX=1 and ISY=0 and wavemakers in the plane $x = 0$, or conversely, one wall extends to both $\pm$ infinity corresponding to a wave tank of infinite width. In all cases it is assumed that the wall(s) are planes of symmetry, and the fluid motion is symmetrical about these planes. Thus the solution for the velocity potential in the fluid domain can be represented by a distribution of sources of known strength, proportional to the normal velocity of the wavemaker, and it is not necessary to solve the integral equation for the velocity potential on the wavemakers. This saves considerable computational time, and also avoids the singular solution that would otherwise occur for bodies of zero thickness in the plane of symmetry.

The geometry of each wavemaker is defined in the .gdf file. In the low-order method (ILOWHI=0) a sufficient number of panels must be included on each wavemaker to ensure a converged solution. In the higher-order method (ILOWHI=1) only one patch is required on each wavemaker. If the wavemakers are rectangular (or quadrilateral), the higher-order analysis can be carried out most easily using the option IGDEF=0, as explained in Section 6.5.

The parameter ISOLVE=-1 is used to indicate that the wavemakers are in planes of symmetry, and that the solution of the integral equation should be skipped. Suitable generalized modes must be defined, as explained in Chapter 8, to represent the normal velocity of each wavemaker. IRAD=0 is recommended, with IMODE(1:6)=0, to avoid computing the 6 rigid-body modes. IDIFF=-1 is required. The separate wavemaker elements are

considered to be part of one 'body', with appropriate generalized modes used to represent the independent motion of each element, and NBODY=1. No other bodies can be present within the fluid domain.

The principal outputs are WAMIT options 6 and 7 (potential and fluid velocity at specified field points). If the field point is on the free surface the potential is equivalent to the wave elevation. No other options are supported. If multiple wavemakers are run together with separate modes for each wavemaker, the parameter INUMOPT6=1 should be specified in the .cfg file to provide separate outputs for each mode. In that case only the complex amplitude is output, with a separate pair of columns for each mode, as indicated in Section 4.9. A post-processor should be used to combine these outputs, for arbitrary combinations of simultaneous motion of all wavemakers.

In the NEWMODES DLL file the subroutine WAVEMAKER is included, to analyze one or more paddle-type wavemaker segments, with pitching motions about a horizontal axis in the lower edge of each wavemaker. The depth of this axis must be specified in the input file WAVEMAKER_DEPTH.DAT. In the subroutine the depth of the axis is the same for all wavemaker elements. The number of wavemakers is arbitrary, but each separate mode of motion corresponds to one patch (or panel) of the geometry, in the same order as these are defined in the .gdf file. In the low-order method this effectively restricts the use of the subroutine to only one panel per wavemaker. Thus it is strongly recommended to use the higher-order method (ILOWHI=1) when using the subroutine WAVEMAKER. NEWMDS=NPATCH must be specified in the pot or cfg file, with the same value as the number of patches in the gdf file. To specify the use of WAVEMAKER the parameter IGENMDS=21 is defined in the CFG file.

TEST23, described in Appendix A23, illustrates the use of this option for a bank of eight wavemakers along the wall $x = 0$, with symmetry about both $x = 0$ and $y = 0$. The generated wave elevations for each wavemaker are evaluated over a square array of $8 \times 8 = 64$ field points. The depth of the horizontal axis is specified by the parameter ZHINGE=-2m in the input file WAVEMAKER_DEPTH.DAT.

Chapter 11

VERSION 6.3S (Second-order module)

This chapter is not completed and will be updated with the release of V6.3S.

WAMIT Version 6.3S designates a program which extends the capabilities of WAMIT Version 6.3 to include the complete analysis of second-order hydrodynamic quantities, as in the previous version of the second-order module, V5.3S. V6.3S has all of the capabilities of V6.3, as described in Chapters 1-10. The extended capabilities of V6.3S include the sum- and difference-frequency components of the second-order forces and moments (Quadratic Transfer Functions, or QTF), the second-order hydrodynamic pressure on the body and in the fluid domain, the second-order wave elevation and the second-order Response Amplitude Operator (RAO), all in the presence of bichromatic and bidirectional waves and one or more structures. The structures can be either freely floating, constrained or fixed.

The list of major updates from V5.3S and V6.1S is as follows.

1. The higher-order method is implemented and the second-order solution can now be evaluated either by the low-order method or higher-order method. All of the geometric input formats described in Chapters 5 and 6 can be used for the second-order solution. When the higher-order method is specified (ILOWHI=1), the second-order solution is represented by B-splines in the same manner as for the linear solution. But the forcing on the body and the free surface are integrated in a piecewise manner as in the low-order method. Using this hybrid approach, the continuous second-order solution is obtained with efficient evaluation of the integral of the forcing.

2. An option for automatic free-surface discretization is implemented. The discretization is made in an optimum manner based on the geometric data in GDF files. This simplifies the use of the second-order extension, particularly for the analysis of multi-body interactions.

3. V6.3S is written in Fortran 90/95. The parameter statements in the earlier versions are removed and arrays are allocated dynamically at runtime.

4. The second-order hydrodynamic pressure can be output at user-specified points in the same manner as the linear pressure.

5. The second-order quadratic pressure force acting along the waterline is output along with the body pressure. This output may be useful when the pressure force is integrated locally to evaluate the cross-sectional force/moment of structures. The geometric data of the waterline segments are output in the PNL file.

The second-order output may not be available in some situations. They include i) the generalized mode (NEWMDS) option is not allowed, ii) fixed/free modes option is not allowed, iii) walls are not allowed and iv) the zero-thickness structure is not allowed. When the second-order RAO of constrained structures are evaluated, the external constraints may not be accounted fully. More specifically, the quadratic interaction between the external constraints and the linear body motion can not be included as an input to the program.

This Chapter describes the special features in input and output of V6.3S which differ from those of V6.3. It is recommended to review the previous chapters, if users are not familiar with V6.3. In the following, the preparation of input files is explained followed by the description of the output from V6.3S. The description of the theoretical background of the second-order analysis can be found in References [9], [17], [26] and other documents cited therein.

All input files prepared for V6.3 are required in order to run V6.3S. These files are CFG, GDF, POT, FRC, optional FNAMES.WAM and, when ILOWHI=1, optional SPL. The input files to V6.3 can be used without modifications, when only the linear output and mean drift force are required.

To evaluate the second-order output, new parameters must be added to the CFG (or CONFIG.WAM) and FRC files. In addition, one or two new input files must be prepared as instructed below. The extensions of new files are .PT2 (Potential Control file 2) and .FDF (Free surface Data File). The filenames of PT2 and FDF can be specified in the optional FNAMES.WAM in order to avoid interactive input during batch runs.

11.1 THE CONFIGURATION FILE (CFG)

The parameter **I2ND** must be specified to evaluate the second-order output.

I2ND=0: Do not compute the second-order solution (0 is the default value).

I2ND=1: Compute the second-order solution for the sum- and difference-frequencies listed in the PT2 file.

When I2ND=1, the dimension of the **NOOUT** array parameter in the CFG file must be increased from 9 to 16. The last 7 options correspond to second-order output as described in the next section. For example, by setting

NOOUT=1 1 1 1 0 1 1 1 1 1 1 1 0 1 1 1

all linear and second-order outputs would be printed in the OUT file except the linear and the second-order pressure on the body surface (IOPTN(5) and IOPTN(13)).

When ILOWHI=0 and I2ND=1, the source formulation must be activated by setting ISOR=1. Execution of the program will be aborted with an appropriate error message, if

ISOR=0.

11.2 THE FORCE CONTROL FILE (FRC)

If I2ND=1 in the CFG file, the dimension of the IOPTN array in FRC must be increased from 9 to 16. The last seven elements control the second-order output. (If I2ND=0, only the first 9 elements would be read in the program. The trailing IOPTNs must be on the same line with IOPTN(9). Otherwise the first IOPTN on the next line would be read in by the program as VCG.) As an example, the Alternative Form 1 of Force Control File is shown below.

```
header
IOPTN(1)...IOPTN(9)IOPTN(10)IOPTN(11)IOPTN(12)...IOPTN(16)
VCG
XPRDCT(1,1) XPRDCT(1,2) XPRDCT(1,3)
XPRDCT(2,1) XPRDCT(2,2) XPRDCT(2,3)
XPRDCT(3,1) XPRDCT(3,2) XPRDCT(3,3)
NBETAH
BETAH(1) BETAH(2) ...  BETAH(NBETAH)
NFIELD
XFIELD(1,1) XFIELD(2,1) XFIELD(3,1)
.
.
XFIELD(1,NFIELD) XFIELD(2,NFIELD) XFIELD(3,NFIELD)
```

The definitions of IOPTN(I), for I=1,...,9 are the same as those of V6.3 as listed in Section 3.3. The definitions of IOPTN(I), for I=10,...,16, and the associated numeric output filenames are shown below.

IOPTN(I) = 0 : do not output parameters associated with option I=10,...,15.

IOPTN(I) = 1 : do output parameters associated with option I=10,...,15.

IOPTN(16) can take one of three value 0, 1, or 2:

IOPTN(16) = 0 : do not output the second-order RAO.

IOPTN(16) = 1 : do output the second-order RAO computed using the 'indirect' exciting force.

IOPTN(16) = 2 : do output the second-order RAO computed using the 'direct' exciting force.

Option	Description	File name
10	Quadratic second-order forces	*frc*.10s, *frc*.10d
11	Total second-order forces by indirect method	*frc*.11s, *frc*.11d
12	Total second-order forces by direct method	*frc*.12s, *frc*.12d
13	Second-order hydrodynamic pressure on the body	*frc*.13s, *frc*.13d
14	Second-order hydrodynamic pressure in the fluid	*frc*.14s, *frc*.14d
15	Second-order wave elevation	*frc*.15s, *frc*.15d
16	Second-order response amplitude operator	*frc*.16s, *frc*.16d

The last letters, 's' and 'd', of the numeric filenames signify that the files contain the sum-frequency output and difference-frequency output, respectively.

When IOPTN(13)=1, the second-order hydrodynamic pressure on the body surface is output. If the body is surface piercing, the normalized hydrodynamic pressure force per unit length at the midpoint of the waterline segments is output as well.

When IOPTN(14)=1 or IOPTN(15)=1, the number of field points NFIELD and their Cartesian coordinates XFIELD should be specified in the FRC file. The second-order wave elevations are computed only for the points on the free surface. (A point is defined to be on the free surface if the vertical distance from the free surface, nondimensionalized by ULEN, is less than 10^{-6} in absolute value.)

The total second-order forces of the output Options 11 and 12 are the sum of the two force components. One component is the quadratic force of Option 10 and the other is the second-order potential force. The latter obtained by the 'indirect' method is added to the quadratic force in the output Option 11. In the output Option 12, the second-order potential force is obtained by the 'direct' method. The output options Option 11 and 12 are the same physical forces computed by different approaches and they must converge to each other with finer discretization. The description of these two methods can be found in References [9] and [17].

When two linear wave periods are the same, the difference frequency quadratic force of Option 10 is the same as the mean drift force of Option 9 and Option 8. Since the second-order potential force is trivial in this case, the output Options 11 and 12 are also the same as the mean drift force. The second-order RAO of Option 16 is set to zero in this case. Although the output options 11d, 12d and 16d are trivial, they are included in the corresponding numerical output files. (Note that, in earlier versions of the second-order module, the output was omitted from Options 11d, 12d and 16d, when two periods are identical.)

11.3 THE POTENTIAL CONTROL FILE 2 (PT2)

It is not necessary to prepare the PT2 file, when all of the 2nd-order output options, from 10 to 16, are suppressed.

The PT2 file contains two set of parameters. One specifies mode indices for which the second-order output is calculated. The other specifies period/wave heading pairs for which

the second-order output is calculated.

The input data in the Potential Control File 2 (PT2) are listed below:

header
IRAD2(1) IDIF2(1)
MODE2(1,1) MODE2(2,1) MODE2(3,1) MODE2(4,1) MODE2(5,1) MODE2(6,1)
IRAD2(2) IDIF2(2)
MODE2(1,2) MODE2(2,2) MODE2(3,2) MODE2(4,2) MODE2(5,2) MODE2(6,2)
.
.
IRAD2(NBODY) IDIF2(NBODY)
MODE2(1,NBODY) MODE2(2,NBODY) MODE2(3,NBODY) ... MODE2(6,NBODY)
IXSUM IXDIF
NSUMP
IPER(1) JPER(1) NBETA2(1)
IBETA(1,1) JBETA(1,1)
.
IBETA(1,NBETA2(1)) JBETA(1,NBETA2(1))
.
.
.
IPER(NSUMP) JPER(NSUMP) NBETA2(NSUMP)
IBETA(NSUMP,1) JBETA(NSUMP,1)
.
IBETA(NSUMP,NBETA2(NSUMP)) JBETA(NSUMP,NBETA2(NSUMP))
NDIFP
IPER(NSUMP+1) JPER(NSUMP+1) NBETA2(NSUMP+1)
IBETA(NSUMP+1,1) JBETA(NSUMP+1,1)
.
IBETA(NSUMP+1,NBETA2(NSUMP+1)) JBETA(NSUMP+1,NBETA2(NSUMP+1))
.
.
.
IPER(NPER2) JPER(NPER2) NBETA2(NPER2)
IBETA(NPER2,1) JBETA(NPER2,1)
.
IBETA(NPER2,NBETA2(NPER2)) JBETA(NPER2,NBETA2(NPER2))

IRAD2(N) is an index for the computation of the radiation solution of the N-th body at the sum- or difference frequency.

IRAD2(N)=-1 Do not compute the second-order radiation solution.

IRAD2(N)= 0 Compute the second-order radiation solution for the modes specified by MODE2.

IRAD2(N)=1: Compute the second-order radiation solution for all modes.

IDIF2(N) is an index for the computation of the second-order diffraction solution at the sum- or difference- frequency.

IDIF2(N)=-1: Do not compute the second-order diffraction solution.

IDIF2(N)=0: Compute the second-order diffraction solution for the modes specified by MODE2.

IDIF2(N)=1: Compute the second-order diffraction solution for all modes.

MODE2(1∼6) represent the modes for which the second-order radiation and diffraction solutions are to be computed.

IXSUM, IXDIF are indices controlling the selection of periods and wave headings to form pairs.

IXSUM=0: Do not compute the sum-frequency second-order solution.

IXSUM=1: Compute the sum-frequency second-order solution only for selected combinations of the wave periods and headings listed below.

IXSUM=2: Compute the sum-frequency second-order solution for all combinations of the wave periods and headings (PER and BETA) listed in the POT file.

IXDIF=0: Do not compute the difference-frequency second-order solution.

IXDIF=1: Compute the difference-frequency second-order solution only for selected combinations of the wave periods and headings listed in the PT2 file.

IXDIF=2: Compute the difference-frequency second-order solution for all combinations of the wave periods and headings (PER and BETA) listed in the POT file.

When IXSUM=1 or IXDIF=1, the number of combinations and indices of selected periods and headings should be specified in the PT2 file through the following parameters:

NSUMP is the total number of combinations of wave periods for the sum-frequency solution.

NDIFP is the total number of combinations of wave periods for the difference-frequency solution.

The dimension NPER2, used in the last arrays in PT2, is the total number of period combinations; NPER2 = NSUMP + NDIFP.

IPER, JPER are indices which identify the 1st-order periods selected from an array PER (1∼ NPER).

NBETA2(I) is the total number of wave-heading pairs for the I-th period combination.

IBETA, JBETA are indices which identify the wave headings selected from an array BETA (1∼NBETA).

11.4 FREE SURFACE DATA FILE (FDF)

It is not necessary to prepare an FDF file if IOPTN(10)=1 but all other second-order output are suppressed.

The FDF file contains all requisite data to perform the integration of the quadratic forcing over the entire free surface exterior to the bodies. The integration is carried out by numerical quadratures over the free surface close to the body. Away from the body, the integration can be performed efficiently using asymptotic approximation of the forcing. For this reason, the free surface is divided into two regions by a 'Partition Circle'. The radius of the partition circle, which will be denoted by R_{part}, should be sufficiently large so that the asymptotic expansion of the forcing is valid outside of this circle.

For better computational efficiency, the region inside R_{part} may be divided into two subregions by an inner circle which is smaller than the partition circle. One subregion is the free surface inside the inner circle and exterior to the bodies. The other is an annular region between the inner and partition circles. The inner region is discretized into quadrilateral panels and the integration is performed piecewise for each panel. The intermediate region may be subdivided into one or more annuli and the integration performed based on the Gauss Chebyshev quadrature in the azimuthal direction and the Gauss Legendre quadrature in the radial direction over each annulus. In the outer region, semi-analytic integration is performed based on the asymptotic approximation of the integrand.

There are two alternative forms for the FDF file. In a simpler form, the free surface panels in the inner region is generated automatically inside the program. In the other form, the user must specify the horizontal coordinates of the panel vertices.

The data in the FDF file should be given relative to the **global** coordinate system.

11.4.1 ALTERNATIVE Form 1 FDF

When this form is used, the discretization of the inner region is carried out automatically inside the program. The data in this form of FDF are

header
PARTR
NPF SCALE
NAL DELR NCIRE NGSP

PARTR is the dimensional radius of the inner circle with the center at the origin of the global coordinate system. PARTR is measured in the same units as the length ULEN.

If PARTR is smaller than 1.25 times the maximum distance from the body surface to the origin of the global coordinate system, the program issues a warning and increases PARTR automatically.

NPF is an integer flag specifying Alternative forms of the FDF file. In Alternative 1 format, NPF must be a **negative** integer.

SCALE is a real number used as a scaling factor of the size of the free surface panels relative to the average length of the waterline panels on the body. For example, if SCALE=1.5, the length scale of free surface panels will be 1.5 times the waterline panels.

The following four input parameters are relevant to the integration on the intermediate region:

NAL is the total number of annuli subdividing the intermediate annular region. When NAL=0, the parameters DELR, NCIRE and NGSP need not be specified.

DELR is the dimensional radial increment of each annulus (in the same units as ULEN).

NCIRE is the integer exponent of 2 such that the total number of nodes is equal to $2^{\text{NCIRE}} + 1$ for the azimuthal integration based on Gauss-Chebyshev quadrature.

NGSP is the number of nodes used for the Gauss-Legendre radial quadrature on each annulus.

The Gauss quadrature over the intermediate region may be more efficient than the integration using quadrilateral panels, especially when the partition circle is large compared to the bodies. However, choosing appropriate values for the parameters NCIRE and NGSP may not be simple. Unlike the piecewise integration using panels, inadequate values of these parameters may produce the result with gross error. It is recommended to check the dependence of the results to these parameters.

11.4.2 ALTERNATIVE Form 2 FDF

The data in this form of FDF should be input in the following form:

header
PARTR
NPF NTCL
NAL DELR NCIRE NGSP
VERX(1,1) VERX(2,1) VERX(3,1) VERX(4,1)
VERY(1,1) VERY(2,1) VERY(3,1) VERY(4,1)
.
.
.
VERX(1,NPF) VERX(2,NPF) VERX(3,NPF) VERX(4,NPF)
VERY(1,NPF) VERY(2,NPF) VERY(3,NPF) VERY(4,NPF)

PARTR: same as Alternative 1 FDF

NPF is the total number of free surface panels defined by the user.

NTCL is the total number of segments on the inner circle

NAL : same as Alternative 1 FDF

DELR: same as Alternative 1 FDF

NCIRE: same as Alternative 1 FDF

NGSP: same as Alternative 1 FDF

VERX(K,I), K=1−4 is the dimensional x coordinate of the K-th vertex of the I-th panel.

VERY(K,I), K=1−4 is the dimensional y coordinate of the K-th vertex of the I-th panel.

For a single body, one quadrant of, half of, or the entire inner region must be discretized according to the symmetry of the problem. Likewise, the number of segments NTCL should be specified over the range of one quadrant, half, or the entire circle depending on the symmetry.

For multiple bodies, the entire inner region must be discretized with respect to the global coordinate system, regardless of the symmetry of the individual bodies. The inner circle must enclose all of the bodies.

When there are multiple waterlines, the distance between each pair of waterlines must be greater than twice the average length of waterline segments. This may require fine discretization of the body near the free surface when the gap between the waterlines are small. In this case, larger value of SCALE must be used in order to avoid fine discretization inside inner circle.

11.4.3 PARTITION CIRCLE AND INNER CIRCLE

As noted above, the radius of the partition circle, R_{part}=PARTR+NAL×DELR, should be sufficiently large so that the asymptotic expansion of the potentials is valid outside of the circle. An appropriate estimation of the radius of the partition circle, R_{part}, is $R_{part} \sim O(h)$ (h=water depth) for shallow water and $R_{part} \sim O(\lambda)$ (λ= longest wavelength involved) for deep water ($h >> \lambda$). The actual constant of proportionality R_{part}/λ may have to be substantially larger than one to achieve accuracy in deep water (see Reference [9] and [12]).

If NAL>0, the radius of the inner circle PARTR should be determined with care. If PARTR is too close to the body, the integration over the intermediate region may not be efficient as intended. The Gauss quadrature converges slowly for the integration of the influence of Rankine sources on the free surface, at nearby points on the body.

If we denote the maximum distance to the body surface from the center of the circle as R_{body}, the following table shows the required number of nodes in the azimuthal integration as PARTR varies. In the table, NCIR=$2^{\text{NCIRE}} + 1$ is the number of nodes of the Gauss-Chebyshev quadrature between $[0, 2\pi]$. The recommended procedure to determine PARTR is i) first to select the parameter NCIRE which would be dependent on the wavelengths of the linear and second-order waves and ii) select R_{part} from the table.

R_{body}/PARTR	NCIRE	NCIR
0.1	3	9
0.4	4	27
0.5	5	33
0.8	6	65
0.9	7	129
0.95	9	257

When field quantities are evaluated on the free surface, such as the second-order wave elevation, similar consideration applys. Thus PARTR should be sufficiently large so that the maximum radial distance to the field points from the origin is substantially less than PARTR. The above table can be used to estimate PARTR after replacing R_{body} with the maximum radial distance to the field points.

11.4.4 RESTRICTION ON VERY LONG WAVES

When k_iR, k_jR or $k_2^{\pm}R$ is less than BOUND in the DATA statement of subroutine FARFS.F and RHSFFS.F, the program will continue to run but neglect the integration over the outer region in the total second-order exciting forces. Here R is the partition radius and k_i, k_j and $k_2^{\pm}$ are wavenumbers associated with the frequencies ω_i, ω_j and $\omega_i \pm \omega_j$. This case may be encountered when a very long wavelength is involved in the difference-frequency solution. If this occurs a warning message will appear in the .OUT file. To avoid this condition the user may either increase the partition radius or increase the spread between first-order wave periods.

In V6.3S, BOUND is set to 10^{-3}. The parameter BOUND should be as small as possible, depending on the range of the double-precision floating-point decimal exponent of the computing system. The source code users can modify this value following the guideline that when the exponent is in the range $\pm$ 64 $\times$ n, the recommended value of BOUND is 10^{-n}.

11.4.5 APPROXIMATION WITHOUT FREE SURFACE FORCING

Users can compute the second-order solution without the evaluation of the free surface integral. In light of the large computational effort for the evaluation of this integral, this option provides an efficient way to compute an approximation to the complete second-order solution. However, since the quality of the approximation depends on the problem, this option must be used with discretion. It is not recommended to use this option for the sum-frequency problem where the free-surface forcing is relative important. (Reference [15] provides an example of computational results showing the relative importance of the free-surface integral for a particular structure.)

To use this option, the parameters NPF, NTCL and NAL in the FDF file should be set equal to zero as follows. (PARTR can be any number.)

header

PARTR
0 0
0

11.4.6 HIGHER-ORDER METHOD (ILOWHI=1)

With the low-order option specified (ILOWHI=0), the body and the free surface are represented by quadrilateral panels. The unknown potential on the body surface is assumed constant over each panel. Similarly the linear or quadratic forcing on the body and/or free surface is assumed constant on each panel and the integration of this forcing is performed in a piecewise manner.

With the higher-order option specified (ILOWHI=1), the potential on the body surface is represented by B-splines in both the linear and second-order analyses. In the linear analysis, the forcing on the body surface is assumed continuous and the integration is carried out by Gauss quadrature. In the second-order analysis, however, the integration of the quadratic forcing on the body and the free surface is performed piecewise in the same manner as for ILOWHI=0.

The body surface is virtually discretized into quadrilateral panels by dividing the space between the knot vectors into smaller line segments. The number of line segments, L, is $L = K + 1$, where K is the order of B-spline. If K is the same for all patches, the virtual panels are the same as the low order geometric data in *gdf*_low.GDF which is output by the program, with the parameter ILOWGDF= $K + 1$. These panels can be visualized using a program such as TECPLOT with the input data *gdf*_pan.DAT, by setting IPLTDAT=$K+1$ in the CFG file.

11.4.7 VISUALIZING FDF

If Alternative Form 1 of FDF is used, the program identifies the panels (virtual panels when ILOWHI=1) on the waterline. Then the free surface panels are generated in a continuous manner starting from the sides of body panels on the waterline by sharing the same vertices. It is advised to arrange the free surface panels in a similar manner near the waterline, when Alternative 2 format is used.

When the Alternative Form 1 of FDF is used, the program outputs the file *fdf*_new.FDF. This is an Alternative Form 2 format of FDF containing the coordinates of the vertices of the free surface panels generated by the program. For both Alternatives of the FDF file, the program output *fdf*_fdf.dat contains the free surface panels in TECPLOT data format. (*fdf* denotes the original filename of Alternative 1 format FDF).

11.5 FNAMES.WAM

When FNAMES.WAM is used to list the input files, the latter should contain the PT2 and FDF file names as well as CFG, GDF, POT, FRC as in the following example:

FNAMES.WAM:
test.cfg
test.gdf
test.pot
test.frc
test.pt2
test.fdf

If all second-order outputs are suppressed, pt2 and fdf can be omitted. If all second-order outputs are suppressed except Option 10, fdf can be omitted.

11.6 DEFINITIONS OF OUTPUT

The non-dimensional definitions of the second-order output from V6.3S are listed in this Section. Note that the second-order outputs satisfy corresponding symmetry relations. For example, the wave excitation Quadratic Transfer Functions (QTF) at the sum- and difference-frequencies satisfy the relations $F_{ij}^{+} = F_{ji}^{+}$, and $F_{ij}^{-} = F_{ji}^{-*}$, respectively. Here $*$ denotes the complex conjugate. (In the difference-frequency problem, $\omega_i \geq \omega_j$ is assumed. Otherwise, the indices are interchanged automatically within the program.)

1. The second-order wave forces and moments at sum- and difference-frequencies are defined as

$$\bar{F}^{+} = \frac{F^{+}}{\rho g L A_i A_j} \quad \text{and} \quad \bar{F}^{-} = \frac{F^{-}}{\rho g L A_i A_j^{*}}$$

$$\bar{M}^{+} = \frac{M^{+}}{\rho g L^2 A_i A_j} \quad \text{and} \quad \bar{M}^{-} = \frac{M^{-}}{\rho g L^2 A_i A_j^{*}}$$

where L=ULEN is the characteristic body length, and A, g and ρ represent the complex first-order incident-wave amplitude, gravitational acceleration and fluid density, respectively.

2.1 The second-order hydrodynamic pressure at sum- and difference-frequencies is defined as

$$\bar{p}^{+} = \frac{p^{+}}{\rho g A_i A_j / L} \quad \text{and} \quad \bar{p}^{-} = \frac{p^{-}}{\rho g A_i A_j^{*} / L}$$

2.2 The second-order hydrodynamic force per unit length acting on the waterline at sum- and difference-frequencies is defined as

$$\bar{f}_w^+ = \frac{f_w^+}{\rho g A_i A_j} \qquad \text{and} \qquad \bar{f}_w^- = \frac{f_w^-}{\rho g A_i A_j^*}$$

3. The second-order wave elevation at sum- and difference-frequencies is defined as

$$\bar{\eta}^+ = \frac{\eta^+}{A_i A_j / L} \qquad \text{and} \qquad \bar{\eta}^- = \frac{\eta^-}{A_i A_j^* / L}$$

4. The second-order response amplitude operator at sum- and difference-frequencies is defined as

$$\bar{\xi}_k^+ = \frac{\xi_k^+}{A_i A_j / L^n} \qquad \text{and} \qquad \bar{\xi}_k^- = \frac{\xi_k^-}{A_i A_j^* / L^n}$$

where $n = 1$ for the translational modes $k = 1, 2, 3$ and $n = 2$ for the rotational modes $k = 4, 5, 6$

11.7 NUMERIC OUTPUT FILES

All requested outputs are listed collectively in the .OUT file and in the numeric output files. The data in the numeric output files are listed as follows:

OPTN.10s, OPTN.11s and OPTN.12s:
PER(i) PER(j) BETA(i) BETA(j) I MOD($\bar{F}_{ij}^+$) PHS($\bar{F}_{ij}^+$) Re($\bar{F}_{ij}^+$) Im($\bar{F}_{ij}^+$)

OPTN.10d, OPTN.11d and OPTN.12d:
PER(i) PER(j) BETA(i) BETA(j) I MOD($\bar{F}_{ij}^-$) PHS($\bar{F}_{ij}^-$) Re($\bar{F}_{ij}^-$) Im($\bar{F}_{ij}^-$)

OPTN.13S:
PER(i) PER(j) BETA(i) BETA(j) M K MOD($\bar{p}^+$) PHS($\bar{p}^+$) Re($\bar{p}^+$) Im($\bar{p}^+$)
(PER(i) PER(j) BETA(i) BETA(j) M K MOD($\bar{f}_w^+$) PHS($\bar{f}_w^+$) Re($\bar{f}_w^+$) Im($\bar{f}_w^+$))

OPTN.13D:
PER(i) PER(j) BETA(i) BETA(j) M K MOD($\bar{p}^-$) PHS($\bar{p}^-$) Re($\bar{p}^-$) Im($\bar{p}^-$)
(PER(i) PER(j) BETA(i) BETA(j) M K (MOD($\bar{f}_w^-$) PHS($\bar{f}_w^-$) Re($\bar{f}_w^-$) Im($\bar{f}_w^-$))

OPTN.14S:
PER(i) PER(j) BETA(i) BETA(j) L MOD($\bar{p}^+$) PHS($\bar{p}^+$) Re($\bar{p}^+$) Im($\bar{p}^+$)

OPTN.14D:
PER(i) PER(j) BETA(i) BETA(j) L MOD($\bar{p}^-$) PHS($\bar{p}^-$) Re($\bar{p}^-$) Im($\bar{p}^-$)

OPTN.15S:
PER(i) PER(j) BETA(i) BETA(j) N MOD($\bar{\eta}^+$) PHS($\bar{\eta}^+$) Re($\bar{\eta}^+$) Im($\bar{\eta}^+$)

OPTN.15D:
PER(i) PER(j) BETA(i) BETA(j) M MOD($\bar{\eta}^-$) PHS($\bar{\eta}^-$) Re($\bar{\eta}^-$) Im($\bar{\eta}^-$)

OPTN.16S:
PER(i) PER(j) BETA(i) BETA(j) I MOD($\bar{\xi}^+$) PHS($\bar{\xi}^+$) Re($\bar{\xi}^+$) Im($\bar{\xi}^+$)

OPTN.16D:
PER(i) PER(j) BETA(i) BETA(j) I MOD($\bar{\xi}^-$) PHS($\bar{\xi}^-$) Re($\bar{\xi}^-$) Im($\bar{\xi}^-$)

Here I is mode index, M indicates the quadrant or half when the body has two or one planes of symmetry, K is the index of the panels on the body, L is the index of the field points.

The numeric output files optn.13s and optn.13d contain the second-order hydrodynamic pressure on the body surface. The points where the pressure is calculated are specified in *gdf*.PNL (see Section 4.9 for the contents in PNL file). If Option 5 is specified, the linear pressure is output for the same set of points.

For the surface-piercing bodies, the second-order pressure force acting on the waterline is of interest. This will be referred to as the waterline pressure force and denoted by $f_w^{\pm}$. When IPNLBPT=0, $f_w^{\pm}$ is output to optn.13s and optn.13d for the surface-piercing bodies. $f_w^{\pm}$ is appended after the output of the pressure($p^{\pm}$) on the body surface.

The PNL file is also extended accordingly and contains the Cartesian coordinates, the length of waterline segments and normal vector corresponding to the points where $f_w^{\pm}$ is calculated. The new additional parameters in the PNL file are shown below.

gdf.PNL: M K XCT YCT ZCT LENGTH n_x n_y n_z $(\mathbf{r}\times\mathbf{n})_x$ $(\mathbf{r}\times\mathbf{n})_y$ $(\mathbf{r}\times\mathbf{n})_z$
where LENGTH is the length of the waterline segments. The definitions of other parameters are the same as those in Section 4.9.

Only the hydrodynamic pressure is output for Option 13 (also Option 5). It would be straightforward to calculate the static pressure, however, using available output from the program. The second-order quadratic hydrostatic pressure, $\bar{P}_{sq}^{\pm}$ normalized in a same manner as the hydrodynamic pressure, can be calculated from the following expression.

$$\bar{P}_{sq}^{\pm} = \frac{1}{4}[(\xi_{4_i}\xi_{6_j}^{\pm} + \xi_{4_j}^{\pm}\xi_{6_i})x + (\xi_{5_i}\xi_{6_j}^{\pm} + \xi_{5_j}^{\pm}\xi_{6_i})y - (\xi_{4_i}\xi_{4_j}^{\pm} + \xi_{6_i}\xi_{6_j}^{\pm})z]$$

where ξ_k is the linear RAO output in optn.4. The subscripts 4,5,6 correspond to the roll, pitch and yaw modes, respectively. ξ^- denotes the complex conjugate of $\xi = \xi^+$. The subscripts $_i$ and $_j$ denote the indices of the periods of the linear incident-wave pair. The normalized position vector of the points on the body surface (x, y and z) can be obtained from the data in the PNL file. Note that the position vector in PNL must be normalized

by ULEN(1) and converted into each body coordinate system before using in the above expression.

The hydrostatic pressure due to the second-order RAO can be calculated from the following expression where $\xi^{\pm}$ is the second-order RAO output optn.16s and optn.16d.

$$\bar{P}^{\pm}_{sp} = \xi^{\pm}_3 + \xi^{\pm}_4 \times y - \xi^{\pm}_5 \times x$$

The linear static pressure can be calculated from the above expression after replacing the second-order RAO with the linear RAO.

The body motion is taken into account in the evaluation of $f^{\pm}_w$ and it includes the effects of both hydrodynamic and static pressure.

11.8 INSTALLATION OF V6.3S-PC Executable

V6.3S-PC Executable can be installed following the same procedure for the installation of V6.3PC Executable as described in Chapter 2. An additional set of standard Test Runs for the second-order analysis is provided in the directory TESTRUNS_2ND.

Chapter 12

THEORY

In this Chapter the theoretical basis for WAMIT is described. Further information can be found in Reference [26], and in the references cited below.

12.1 THE BOUNDARY-VALUE PROBLEM

Figure 12.1 illustrates a three-dimensional body interacting with plane progressive waves in water of finite water depth H. The objective of WAMIT is to evaluate the unsteady hydrodynamic pressure, loads and motions of the body, as well as the induced pressure and velocity in the fluid domain. The free-surface and body-boundary conditions are linearized, the flow is assumed to be potential, free of separation or lifting effects. A harmonic time dependence is adopted.

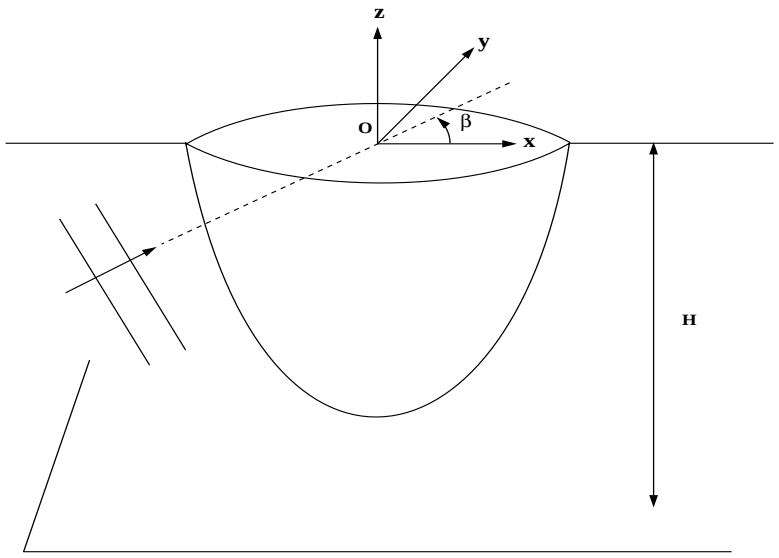


Figure 12.1: Sketch defining the coordinates, fluid depth, and wave-heading angle.

The Cartesian coordinate system (x, y, z) defined in Figure 12.1 is stationary relative to the undisturbed position of the free surface and body. Its origin may lie on, above or below the free surface as the particular application requires. The body geometry input to WAMIT is defined relative to that system. Here, the origin is located on the free surface for the sake of simplicity.

The assumption of a potential flow permits the definition of the flow velocity as the gradient of the velocity potential Φ satisfying the Laplace equation

$$\nabla^2 \Phi = 0 \tag{12.1}$$

in the fluid domain. The harmonic time dependence allows the definition of a complex velocity potential φ, related to Φ by

$$\Phi = \mathrm{Re}(\varphi e^{i\omega t}), \tag{12.2}$$

where Re denotes the real part, ω is the frequency of the incident wave and t is time. The ensuing boundary-value problem will be expressed in terms of the complex velocity potential φ, with the understanding that the product of all complex quantities with the factor $e^{i\omega t}$ applies. The linearized form of the free-surface condition is

$$\varphi_z - K\varphi = 0, \qquad z = 0 \tag{12.3}$$

where $K = \omega^2/g$, and g is the acceleration of gravity. The velocity potential of the incident wave is defined by

$$\varphi_0 = \frac{igA}{\omega} \frac{\cosh[\nu(z+H)]}{\cosh \nu H} e^{-i\nu x \cos\beta - i\nu y \sin\beta}, \tag{12.4}$$

where the wavenumber ν is the real root of the dispersion relation

$$\frac{\omega^2}{g} = \nu \tanh \nu H, \tag{12.5}$$

and β is the angle between the direction of propagation of the incident wave and the positive $x-$axis as defined in Figure 12.1.

The linearization of the problem permits the decomposition of the velocity potential φ into the radiation and diffraction components

$$\varphi = \varphi_R + \varphi_D, \tag{12.6}$$

$$\varphi_R = i\omega \sum_{j=1}^{6} \xi_j \varphi_j, \tag{12.7}$$

$$\varphi_D = \varphi_0 + \varphi_7. \tag{12.8}$$

The constants ξ_j denote the complex amplitudes of the body oscillatory motion in its six rigid-body degrees of freedom, and φ_j the corresponding unit-amplitude radiation potentials. The velocity potential φ_7 represents the *scattered* disturbance of the incident wave by the body fixed at its undisturbed position. We will refer to the sum (12.8) as the *diffraction* potential φ_D.

On the undisturbed position of the body boundary, the radiation and diffraction potentials are subject to the conditions,

$$\varphi_{jn} = n_j, \tag{12.9}$$

$$\varphi_{Dn} = 0, \tag{12.10}$$

where $(n_1, n_2, n_3) = \mathbf{n}$ and $(n_4, n_5, n_6) = \mathbf{x} \times \mathbf{n}$, $\mathbf{x} = (x, y, z)$. The unit vector $\mathbf{n}$ is normal to the body boundary and *points out of the fluid domain.*

The boundary value problem must be supplemented by a condition of outgoing waves applied to the velocity potentials φ_j, $j = 1, \ldots, 7$.

12.2 INTEGRAL EQUATIONS FOR THE VELOCITY POTENTIAL

In WAMIT the boundary value problems (12.1-10) are solved by using Green's theorem to derive integral equations for the radiation and diffraction velocity potentials on the body boundary. The integral equation satisfied by the radiation velocity potentials φ_j on the body boundary takes the form

$$2\pi\varphi_j(\mathbf{x}) + \iint_{S_b} \varphi_j(\boldsymbol{\xi}) \frac{\partial G(\boldsymbol{\xi};\mathbf{x})}{\partial n_\xi} d\boldsymbol{\xi} = \iint_{S_b} n_j G(\boldsymbol{\xi};\mathbf{x}) d\boldsymbol{\xi}. \tag{12.11}$$

where S_b denotes body wetted surface at calm water.

The corresponding equation for the total diffraction velocity potential φ_D is

$$2\pi\varphi_D(\mathbf{x}) + \iint_{S_b} \varphi_D(\boldsymbol{\xi}) \frac{\partial G(\boldsymbol{\xi};\mathbf{x})}{\partial n_\xi} d\boldsymbol{\xi} = 4\pi\varphi_0(\mathbf{x}). \tag{12.12}$$

The diffraction potential may also be obtained from equation (12.8) after solving for the scattered potential φ_7. The equation for the scattered velocity potential is

$$2\pi\varphi_7(\mathbf{x}) + \iint_{S_b} \varphi_7(\boldsymbol{\xi}) \frac{\partial G(\boldsymbol{\xi};\mathbf{x})}{\partial n_\xi} d\boldsymbol{\xi} = -\iint_{S_b} \frac{\partial \varphi_o(\xi)}{\partial n} G(\boldsymbol{\xi};\mathbf{x}) d\boldsymbol{\xi}. \tag{12.13}$$

From the computational point of view, equation (12.12) has some advantages over equation (12.13) in terms of cpu time and the requirement of storage space, because of the relative simplicity of the right-hand side.

The Green function $G(\mathbf{x};\boldsymbol{\xi})$ is referred to as the wave source potential. It is the velocity potential at the point $\mathbf{x}$ due to a point source of strength -4π located at the point $\boldsymbol{\xi}$. It satisfies the free-surface and radiation conditions, and in infinite water depth is defined by

$$G(\mathbf{x};\boldsymbol{\xi}) = \frac{1}{r} + \frac{1}{r'} + \frac{2K}{\pi}\int_0^\infty dk\, \frac{e^{k(z+\zeta)}}{k-K} J_0(kR) \tag{12.14}$$

$$r^2 = (x-\xi)^2 + (y-\eta)^2 + (z-\zeta)^2 \tag{12.15}$$

$$r'^2 = (x-\xi)^2 + (y-\eta)^2 + (z+\zeta)^2, \tag{12.16}$$

where $J_0(x)$ is the Bessel function of zero order. In finite depth, the Green function is defined by

$$G(\mathbf{x};\boldsymbol{\xi}) = \frac{1}{r} + \frac{1}{r''} + 2\int_0^\infty dk\, \frac{(k+K)\cosh k(z+H)\cosh k(\zeta+H)}{k\sinh kH - K\cosh kH}\, e^{-kH} J_0(kR) \tag{12.17}$$

$$(r'')^2 = (x-\xi)^2 + (y-\eta)^2 + (z+\zeta+2H)^2. \tag{12.18}$$

In both expressions (12.14) and (12.17) the Fourier $k-$integration is indented above the pole on the real axis in order to enforce the radiation condition. Efficient algorithms for the evaluation of the infinite and finite-depth wave-source potentials and their spatial derivatives, have been developed in [1] and [11]. These algorithms are coded in the subroutines VGRN00 and HGRN00 which are included in WAMIT. (These subroutines supersede the earlier versions of the subroutines FGRN89, HGRN89 and IGRN89 or VGRN94, HGRN94, VGRN98 and HGRN98.)

Special attention must be given to the singular components of the Green function for small values of r, r' and r''. The source-like singularities $1/r$, $1/r'$ and $1/r''$ and their normal derivatives can be integrated analytically over a quadrilateral panel, as described in [2]. In addition, the ascending series expansion of the wave source potential for small values of r' (Ref. [1], eq.(5)), reveals the presence of the logarithmic singularity,

$$G(\mathbf{x};\boldsymbol{\xi}) = \frac{1}{r} + \frac{1}{r'} - 2K\log K(r' + |z+\zeta|) + O(1), \tag{12.19}$$

(The derivation of this result in [1] is for the infinite-depth case, but it can be shown from the analysis of the finite-depth case in the same reference that the same singularity applies.) Provision has been made in WAMIT to permit the logarithmic singularity and its derivatives to be integrated analytically in the solution of the integral equations when the source and field points are close to each other and to the free surface. Further details are given in Section 12.4.

12.3 INTEGRAL EQUATIONS FOR THE SOURCE FORMULATION

In this Section a brief description is given of the source distribution method, which is used to derive the fluid velocity components on the body surface in the low-order method. The velocity are required to evaluate the mean second-order pressure, from which the mean drift forces and moments can be evaluated in the manner described in Section 12.7.

In source distribution method, the velocity potential is expressed by a distribution of sources only

$$\varphi(\mathbf{x}) = \iint_{S_b} \sigma(\boldsymbol{\xi}) G(\mathbf{x};\boldsymbol{\xi}) d\boldsymbol{\xi} \tag{12.20}$$

After discretizing the body boundary with plane panels with constant source strength on each panel, the potential can be expressed by

$$\varphi(\mathbf{x}_i) = \sum_{j=1}^{N} \sigma(\mathbf{x}_j) \iint_{S_j} G(\mathbf{x}_i;\boldsymbol{\xi}) d\boldsymbol{\xi} \tag{12.21}$$

Denote the normal vector as $\vec{n}$ and the two tangential unit vectors as $\vec{s}$ and $\vec{t}$ on each panel. The three components of the velocity are then given in the $(\vec{n},\vec{s},\vec{t})$ coordinate system as follows:

$$\varphi_n(\mathbf{x}_i) = 2\pi\sigma(\mathbf{x}_i) + \sum_{j=1}^{N} \sigma(\mathbf{x}_j) \iint_{S_j} G_n(\mathbf{x}_i;\boldsymbol{\xi}) d\boldsymbol{\xi} \tag{12.22}$$

$$\varphi_s(\mathbf{x}_i) = \sum_{j=1}^{N} \sigma(\mathbf{x}_j) \iint_{S_j} G_s(\mathbf{x}_i;\boldsymbol{\xi}) d\boldsymbol{\xi} \tag{12.23}$$

$$\varphi_t(\mathbf{x}_i) = \sum_{j=1}^{N} \sigma(\mathbf{x}_j) \iint_{S_j} G_t(\mathbf{x}_i;\boldsymbol{\xi}) d\boldsymbol{\xi} \tag{12.24}$$

12.4 DISCRETIZATION OF THE INTEGRAL EQUATIONS IN THE LOW-ORDER METHOD (ILOWHI=0)

The mean position of the body wetted surface is approximated by a collection of quadrilaterals. Each quadrilateral is defined by four vertices, lying on the body surface. Their Cartesian coordinates are input to WAMIT. They are numbered in the counter-clockwise direction when the panel is viewed from the fluid domain. Instructions on how to input the vertex coordinates are given in Chapter 5.

In general the quadrilaterals defined above are not plane, but if a sufficiently fine discretization is used for a boundary surface with continuous curvature, each element will

approach a plane surface. In this circumstance a plane approximation of the general quadrilateral is defined by the midpoints of each side, which always lie in the same plane. Each *panel* is defined by projecting the four vertices onto this plane. If the coordinates of two adjacent vertices coincide, the quadrilateral panel reduces to a triangular panel.

For bodies of general shape, gaps may exist between panels. Experience suggests that they do not significantly affect the accuracy of the velocity potential and the hydrodynamic forces.

The radiation and diffraction velocity potentials are taken to be constant over each panel. The discretization errors associated with the selection of plane panels and a piecewise constant variation of the velocity potential are of the same order, if the integration of the singular components of the wave source potential over the panels are carried out with sufficient accuracy.

Based on this discretization, the continuous integral equations (12.11) and (12.12) can be reduced to a set of linear simultaneous equations for the values of the velocity potentials over the panels. For the radiation potentials we obtain

$$2\pi\varphi(\mathbf{x}_i) + \sum_{k=1}^{N} D_{ik}\varphi_k = \sum_{k=1}^{N} S_{ik}\left(\frac{\partial\varphi}{\partial n}\right)_k, \tag{12.25}$$

where $i = 1, \ldots, N$, N being the number of panels. For the total diffraction potential

$$2\pi\varphi(\mathbf{x}_i) + \sum_{k=1}^{N} D_{ik}\varphi_k = 4\pi\varphi_0(\mathbf{x_i}). \tag{12.26}$$

The matrices D_{ik} and S_{ik} are defined by

$$D_{ik} = \iint_{s_k} \frac{\partial G(\boldsymbol{\xi}, \mathbf{x_i})}{\partial n_\xi} d\boldsymbol{\xi}, \tag{12.27}$$

$$S_{ik} = \iint_{s_k} G(\boldsymbol{\xi}, \mathbf{x_i}) d\boldsymbol{\xi}, \tag{12.28}$$

where s_k denotes the surface of the $k-$th panel. The 'collocation' points $\mathbf{x_i}$, where the integral equations are enforced, are located at the panel centroids.

The rest of this section describes the computation of the influence coefficients D_{ik} and S_{ik}.

If the index IDIAG=0 in the potential control file (see Section 3.1), the characteristic length scale DIAG(2) for a given panel is equal to the square root of its area; if IDIAG=1, DIAG(2) is equal to the length of the largest diagonal of the panel. The setting of the index IDIAG applies to all the panels. The selection IDIAG=1 is more conservative, and results in an increase in the number of pairs of panels for which the Rankine singularities are integrated analytically. The Rankine and logarithmic singularities of the wave source potential are subtracted and integrated analytically over the panels if the distance between the centroids of the $i-$th and $j-$th panels is less than 10×DIAG(2). This comparison is

based on the maximum DIAG(2) of the pair of panels. Assuming that the collocation point is located on the $i-$th panel, it is reflected about the free surface and the bottom (if present). The distance of its reflections from the centroid of the j-th panel are denoted by r' (above the free surface) and r'' (below the bottom). The integrals of the $1/r'$ and $1/r''$ singularities are evaluated as for the $1/r$ singularity, using analytic integration over the $j-$th panel, when these distances are less than $10\times$DIAG(2). When the index ILOG=1 the logarithmic singularity is subtracted and integrated analytically along with $1/r'$. When any of the Rankine and logarithmic singularities is not subtracted and integrated analytically, it is incorporated in the evaluation of the wave source potential by VGRN00.

The index ILOG, also set in the potential control file, controls the analytical integration of the logarithmic singularity. When ILOG=1, the logarithmic singularity is subtracted and integrated analytically for the same pairs of panels for which the Rankine singularity $1/r'$ is integrated analytically. When ILOG=0 the singularity is included in the evaluation of the wave-source potential and integrated by quadrature.

The integrals of the $1/r, 1/r', \log[r' - (z + \zeta)]$ and $1/r''$ singularities over the panels are frequency independent and are carried out once for all frequencies for the relevant pairs of panels. These integrals are stored in a file for use at each frequency. The evaluation of the Rankine integrals is accompanied by the set-up of an integer array of indices to indicate which singularities have been subtracted for each pair of panels. These indices are input to VGRN00, which in turn evaluates the appropriate component of the wave source potential.

The analytic integration of the Rankine source potentials and their derivatives follows the theory outlined in [2]. Depending on the radial distance to the field point, relative to the panel length-scale DIAG(2), the evaluations are performed either from the exact formulation or from multipole expansions including either second- or fourth-moments of the panel area. The radial distances for transitions between these three algorithms are based on the requirement of six-decimal relative accuracy. The formulae used for the analytic integration of the logarithmic singularity are derived in [6].

The integration over each panel of the regular part of the wave source potential (evaluated in VGRN00) is carried out by quadrature. Two alternative quadrature formulae are used in WAMIT, depending on the input parameter IQUAD. If IQUAD= 0 the centroid integration formula is used, where the integral over each panel is approximated by multiplying the value of the integrand at the centroid by the panel area. If IQUAD= 1 the Gauss integration formula is used with four nodes and corresponding weights over each panel, by mapping their surface to a unit square using a bilinear transformation. The wave source potential can be integrated over the panels by either method. When the centroid integration is selected, WAMIT exploits the symmetry of the wave source potential with respect to the position of the source and field points to evaluate the wave source potential once for each pair of panels. Thus, if there are N panels and no planes of symmetry, $N^2/2$ calls are made to VGRN00. The selection of the four-node Gauss quadrature formula requires $4N^2$ calls to VGRN00 since the wave source potential symmetries cannot be utilized. For a sufficiently large number of panels, the use of the centroid integration has been found to be sufficiently accurate, and is recommended.

12.5 DISCRETIZATION OF THE INTEGRAL EQUATIONS IN THE HIGHER-ORDER METHOD (ILOWHI=1)

The mean body surface is defined by 'patches', as explained in Chapter 6. Each patch must be a smooth continuous surface. The Cartesian coordinates $\mathbf{x} = (x, y, z)$ of a point on the patch are expressed in term of two parametric coordinates $\mathbf{u} = (u, v)$. The latter is normalized so that they vary in ± 1. Details of geometric description of the body surface are provided in Section 6.1.

The velocity potential on each patch is represented by a product of B-spline basis functions $U(u)$ and $V(v)$ as shown in equation (6.4). The total number of B-spline basis functions on each patch is M_u times M_v.

Upon substituting equations (6.3) and (6.4), (12.11) can be expressed in a form

$$\begin{aligned} 2\pi \sum_{m=1}^{M_v} \sum_{l=1}^{M_u} (\phi_j)_k \mathbf{U}_k(\mathbf{u}_f) \quad &+ \quad \sum_{n=1}^{N_p} \iint d\mathbf{u} \sum_{m=1}^{M_v} \sum_{l=1}^{M_u} (\phi_j)_k \mathbf{U}_k(\mathbf{u}) \frac{\partial G(\mathbf{u}; \mathbf{u_f})}{\partial n(\mathbf{u})} J(\mathbf{u}) \\ &= \sum_{n=1}^{N_p} \iint d\mathbf{u} \frac{\partial \phi_j}{\partial n}(\mathbf{u}) G(\mathbf{u}; \mathbf{u_f}) J(\mathbf{u}) \end{aligned} \tag{12.29}$$

where, $\mathbf{U}_k = U_l(u)V_m(v)$, $\iint d\mathbf{u} = \int_{-1}^{1} dv \int_{-1}^{1} du$, $J(\mathbf{u})$ is Jacobian and N_p denotes the number of patches.

Following the Galerkin method, (12.29) is multiplied by $\mathbf{U}_i$ and integrated over each patch. This results in the linear system

$$2\pi d_{ik}^H (\phi_j)_k + \sum_{k=1}^{N} D_{ik}^H (\phi_j)_k = S_i^H \tag{12.30}$$

for the radiation potential. Similarly, from (12.12), we obtain

$$2\pi d_{ik}^H (\phi_D)_k + \sum_{k=1}^{N} D_{ik}^H (\phi_D)_k = I_i^H \tag{12.31}$$

for the diffraction potential.

In the above equations, $(\phi_j)_k$ and $(\phi_D)_k$ are unknown coefficients of basis function for the radiation and diffraction potentials, respectively. The matrices d_{ik}^H and D_{ik}^H and vectors S_i^H and I_i^H are defined by

$$d_{ik}^H = \iint d\mathbf{u}_f \mathbf{U}_i(\mathbf{u}_f) \mathbf{U}_k(\mathbf{u}_f) \tag{12.32}$$

$$D_{ik}^H = \iint d\mathbf{u}_f \mathbf{U}_i(\mathbf{u}_f) \iint d\mathbf{u} \mathbf{U}_k(\mathbf{u}) \frac{\partial G(\mathbf{u}; \mathbf{u}_f)}{\partial n(\mathbf{u})} J(\mathbf{u}) \tag{12.33}$$

$$S_i^H = \iint d\mathbf{u}_f \mathbf{U}_i(\mathbf{u}_f) \iint d\mathbf{u} \frac{\partial \phi_j}{\partial n}(\mathbf{u}) G(\mathbf{u}; \mathbf{u}_f) J(\mathbf{u}) \tag{12.34}$$

$$I_i^H = \iint d\mathbf{u}_f \mathbf{U}_i(\mathbf{u}_f)\varphi_0(\mathbf{u}_f) \tag{12.35}$$

As explained in Chapter 6, a set of B-spline basis functions is defined by the order of the polynomial (K_u and K_v) and the number of panels (N_u and N_v). In general, the basis function has nonzero value over a part of the patch. For example, $U_l(u)V_m(v)$ is nonzero on the panels from $l - K_u + 1$- (or 1, if $l - K_u + 1 < 1$) to l-th panels (or N_u, if $l > N_u$) in u and $m - K_v + 1$- (or 1, if $m - K_v + 1 < 1$) to m-th (or N_v, if $m > N_v$) panels in v.

The integration in $\mathbf{u}_f$ over each panel is referred to as the 'outer' integration. This is carried out by Gauss-Legendre quadrature. The order of the quadrature is specified by the input parameters IQUO and IQVO in the SPL file, for u and v respectively, or by the parameter IQUADO in CONFIG.WAM. The order of the basis functions are specified by the parameters KU and KV in the SPL file, or KSPLIN in CONFIG.WAM.

The *inner* integrations in $\mathbf{u}$ in equations (12.33) and (12.34) are carried out as described below, for each abscissa of the Gaussian quadrature for the outer integral, $\mathbf{u}_f$.

The integration of the regular part of the wave source potential (evaluated in VGRN00) is carried out by Gauss-Legendre quadrature. The order of the inner quadrature is specified by the input parameters IQUI and IQVI in the SPL file or by the parameter IQUADI in CONFIG.WAM. Numerical tests suggest that the order of the quadrature should be one order higher than the order of the basis function.

The integrals involving the Rankine source and normal dipole are evaluated in the manner explained next. If the field point $\mathbf{u}_f$ is on the panel, the integrand is singular at this point. Otherwise the integrand is regular throughout the domain of the panel. In the singular case, the panel is subdivided into a small square subdomain centered at the field point $\mathbf{u}_f$, and one or more rectangular subdivisions adjoining the square as required to cover the remainder of the panel. The integrals over the latter subdivisions are treated in the same manner as for the other panels where the integrand is regular.

The integrals where the integrand is regular are evaluated by Gauss-Legendre quadrature. If the field point is near the panel, the panel is subdivided into four smaller panels. This subdivision is repeated until the size of the subdivided panel is less than a prescribed multiple of the physical distance from $\mathbf{u}_f$ to the centroid of the panel. For this purpose the size of the panel is defined as the maximum physical length from the center of the panel to its four vertices. The maximum size permitted without further subdivision is $1/\sqrt{2.25} = 2/3$. In some cases a large number of subdivisions are required, particularly when $\mathbf{u}_f$ is close to the sides or vertices of the panel. In this case, the program terminates subdivision after the domain is subdivided into 2048 subdomains. The program issues a warning message to the monitor and error log file but continues without interruption. Experience indicates that this warning message is most likely to occur when the mapping of a physical surface onto a patch is singular, as at the poles of a spherical or spheroidal surface, and that the accuracy of most relevant hydrodynamic outputs are not affected significantly by this problem.

The singular integral over the square subdomain centered at $\mathbf{u}_f$ is explained below. The integration of the dipole is defined in the 'principal-value' sense excluding a vanishingly small area around the origin. With this definition for the dipole integral, both the source

and dipole distributions can be evaluated in a same manner.

After an appropriate normalization of the length and use of a local coordinates with the origin at (u_f, v_f), the integral takes a form

$$I = \int_{-1}^{1}\int_{-1}^{1} \frac{A(u,v)}{|\mathbf{x}(u,v)|} du dv \tag{12.36}$$

where $A(u,v)$ is regular function. $|\mathbf{x}(u,v)|$ is the distance between the source and field points in physical space.

For simplicity, we consider only a case where $|\frac{\partial \mathbf{x}}{\partial u}| = 1$, $|\frac{\partial \mathbf{x}}{\partial v}| = 1$ and $\frac{\partial \mathbf{x}}{\partial u} \cdot \frac{\partial \mathbf{x}}{\partial v} = 0$ but the analysis below can be applied directly to the general case. Since $|\mathbf{x}|/\sqrt{u^2+v^2}$ is regular and thus (12.36) can be expressed in the form

$$I = \int_{-1}^{1}\int_{-1}^{1} \frac{f(u,v)}{\sqrt{u^2+v^2}} du dv \tag{12.37}$$

where $f(u,v)$ is regular.

The singularity at the origin is removed by subdividing the square domain into 4 isosceles triangles with a common vertex at the origin and evaluating the integral separately over each triangle. For example, the integral over a triangle with a side on $u = 1$ is

$$I^{(1)} = \int_0^1 du \int_{-u}^{u} dv \frac{f(u,v)}{\sqrt{u^2+v^2}} \tag{12.38}$$

After the change of variables $u = p$, $v = pq$, we have

$$I^{(1)} = \int_0^1 dp \int_{-1}^{1} dq \frac{f(p,pq)}{\sqrt{1+q^2}} \tag{12.39}$$

After adding the contributions from other three triangles, we have

$$I = \int_{-1}^1 dp \int_{-1}^{1} dq \frac{f(p,pq) + f(pq,p)}{\sqrt{1+q^2}} \tag{12.40}$$

Next we remove the square root function from (12.40). By change of the variables $q = \sinh(as)$, we have

$$I = a \int_{-1}^1 dp \int_{-1}^{1} ds f(p, p\sinh as) + f(p\sinh as, p) \tag{12.41}$$

where $a = \sinh^{-1} 1$. The integral (12.41) is evaluated by applying the Gauss-Legendre quadrature in p and s coordinates.

The integral of the log singularity in the free-surface component of the source potential can be evaluated either in a similar manner as for the Rankine source, or together with the regular part of the wave source potential. The index ILOG, set in the potential control file or in CONFIG.WAM controls these options. When ILOG=1, the logarithmic singularity is subtracted from the wave-source potential and integrated separately. When ILOG=0 it is included in the evaluation of the wave-source potential and integrated by the same quadrature.

12.6 SOLUTION OF THE LINEAR SYSTEMS

The procedure used to solve the linear systems (12.25-6) or (12.30-1) is essentially the same. The dimension of these linear systems is denoted by NEQN (number of equations). In the low-order method NEQN is the same as the number of panels. In the higher-order method NEQN depends on the number of patches, panels, and on the order of the basis functions, as explained in Section 12.5.

If the body geometry has one or two planes of symmetry, half or a quarter of its wetted surface is represented with panels, respectively. Each of the radiation potentials is symmetric or anti-symmetric with respect to the symmetry planes, depending on the rigid-body mode, and the diffraction potential is decomposed into similar components. Thus for one or two planes of symmetry either two or four separate components of the diffraction problem are defined on the basis of their symmetry or anti-symmetry with respect to the x- and y-planes. With this decomposition the dimensions of all linear systems are the same as NEQN.

Flow symmetries and anti-symmetries are enforced in the solution of the integral equations by the method of images. The collocation point $\mathbf{x_i}$ in the argument of the wave source potential, is reflected about the geometry symmetry planes accompanied by a multiplication by $+1$ or -1 for symmetric and antisymmetric flow, respectively.

To avoid unnecessary computations, the architecture of WAMIT permits the analysis of any desired sub-set of the rigid-body modes and of the corresponding diffraction components, based on the settings of the MODE(I) indices in the potential control file (see Section 3.1). For example, if only the heave mode is specified in conjunction with the solution of the diffraction problem, and if there are planes of symmetry, only the symmetric component of the diffraction potential is evaluated. For this reason it is necessary to specify the complete diffraction solution (IDIFF= 1) to evaluate field data (free surface elevation, pressure, and fluid velocity) or to evaluate the drift forces.

With the default value of the input parameter ISOLVE=0, WAMIT solves the linear systems in the radiation and diffraction problems by means of a special out-of-core iterative solver. The maximum number of iterations is controlled by the parameter MAXITT in the config.wam file (Section 3.7), with the default value equal to 35. When convergence is not archived within the limit, a warning message is printed to the on monitor, and the computation continues without interruption. If the number of iterations displayed in the output is equal to MAXITT, this also indicates that convergence does not occur.

A supplementary direct solver for the linear systems may be used for the case for which the iterative solver is nonconverging or slowly converging. It is based on standard Gauss reduction, with partial pivotting. The LUD algorithm is employed for efficiency in solving several linear systems simultaneously, with different right-hand sides. The direct solver also can be used to confirm the iterative solution, and in cases where the number of equations (NEQN) is relatively small the direct solver can result in reduced computing time, particularly if the number of right-hand sides is large.

WAMIT also includes a block-iterative solver to provide increased options in the solution of the linear system. This solver is based on the same algorithm as the iterative solver,

but the local LU decompositions are performed for specified blocks adjacent to the main diagonal. At each iteration the back substitution is performed for each block, at each stage of iterations. This accelerates the rate of convergence, and as the dimension of the blocks increases the limiting case is the same as the direct solver. The opposite limit is the case when the dimension of the blocks is one, which is the result of setting ISOLVE=NEQN; in this case the result is identical to using the iterative solver without blocks (ISOLVE=0).

The convergence of the iterative method is quite different for the low- and higher-order methods. The primary reason for the slow convergence of the higher-order method is that the linear system loses diagonal dominance as the order of basis function increases as shown in the expression for d_{ik}^H in (12.32).

Usually in the low-order method the number of iterations required to obtain convergence is in the range 10-20, and the iterative solver is most effective. In the higher-order method experience indicates that the convergence rate is reduced, and it is generally advisable to use the direct solver (ISOLVE= 1) or, in cases where NEQN is very large, the block-iterative solver (ISOLVE> 1). For special problems where hydrodynamic resonances occur, the convergence rate is reduced and it is necessary to use the direct or block-iterative solver also with the low-order method.

Experience using the low-order method has shown that slow convergence is infrequent, and limited generally to special applications where there either is a hydrodynamic resonance in the fluid domain, as in the gap between two adjacent barges, or in the non-physical domain exterior to the fluid volume. An example of the latter is a barge of very shallow draft, where the irregular frequencies are associated with non-physical modes of resonant wave motion inside the barge. These types of problems can often be overcome by modifying the arrangement of the panels or increasing the number of panels.

For the low-order method various convergence tests have been published in References 5, 6, 9, 10 and 12, based on applications of WAMIT. The accuracy of the evaluated quantities has been found to increase with increasing numbers of panels, thus ensuring the convergence of the discretization scheme. The condition number of the linear systems is relatively insensitive to the order of the linear systems, and sufficiently small to permit the use of single-precision arithmetic. Convergence tests for the higher-order method are reported in References 18, 19, 20, 24 and 25.

12.7 MEAN DRIFT FORCE AND MOMENT BASED ON PRESSURE INTEGRATION

The instantaneous position vector ($\vec{X}$) in an inertial coordinate system of the point fixed on the body fixed coordinate system ($\vec{x}$) is given by

$$\vec{X} = \vec{x} + \vec{\xi} + \vec{\alpha} \times \vec{x} + H\vec{x} \tag{12.42}$$

For the present purpose we consider only the first order quantities of the translational modes $\vec{\xi}$, the rotational modes $\vec{\alpha}$ and the velocity potential ϕ. The roll-pitch-yaw sequence of rotations is adopted, and the transformation matrix is given by

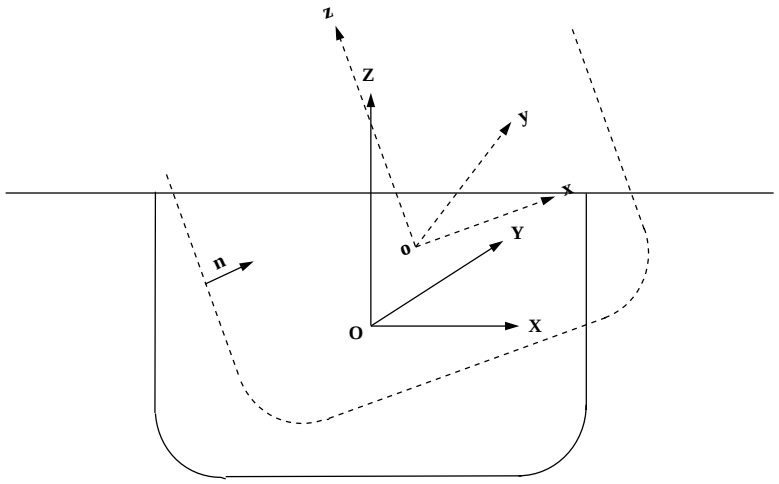


Figure 12.2: Coordinate systems fixed with respect to the body (dashed lines) and its mean position (solid lines)

$$\mathrm{H} = \frac{1}{2}\begin{pmatrix} -({\alpha_2}^2 + {\alpha_3}^2) & 0 & 0 \\ 2\alpha_1\alpha_2 & -({\alpha_1}^2 + {\alpha_3}^2) & 0 \\ 2\alpha_1\alpha_3 & 2\alpha_2\alpha_3 & -({\alpha_1}^2 + {\alpha_2}^2) \end{pmatrix} \tag{12.43}$$

The normal vector in the inertial coordinate system can be expressed by

$$\vec{N} = \vec{n} + \vec{\alpha} \times \vec{n} + H\vec{n} \tag{12.44}$$

where $\vec{n}$ is the normal vector on the body boundary in the body fixed coordinate system.

The pressure at the instantaneous position $\vec{X}$ given in the equation (12.42) can be expressed by

$$\begin{aligned} P = \quad & -\rho[g(z + Z_o) + (\phi_t + g(\xi_3 + \alpha_1 y - \alpha_2 x)) \\ & +(\tfrac{1}{2}\nabla\phi \cdot \nabla\phi + (\vec{\xi} + \vec{\alpha} \times \vec{x}) \cdot \nabla\phi_t + gH\vec{x} \cdot \nabla z)] \end{aligned} \tag{12.45}$$

where Z_o is the vertical coordinate of O relative to the free-surface.

The mean forces and moments are obtained by time averaging the following expressions for the forces and moments:

$$\vec{F}^{(2)} = \iint_{\tilde{S}_b} \vec{N} P ds \tag{12.46}$$

$$\vec{M}^{(2)} = \iint_{\tilde{S}_b} (\vec{X} \times \vec{N}) P ds \tag{12.47}$$

In equations (12.46) and (12.47), the integrations are over the exact body wetted surface $\tilde{S}_b$.

After substituting (12.42), (12.44), and (12.45) and integrating the hydrostatic components,

$$\begin{aligned}
\vec{F}^{(2)} &= \frac{1}{2}\rho g \int_{WL} \vec{n}'[\zeta^2 - 2\zeta(\xi_3 + \alpha_1 y - \alpha_2 x)]dl \\
&+ \frac{1}{2}\rho g \vec{k} \int_{WL} \frac{n_z}{\sqrt{1-n_z^2}}[\zeta - (\xi_3 + \alpha_1 y - \alpha_2 x)]^2 dl \\
&- \rho \iint_{S_b} \vec{n}(\frac{1}{2}\nabla\phi \cdot \nabla\phi + (\vec{\xi} + \vec{\alpha} \times \vec{x}) \cdot \nabla\phi_t)ds \\
&+ \vec{\alpha} \times (-\rho \iint_{S_b} \vec{n}\phi_t ds) \\
&+ \vec{F}_S^{(2)}
\end{aligned} \tag{12.48}$$

where $\vec{F}_S^{(2)} = -\rho g A_{wp}[\alpha_1\alpha_3 x_f + \alpha_2\alpha_3 y_f + \frac{1}{2}(\alpha_1^2 + \alpha_2^2)Z_o]\vec{k}$

$$\begin{aligned}
\vec{M}^{(2)} &= \frac{1}{2}\rho g \int_{WL} (\vec{x} \times \vec{n}')[\zeta^2 - 2\zeta(\xi_3 + \alpha_1 y - \alpha_2 x)]dl \\
&+ \frac{1}{2}\rho g \int_{WL} \frac{n_z}{\sqrt{1-n_z^2}}(\vec{x} \times \vec{k})[\zeta - (\xi_3 + \alpha_1 y - \alpha_2 x)]^2 dl \\
&- \rho \iint_{S_b} (\vec{x} \times \vec{n})(\frac{1}{2}\nabla\phi \cdot \nabla\phi + (\vec{\xi} + \vec{\alpha} \times \vec{x}) \cdot \nabla\phi_t)ds \\
&- \rho\vec{\alpha} \times \iint_{S_b} (\vec{x} \times \vec{n})\phi_t ds \\
&+ \vec{\xi} \times (-\rho \iint_{S_b} \vec{n}\phi_t ds) \\
&+ \vec{M}_S^{(2)}
\end{aligned} \tag{12.49}$$

$$\begin{aligned}
\vec{M}_S^{(2)} = \rho g\{ \quad & [\quad (\xi_2\alpha_2 - \xi_3\alpha_3)A_{wp}x_f - \xi_2\alpha_1 A_{wp} y_f \\
& + \quad \alpha_1\alpha_2 A_{wp} x_f Z_o - (\frac{3}{2}\alpha_1^2 + \frac{1}{2}\alpha_2^2)A_{wp} y_f Z_o \\
& - \quad \xi_2\xi_3 A_{wp} - \xi_3\alpha_1 A_{wp} Z_o \\
& + \quad \alpha_2\alpha_3(L_{11} - L_{22}) - 2\alpha_1\alpha_3 L_{12} \\
& + \quad \alpha_1\alpha_2 \forall x_b - \frac{1}{2}(\alpha_1^2 + \alpha_3^2)\forall y_b]\vec{i} \\
& [\quad -\xi_1\alpha_2 A_{wp} x_f + (\xi_1\alpha_1 - \xi_3\alpha_3)A_{wp} y_f \\
& + \quad (\frac{1}{2}\alpha_1^2 + \frac{3}{2}\alpha_2^2)A_{wp} x_f Z_o - \alpha_1\alpha_2 A_{wp} y_f Z_o \\
& + \quad \xi_1\xi_3 A_{wp} - \xi_3\alpha_2 A_{wp} Z_o \\
& + \quad \alpha_1\alpha_3(L_{11} - L_{22}) + 2\alpha_2\alpha_3 L_{12}
\end{aligned}$$

$$+ \quad \frac{1}{2}(\alpha_2^2 + \alpha_3^2)\forall x_b]\vec{j}\} \tag{12.50}$$

ζ is the first order runup, A_{wp} is the waterplane area and $\forall$ is the volume of the body. In addition (x_f, y_f) are the coordinates of the center of flotation, (x_b, y_b, z_b) are the coordinates of the center of buoyancy, $(\vec{i}, \vec{j}, \vec{k})$ are positive unit vectors relative to the x, y, z coordinates, and $L_{ij} = \int_{wp} x_i x_j ds$ denotes the moments of the waterplane area.

12.8 REMOVAL OF IRREGULAR FREQUENCIES

The irregular frequency effects are removed from the velocity potential and the source strength using the extended boundary integral equations. The details of discussion on the method are provided in Reference [8] and [16].

The extended boundary integral equation for the potential formulation (12.11) takes a form

$$2\pi\varphi(\mathbf{x}) + \iint_{S_b} \varphi(\boldsymbol{\xi})\frac{\partial G(\mathbf{x};\boldsymbol{\xi})}{\partial n_\xi}d\boldsymbol{\xi} + \iint_{S_i} \varphi'(\boldsymbol{\xi})\frac{\partial G(\mathbf{x};\boldsymbol{\xi})}{\partial n_\xi}d\boldsymbol{\xi} = \iint_{S_b} \frac{\partial \varphi(\boldsymbol{\xi})}{\partial n_\xi}G(\mathbf{x};\boldsymbol{\xi})d\boldsymbol{\xi} \qquad \mathbf{x} \in S_b \tag{12.51}$$

$$-4\pi\varphi'(\mathbf{x}) + \iint_{S_b} \varphi(\boldsymbol{\xi})\frac{\partial G(\mathbf{x};\boldsymbol{\xi})}{\partial n_\xi}d\boldsymbol{\xi} + \iint_{S_i} \varphi'(\boldsymbol{\xi})\frac{\partial G(\mathbf{x};\boldsymbol{\xi})}{\partial n_\xi}d\boldsymbol{\xi} = \iint_{S_b} \frac{\partial \varphi(\boldsymbol{\xi})}{\partial n_\xi}G(\mathbf{x};\boldsymbol{\xi})d\boldsymbol{\xi} \qquad \mathbf{x} \in S_i \tag{12.52}$$

Here $\varphi(\mathbf{x})$ is the unknown velocity potential on the body surface S_b and $\varphi'(\mathbf{x})$ is the nonphysical velocity potential on the interior free surface S_i.

The corresponding equation for the source formulation (12.22) takes a form

$$2\pi\sigma(\mathbf{x}) + \iint_{S_b} \sigma(\boldsymbol{\xi})\frac{\partial G(\mathbf{x};\boldsymbol{\xi})}{\partial n_x}d\boldsymbol{\xi} + \iint_{S_i} \sigma(\boldsymbol{\xi})\frac{\partial G(\mathbf{x};\boldsymbol{\xi})}{\partial n_x}d\boldsymbol{\xi} = \frac{\partial \varphi(\boldsymbol{\xi})}{\partial n_x} \qquad \mathbf{x} \in S_b \tag{12.53}$$

$$-4\pi\sigma(\mathbf{x}) + \iint_{S_b} \sigma(\boldsymbol{\xi})\frac{\partial G(\mathbf{x};\boldsymbol{\xi})}{\partial n_x}d\boldsymbol{\xi} + \iint_{S_i} \sigma(\boldsymbol{\xi})\frac{\partial G(\mathbf{x};\boldsymbol{\xi})}{\partial n_x}d\boldsymbol{\xi} = \frac{\partial \varphi(\boldsymbol{\xi})}{\partial n_x} \qquad \mathbf{x} \in S_i \tag{12.54}$$

12.9 INTEGRAL EQUATION FOR BODIES WITH ZERO THICKNESS

Green's integral equations in Section 12.2 tend to be singular as the thickness of the body decreases. To avoid this singularity in the discretized problem, the panel size should be of the same order as the thickness, or smaller, in order to render the linear system well-conditioned. Thus the size of the linear system becomes large as the thickness decreases.

An alternative form, which is nonsingular, can be obtained from Green's integral equation for the limit when the thickness vanishes. In this modified form of the integral equation, the velocity potential is represented by a distribution of dipoles only, without sources. The dipole strength is equal to the difference of the velocity potential on two opposite sides of the zero-thickness surface, denoted by $\Delta\phi$ below.

When bodies consist partly of thin parts, a combination of Green's integral equation and the dipole distribution can be applied. It takes the following form:

$$2\pi\phi(\mathbf{x}) + \iint_{S_b} \phi G_{n_\xi} dS_\xi + \iint_{S_d} \Delta\phi G_{n_\xi} dS_\xi = \iint_{S_b} \phi_{n_\xi} G dS_\xi \tag{12.55}$$

when $\mathbf{x}$ is on the normal body surface S_b and

$$\iint_{S_b} \phi G_{n_\xi n_x} dS_\xi + \iint_{S_d} \Delta\phi G_{n_\xi n_x} dS_\xi = -4\pi\phi_{n_x} + \iint_{S_b} \phi_{n_\xi} G_{n_x} dS_\xi \tag{12.56}$$

when $\mathbf{x}$ is on the surface of zero thickness S_d.

In the low-order method this integral equation can be solved by WAMIT, to solve for the velocity potential on S_b and for $\Delta\phi$ on S_d. This extension is restricted to the low-order method (ILOWHI=0) and to the potential solution (ISOR=0). Instructions for using this option are in Section 5.4, and TEST09 is an example of its use (Section A.9).

12.10 MEAN DRIFT FORCE AND MOMENT USING CONTROL SURFACE

The mean drift force and moment are evaluated by one the following two alternatives depending on the value of ICTRSURF in the CFG file.

Alternative 1:

$$\begin{aligned}
\vec{F}^{(2)} = \; & - \frac{1}{2}\rho g \int_{CL} \vec{n}'\zeta^2 dl - \rho g \vec{k} \int_{WL} \zeta[(\vec{\xi} + \vec{\alpha} \times \vec{x}) \cdot \vec{n}']dl \\
& + \frac{1}{2}\rho g \vec{k} \int_{WL} \frac{n_z}{\sqrt{1-n_z^2}} [\zeta - (\xi_3 + \alpha_1 y - \alpha_2 x)]^2 dl \\
& - \rho \iint_{S_c} [\nabla\phi \frac{\partial\phi}{\partial n} - \frac{1}{2}\vec{n}(\nabla\phi \cdot \nabla\phi)]ds
\end{aligned}$$

$$+ \quad \rho\vec{k}\iint_{S_f}(\zeta\frac{\partial\phi_t}{\partial z}+\frac{1}{2}\nabla\phi\cdot\nabla\phi)ds+\vec{F}_S^{(2)} \tag{12.57}$$

$$\begin{aligned}\vec{M}^{(2)}= \quad & -\quad \frac{1}{2}\rho g\int_{CL}(\vec{x}\times\vec{n}')\zeta^2 dl-\rho g\int_{WL}\zeta[(\vec{\xi}+\vec{\alpha}\times\vec{x})\cdot\vec{n}'](\vec{x}\times\vec{k})dl\\ & +\quad \frac{1}{2}\rho g\int_{WL}\frac{n_z}{\sqrt{1-n_z^2}}(\vec{x}\times\vec{k})[\zeta-(\xi_3+\alpha_1 y-\alpha_2 x)]^2 dl\\ & -\quad \rho\iint_{S_c}[(\vec{x}\times\nabla\phi)\frac{\partial\phi}{\partial n}-\frac{1}{2}(\vec{x}\times\vec{n})(\nabla\phi\cdot\nabla\phi)]ds\\ & +\quad \rho\iint_{S_f}(\vec{x}\times\vec{k})(\zeta\frac{\partial\phi_t}{\partial z}+\frac{1}{2}\nabla\phi\cdot\nabla\phi)ds+\vec{M}_S^{(2)}\end{aligned} \tag{12.58}$$

Alternative 2:

$$\begin{aligned}\vec{F}^{(2)} \quad = \quad & \frac{1}{2}\rho g\int_{WL}\vec{n}'\zeta^2 dl-\rho g\iint_{S_f}\zeta\nabla'\zeta ds\\ & -\quad \rho g\vec{k}\int_{WL}\zeta[(\vec{\xi}+\vec{\alpha}\times\vec{x})\cdot\vec{n}']dl\\ & +\quad \frac{1}{2}\rho g\vec{k}\int_{WL}\frac{n_z}{\sqrt{1-n_z^2}}[\zeta-(\xi_3+\alpha_1 y-\alpha_2 x)]^2 dl\\ & -\quad \rho\iint_{S_c}[\nabla\phi\frac{\partial\phi}{\partial n}-\frac{1}{2}\vec{n}(\nabla\phi\cdot\nabla\phi)]ds\\ & +\quad \rho\vec{k}\iint_{S_f}(\zeta\frac{\partial\phi_t}{\partial z}+\frac{1}{2}\nabla\phi\cdot\nabla\phi)ds+\vec{F}_S^{(2)}\end{aligned} \tag{12.59}$$

$$\begin{aligned}\vec{M}^{(2)} \quad = \quad & \frac{1}{2}\rho g\int_{WL}(\vec{x}\times\vec{n}')\zeta^2 dl-\rho g\iint_{S_f}\zeta(\vec{x}\times\nabla'\zeta)ds\\ & -\quad \rho g\int_{WL}\zeta[(\vec{\xi}+\vec{\alpha}\times\vec{x})\cdot\vec{n}'](\vec{x}\times\vec{k})dl\\ & +\quad \frac{1}{2}\rho g\int_{WL}\frac{n_z}{\sqrt{1-n_z^2}}(\vec{x}\times\vec{k})[\zeta-(\xi_3+\alpha_1 y-\alpha_2 x)]^2 dl\\ & -\quad \rho\iint_{S_c}[(\vec{x}\times\nabla\phi)\frac{\partial\phi}{\partial n}-\frac{1}{2}(\vec{x}\times\vec{n})(\nabla\phi\cdot\nabla\phi)]ds\\ & +\quad \rho\iint_{S_f}(\vec{x}\times\vec{k})(\zeta\frac{\partial\phi_t}{\partial z}+\frac{1}{2}\nabla\phi\cdot\nabla\phi)ds+\vec{M}_S^{(2)}\end{aligned} \tag{12.60}$$

In these equations S_c is the submerged part of the control surface, S_f is the part of the control surface on the free surface, WL is the body waterline and CL is the common boundary of S_c and S_f. $\vec{n}'$ denotes the two-dimensional normal vector in the horizontal plane on the body waterline, pointing into the body, ∇' is the gradient in the horizontal

plane, and $\vec{k}$ is the unit vector pointing vertically upward. $\vec{F}_S^{(2)}$ and $\vec{M}_S^{(2)}$ are the same as those in the equation (12.48) and (12.49). The derivations of the equations (12.57-12.60) from (12.48-12.49) are given in Reference [28].

12.11 INTERNAL TANK EFFECTS

The solution for the velocity potential in each tank, and the resulting forces and moments, are computed in a similar manner as for the exterior domain outside the body or bodies. Since there is no diffraction potential to consider, the velocity potential ϕ in each tank is

$$\phi = i\omega \sum \xi_j \phi_j \tag{12.61}$$

and the first-order pressure at a fixed point on the tank surface is given by

$$P = -\rho g(z - Z_T + \xi_3 + \alpha_1 y - \alpha_2 x) - \rho \phi_t \tag{12.62}$$

Here Z_T is the vertical coordinate of the tank free surface above the origin.

The solution for the velocity potential ϕ in each tank is computed simultaneously with the potential in the exterior fluid domain outside the hull, using one extended linear system which includes all of the fluid domains (exterior plus interior of all tanks). The principal modification is to impose the condition that there is no influence between the separate fluid domains. Thus the elements in the extended influence coefficient matrix are set equal to zero if the row and column correspond to different domains. Further details are given in Reference [27].

The force and moment exerted by the tank fluid on the vessel are given by

$$\mathbf{F} = \int\int_{S_T} P\mathbf{N} dS = \int\int_{S_T} P\left(\mathbf{n} + \alpha \times \mathbf{n}\right) dS \tag{12.63}$$

$$\mathbf{M} = \int\int_{S_T} P\left(\mathbf{X} \times \mathbf{N}\right) dS = \int\int_{S_T} P\left(\mathbf{x} + \xi + \alpha \times \mathbf{x}\right) \times \left(\mathbf{n} + \alpha \times \mathbf{n}\right) dS \tag{12.64}$$

where $\mathbf{n}$ is the normal pointing out of the tank (away from the fluid domain of the tank) and double integrals are over the submerged surface of the tank.

After some vector analysis these equations give relations similar to (12.46-12.47) for the contributions from the hydrostatic pressure:

$$C_T(3,3) = \rho g \int\int_{S_T} n_3 dS$$

$$C_T(3,4) = \rho g \int\int_{S_T} y n_3 dS$$

$$C_T(3,5) = -\rho g \int\int_{S_T} x n_3 dS$$

$$C_T(4,4) = \rho g \int\int_{S_T} y^2 n_3 dS - \rho g \forall_T z_c$$

$$C_T(4,5) = -\rho g \int\int_{S_T} xyn_3 dS$$

$$C_T(4,6) = \rho g \forall_T x_c$$

$$C_T(5,5) = \rho g \int\int_{S_T} x^2 n_3 dS - \rho g \forall_T z_c$$

$$C_T(5,6) = \rho g \forall_T y_c$$

All other elements of the matrix C_T are equal to zero. Here

$$\forall_T = \text{VOLTANK}(1) = \int\int_{S_T} xn_1 dS$$

$$\forall_T = \text{VOLTANK}(2) = \int\int_{S_T} yn_2 dS$$

$$\forall_T = \text{VOLTANK}(3) = \int\int_{S_T} (z - Z_T) n_3 dS$$

$$x_c = \frac{1}{2\forall_T} \int\int_{S_T} x^2 n_1 dS$$

$$y_c = \frac{1}{2\forall_T} \int\int_{S_T} y^2 n_2 dS$$

$$z_c = \frac{1}{2\forall_T} \int\int_{S_T} (z^2 - Z_T^2) n_3 dS$$

Subroutine GEOMSTAT evaluates the hydrostatic parameters of the hull and tanks separately. Thus VOL, C(i,j) are evaluated for the hull ignoring tank patches/panels and their values are the same with or without tanks, as defined in Section 4.1. The corresponding tank parameters VOLTNK(1:3,1:NTANK) and CTANK(1:9,1:NTANK) are evaluated separately for each tank (the second index is omitted for simplicity of notation):

$$\text{CTANK}(1) = \int\int_{S_T} n_3 dS$$

$$\text{CTANK}(2) = -\int\int_{S_T} xn_3 dS$$

$$\text{CTANK}(3) = \int\int_{S_T} yn_3 dS$$

$$\text{CTANK}(4) = \int\int_{S_T} x^2 n_1 dS$$

$$\text{CTANK}(5) = \int\int_{S_T} y^2 n_2 dS$$

$$\text{CTANK}(6) = \int\int_{S_T} (z^2 - Z_T^2) n_3 dS$$

$$\text{CTANK}(7) = \int\int_{S_T} x^2 n_3 dS$$

$$\text{CTANK}(8) = -\int\int_{S_T} xyn_3 dS$$

$$\text{CTANK}(9) = \int\int_{S_T} y^2 n_3 dS$$

It is necessary to consider the implications of planes of symmetry with respect to the tanks. If all of the tanks are symmetric about a plane of symmetry, then it is appropriate to use that option (assuming the hull is also symmetric about the same plane). Thus, for example, for a hull with symmetry about y=0, and with all tanks symmetric about the same plane, it is appropriate to set IS(2)=1. But if there are two or more tanks along the x-axis, IS(1)=0 is required. When planes of symmetry are appropriate, the following integrals (which are originally evaluated over one half or one quarter of the tank, with nonzero values), should be set equal to zero:

If IS(1)=1, CTANK(2) = 0, CTANK(4) = 0, CTANK(8) = 0

If IS(2)=1, CTANK(3) = 0, CTANK(5) = 0, CTANK(8) = 0

All other elements are multiplied by IMUL=2 or 4 to account for the planes of symmetry, in the same manner as C(I,J).

In GETIF, after CALL OPHEAD to output the hydrostatic matrix for the hull, the subroutine TANKFS is called when tanks are present. In TANKFS the following assignments are made for the hydrostatic restoring coefficients, where the extra terms added for the tanks are summed over all tanks associated with each body:

$$C(3,3) = C(3,3) + (\rho_T/\rho)\text{CTANK}(1)$$

$$C(3,4) = C(3,4) + (\rho_T/\rho)\text{CTANK}(3)$$

$$C(3,5) = C(3,5) + (\rho_T/\rho)\text{CTANK}(2)$$

$$C(4,4) = C(4,4) + (\rho_T/\rho)\left[\text{CTANK}(9) - \tfrac{1}{2}\text{CTANK}(6)\right]$$

$$C(4,5) = C(4,5) + (\rho_T/\rho)\text{CTANK}(8)$$

$$C(5,5) = C(5,5) + (\rho_T/\rho)\left[\text{CTANK}(7) - \tfrac{1}{2}\text{CTANK}(6)\right]$$

Also, if IALTFRC=2,

$$C(4,6) = C(4,6) + \tfrac{1}{2}(\rho_T/\rho)\text{CTANK}(4)$$

$$C(5,6) = C(5,6) + \tfrac{1}{2}(\rho_T/\rho)\text{CTANK}(5)$$

The extra terms in $C(4,6)$ and $C(5,6)$ are omitted for a freely-floating body since these are balanced by the vessel's corresponding buoyancy moments.

Next consider the inertia forces and moments due to the body mass. If IALTFRC=1 the body mass m is calculated from VOL and all of the inertia coefficients are propotional to m. Since VOL is the total displaced volume of the hull, the static mass of the fluid in the tanks is included in m. However the same inertia effects are represented (more

correctly for dynamic conditions) by the added mass of the tank, to be discussed below. For this reason, if IALTFRC=1, the mass matrix BFRCND is reduced by the sum $\sum(\rho_T/\rho)\times$VOLTANK/VOLM, where the sum is over all tanks. Likewise, the terms $-mgz_g$ in the restoring coefficients $C(4,4)$ and $C(5,5)$ are reduced. If IALTFRC=2 these corrections are not made, and the external mass matrix should exclude the fluid in the tanks. If IALTFRC=1 the radii of gyration should be estimated ignoring the fluid in the tanks.

The added mass and damping coefficients are evaluated globally, by integrating the corresponding components of the pressure over both the external hull surface and the internal tank surfaces. The only modification for tank panels/patches is to multiply their contributions by RHOTANK, the relative density of the tank fluid compared to the external fluid. Since there is no radiation from the tanks, the damping coefficients should be zero. In test calculations they are generally very small, except near tank resonances. (A useful check is to verify that the damping coefficients of the hull with tanks are equal to the damping coefficients without tanks.)

Special attention is required for the vertical modes (heave, roll, pitch), where there is a fictitious hydrostatic contribution to the added mass. First consider heave, where the relevant boundary conditions are $\phi_n = n_3$ on S_b and $K\phi - \phi_z = 0$ on the free surface. Here $K = \omega^2/g$. Thus the heave potential is given by $\phi_3 = z + 1/K$, where z is a local vertical coordinate with its origin in the tank free surface. The heave added-mass coefficient is

$$A_{33} = \rho \int\int_{S_T} n_3\phi_3 dS = \rho\forall_T + C_T(3,3)/(gK) \tag{12.65}$$

Also derivable from the same potential are

$$A_{34} = \rho \int\int_{S_T} (yn_3 - zn_2)\phi_3 dS = \rho\forall_T y_c + C_T(3,4)/(gK) \tag{12.66}$$

$$A_{35} = \rho \int\int_{S_T} (zn_1 - xn_3)\phi_3 dS = -\rho\forall_T x_c + C_T(3,5)/(gK) \tag{12.67}$$

Since $\omega^2 = gK$, the last terms are cancelled by the hydrostatic restoring coefficients. Thus, in the limit $\omega \to 0$, there are no contributions to the equations of motion for the LHS elements associated with the vertical force or vertical translation, as expected on physical grounds.

In the classical hydrostatic analysis of ships, the tank free-surface effect is evaluated by considering the second moments of the tank free surface about a local origin at the centroid of the free surface, whereas in the expressions for C_T these moments are about the global origin. This difference can also be explained in terms of the corresponding added mass coefficients, in an analogous manner to the analysis above.

Chapter 13

THE F2T UTILITY

The Fortran utility F2T (Frequency-to-Time-domain) is a post-processor to transform frequency-domain WAMIT output to time-domain impulse-response functions (IRFs). This program is intended to provide a utility which can be used for general purposes, based on standard WAMIT outputs. This program accepts as input all of the first-order (linear) outputs from WAMIT, including any combinations of Options 1-7 (added mass/damping, Haskind exciting forces, Diffraction exciting forces, RAO's, body pressures/velocities, field-point pressures/velocities). In principle there are no restrictions regarding the numbers of rigid-body modes, generalized modes, or bodies. The computed IRFs are saved in output files which are analogous to the input files for each option and use the same filename extensions.

The Fourier transforms from the frequency-domain to the time-domain are evaluated in F2T by Filon numerical integration. This method provides relatively accurate results for large values of the time variable t. A fundamental requirement is that the frequency-domain data must be evaluated by WAMIT for a large number N of uniformly-spaced frequencies ω_n where n=1,2,3,...,N. Special attention is necessary to ensure that the increment $\Delta\omega = \omega_{n+1} - \omega_n$ is sufficiently small (to preserve the accuracy of the numerical integration) and that the highest finite frequency ω_N is sufficiently large to span the physically-significant range of frequencies for the application (or from the mathematical standpoint, to ensure that the truncated Fourier integrals are reasonable approximations of the infinite integrals). In view of the need to include high frequencies in the WAMIT analysis it is usually advisable to use the irregular-frequency option (IRR=1), unless the body is submerged or its waterplane area is very small. The requirement of accuracy over a broad range of frequencies means that either a large number of low-order panels should be used, or alternatively that the higher-order method is used with appropriate control of the panel subdivision indices NU,NV or the global parameter PANEL_SIZE.

Reference [26] contains illustrative results from the use of F2T, including comparisons with the results from the time-domain panel program TiMIT.

13.1 DEFINITIONS OF RADIATION AND DIFFRACTION OUTPUTS

The outputs from WAMIT and F2T are considered to be of either the radiation or diffraction type, depending on whether they are caused by forced motions in calm water or by incident waves, respectively. The simplest physical distinction between these two types is in terms of the incident wave amplitude: if the response is proportional to the wave amplitude it is of the diffraction type, and vice versa.

The added-mass and damping coefficients (Option 1) are of the radiation type, whereas the exciting forces and RAO's (Options 2,3,4) are of the diffraction type. Except as noted in Section 13.5, the pressures and fluid velocities on the body (Option 5) and in the fluid (Options 6,7) are of the diffraction type, since these are defined in the WAMIT convention as total responses with the body free to respond (or fixed) in incident waves.

A fundamental distinction between the two types of outputs is in terms of their limits at infinite frequency or zero period. In this limit the radiation outputs are generally real and nonzero, corresponding to the added mass, pressure, and fluid velocity induced by forced motions of the body without wave effects on the free surface. Conversely, in the same limit there are no diffraction effects since the 'incident waves' have vanishingly small wavelength and cause no disturbance of either the body or the fluid.

When radiation IRFs are evaluated it is necessary to evaluate the corresponding frequency-domain coefficients for $\omega_0 = 0$ and $\omega_{N+1} = \infty$, using the special instructions in the WAMIT User Manual (Section 3.1, page 3-9 of the V6.1 User Manual). Thus, in V6.1, two WAMIT runs are required. V6.1 also restricts the $\omega_0 = 0$ and $\omega_{N+1} = \infty$ evaluations to Option 1, so that V6.1 cannot be used with F2T to analyse radiation type outputs in Options 5,6,7. WAMIT V6.2 removes this restriction, and will also permit the analysis of all frequency-domain outputs to be made in a single run. To simplify the instructions below, Version 6.2 or higher is assumed, and instructions are given for making only one WAMIT run. (Users of V6.1 should make two separate runs, one for finite wave periods and one for zero and infinity, and for the latter only Option 1 should be specified.)

13.2 ACQUIRING INPUT DATA FOR F2T WITH WAMIT

The frequency-domain input data for F2T is evaluated by WAMIT. The algorithms used to evaluate the Fourier transforms in F2T require that the input data is restricted to a uniformly-spaced set of frequencies $\omega_n = n\Delta\omega$, where (n=1,2,3,...,NPER), augmented by the limiting frequencies 0 and ∞. In the WAMIT run this is done most easily by setting the parameter IPERIO=2 in the configuration file (inputs are radian frequencies), and by using the option to write the data -NPER and ω_1, $\Delta\omega$ on the lines normally used to specify NPER and the array PER. The simplest procedure is to make one run, with $\omega_1 = -\Delta\omega$, so that both infinite and zero wave periods are included in the run. (Note in this case that NPER must include the total number of wave periods including zero and infinity.)

The input files used for the tests of the ISSC TLP can be regarded as an example. These

files are listed in Appendix A14a. They are essentially the same as for TEST14, with the exception of IPERIO=2 in TEST14.CFG, and the specification of the input frequencies.

When the input files to F2T are read, the data is sorted so that the frequencies are listed in ascending order, regardless of their order in the WAMIT output files. Thus the order of the periods (-1.0, 0.0) is irrelevant, and it is possible to patch together two or more separate sets of output files from WAMIT, e.g. one with a coarse set of wave frequencies and the other with intermediate frequencies, to provide a finer set, without concern regarding their order.

13.3 HOW TO USE F2T

The program F2T can be executed after the appropriate WAMIT output files are available. The user must specify the filenames of these files and a small number of input parameters, either interactively in response to appropriate runtime prompts or by preparing the special input file `inputs.f2t`. The following example of this special input file corresponds to the TEST14a test run described above:

```
header line for inputs.f2t control file, TLP example
TEST14a
0 0 1 (IRAD IDIFF NUMHDR)
0 0 0 (INUMOPT5 INUMOPT6 INUMOPT7)
0.2 100 (DT NT time step and number of time steps)
0 (IOUTFCFS, output both cosine and sine transforms)
43.125 9.80665 ULEN GRAV
```

These inputs are described for each line as follows:

Line 1 is an ASCII header dimensioned CHARACTER*72 as in most WAMIT input files. This line should be used to insert a brief description of the file.

Line 2 is a list of the filenames (not the extensions) of the primary and secondary WAMIT output files. F2T attempts to open all numeric output files with the same filenames, and includes all of these files in the analysis. Thus the determination of which options to be included depends on the available WAMIT output files. In this example where the TEST14.FRC control file was used as in the standard WAMIT test runs, Options 1,2,3,4 will be included in the F2T analysis. If all of the input data is included in the primary file it is not necessary to list other filenames. Additional secondary files can also be included, up to a maximum limit of 256 ASCII characters for the complete line. At least one blank space must be used to separate each filename.

Lines 3 and 4 contain the six WAMIT control parameters identified by the comments in parenthesis. These parameters must have the same values as in the WAMIT runs. (No distinction is made between IRAD,IDIFF=0 or 1, and the only important value to specify correctly is -1. For any input values of IRAD,IDIFF other than -1 the results are the same as for 0 or 1.) NUMHDR, which is optional in WAMIT with the default value 0, must be specified heree with the value 0 (no headers) or 1 (one line of headers) to indicate the

presence or absence of a header line in the WAMIT numeric output files. INUMOPT5, the optional configuration parameter introduced in WAMIT Version 6.1 to permit outputting separate components of the body pressure and velocity, and also INUMOPT6, INUMOPT7 introduced in Version 6.2 must be specified here with the value 0 (default in WAMIT) or the separate-components values 1.

Line 5 contains the time step and number of time steps for the computation and tabulation of the time-domain response functions. The radiation IRF'S are computed and tabulated for t=0 and for NT positive times DT, 2DT, 3DT, ..., NT∗DT. The diffraction IRFs are evaluated for both positive and negative times, starting with −NT∗DT and ending with +NT∗DT.

Line 6 contains the optional parameter IOUTFCFS, with the following options for its value:

IOUTFCFS=1: output only the cosine transform of radiation irf's

IOUTFCFS=2: output only the sine transform of radiation irf's

IOUTFCFS=0 (or any other integer except 1 or 2): output both cosine and sine transforms

(These transforms are redundant, as explained below.) The default value IOUTFCS=0 is used if line 6 is missing from the file. Thus the use of this parameter is optional.

Line 7 contains the optional parameters ULEN and GRAV, which are the same characteristic length scale and gravitational acceleration parameters as input in the GDF file. These parameters are only required when Options 5, 6, or 7 are included, and when the radiation outputs are specified, as explained in Section 13.5 below. In all other circumstances the parameters ULEN and GRAV can be omitted from `inputs.f2t`. **When ULEN and GRAV are included in `inputs.f2t` it is essential to also include IOUTFCS on line 6.**

The use of the special file `inputs.f2t` is optional. If this file does not exist, or if the first five lines cannot be read with the appropriate data, the user is prompted to specify all of the above inputs interactively. The special file can also be used in a partial form with some but not all of the above lines, but the lines included must be in the same order as above. This permits the user to interactively input different values of the time step and number, simply by omitting Line 5 from the special file.

The numeric data in the special file is read with free format READ statements, separately for each line. Any additional text on the same lines is ignored, so that comments may be inserted as in the example above. The filenames on Line 2 are read as ASCII text of unknown length (maximum of 256 characters, all on one line) and no additional comments may be included on this line.

13.4 OUTPUT FILES

The output files from F2T are in two complementary formats with duplication of the output data in the two formats. The filename assigned to all of the output files is *primary*, with

different extensions. The first set of output files have appended filenames including _IR followed by the same extensions as the WAMIT output files. The second set have the appended filenames including _JR. The first set follow the same format as the WAMIT numeric output files of the same number, except that the period is replaced by the time step and the WAMIT force coefficients are replaced by their Fourier cosine and sine transforms. Different modes and mode combinations are listed on separate lines with the identifying mode indices, just as in the numeric output files of WAMIT.

To facilitate plotting and separation of the different modes and wave angles (BETA), all of the Fourier cosine/sine transforms are listed on one line in the output files denoted by _JR, in the same order of mode combinations but without explicit mode indices. The cosine/sine transforms are listed as pairs, unless one or the other is ommitted by setting IOUTFCFS equal to 1 or 2 as explained in the following paragraph. Column one of the _JR file contains the value of time t.

Either the cosine transforms of the added mass or the sine transforms of the damping can be used to evaluate the radiation IRFs (cf. equations 3 and 4 below). These two sets of data can be checked to verify their accuracy and consistency, in much the same way that the Haskind and Diffraction exciting forces or cross-coupling coefficients are compared. Alternatively, to achieve more compact output files, one of these transforms can be omitted using the parameter IOUTFCFS.

One more output file is produced with the extension .KR1, containing the impulse-response functions K_{ij} which are evaluated in TiMIT. These alternative IRF's are evaluated in F2T by numerical differentiation of the IRF's L_{ij} which are defined below.

The diffraction files _JR are different from the radiation files in two respects, to facilitate their use. First, the time steps begin with $-$NT$*$DT, and end with $+$NT$*$DT. Secondly, the cosine and sine transforms are combined (adding for $t < 0$ and subtracting for $t > 0$) to give the actual IRFs for the corresponding exciting forces and RAOs (cf. equation 8 below).

For practical purposes the .JRn files will be most useful, and the .IRn files may be useful only to clarify the identity of the different columns in the .JRn files.

Some experience and/or trial computations will be needed to determine appropriate values of the input frequencies and time steps. The dimensions of these parameters correspond to GRAV in the WAMIT run.

13.5 OPTIONS 5,6,7

The F2T utility has been developed primarily for use with Options 1 to 4 (global forces and RAO's). Local pressures, velocities, and wave elevations have not been tested, and these may be difficult to transform accurately, due to limited or non-convergence of the Fourier transforms at high frequencies.

If IDIFF$=-1$ is specified in the WAMIT run, signifying that there are no incident waves, the outputs from Options 5,6,7 are the total responses from superposition of all specified

radiation modes; in this case all of the available outputs are of the radiation type. In this case, if more than one mode is considered, the output is for nonzero finite frequencies only and is not suitable for transform to the time domain. If the configuration parameters INUMOPT5, INUMOPT6, INUMOPT7 = 1 in the WAMIT run, the corresponding Option 5,6,7 outputs are separated into radiation components for each mode of forced motion plus the diffraction component, as explained in Sections 4.9 and 4.12; in this case F2T analyses the radiation components and diffraction components separately, according to their types.

The procedure using INUMOPT5, INUMOPT6, INUMOPT7 = 1, as described in Section 4.12, should be followed if F2T is used to transform the radiation components of the pressures, velocities, and wave elevations.

13.6 THEORY

The fundamental relations between the time- and frequency-domain express the added-mass coefficient A_{ij} and damping coefficient B_{ij} in terms of Fourier transforms of the impulse-response function $L_{ij}(t)$:

$$A_{ij}(\omega) - A_{ij}(\infty) = \int_0^\infty L_{ij}(t) \cos \omega t \, dt \tag{13.1}$$

$$B_{ij}(\omega) = \omega \int_0^\infty L_{ij}(t) \sin \omega t \, dt \tag{13.2}$$

The inverse-transforms of (13.1-2) give complementary relations for the impulse-response function:

$$L_{ij}(t) = \frac{2}{\pi} \int_0^\infty [A_{ij}(\omega) - A_{ij}(\infty)] \cos \omega t \, d\omega \tag{13.3}$$

$$L_{ij}(t) = \frac{2}{\pi} \int_0^\infty \frac{B_{ij}(\omega)}{\omega} \sin \omega t \, d\omega \tag{13.4}$$

Similar relations exist for the exciting forces and RAOs. Define one of these quantities by the complex function $X_i(\omega)$ The corresponding impulse-response function is real, denoted by $K_i(t)$. The appropriate physical ranges are $(0 \le \omega < \infty)$ and $(-\infty < t < \infty)$. Then the complex Fourier transform pairs are as follows:

$$X_i(\omega) = \int_{-\infty}^\infty K_i(t) \mathrm{e}^{-i\omega t} \, dt \tag{13.5}$$

and

$$2\pi K_i(t) = \int_{-\infty}^\infty X_i(\omega) \mathrm{e}^{i\omega t} \, d\omega \tag{13.6}$$

Formally, since K_i is real, $X_i(-\omega) = X_i^*(\omega)$, and thus

$$2\pi K_i(t) = \int_0^\infty \left[X_i(\omega)\mathrm{e}^{i\omega t} + X_i^*(\omega)\mathrm{e}^{-i\omega t}\right]\, d\omega \tag{13.7}$$

or

$$K_i(t) = \frac{1}{\pi}\int_0^\infty \left[\mathrm{Re}(X_i)\cos\omega t - \mathrm{Im}(X_i)\sin\omega t\right]\, d\omega \tag{13.8}$$

The principal task is to evaluate (13.3), (13.4), and (13.8). This is done by truncating the infinite integrations at the largest value of the evaluated frequency, and using Filon quadratures to evaluate the resulting finite integrals. A truncation correction is derived below, and applied to (13.3).

Usually the most significant truncation error is associated with the transform of the added-mass (13.3). From partial integration of (13.1) it follows that

$$A_{ij}(\omega) - A_{ij}(\infty) = -\frac{1}{\omega}\int_0^\infty L'_{ij}(t)\sin\omega t\, dt \simeq -L'(0)\omega^{-2} \tag{13.9}$$

where the neglected integral is of order ω^{-3}. If (13.3) is truncated at a finite frequency $\omega_N = \Omega$, the truncation correction is

$$\Lambda_{ij}(t) = \frac{2}{\pi}\int_\Omega^\infty \left[A_{ij}(\omega) - A_{ij}(\infty)\right]\cos\omega t\, d\omega \tag{13.10}$$

This can be approximated, using (13.9), if Ω is sufficiently large:

$$\Lambda_{ij}(t) \simeq -\frac{2}{\pi}L'_{ij}(0)\int_\Omega^\infty \omega^{-2}\cos\omega t\, d\omega = -\frac{2}{\pi\Omega}L'_{ij}(0)\left[\cos\Omega t + \Omega t\,\mathrm{si}(\Omega t)\right] \tag{13.11}$$

Here we follow the notation of Abramowitz & Stegun (equation 5.2.26) for the sine integral

$$\int_z^\infty \frac{\sin t}{t}dt = -\mathrm{si}(z)$$

The constant $L'_{ij}(0)$ can be evaluated from the fact that $L_{ij}(0) = 0$, and thus

$$\Lambda_{ij}(0) = -\frac{2}{\pi}\int_0^\Omega \left[A_{ij}(\omega) - A_{ij}(\infty)\right]\, d\omega \simeq -\frac{2}{\pi\Omega}L'_{ij}(0) \tag{13.12}$$

Combining (13.11) and (13.12) gives the truncation correction in (13.10) in the form

$$\Lambda_{ij}(t) \simeq -\frac{2}{\pi}\left[\cos\Omega t + \Omega t\,\mathrm{si}(\Omega t)\right]\int_0^\Omega \left[A_{ij}(\omega) - A_{ij}(\infty)\right]\, d\omega \tag{13.13}$$

However this procedure suffers from the slow algebraic convergence of the last integral. An alternative procedure, which is adopted in F2T, is based instead on diffentiating (13.4) to give the relations

$$L'_{ij}(0) = \frac{2}{\pi}\int_0^\infty B_{ij}(\omega)\, d\omega \simeq \frac{2}{\pi}\int_0^\Omega B_{ij}(\omega)\, d\omega \tag{13.14}$$

$$\Lambda_{ij}(t) \simeq -\frac{4}{\pi^2\Omega}\left[\cos\Omega t + \Omega t\,\mathrm{si}(\Omega t)\right]\int_0^\Omega B_{ij}(\omega)\, d\omega \tag{13.15}$$

Equations (13.12) and (13.14) are complementary, but (13.14) is more robust since the integrand of (13.14) is positive-definite and converges to zero more rapidly than the integrand of (13.12).

Chapter 14

MEAN DRIFT FORCES USING CONTROL SURFACE

Starting with V6.3, option .9c is included in WAMIT to evaluate the mean drift forces (and moments). In this option, the mean drift forces are evaluated from the momentum flux through a control surface surrounding the body. If IOPTN(9)>0 in the Force Control File, and ICTRSURF>0 in the Configuration File, the drift forces are evaluated both from the pressure-integration method (with output in the .9 numeric output file) and from the control-surface momentum-flux method (with output in the .9c file). When NBODY> 1, a control surface surrounding each body is required, and the drift force acting on each body is evaluated separately as in Option 9.

The advantages of using the control surface are i) all six components of the mean drift forces and moments, on a single body or on each body in the multiple body interaction, are evaluated as in the pressure integration, and ii) the computational results are more accurate than the pressure integration method when the body surface is not smooth, especially for bodies with sharp corners. The disadvantages are i) the user must specify the control surface as an additional input, and ii) the evaluation of the momentum flux at a sufficiently large number of field points on the control surfaces increases the run time of the FORCE module. This option is recommended when the accuracy of the mean forces and moments evaluated by pressure integration is uncertain, due to slow or lack of convergence with respect to the discretization of the body.

The drift forces and moments using a control surface are evaluated by one of two alternatives, depending on the integer parameter ICTRSURF in the configuration file. When ICTRSURF=1, Alternative 1 is used, based on equations (12.57) and (12.58) When ICTRSURF=2, Alternative 2 is used, based on equations (12.59) and (12.60). These two alternatives are analytically equivalent. Alternative 1 is derived by transfer of the integral on the waterline (WL) in Alternative 2 to the line integral along the intersection of the free surface and the control surface (CL). Alternative 1 is generally more accurate for the horizontal forces and yaw moment, because it does not include a waterline integral. However, Alternative 2 must be used when CL is very close to the body. This is because the evaluation of the momemtum flux (pressure and/or velocity) very close to the body is

either inaccurate or not possible. An example for which the Alternative 2 may be required is when the gap between two adjacent bodies is very small.

To evaluate the mean drift forces and moments using this option, i) the parameter **ICTRSURF** must be specified in the CFG file, as explained in Section 3.7, and ii) the **CSF**(Control Surface File) defining the geometry of the control surface must be prepared. The CSF file must have the same filename as the corresponding geometric data file for the body, with the extension .csf, *i.e. gdf*.csf.

The control surface must be a closed surface surrounding the body in the fluid. In general, for a floating body which intersects the free surface, the control surface must start from the body's waterline, either extending outward on the free surface or downward away from the waterline into the fluid. Simple examples include a hemisphere or circular cylinder with sufficiently large dimensions so that the body is entirely within the interior of this surface, together with the intermediate portion of the free surface between the outer control surface and the body waterline. For multiple bodies, the control surface for each body should not include or intersect with other bodies, but it can intersect with other control surfaces.

In principle, the position and shape of the control surface are arbitrary. From a practical standpoint, the control surface should be sufficiently far from the body to ensure robust evaluation of the field velocity and pressure, but not so far as to require a very large number of field point evaluations. Simple geometrical description of the control surface is usually desirable.

If the Alternative 1 method is used, there is no contribution to the horizontal drift force and vertical drift moment from any part of the control surface that is in the plane of the free surface ($z = 0$). Thus, if these are the only required components of the drift forces, the control surface can be completely separated from the body surface without the need to include the intermediate portion of the free surface. This simplifies the definition of the control surface, especially for bodies with complicated geometry of the waterline. In addition, the numerical errors are generally smaller for field points that are not too close to the body surface.

14.1 CONTROL SURFACE FILE (CSF)

The geometry of the control surface can be described in the same manner as the body geometry. Similar options exist to define the control surface, and different options can be used for the control surface and for the body. In the low-order method, specified by inputting the parameter ILOWHICSF=0 on line 2 of the CSF file, the control surface is described with quadrilateral panels in the same manner as is described for the body in Chapter 5. In the higher-order method, the parameter ILOWHICSF=1 on line 2 of the CSF file and the control surface is described in the same manner as is described for the body in Chapter 6, using any of the options available for higher-order representation of the body surface, including flat panels, B-splines, MS2 files and analytically using subroutines in GEOMXACT. The format of the CSF file is almost identical with the GDF file. The

principal difference is on line 2, where the parameters ULEN and GRAV in the GDF file are replaced by ILOWHICSF. Also, when ILOWHICSF=1, the parameter PSZCSF is specified in the CSF file to control the accuracy of the numerical integration.

The control surface is defined in terms of the body coordinates. The normal vector is defined to point into the interior of the control surface (toward the fluid in the domain between the control surface and the body). If part of the control surface coincides with the plane of the free surface, the normal on this surface is positive downwards.

The accuracy of the numerical integration of the momentum flux depends not only on the accuracy of the field quantities on the control surface, but also on the discretization of the control surface. If a low-order control surface is used, the integral of the momentum flux is calculated as the sum of the product of the flux at the centroid of each panel and the area of the panel. If a higher-order control surface is used, the control surface is subdivided into 'higher-order' panels. On each panel, the momentum flux is calculated based on Gauss quadrature assuming quadratic variation of the flux.

The input files for the test runs 5, 13, and 22, shown in Appendix A, illustrate the different methods for defining the control surfaces, as noted below. As noted above, it is possible to use different values of ILOWHI and ILOWHICSF. In TEST05, which uses the low-order analysis for two bodies, the control surface around the cylinder is defined by low-order panels while the control surface around the spheroid is defined by the higher-order analytical method using a special subroutine in the GEOMXACT DLL file.

Since the description of the control surface in FORCE is completely separate from the solution in POTEN, arbitrary combinations of ILOWHI in POTEN and ILOWHICSF in FORCE can be used together. One or two planes of symmetry can be used to simplify the definition of the control surface, even when the body is not symmetric with respect to the same plane(s). However the opposite is not true: if the body is defined with one or two planes of symmetry, the control surface must use the same plane(s) of symmetry. (The case NBODY>1 is an exception, since symmetric bodies are reflected and analyzed without any planes of symmetry; thus it is possible in this case to use a non-symmetric control surface with a symmetric body, but this is not likely to be useful.)

14.2 LOW-ORDER CONTROL SURFACE FILE

In the low-order method the control surface is represented by an ensemble of panels. The CSF file contains a description of this discretized surface in the same format as in the GDF file except that ULEN and GRAV are not specified in CSF.

The data in the CSF file can be input in the following form:

```
header
0 (ILOWHICSF)
ISXCSF ISYCSF
NPANCSF
X1(1) Y1(1) Z1(1) X2(1) Y2(1) Z2(1) X3(1) Y3(1) Z3(1) X4(1) Y4(1) Z4(1)
```

```
X1(2) Y1(2) Z1(2) X2(2) Y2(2) Z2(2) X3(2) Y3(2) Z3(2) X4(2) Y4(2) Z4(2)
.
.
.
.   .   . . . . . .   .   . . . . X4(NPANCSF) Y4(NPANCSF) Z4(NPANCSF)
```

'header' denotes a one-line ASCII header dimensioned CHARACTER∗72.

ISXCSF, **ISYCSF** are the geometry symmetry indices which have integer values 0 or 1. If ISXCSF and/or ISYCSF =1, $x = 0$ and/or $y = 0$ is a geometric plane of symmetry, and the input data are restricted to one quadrant or one half of the control surface. Conversely, if ISXCSF=0 and ISYCSF=0, the complete control surface must be represented by panels.

ISXCSF = 1: The $x = 0$ plane is a geometric plane of symmetry.
ISXCSF = 0: The $x = 0$ plane is not a geometric plane of symmetry.

ISYCSF = 1: The $y = 0$ plane is a geometric plane of symmetry.
ISYCSF = 0: The $y = 0$ plane is not a geometric plane of symmetry.

For all values of ISXCSF and ISYCSF, the (x, y) axes are understood to belong to the body system of the corresponding GDF file and the panel data are always referenced with respect to this system.

NPANCSF is equal to the number of panels with coordinates defined in this file.

XI(J), YI(J), ZI(J) are the Cartesian coordinates (x, y, z) of I-th vertex of the J-th panel. The four vertices of a panel are specified in the anti-clockwise direction when the panel is viewed from outside of the control surface, as in the case of the body surface illustrated in Figure 5.1.

Further description of the data in the CSF file can be found in Section 5.1, which describes the low-order GDF file.

14.3 HIGHER-ORDER CONTROL SURFACE FILE

In the higher-order method the first part of the CSF file is as follows:

```
header
1 (ILOWHICSF)
ISXCSF  ISYCSF
NPATCSF  ICDEF  PSZCSF
```

Subsequent data may be included in the CSF file after these four lines, depending on the manner in which the geometry of the control surface is represented, in the same manner as for the GDF file (See Sections 6.5-6.8.)

header, **ISXCSF** and **ISYCSF** are the same as those in the CSF file for ILOWHI=0 above.

NPATCSF is equal to the number of patches used to describe the control surface. If one or two planes of symmetry are specified, NPATCSF is the number of patches required to discretize a half or one quadrant of the whole of the control surface, respectively.

ICDEF is an integer parameter which is used to specify the manner in which the geometry of the control surface is defined. Four specific cases are relevant, corresponding respectively to the representations explained in Sections 6.5, 6.6, 6.7 and 6.8:

ICDEF = 0: The geometry of each patch is a flat quadrilateral, with vertices listed in the CSF file (cf. Section 6.5).

ICDEF = 1: The geometry of each patch is represented by B-splines, with the corresponding data in the CSF file (cf. Section 6.6).

ICDEF = 2: The geometry is defined by inputs from a MultiSurf .ms2 file (cf. Section 6.7).

ICDEF < 0 or > 2: The geometry of each patch is represented explicitly by a subroutine in the library GEOMXACT, with optional data in the CSF file (cf. Section 6.8).

In the last case (ICDEF < 0 or > 2), the parameter ICDEF is used in the same manner as IGDEF, to select the appropriate subroutine. Any of the existing subroutines which are normally used to define body geometry can be used to define the control surface, for example specifying ICDEF=-1 defines the control surface as a circular cylinder with specified radius and draft. It is important to use different subroutines for the body geometry and control surface, with ICDEF$\neq$IGDEF. (The reason for this restriction is that, when parameters are input by the same subroutine from the GDF file and CSF file, these parameters may be overwritten. If it is desired to use the same subroutine for both geometries, a duplicate copy of the subroutine with a different name and assigned value of IGDEF should be added to GEOMXACT.)

PSZCSF is a parameter which controls the accuracy of the numerical integration over the control surface, in the same manner that PANEL_SIZE is used on the body (cf. Chapter 6). Thus the control surface is subdivided into elements with the approximate length scale of each element equal to PSZCSF. If the parameter PSZCSF is negative, the subdivision of the control surface is determined by the parameters NU,NV in the file *gdf*.CSP, in an analogous manner to the use of the same parameters in the spline control file *gdf*.SPL (Section 6.11). TEST22 is an example where the latter procedure is used.

14.4 OUTPUT

The mean forces and moments using a control surface are output in the OUT file and in the numeric output file optn.**9c** (or *frc*.**9c**) in the same format as optn.9. When IPLTDAT> 0 in the CFG file, the auxiliary file *gdf*_**csf.dat** is output which can be used for visualization of the control surface using Tecplot. When ILOWHICSF= 1 and ILOWGDF> 0 in the CFG file, *gdf*_**low.csf** is created which contains the data of the control surface in the format for the low order method.

TEST05, TEST13 and TEST22 show examples of using the control surface in the evaluation of the mean forces and moments.

REFERENCES

1. J.N. Newman. "Algorithms for the Free-Surface Green Function". *Journal of Engineering Mathematics,* Vol. 19, pp. 57-67, 1985.

2. J. N. Newman. "Distribution of Sources and Dipoles over a Quadrilateral". *Journal of Engineering Mathematics,* Vol. 20, pp. 113-126, 1986.

3. J. N. Newman. *Marine Hydrodynamics.* MIT Press, 1977.

4. J. N. Newman. "The Drift Force and Moment on Ships in Waves". *Journal of Ship Research,* Vol. 11, pp. 51-60, 1967.

5. F. T. Korsmeyer, C.-H. Lee, J. N. Newman and P. D. Sclavounos. "The Analysis of Wave Effects on Tension-Leg Platforms". *Invited paper of OMAE '88 Conference*, Houston, TX, 1988.

6. J. N. Newman, and P. D. Sclavounos. "The Computation of Wave Loads on Large Offshore Structures". *BOSS '88 Conference*, Trondheim, Norway, 1988.

7. R. Eatock Taylor and E. R. Jefferys. "Variability of Hydrodynamic Load Predictions for a Tension Leg Platform". *Ocean Engineering,* Vol. 13, No. 5, pp. 449-490, 1986.

8. X. Zhu, "Irregular Frequency Removal from the Boundary Integral Equation for the Wave-body Problem", Master Thesis, Dept. of Ocean Eng., MIT, 1994.

9. C.-H. Lee, J.N. Newman, M.-H. Kim & D.K.P. Yue. "The computation of second-order wave loads". *OMAE '91 Conference*, Stavanger, Norway, 1991.

10. C.-H. Lee, and J.N. Newman. "First- and second-order wave effects on a submerged spheroid," *Journal of Ship Research*, 1991.

11. J.N. Newman. "The approximation of free-surface Green functions," in *Wave Asymptotics,* P. A. Martin & G. R. Wickham, editors, Cambridge University Press, 1992.

12. J.N. Newman and C.-H. Lee "Sensitivity of wave loads to the discretization of bodies" *BOSS '92*, London, England, 1992

13. J.N. Newman "Wave effects on deformable bodies", *Applied Ocean Research,* **16**, 1, 47-59, 1994

14. C.-H. Lee and X. Zhu "Second-order diffraction and radiation solutions on floating bodies" *8th Int'l Workshop on Water Waves and Floating Bodies, St. John's, Newfoundland, Canada* 1993

15. C.-H. Lee, and J.N. Newman. "Second-order Wave Effects on Offshore Structures" *BOSS'*94, MIT, 1994

16. C.-H. Lee, J.N. Newman and X. Zhu "An extended boundary-integral-equation method for the removal of irregular-frequency effects," *International Journal for Numerical Methods in Fluids*, **23**, 637-660, 1996.

17. C.-H. Lee, "WAMIT Theory Manual", Report 95-2, Dept. of Ocean Engineering, MIT, 1995

18. H. Maniar, "A three dimensional higher-order panel method based on B-splines. Ph.D. Thesis, Department of Ocean Engineering, MIT, Cambridge, Massachusetts, 1995.

19. C.-H. Lee, H. Maniar, J.N. Newman and X. Zhu, "Computation of wave loads using a B-spline panel method" *Proceedings*, 21st Symposium on Naval Hydrodynamics, Trondheim, Norway, 1996.

20. C.-H. Lee and J. N. Newman, "HIPAN V2.1 User Manual", MIT, 1999

21. J.E. Kerwin and C.S. Lee, "Prediction of Steady and Unsteady Marine Propeller Performance by Numerical Lifting-Surface Theory." *Transactions*, Society of Naval Architects and Marine Engineers, 86, 1978.

22. M. E. Mortenson *Geometric Modeling*, Second Edition, Wiley, 1997. Digital Press, 1999.

23. M. Etzel and K. Dickinson, *Digital Visual Fortran Programmer's Guide*, Digital Press, 1999.

24. C.-H. Lee, J. S. Letcher, Jr., R. G. Mack II, J. N. Newman, D. M. Shook and E. Stanley. "Integation of Geometry Definition and Wave Analyis Software". *OMAE 2002 Conference*, Oslo, June 2002.

25. C.-H. Lee and J. N. Newman, "Boundary-Element Methods in Offshore Structure Analysis". *OMAE 2001 Conference*, Rio de Janeiro, 2001. Also published in *Journal of Offshore Mechanics and Arctic Engineering,* Vol. 124, pp 81-89 (2002).

26. C.-H. Lee and J. N. Newman, "Computation of wave effects using the panel method". In *Numerical Modeling in Fluid-Structure Interaction,* Edited by S. Chakrabarti. WIT Press, 2004.

27. Newman, J.N., "Wave Effects on Vessels with Internal Tanks". *20th International Workshop on Water Waves and Floating Bodies*, Spitsbergen, Norway, 2005.

28. Lee, C.-H., "Evaluation of quadratic forces using control surfaces". *2005 WAMIT Consortium Report*, www.wamit.com\report05.pdf, 2005.

Appendix A

DESCRIPTION OF TEST RUNS

WAMIT V6 includes 23 standard test runs, including 9 low-order and 14 higher-order applications. These are designed to illustrate various different options and features of WAMIT, and to help users to develop appropriate input files for their own purposes.

The following table gives relevant features of each test run. In this table the first column *tst* denotes the name of the test run. All of the corresponding input/output files (except FNAMES.WAM) are assigned the filenames TEST*tst*. (For example, the input POT file for the first test run listed below is TEST01.POT.) The first character of *tst* is 0 for low-order test runs (ILOWHI=0), and ≥1 for higher-order test runs (ILOWHI=1). Higher-order test runs which are identical except for different input options are assigned the same number with a letter suffix. Thus TEST11 and TEST11a-c describe the same physical problem using different options to represent the geometry (B-splines, exact analytic formulae, MultiSurf, uniform and nonuniform mapping). In TEST14, the ISSC TLP is analysed and the use of the fixed mode option is illustrated. In TEST14A, the same geometry is analysed for a large number of input frequencies including zero and infinite frequencies and the outputs are postprocessed by the F2T utility. In TEST16 the barge is defined by the subroutine BARGE (IGDEF=-5), and in TEST16a the patches are defined by flat panels (IGDEF=0). Tests 17, 17a, and 17b illustrate alternative methods for analyzing a body with moonpools, as explained in detail in the corresponding section below. TEST23 illustrates the use of WAMIT to compute the radiated wavefield from a bank of wavemakers.

tst	geometry	ILOWHI	other parameters
01	Circular cylinder	0	
02	Circular cylinder	0	IRR=3
03	Circular cylinder	0	ISOR=1
04	Barge near wall	0	ISY=-1
05	Cylinder & spheroid	0	NBODY=2
06	ISSC TLP (coarse)	0	NPAN=128
07	ISSC TLP (fine)	0	NPAN=1012
08	Elastic column	0	NEWMDS=4
09	Spar with strakes	0	NPAND=288
11	Circular cylinder	1	IGDEF=1
11a	Circular cylinder	1	IGDEF=-1, INONUMAP=0
11b	Circular cylinder	1	IGDEF=-1, INONUMAP=1
11c	Circular cylinder	1	IGDEF=2
12	Circular cylinder	1	IGDEF=-1, INONUMAP=0, IRR=1
13	Cylinder & spheroid	1	NBODY=2
14	ISSC TLP	1	IGDEF=-9
15	Semi-sub	1	IGDEF=-10
16	Elastic barge	1	IGDEF=-5
16a	Elastic barge	1	IGDEF=0
17	Cylinder & moonpool	1	IGDEF=-7,
17a	Cylinder & moonpool	1	IGDEF=-7, NEWMDS=2, IDAMP=0
17b	Cylinder & moonpool	1	IGDEF=-7, NEWMDS=2, IDAMP=1
18	Elastic column	1	IGDEF=-1, NEWMDS=4
19	Catamaran barge	1	IGDEF=0
20	MultiSurf barge	1	IGDEF=2
21	Spar with strakes	1	IGDEF=-12
22	FPSO with 2 tanks	1	IGDEF=-21
23	Bank of wavemakers	1	IGDEF=0
24	Motions of a hinged vessel	1	IGDEF=-32

All of the required input files for each test run, and the labeled output file (*.out) are included with the WAMIT software on the CD-ROM disk provided to licensed users. The same files can be downloaded with the demonstration programs from the web site `http:://www.wamit.com`. The input files for Test Run *tst* are named with the filename 'test*tst*' followed by the extensions *.gdf*, *.pot*, and *.frc*. The corresponding files **fnames** and **config** are given the same filenames with the extension *.tst*. Thus, before running TEST*tst*, the user should copy two files as follows:

```
copy fnames.tst fnames.wam
```

as explained in Chapter 2. Alternatively, the batch file **runtests.bat** can be used to run all tests in succession.

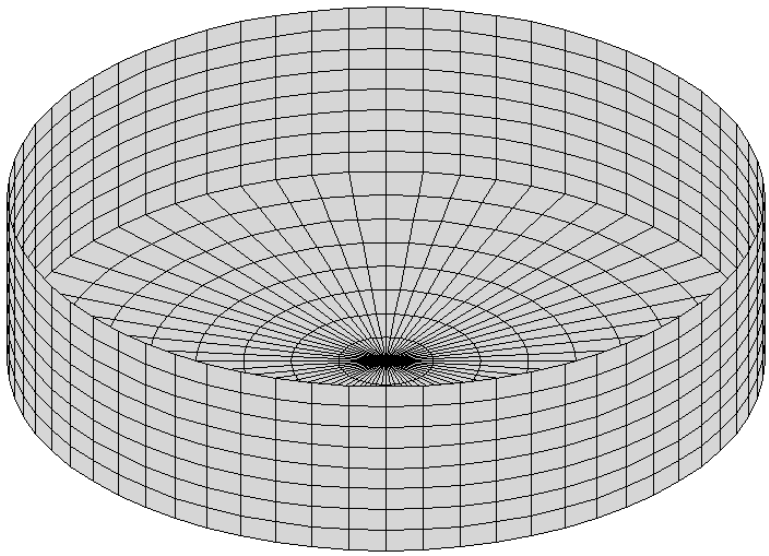
In all of the test runs metric units are used, and the gravitational acceleration is set equal to 9.80665 meters-per-second^2.

Each test run is described briefly in the following sections. Also included in these sections are perspective illustrations of the complete underwater geometry, including reflections about the indicated planes of symmetry, and abbreviated listings of the input files. For the low-order tests (01-09) the perspective figures show the subdivisions into panels. For the higher-order tests (11-22) two perspective figures are included, to show the subdivisions into patches (upper or left) and into panels (lower or right).

A.1 TRUNCATED VERTICAL CYLINDER – TEST01

The added-mass and damping coefficients, exciting forces, motions, wave elevations, field pressures, field velocities and drift forces are evaluated for a freely floating truncated vertical circular cylinder of radius 1 meter and draft 0.5 meters, in infinite water depth for three wave periods and one wave heading.

The origin of the coordinate system is located at the intersection of the vertical axis of the cylinder and the undisturbed position of the free surface. Using two planes of symmetry, only the first quadrant of the surface of the cylinder is discretized with 256 panels. 16, 8, and 8 panels are distributed in the azimuthal, radial, and vertical directions with equal spacing. The characteristic length is set equal to the radius of the cylinder. The cylinder center of gravity is located at the origin of the coordinate system, and the radii of gyration relative to its axes are taken equal to 1 meter.



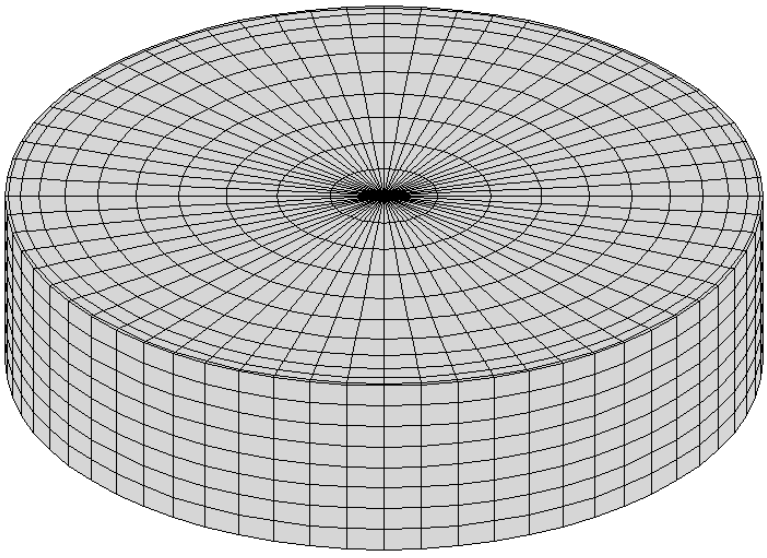
```
TEST01.GDF: (lines 1-8 only):
TEST01.GDF  --  circular cylinder, R=1, T=0.5, ILOWHI=0
    1.000000       9.806650     ULEN, GRAV
           1            1       ISX, ISY
         256                    NEQN
   0.0000000E+00  0.0000000E+00 -0.5000000
   0.0000000E+00  0.0000000E+00 -0.5000000
   0.1243981      1.2252143E-02 -0.5000000
   0.1250000      0.0000000E+00 -0.5000000

TEST01.POT:
TEST01.POT -- cylinder R=1, T=0.5, ILOWHI=0, IRR=0
 -1. 0. 0. 0. 0.                HBOT, XBODY(1-4)
 1           1                  IRAD, IDIFF
 1  1  1  1  1  1               IMODE(1-6)
 3                             NPER (array PER follows)
 8.971402 2.006403 1.003033
 1                             NBETA (array BETA follows)
 0.

TEST01.FRC:
TEST01.FRC  Circular cylinder, ILOWHI=0, IRR=0
 1    1    1    1    0    1    1    2    0   IOPTN(1-9)
 0.000000                                    VCG
 1.000000      .0000000      .0000000
 .0000000      1.000000      .0000000
 .0000000      .0000000      1.000000        XPRDCT
 0                                           NBETAH
 2                                           NFIELD
 1.5  0.  0.
 1.5  0. -0.5                                (end of file)

test01.cfg:
 maxscr=1024  (assign a maximum block of 1024*1024 RAM for scratch LHS)
 ISOR=1       (omit ISOR in POT file, include source formulation)
 ISOLVE=0     (use iterative solver)
 ISCATT=0     (solve for total diffraction potential, not scattering)
 IQUAD=0      (omit IQUAD in POT file, use single-node quadrature)
 ILOG=1       (omit ILOG in POT file, integrate log singularity)
 IDIAG=0      (omit IDIAG in POT file, panel length based on area)
 IRR=0        (omit IRR in POT file, no irregular-frequency removal)
 MONITR=0     (do not write FORCE output data to monitor)
 NUMHDR=1     (write headers to numeric output files)
 USERID_PATH=\WAMITv6   (directory for *.exe, *.dll, and userid.wam)
```

A.2 IRREGULAR-FREQUENCY REMOVAL – TEST02

This test run illustrates the use of the irregular-frequency option described in Chapter 9. The geometry and most other inputs are the same as in TEST01, as described above. The parameter IRR= 3 is set to use automatic panelization of the interior free surface. Wave periods are chosen so that the wave frequencies are near the first and second irregular frequencies of the cylinder.

The maximum number of iteration MAXITT is increased to 100 in the configuration file for this test run.

Input GDF is the same as TEST01.GDF. The following figure shows additional panels on the free surface which are generated in WAMIT.



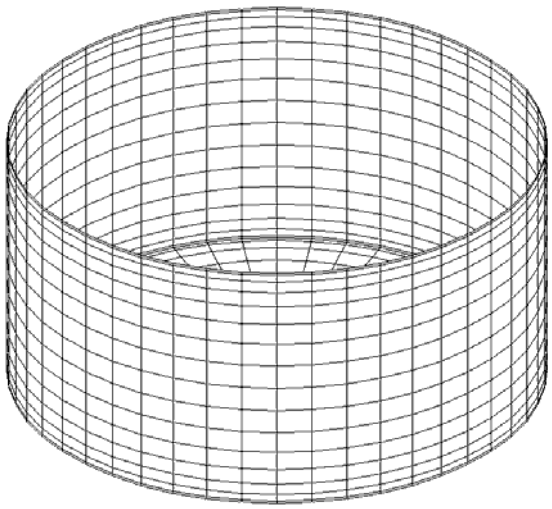
```
TEST02.GDF: (lines 1-8 only):
TEST02.GDF  circular cylinder, R=1, T=0.5, ILOWHI=0, IRR=3,
    1.000000       9.806650
            1           1
          256
   0.0000000E+00  0.0000000E+00 -0.5000000
   0.0000000E+00  0.0000000E+00 -0.5000000
   0.1243981      1.2252143E-02 -0.5000000
   0.1250000      0.0000000E+00 -0.5000000

TEST02.POT:
TEST02.POT -- Circular cylinder, ILOWHI=0, IRR=3
 -1.  0.0  0.0  0.0  0.0
 1           1
 1  1  1  1  1  1
 2
 1.182288    1.003025
 1
 0.0

TEST02.FRC:
TEST02.FRC  -- Circular cylinder, ILOWHI=0, IRR=3
 1    1    1    1    0    1    1    1    1
 0.000000
 1.000000      .0000000      .0000000
 .0000000      1.000000      .0000000
 .0000000      .0000000      1.000000
 0
 2
 1.5 0.0  0.0
 1.5 0.0 -0.5
test02.cfg:
 IRR=3
 ISOR=1
 ISOLVE=4
 IQUAD=0
 ILOG=1
 IDIAG=0
 NEWMDS=0
 MONITR=0
 NUMHDR=1
 maxscr=1024
 USERID_PATH=\WAMITv6   (directory for *.exe, *.dll, and userid.wam)
 MAXITT=100
```

A.3 LOCAL PRESSURE DRIFT FORCE – TEST03

This test run is used to illustrate the use of the source formulation (Section 5.2) to determine the mean drift force and moment from local pressure integration. The motions and the drift forces are evaluated for a freely floating truncated vertical circular cylinder of radius 1 meter and draft 1 meter, in a water depth of 7.14 meter for four wave periods and one wave heading.

The origin of the global coordinate system is located at the intersection of the vertical axis of the cylinder and the undisturbed position of the free surface. The origin of the body fixed coordinate system is shifted -0.515 meters under the free-surface. Using two planes of symmetry, the first quadrant of the surface of the cylinder is discretized with 288 panels. 12, 8, and 16 panels are distributed in the azimuthal, radial, and vertical directions with cosine spacing at the free surface and corner. The characteristic length is set equal to the radius of the cylinder. The cylinder center of gravity is located at the origin of the body coordinate system, and the radii of gyration relative to its axes are shown in the FRC file.

Generally speaking, the evaluation of mean drift forces is more accurate when the momentum conservation method is used, since this does not depend on local velocities on the body surface. However the momentum method cannot be used to evaluate the vertical component of the drift force. Tests with larger number of panels are reported in Reference [12] to show the sensitivity of the results to the discretization of the body. The vertical drift force predicted by the mean pressure integration shows slow convergence near the heave resonance frequency (KL=0.66), due to cancelation between two large contributions of opposite signs (the second integration in equations (12.47) and (12.48)), when the heave motion amplitude is large.



```
TEST03.GDF: (lines 1-8 only):
TEST03.GDF  Cylinder, R=T=1, ILOWHI=0, ISOR=1
    1.000000       9.806650
            1           1
          288
   0.0000000E+00  0.0000000E+00 -0.4850000
   0.0000000E+00  0.0000000E+00 -0.4850000
   0.1934213      2.5464399E-02 -0.4850000
   0.1950903      0.0000000E+00 -0.4850000

TEST03.POT:
TEST03.POT  Cylinder, R=T=1, ILOWHI=0, ISOR=1
    7.14  0.0  0.0  -0.515 0.0
    1           1
    1  1  1  1  1  1
    4
    2.837491 2.398118 2.006409 1.638226
    1
    0.0

TEST03.FRC:
TEST03.FRC  Cylinder, R=T=1, ILOWHI=0, ISOR=1
  0    0    0    1    0    0    0     1     1
  0.000000
  0.742000      0.000000      0.000000
  0.000000      0.742000      0.000000
  0.000000      0.000000      1.000000
  0
  0

test03.cfg:
 maxscr=1024
 ISOR=1
 ISOLVE=0
 ISCATT=0
 IQUAD=0
 ILOG=0
 IDIAG=0
 IRR=0
 MONITR=0
 NUMHDR=1
 USERID_PATH=\WAMITv6   (directory for *.exe, *.dll, and userid.wam)
```

A.4 BODY NEAR A WALL – TEST04

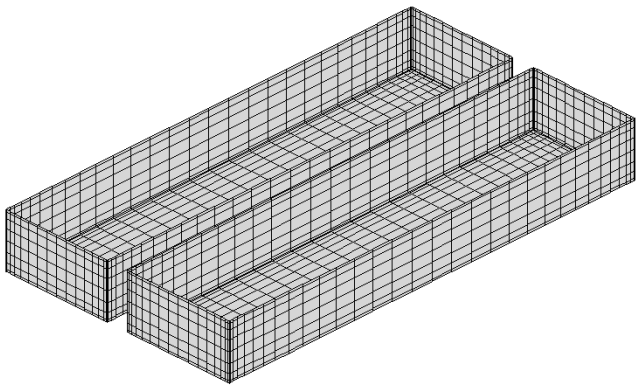
The option to analyze a body near one or two vertical walls is described in Section 5.3. In this test run a rectangular barge of length 80m, beam 20m, draft 10m is positioned with its longitudinal axis parallel to one wall, separated by a gap of 2m. Incident head waves are considered, and computations are made of the surge, heave, and pitch coefficients, RAO's, and drift force and moment in incident waves which propagate parallel to the wall (BETA=0).

In the GDF file one half of the barge is discretized, forward of the midship section $x = 0$. Both the port and starboard sides of the barge are included in the GDF file, hence the appropriate symmetry indices for this case are ISX=1, ISY=-1.

Since the incident waves propagate parallel to the wall this problem is identical to the 'barge catamaran' studied in [6], and in TEST19; the only modifications required in the latter case are (1) a lateral offset equal to the sum of the half-beam and gap must be added to the y-coordinates of the panels in the GDF file; (2) ISY=1; and (3) the forces and moments calculated for the catamaran are the total acting on both hulls. The definition of the incident-wave amplitude differs between these different problems, however, due to the convention for the wave amplitude in the presence of a wall (Section 5.3). In the present case, where the incident-wave angle is zero and the waves propagate parallel to the wall, the wave system in the absence of the body is a progressive wave with total physical amplitude $2A$.

It also is possible to replicate the present results with the NBODY option, specifying two independent hulls in place of the rigid constraint implied by the catamaran; this is a less efficient computational approach since the planes of symmetry are not exploited.

The figure below shows the catamaran configuration or, equivalently, the original hull plus its image with respect to the wall.



```
TEST04.GDF: (lines 1-8 only):
TEST04.GDF -- Barge near wall, ILOWHI=0
    40.00000       9.806650
            1           -1
          640
    3.920686       10.00000     -0.3806022
   0.0000000E+00   10.00000     -0.3806022
   0.0000000E+00   10.00000      0.0000000E+00
    3.920686       10.00000      0.0000000E+00

TEST04.POT:
TEST04.POT -- Barge near wall, ILOWHI=0
 -1. 0. 12. 0. 0.
 0           0
 1  0  1  0  1  0
 3
 6.  7.  8.
 1
 0.0

TEST04.FRC:
TEST04.FRC -- Barge near wall, ILOWHI=0
 1    1    1    1    0    0    0    0    1
 3.0
 20.00000      0.000000      0.000000
 0.000000      5.000000      0.000000
 0.000000      0.000000      20.00000
 0
 0

test04.cfg:
 maxscr=1024
 ISOR=1
 ISOLVE=0
 ISCATT=0
 IQUAD=0
 ILOG=0
 IDIAG=0
 IRR=0
 MONITR=0
 NUMHDR=1
 USERID_PATH=\WAMITv6   (directory for *.exe, *.dll, and userid.wam)
```

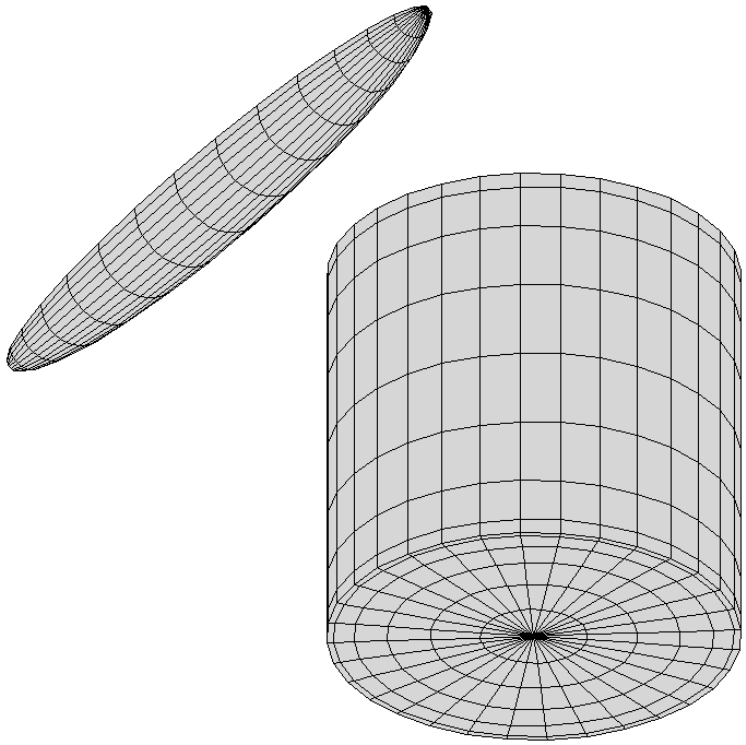
A.5 MULTIPLE BODIES – TEST05

The NBODY option described in Chapter 7 is illustrated in this test run. Body one is a circular cylinder of radius 1 meter and draft 2 meters. Body 2 is a spheroid of length 4 meters and maximum radius 0.25 meters. The gap between these two bodies is set equal to the beam of the spheroid (0.5 meters) and the origin of the global coordinate system is located at the mid-point of this gap. The relative locations of the two bodies and the orientation of the spheroid are specified in the GGDF file. One quadrant of the cylinder is discretized with 112 panels. 8,6 and 8 panels are distributed in the azimuthal, radial, and vertical directions using cosine spacing in radial and vertical directions. One quadrant of the spheroid is discretized with 64 panels. 8 and 8 panels are distributed in the longitudinal, and transverse directions using cosine spacing in the longitudinal direction.

The Alternative 1 input format is used for the POTEN subprogram and the Alternative 3 input format is used for FORCE. The separate FRC files TEST05C, TEST05S are used with IALTFRC=3. The vector IALTFRCN is included in TEST05.CFG to indicate that IALTFRC=1 in the separate FRC files for each body.

The added-mass and damping coefficients, exciting forces, motions, wave elevations, field pressures and field velocities, and drift forces are evaluated in infinite water depth for two wave periods and one wave heading.

The option is used to evaluate the mean drift force and moment using a control surface, following the instructions in Chapter 14. The parameter ICTRSURF=1 is assigned in the CFG file. The control surfaces surrounding the cylinder and spheroid are defined by the input files TEST05c.csf and TEST05s.csf. In order to illustrate the alternatives, the control surface for the cylinder uses low-order panels (ILOWHICSF=0) and the control surface for the spheroid is generated with the higher-order (ILOWHICSF=1) subroutine ELLIPSOID_CS in the GEOMXACT DLL library. The corresponding output for the mean drift force and moment is contained in the file TEST05.9c. It should be noted that the higher-order control surface for the spheroid does not include the intermediate free surface patch, and thus the horizontal drift force is correct whereas the vertical drift force is not complete. The reason for omitting the free surface patch here is that the low-order solution for the body does not give a sufficiently robust evaluation of field velocities and wave elevations at points on the free surface that are very close to the body. The low-order control surface is more suitable for use with low-order body representations, in this respect, provided the panels on the free surface have dimensions similar to the dimensions of the adjacent panels on the body.



```
TEST05.GDF:
TEST05.GDF -- GGDF file for use with IGDFPOT_OPTIONS=0
 -1. 9.80665
 2
 test05c.gdf
 1.25 0.0 0.0 0.0
 1   1   1   1   1   1
 test05s.gdf
 -0.5 0.0 0.0 90.0
 1   1   1   1   1   1

TEST05c.GDF: (lines 1-8 only):
Cylinder R=1 T=2 8*(6+8)
    1.000000       9.806650
             1           1
           112
   0.0000000E+00  0.0000000E+00  -2.000000
   0.0000000E+00  0.0000000E+00  -2.000000
   0.2538459      5.0493091E-02  -2.000000
   0.2588190      0.0000000E+00  -2.000000

TEST05s.GDF: (lines 1-8 only):
```

```
Spheroid, Slendernes =0.125 Halflength=2m 8*8
    2.000000       9.806650
            1           1
           64
    2.000000     -0.0000000E+00 -0.0000000E+00
    1.961571     -7.9460625E-09 -4.8772585E-02
    1.961571      9.5150545E-03 -4.7835436E-02
    2.000000      0.0000000E+00 -0.0000000E+00

TEST05.POT:
TEST05.POT -- Cylinder + spheroid, ILOWHI=0
 -1.0 0.0 0.0 0.0 0.0
 0   0
 0  0  0  0  0  0
 2
 1.5 2.0
 1
 0.0

TEST05.FRC:
TEST05.FRC -- Cylinder + spheroid, ILOWHI=0
  1  1  1  1  0  1  1  1  1
  1.0
 test05c.frc
 test05s.frc
  0
  1
  0. 0. 0.

TEST05c.FRC:
CYL.FRC
 0    0    0    0    0    0    0    0    0
 0.000000
 1.000000      .0000000      .0000000
 .0000000      1.000000      .0000000
 .0000000      .0000000      1.000000
 0
 0

TEST05s.FRC:
SPD.FRC
 0    0    0    0    0    0    0    0    0
 0.000000
 1.000000      .0000000      .0000000
```

```
 .0000000      1.000000      .0000000
 .0000000      .0000000      1.000000
 0
 0

TEST05c.csf:
ccylinder R=1.2 T=2.2 -- analytic CONTROL SURFACE (npatch=3)
    0        ILOWHICSF
    1     1  ISX ISY
    160      NPAN
  0.12000E+01  0.00000E+00  0.00000E+00
  0.12000E+01  0.00000E+00 -0.27500E+00
  0.11769E+01  0.23411E+00 -0.27500E+00
  0.11769E+01  0.23411E+00  0.00000E+00

TEST05s.csf:
ELLIPSOID CONTROL SURFACE for ellisoid GDF-- igdef=-1003 without free surface portion
1   ILOWHICSF
1  1        ISX  ISY
1  -1003  0.5 NPATCH  IGDEF PSZCSF
2           NLINES
2.2 0.3 0.3    A, B, C
2.0 0.25 (x and y maximum of ellipsoid GDF)

test05.cfg:
 IPLTDAT=1
 maxscr=1024
 ISOR=0
 IQUAD=0
 ILOG=0
 IDIAG=0
 IRR=0
 NUMHDR=1
 NOOUT=0 1 1 1 0 1 1 0 0
 IALTPOT = 1    !  GDF names in GGDF file
 IALTFRC = 3    !  Alternative Form 3 FRC
 IALTFRCN= 1 1
 ICTRSURF=1     !  Evaluate control surface drift forces
 USERID_PATH=\WAMITv6   (directory for *.exe, *.dll, and userid.wam)
```

A.6 THE ISSC TENSION-LEG PLATFORM – TEST06

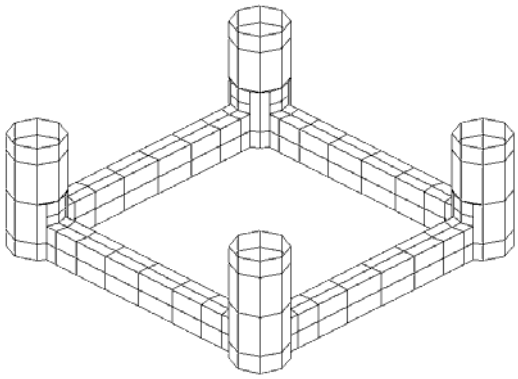
The added-mass, damping coefficients, exciting forces, motions and wave loads are evaluated for the ISSC Tension-Leg-Platform, in a finite water depth of 450 meters, for three wave periods and one wave heading. The TLP consists of four circular cylindrical columns and four rectangular pontoons as shown in the plots of the panel discretization. The radius of each column is 8.435 meters. The width and height of each pontoon are 7.5 meters and 10.5 meters, respectively. The distance between the centers of adjacent columns is 86.25 meters. Further information is given by Eatock Taylor and Jefferys [7].

Two planes of symmetry are used with 128 panels in one quadrant. Thus there are a total of 512 panels on the complete surface. The origin of the coordinate system is located at the intersection of the undisturbed free surface and the two planes of symmetry. The characteristic length is set equal to 43.125 meters, which corresponds to half of the distance between the centers of adjacent columns.

Only head seas are considered, with $\beta = 0$ specified in the .pot file. For this reason, only the modes (surge, heave, pitch) are analyzed with IRAD=IDIFF=0, and these modes are specified on line 3.

In the .frc file the horizontal modes (1,2,6) are free and the vertical modes (3,4,5) are fixed, to represent a TLP moored by vertical tendons. The Alternative 1 form is used, with the result that the body mass is evaluated as if the TLP is freely floating (see Section 3.5).

The output shows the conventional response amplitude operator for surge, and the wave loads for heave and pitch.



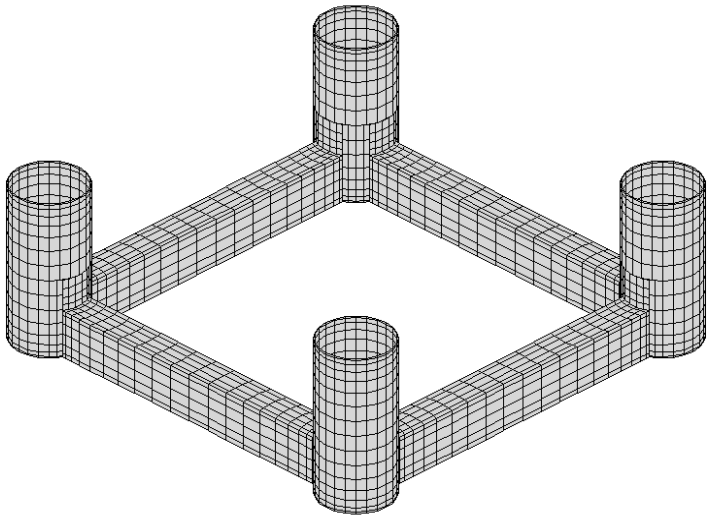
```
TEST06.GDF: (lines 1-8 only):
TEST06.GDF -- ISSC TLP, coarse discretization
  43.125     9.80665
   1           1
          128
                 49.09267       37.15733        0.00000
                 49.09267       37.15733       -5.12567
                 51.56456       43.12500       -5.12567
                 51.56456       43.12500        0.00000

TEST06.POT:
TEST06.POT -- ISSC TLP, coarse discretization
 450. 0. 0. 0. 0.
 0  0
 1  0  1  0  1  0
 3
 5. 10. 15.
 1
 0.

TEST06.FRC:
TEST06.FRC -- ISSC TLP, coarse discretization
 1  1  1 -1  1  0  0  0  0  IOPTN (IOPTN(4)<0 signifies fixed modes)
 6                         NDFR
 1 1 0 0 0 1               IMODE
 3.0                       VCG
 38.876     0.        0.
  0.       38.876     0.
  0.        0.       42.420
  0                        NBETAH
  0                        NFIELD

test06.cfg:
 maxscr=1024
 ISOR=0
 ISOLVE=0
 ISCATT=0
 IQUAD=0
 ILOG=0
 IDIAG=0
 IRR=0
 MONITR=0
 NUMHDR=1
 NOOUT= 1 1 1 1 0 1 1 1 1
 USERID_PATH=\WAMITv6   (directory for *.exe, *.dll, and userid.wam)
```

A.7 THE ISSC TENSION-LEG PLATFORM – TEST07

This test run is intended to refine the analysis of the ISSC TLP described in TEST06. 1012 panels are used on each quadrant, resulting in 4048 panels for the complete structure. The block iterative solver is used (ISOLVE=4) to provide a relatively fast but robust solution. For the sake of variety in the analysis of the diffraction problem, the solution for the scattered potential is computed (ISCATT=1).

Alternative form 2 of .FRC is used, but the mass is assumed to be equal to the displacement computed by WAMIT. Note that the displacement is about 4% greater than for Test Run 2, due to the more accurate description of the columns. (The panel vertices are defined to lie on the exact circular cylinder surface, hence the flat panels define a surface with less displaced volume than the exact body.)

Comparisons should be made with the output files from TEST06 to judge the convergence of the results with increasing numbers of panels.



```
TEST07.GDF: (lines 1-8 only):
TEST07.GDF  ISSC TLP -- ILOWHI=0, fine discretization
     43.125   9.806650
     1       1
     1012
     49.09267      37.15733       0.00000
     49.09267      37.15733      -0.33626
     50.43388      38.90522      -0.33626
     50.43388      38.90522       0.00000

TEST07.POT:
TEST07.POT  ISSC TLP -- ILOWHI=0, fine discretization
 450. 0. 0. 0. 0.
 0  0
 1  0  1  0  1  0
 3
 5. 10. 15.
 1
 0.

TEST07.FRC:
TEST07.FRC  ISSC TLP -- ILOWHI=0, fine discretization -- IALTFRC=2
 1  1  1 -2  0  0  0  0  0  IOPTN (IOPTN(4)<0 signifies fixed modes)
 6                          NDFR
 1 1 0 0 0 1                IMODE
 1.                         RHO
 0. 0. 3.0                  XCG
 1                          IMASS
    53066.4      0.        0.        0.     159199.2         0.
        0.   53066.4       0.  -159199.2         0.          0.
        0.       0.    53066.4       0.          0.          0.
        0. -159199.2       0.  8.0201552E7       0.          0.
   159199.2      0.        0.        0.   8.0201552E7        0.
        0.       0.        0.        0.          0.  9.54906731E7
 0                           IDAMP
 0                           ISTIFF
 0                           NBETAH
 0                           NFIELD
```

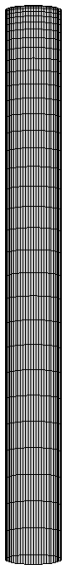
```
test07.cfg:
 maxscr=1024
 ISOR=0
 ISOLVE=4
 ISCATT=1
 IQUAD=0
 ILOG=0
 IDIAG=0
 IRR=0
 MONITR=0
 NUMHDR=1
 IALTFRC=2
 USERID_PATH=\WAMITv6   (directory for *.exe, *.dll, and userid.wam)
```

A.8 ELASTIC COLUMN WITH GENERALIZED MODES – TEST08

This test run evaluates the force coefficients and RAO's for a bottom-mounted vertical cylinder of circular cross-section, with four bending modes defined by shifted Jacobi polynomials. The hydroelastic analysis of these bending modes is analyzed using the generalized body mode option described in Chapter 8. Further details are given in Reference [13]. The cylinder extends from the free surface, where it is free, down to the bottom, at a depth of 200m, where it is clamped. The cylinder radius is 10m. Since the cylinder is clamped at the bottom the six rigid-body modes are all fixed, and specified by the values MODE(j)=0 in the POT file.

External mass and stiffness matrices are defined in the (Alternative 2) FRC file. The cylinder is considered to have a constant distributed mass equal to half of the displaced mass of fluid, and also a concentrated mass at the free surface equal to the displaced mass. The stiffness factor EI for the beam equation is assumed constant with the value $0.41 m_0 h^3$, where m_0 is the concentrated mass and h is the fluid depth. No matrix elements are required for the square submatrix $(i, j) \leq 6$ since the body is fixed in these modes. Further details for this case are given in [13].

The cylinder geometry is defined with two planes of symmetry and 512 panels on one quadrant. The length scale ULEN is specified as 1.0 to simplify the definitions of modes and output quantities. The generalized modes are defined in the subroutine DEFMOD.FOR, which is distributed to licensed users with the test files on the CD-ROM disk. The use of DEFMOD is described in Chapter 8. The output file from DEFMOD, TEST08.MOD, is included with the test files so that this test can be run without prior use of DEFMOD. Only one wave period is considered here, which coincides with resonant bending motion of the cylinder.



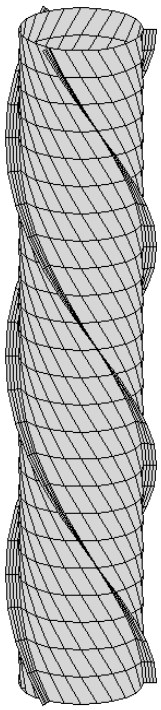
```
TEST08.GDF: (lines 1-8 only):
TEST08.GDF vertical cylinder, 16*32, cosine spacing at free surface
     1.0000        9.80665
           1           1
         512
    10.0000       0.000000       -200.000
    9.95185       0.980171       -200.000
    9.95185       0.980171       -190.186
    10.0000       0.000000       -190.186

TEST08.POT:
TEST08.POT -- bending of vertical column at resonance, 200m depth
      200.0  0.0  0.0  0.0  0.0
           0  0  0
           0   0
       0   0   0   0   0   0
           1
           6.5
           1
           0.0

TEST08.FRC:
TEST08.FRC file, vertical column with 4 bending modes
         1    1    1    1    0    0    0    0   0
        1.0
        .0000000      .0000000      1.000000
    1
  0.   0.   0.   0.   0.   0.        0.        0.        0.        0.
  0.   0.   0.   0.   0.   0.        0.        0.        0.        0.
  0.   0.   0.   0.   0.   0.        0.        0.        0.        0.
  0.   0.   0.   0.   0.   0.        0.        0.        0.        0.
  0.   0.   0.   0.   0.   0.        0.        0.        0.        0.
  0.   0.   0.   0.   0.   0.        0.        0.        0.        0.
  0.   0.   0.   0.   0.   0.    69115.    62832.    62832.    62832.
  0.   0.   0.   0.   0.   0.    62832.    67320.    62832.    62832.
  0.   0.   0.   0.   0.   0.    62832.    62832.    66323.    62832.
  0.   0.   0.   0.   0.   0.    62832.    62832.    62832.    65688.
    0
    1
  0.   0.   0.   0.   0.   0.        0.        0.        0.         0.
  0.   0.   0.   0.   0.   0.        0.        0.        0.         0.
  0.   0.   0.   0.   0.   0.        0.        0.        0.         0.
  0.   0.   0.   0.   0.   0.        0.        0.        0.         0.
  0.   0.   0.   0.   0.   0.        0.        0.        0.         0.
```

```
  0.   0.   0.   0.   0.   0.        0.        0.        0.         0.
  0.   0.   0.   0.   0.   0.   103044.   412177.   824354.   1339575.
  0.   0.   0.   0.   0.   0.   412177.  4430902.  9789203.  16487078.
  0.   0.   0.   0.   0.   0.   824354.  9789203. 37899671.  64382041.
  0.   0.   0.   0.   0.   0.  1339575. 16487078. 64382041. 162406554.
    0
    0

test08.cfg:
 MAXSCR=1024
 NUMHDR=1
 NUMNAM=0
 ISOR=0
 IRR=0
 MONITR=0
 newmds=4
 IALTPOT=1
 IALTFRC=2
 USERID_PATH=\WAMITv6   (directory for *.exe, *.dll, and userid.wam)
```

A.9 SPAR WITH THREE STRAKES – TEST09

This test run analyzes a circular cylinder with three spiral strakes. The strakes are modeled as zero-thickness dipole panels, following the method described in Section 5.4. The radius of the cylinder is 18m and the draft is 200m. The strake width is 3.7m. There are no planes of symmetry, due to the twist of the strakes. A total of 960 panels are used, including 672 on the cylinder plus 288 on the strakes. The excerpts from the GDF file include the first body panel and also the first dipole panel. A perspective view of the discretized structure is shown in the figure.

The FORCE run includes all options which can be evaluated without using the source formulation (ISOR=1), since the latter option cannot be used with dipole panels. The body pressure file TEST09.5p includes the pressure on the body panels, and the pressure jump on the dipole panels. The corresponding panel centroids are listed in the output file TEST09.PNL.



```
TEST09.GDF: (lines 1-8 and the first 5 lines for dipole panels)
SPAR  R, D, W, T, NS, TWIST=   18.00  200.00    3.70  0.000000    3  1.000
   18.00000       9.806650
           0           0
         672
   18.00000      0.0000000E+00  0.0000000E+00
   17.38667       4.658743      -8.333333
   15.58846       9.000000      -8.333333
   17.38667       4.658743      0.0000000E+00
         .
         .
         .
        288
   18.00000      0.0000000E+00  0.0000000E+00
   17.38667       4.658743      -8.333333
   18.28015       4.898150      -8.333333
   18.92500      0.0000000E+00  0.0000000E+00
         .
         .
         .


TEST09.POT:
TEST09.POT (Spar with three strakes)
 -1. 0. 0. 0. 0.                HBOT, XBODY(1-4)
 1           1                  IRAD, IDIFF
 1  1  1  1  1  1               IMODE(1-6)
 3                              NPER (array PER follows)
 0.1 0.5 1.0
 2                              NBETA (array BETA follows)
 0.0 45.                        (end of file)

TEST09.FRC:
TEST09.FRC (Spar with three strakes)
 1    1    1    1    1    1    1    2    0   IOPTN(1-9)
 0.000000                                   VCG
 1.000000      .0000000      .0000000
 .0000000      1.000000      .0000000
 .0000000      .0000000      1.000000       XPRDCT
 0                                          NBETAH
 2                                          NFIELD
 23.  0.  0.
 15.  15. -0.5                               (end of file)
```

```
test09.cfg:
 MAXSCR=1000
 ISOR=0
 ISOLVE=0
 ISCATT=0
 IQUAD=0
 ILOG=0
 IDIAG=0
 IRR=0
 MONITR=0
 NUMHDR=1
 IPERIO=3
 NOOUT= 1 1 1 1 0 1 1 1 1
 USERID_PATH=\WAMITv6   (directory for *.exe, *.dll, and userid.wam)
```

A.11 CIRCULAR CYLINDER – TEST11

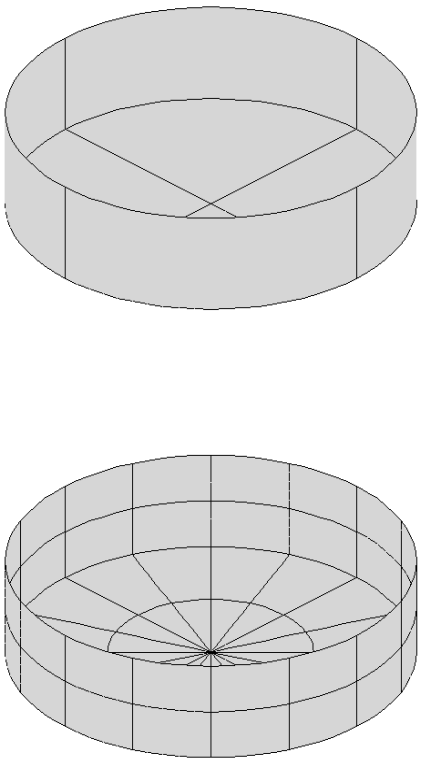
The same cylinder used for the low-order TEST01 is used here with the higher-order option (ILOWHI=1). Two alternatives are used for the geometry.

In TEST11 the geometry is defined by B-splines (IGDEF=1). The parameters, knot vectors, and coefficients for each patch are contained in the file TEST11.GDF. It should be noted that the circular patches and boundaries cannot be fit exactly with B-splines; however the geometric errors are generally much smaller in this case, compared to the flat-panel representation in TEST01. For example the maximum error of any point output in the data file test11.pnl is less than 3E-5, and the maximum error in the computed volume is 1E-5. By comparison, using the flat-panel discretization in TEST01, the maximum error in the computed volume is 3E-3. Thus, when the higher-order method is used, the principal errors in the results should be associated with the approximation of the potential by B-splines, as opposed to the representation of the geometry. This approximation can be systematically refined by increasing the number of panels, or by using the PANEL-SIZE option in the CONFIG.WAM (or CFG) file and reducing the value of this parameter.

In TEST11a the geometry is defined analytically by the GEOMXACT.F subroutine CIRCCYL (IGDEF=-1). The radius and draft of the cylinder are input in TEST11a.GDF. The parameter INONUMAP=0 specifies uniform mapping. Comparison of the output files with TEST01 and TEST11 confirms the statements above regarding accuracy. Most of the output data from TEST11 and TEST11a agree to at least five decimals, except for the third wave period which coincides with an irregular frequency.

In TEST11b the geometry is defined analytically, in the same manner as for TEST11a, except that nonuniform mapping is specified by the parameter INONUMAP=1 as explained in Section 6.8. This modification gives a more accurate solution near the corner and waterline, which is particularly beneficial for the pressure drift force evaluation. Comparison between the outputs for the momentum and pressure drift force shows that the results are more consistent in this case, compared to the use of uniform mapping in TEST11a. More extensive comparisons for the same geometry are included in Reference 24.

TEST11c illustrates the use of the option IGDEF=2, where the geometry is described by MultiSurf (see Section 6.7 and Appendix 2). In this case the same nonuniform mapping is used as in TEST11b, using the relabeling technique in MultiSurf. Comparison of the results with TEST11b indicates that they are practically identical.



```
TEST11.GDF: (lines 1-8 only):
TEST11  cylinder R=1 T=0.5 defined by B-splines (IGDEF=1)
1. 9.80665  ULEN GRAV
1  1        ISX  ISY
2  1        NPATCH, IGDEF
           4           2
           4           4
  -1.00000000000000       -1.00000000000000       -1.00000000000000
  -1.00000000000000      -0.500000000000000       0.000000000000000E+000

TEST11.SPL:
TEST11  cylinder R=1 T=0.5 defined by B-splines (IGDEF=1)
4  2        NU NV (Patch 1, side   u azimuthal v vertical)
4  2        NU NV (Parch 2, bottom u azimuthal v radial)

TEST11.POT:
TEST11.POT  Cylinder R=1, T=0.5, igdef=1
 -1.
 1           1               IRAD, IDIFF
```

```
 3                              NPER (array PER follows)
8.971402 2.006403 1.003033
 1                              NBETA (array BETA follows)
 0.
 1                              NBODY
 test11.gdf
 0. 0. 0. 0.                    XBODY
 1  1  1  1  1  1              IMODE(1-6)
 0                              NEWMDS

TEST11.FRC:
TEST11.FRC  Cylinder R=1, T=0.5, igdef=1
 1    1    1    1    3    1  1  2 2     IOPTN(1-9)
 0.000000                                VCG
 1.000000      .0000000      .0000000
 .0000000      1.000000      .0000000
 .0000000      .0000000      1.000000    XPRDCT
 0                                       NBETAH
 2                                       NFIELD
1.5 0.  0.
1.5 0. -0.5                             (end of file)

test11.cfg:
 ILOWHI=1
 IALTPOT=2
 IRR=0
 ISOLVE=2
 KSPLIN=3
 IQUADO=3
 IQUADI=4
 IPERIO=1
 MONITR=0
 NOOUT= 1 1 1 1 0 1 1 1 1
 NUMHDR=1
 USERID_PATH=\WAMITv6   (directory for *.exe, *.dll, and userid.wam)
```

```
TEST11A.GDF: (lines 1-8 only):
TEST11a cylinder R=1 T=0.5 -- analytic geometry, uniform mapping
1. 9.80665  ULEN GRAV
1  1        ISX  ISY
2  -1       NPATCH  IGDEF
2           NLINES
1.0 0.5     RADIUS, DRAFT
0           INONUMAP (uniform mapping)

 TEST11A.SPL:
TEST11a.spl - cylinder R=1 T=0.5 -- analytic geometry (npatch=2)
4  2        NU NV (Patch 1, side   u azimuthal v vertical)
4  2        NU NV (Parch 2, bottom u azimuthal v radial)

TEST11A.POT:
TEST11A.POT  Cylinder R=1, T=0.5, igdef=-1
 -1.
 1           1               IRAD, IDIFF
 2                           NPER (array PER follows)
8.971402 2.006403
 2                           NBETA (array BETA follows)
 0. 45.
 1                           NBODY
 test11a.gdf
 0. 0. 0. 0.                 XBODY
 1  1  1  1  1  1            IMODE(1-6)
 0                           NEWMDS


TEST11A.FRC:
TEST11a.FRC  Cylinder R=1, T=0.5, igdef=-1
 1    1    1    1    3    1  1  2 2     IOPTN(1-9)
 0.000000                                VCG
 1.000000      .0000000      .0000000
 .0000000      1.000000      .0000000
 .0000000      .0000000      1.000000    XPRDCT
 0                                       NBETAH
 2                                       NFIELD
1.5 0.  0.
1.5 0. -0.5                              (end of file)

test11a.cfg:
 ILOWHI=1
 IALTPOT=2
```

```
IRR=0
ISOLVE=2
KSPLIN=3
IQUADO=3
IQUADI=4
IPERIO=1
MONITR=0
NUMHDR=1
NOOUT= 1 1 1 1 0 1 1 1 1
USERID_PATH=\WAMITv6   (directory for *.exe, *.dll, and userid.wam)
```

```
TEST11B.GDF: (lines 1-8 only):
TEST11a cylinder R=1 T=0.5 -- analytic geometry, nonuniform mapping
1. 9.80665  ULEN GRAV
1  1        ISX  ISY
2  -1       NPATCH  IGDEF
2           NLINES
1.0 0.5     RADIUS, DRAFT
1           INONUMAP  (nonuniform mapping)

TEST11B.POT:
TEST11B.POT  Cylinder R=1, T=0.5, igdef=-1
 -1.
 1           1               IRAD, IDIFF
 2                            NPER (array PER follows)
8.971402 2.006403
 2                            NBETA (array BETA follows)
 0. 45.
 1                            NBODY
 test11b.gdf
 0. 0. 0. 0.                  XBODY
 1  1  1  1  1  1             IMODE(1-6)
 0                            NEWMDS

 TEST11B.SPL:
TEST11b.spl - cylinder R=1 T=0.5 -- analytic geometry (npatch=2)
4  2        NU NV (Patch 1, side   u azimuthal v vertical)
4  2        NU NV (Parch 2, bottom u azimuthal v radial)

TEST11B.FRC:
TEST11B.FRC  Cylinder R=1, T=0.5, igdef=-1
 1    1    1    1    3    1  1  2 2     IOPTN(1-9)
 0.000000                                  VCG
 1.000000      .0000000      .0000000
 .0000000      1.000000      .0000000
 .0000000      .0000000      1.000000      XPRDCT
 0                                         NBETAH
 2                                         NFIELD
1.5 0.  0.
1.5 0. -0.5                                (end of file)

test11B.cfg:
 ILOWHI=1
 IALTPOT=2
 IRR=0
```

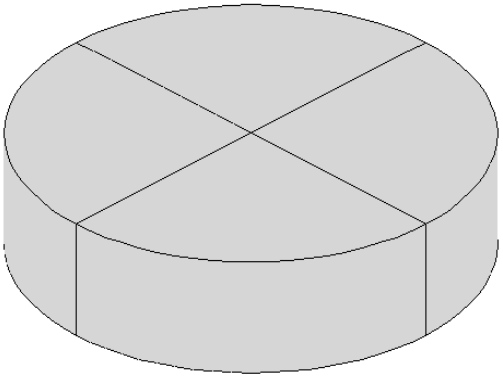
```
ISOLVE=2
KSPLIN=3
IQUADO=3
IQUADI=4
IPERIO=1
MONITR=0
NUMHDR=1
NOOUT= 1 1 1 1 0 1 1 1 1
USERID_PATH=\WAMITv6   (directory for *.exe, *.dll, and userid.wam)
```

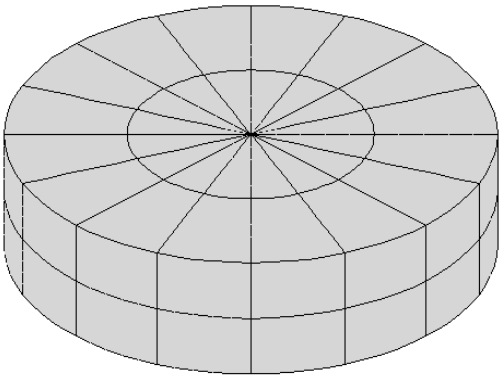
```
TEST11C.GDF: (lines 1-8 only):
TEST11 cylinder R=1 T=0.5 -- MultiSurf .ms2 input, nonuniform mapping
1. 9.80665  ULEN GRAV
1  1        ISX  ISY
2  2        NPATCH  IGDEF
3           NLINES
TEST11C.MS2    (name of .ms2 file)
*           default wetted surface (use all patches)
0  0  0     default settings: FAST, DivMult, outward normal

TEST11C.POT:
TEST11C.POT  Cylinder R=1, T=0.5, igdef=-1
 -1.
 1           1               IRAD, IDIFF
 2                            NPER (array PER follows)
8.971402 2.006403
 2                            NBETA (array BETA follows)
 0. 45.
 1                            NBODY
 test11c.gdf
 0. 0. 0. 0.                  XBODY
 1  1  1  1  1  1             IMODE(1-6)
 0                            NEWMDS

 TEST11C.SPL:
TEST11c.spl - cylinder R=1 T=0.5 -- analytic geometry (npatch=2)
4  2        NU NV (Patch 1, side   u azimuthal v vertical)
4  2        NU NV (Parch 2, bottom u azimuthal v radial)

TEST11C.MS2:
MultiSurf 1.23
// Truncated cylinder for base HIPAN example
// Full cosine spacing on side_surf using type-3 BLoftSurf (analytic)
// Half cosine spacing on bottom
Units: m kg
Symmetry:  x y
Extents:  -1.000 -1.000 -0.500 1.000 1.000 0.000
View:  -20.00 120.00 0
Places: 3
Layers: FFFFFFFF FFFFFFFF FFFFFFFF FFFFFFFF FFFFFFFF FFFFFFFF FFFFFFFF FFFFFFFF
DivMult: 1
BeginModel;
AbsPoint top 14 1 / 0.000 0.000 0.000 ;
EulerFrame F0 15 1 / top * 0.0000 0.0000 0.0000 ;
```

```
FrameAbsPt draft 15 1 / F0 0.000 0.000 -0.500 ;
FrameRelPt radius 15 1 / F0 draft 1.000 0.000 0.000 ;
Line axis 6 1 1x1 / * draft top ;
Helix chine 11 1 8x4 / * radius axis 0.000000 90.0000 ;
Plane2 top_plane 7 1 / top draft ;
ProjCurve top_edge 11 1 8x4 / * chine top_plane ;
ObjectList wetted_surfs /
    { side_surf bottom_surf } ;
BLoftSurf bottom_surf 6 3 8x4 4x4 1 / * 2
    { chine chine draft } ;
BLoftSurf side_surf 2 3 8x4 4x4 1 / * 3
    { top_edge top_edge chine chine } ;
EndModel;

TEST11C.FRC:
TEST11C.FRC  Cylinder R=1, T=0.5, igdef=-1
 1    1    1    1    3    1  1  2 2     IOPTN(1-9)
 0.000000                                    VCG
 1.000000      .0000000      .0000000
 .0000000      1.000000      .0000000
 .0000000      .0000000      1.000000        XPRDCT
 0                                           NBETAH
 2                                           NFIELD
1.5 0.  0.
1.5 0. -0.5                                  (end of file)

test11C.cfg:
 ILOWHI=1
 IALTPOT=2
 IRR=0
 ISOLVE=2
 KSPLIN=3
 IQUADO=3
 IQUADI=4
 IPERIO=1
 MONITR=0
 NUMHDR=1
 NOOUT= 1 1 1 1 0 1 1 1 1
 USERID_PATH=\WAMITv6   (directory for *.exe, *.dll, and userid.wam)
```

A.12 IRREGULAR-FREQUENCY REMOVAL – TEST12

TEST12 is the higher-order analog of TEST02, intended to illustrate the removal of irregular-frequency effects using the higher-order method. As in TEST11a, the geometry is defined analytically (IGDEF=-1) and the dimensions are input in the file TEST12.GDF. In this case NPATCH=3 is specified, where the additional patch corresponds to the interior free surface as required for the irregular-frequency option. The figures below illustrate the extra patch and panels.





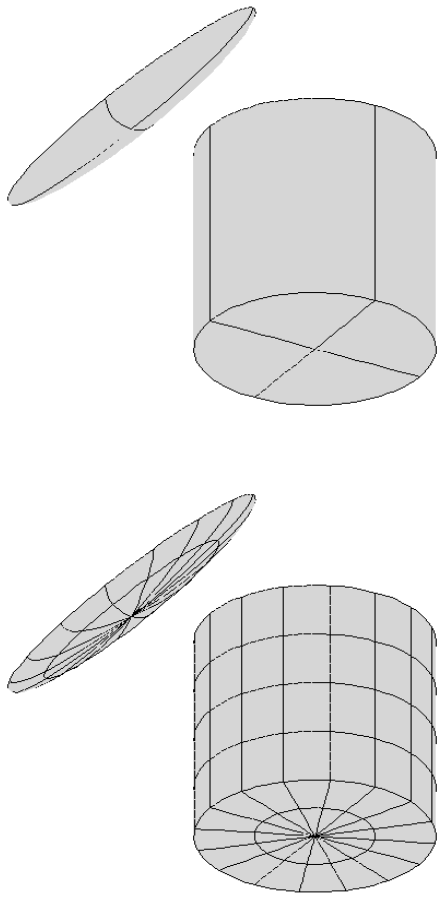
```
TEST12.GDF:
TEST12 cylinder R=1 T=0.5 -- analytic geometry (npatch=3)
1. 9.80665  ULEN GRAV
1  1        ISX  ISY
3  -1       NPATCH  IGDEF
2           NLINES
1.0 0.5     RADIUS, DRAFT
0           INONUMAP (uniform mapping)

TEST12.SPL:
TEST12.spl - cylinder R=1 T=0.5 -- analytic geometry (npatch=3)
4  2        NU NV (Patch 1, side   u azimuthal v vertical)
4  2        NU NV (Parch 2, bottom u azimuthal v radial)
4  2        NU NV (Parch 3, interior free surface)

TEST12.POT:
TEST12.POT  Cylinder R=1, T=0.5, igdef=-1, npatch=3 (IRR=1)
 -1.
 1           1               IRAD, IDIFF
 3                           NPER (array PER follows)
8.971402 2.006403 1.003033
 2                           NBETA (array BETA follows)
 0. 45.
 1                           NBODY
 test12.gdf
 0. 0. 0. 0.                 XBODY
 1  1  1  1  1  1            IMODE(1-6)
 0                           NEWMDS

TEST12.FRC:
TEST12.FRC  Cylinder R=1, T=0.5, igdef=-1 (irr=1)
 1    1    1    1    3    1  1  2 2     IOPTN(1-9)
 0.000000                                  VCG
 1.000000      .0000000      .0000000
 .0000000      1.000000      .0000000
 .0000000      .0000000      1.000000      XPRDCT
 0                                         NBETAH
 2                                         NFIELD
1.5 0.  0.
1.5 0. -0.5                                (end of file)

test12.cfg:
 ILOWHI=1
 IALTPOT=2
```

```
IRR=1
ILOG=1
ISOLVE=1
KSPLIN=3
IQUADO=3
IQUADI=4
IPERIO=1
MONITR=0
NUMHDR=1
NOOUT= 1 1 1 1 0 1 1 1 1
USERID_PATH=\WAMITv6   (directory for *.exe, *.dll, and userid.wam)
```

A.13 MULTIPLE BODIES – TEST13

This test uses the same cylinder and spheroid as in the low-order TEST05. IALTPOT=2 is used to specify the separate input files TEST13C.GDF and TEST13S.GDF. TEST13C uses IGDEF=-1 as in TEST11a. TEST13S uses the ELLIPSOID subroutine (IGDEF=-4) with the semi-axes (2.0, 0.25, 0.25) specified in TEST13S.GDF. The same separate FRC files TEST05C, TEST05S are used with IALTFRC=3. The vector IALTFRCN is included in TEST13.CFG to indicate that IALTFRC=1 in the separate FRC files for each body. (Normally it is necessary to duplicate the FRC files for analogous runs, as for example in TEST01 and TEST11, since the output filenames are assigned based on the FRC filename. This is not necessary for individual FRC files for each body when multiple bodies are analyzed, since these do not affect the output filenames.)

The option is used to evaluate the mean drift force and moment using a control surface, following the instructions in Chapter 14. The parameter ICTRSURF=1 is assigned in the CFG file. The control surfaces surrounding the cylinder and spheroid are defined by the input files TEST13c.csf and TEST13s.csf. These control surfaces are generated by the subroutines CIRCYL_CS and ELLIPSOID_CS in the GEOMXACT DLL library. The surfaces generated by these subroutines include the portion of the free surface between the body and outer control surface. The corresponding output for the mean drift force and moment is contained in the file TEST13.9c.



```
TEST13c.GDF:
TEST13C cylinder R=1 T=2 -- analytic geometry (npatch=2)
1. 9.80665  ULEN GRAV
1  1        ISX  ISY
2  -1       NPATCH  IGDEF
2           NLINES
1.0 2.0     RADIUS, DRAFT
0           INONUMAP (uniform mapping)

TEST13s.GDF:
TEST13S spheroid  a=2, b=c=0.25 -- igdef=-4
1. 9.80665  ULEN GRAV
1  1        ISX  ISY
1  -4       NPATCH  IGDEF
1           NLINES
2.0 0.25 0.25    A, B, C
```

```
TEST13C.SPL:
TEST13C cylinder R=1 T=2 -- analytic geometry (npatch=2)
 4  4        NU NV (side)
 4  2        NU NV (bottom)

TEST13S.SPL:
TEST13S spheroid A=2 B=C=.25 -- analytic geometry (npatch=1)
 4  2        NU NV

TEST13.POT:
TEST13.POT -- Cylinder + spheroid, ILOWHI=1
 -1.
 1           1               IRAD, IDIFF
 2                            NPER (array PER follows)
 1.5 2.0
 1                           NBETA (array BETA follows)
 0.
 2                            NBODY
 test13c.gdf
 1.25 0.0 0.0 0.0             XBODY
 1  1  1  1  1  1             IMODE(1-6)
 0                            NEWMDS
 test13s.gdf
 -0.5 0.0 0.0 90.0            XBODY
 1  1  1  1  1  1             IMODE(1-6)
 0                            NEWMDS

TEST13.FRC:
TEST13.FRC -- Cylinder + spheroid, ILOWHI=1
  1  1  1  1  0  1  1  1  1
  1.0
 test05c.frc
 test05s.frc
  0
  1
  0. 0. 0.

TEST05c.FRC:
CYL.FRC
 0    0    0    0    0    0    0    0    0
 0.000000
 1.000000      .0000000      .0000000
 .0000000      1.000000      .0000000
 .0000000      .0000000      1.000000
```

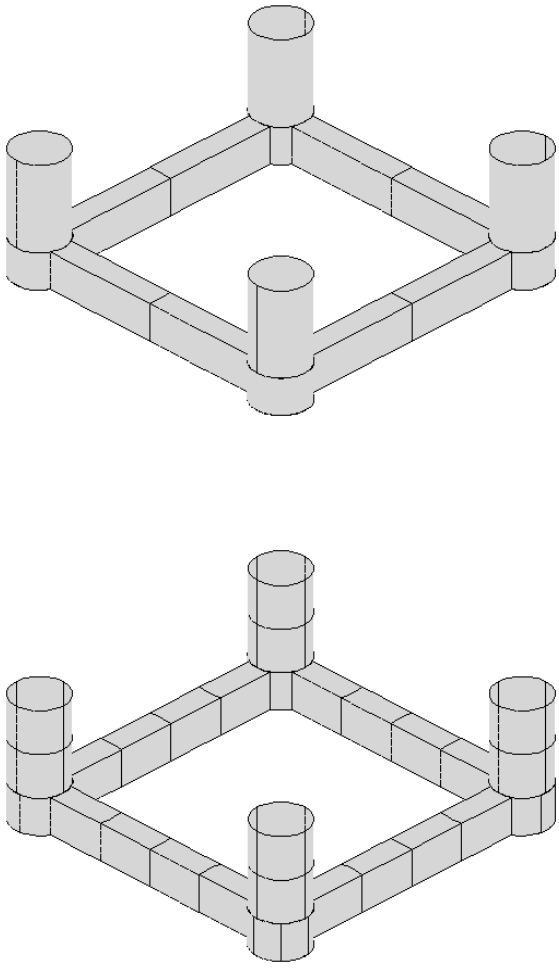
```
 0
 0

TEST05s.FRC:
SPD.FRC
 0    0    0    0    0    0    0    0    0
 0.000000
 1.000000      .0000000      .0000000
 .0000000      1.000000      .0000000
 .0000000      .0000000      1.000000
 0
 0

TEST13c.csf:
cylinder R=1.2 T=2.2 -- analytic CONTROL SURFACE (npatch=3)
1   ILOWHICSF
1  1        ISX  ISY
3 -1001  1. NPATCH  ICDEF  PSZCSF
2
1.2 2.2  1.0     RADIUS, DRAFT, Inner radius
0           UNIFORM MAPPING

TEST13s.csf:
ELLIPSOID CONTROL SURFACE defined by subroutine ELLIPSOID_CS
1   ILOWHICSF
1  1        ISX  ISY
2 -1003  1. NPATCH  IGDEF PSZCSF
2           NLINES
2.2 0.3 0.3    A, B, C  (semi-axes of outer control surface)
2.0 0.25               (semi-axes of body waterline)

test13.cfg:
 IPLTDAT=4
 maxscr=1024
 ILOWHI=1
 IALTPOT=2
 IRR=0
 ISOLVE=2
 KSPLIN=3
 IQUADO=3
 IQUADI=4
 IPERIO=1
 NUMHDR=1
 NOOUT=0 1 1 1 0 1 1 1 1
```

```
IALTFRC = 3    !  Alternative Form 3 FRC
IALTFRCN= 1 1
ICTRSURF=1     !  Evaluate control surface drift forces
USERID_PATH=\WAMITv6   (directory for *.exe, *.dll, and userid.wam)
```

A.14 ISSC TLP – TEST14

The subroutine TLP (IGDEF=-9) is used to generate the ISSC TLP with the dimensions specified in TEST14.GDF. Except for the geometry, the inputs correspond to the low-order test runs TEST06 and TEST07.

For TEST14A, TEST14A.CFG TEST14A.POT and TEST14A.FRC are used to output data to be used as input to F2T. TEST14.GDF and TEST14.SPL are used without change. TEST14A.POT has zero and infinite frequencies and 98 uniformly spaced frequencies. In CFG, IPERIO=2 is specified. In FRC, IOPTN(4) is set to output RAOs.



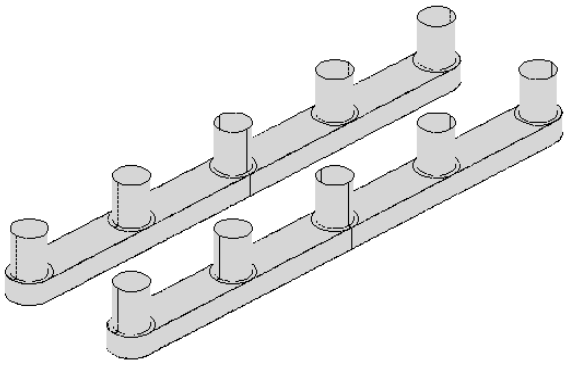
```
TEST14.GDF:
TEST14 -- ISSC TLP (ILOWHI=1)
43.125  9.80665          ULEN GRAV
1  1                     ISX  ISY
12 -9                    NPATCH, IGDEF
2                        NLINES
8.435  35.    43.125     RADIUS  DRAFT  HSPACE
7.5    10.5              WIDTH   HEIGHT

TEST14.SPL:
TEST14 -- ISSC TLP (ILOWHI=1)
1  2        NU NV patch 1
1  2        NU NV patch 2
1  2        NU NV patch 3
1  2        NU NV patch 4
1  2        NU NV patch 5
1  2        NU NV patch 6
1  2        NU NV patch 7
1  2        NU NV patch 8
4  1        NU NV patch 9
4  2        NU NV patch 10
4  1        NU NV patch 11
1  1        NU NV patch 12

TEST14.POT:
TEST14 -- ISSC TLP (ILOWHI=1)
 450
 0         0                 IRAD, IDIFF
 3                           NPER (array PER follows)
 5. 10. 15.
 1                           NBETA (array BETA follows)
 0.
 1                           NBODY
 test14.gdf
 0. 0. 0. 0.                 XBODY
 1  0  1  0  1  0            IMODE(1-6)
 0                           NEWMDS

TEST14.FRC:
TEST14 -- ISSC TLP (ILOWHI=1, IALTFRC=2)
 1  1  1 -2  0  0  0  0  0  IOPTN (IOPTN(4)<0 signifies fixed modes)
 6                          NDFR
 1 1 0 0 0 1                IMODE
 1.                         RHO
```

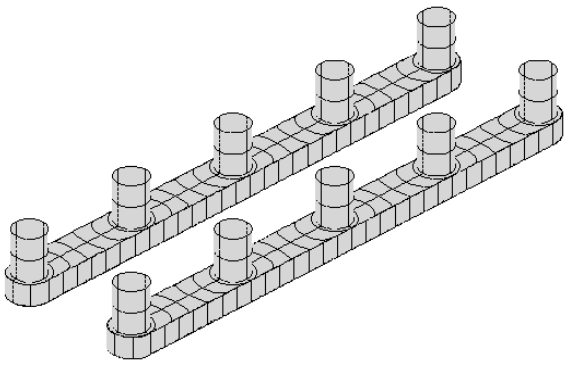
```
 0. 0. 3.0                  XCG
 1                          IMASS
    53066.4      0.        0.        0.     159199.2         0.
        0.   53066.4       0.  -159199.2         0.          0.
        0.       0.    53066.4       0.          0.          0.
        0. -159199.2       0.  8.0201552E7       0.          0.
   159199.2      0.        0.        0.   8.0201552E7        0.
        0.       0.        0.        0.          0.  9.54906731E7
 0                          IDAMP
 0                          ISTIFF
 0                          NBETAH
 0                          NFIELD

test14.cfg:
 MAXSCR=1024
 ILOWHI=1
 IALTPOT=2
 IRR=0
 ISOLVE=1
 IQUADI=4
 IQUADO=3
 KSPLIN=3
 NUMHDR=1
 IALTFRC=2
 USERID_PATH=\WAMITv6   (directory for *.exe, *.dll, and userid.wam)
```

```
TEST14A.POT:
TEST14A -- ISSC TLP (ILOWHI=1)
 450
 0         0                  IRAD, IDIFF
 -101                           NPER (array PER follows)
 -0.05 0.05
 1                              NBETA (array BETA follows)
 0.
 1                              NBODY
 test14.gdf
 0. 0. 0. 0.                    XBODY
 1  0  1  0  1  0              IMODE(1-6)
 0                              NEWMDS

TEST14A.FRC:
TEST14A -- ISSC TLP (ILOWHI=1, IALTFRC=2)
 1  1  1  1  0  0  0  0  0  IOPTN
 1.                          RHO
 0. 0. 3.0                   XCG
 1                           IMASS
    53066.4      0.        0.        0.     159199.2         0.
        0.   53066.4       0.  -159199.2         0.          0.
        0.       0.    53066.4       0.          0.          0.
        0. -159199.2       0.  8.0201552E7       0.          0.
   159199.2      0.        0.        0.   8.0201552E7        0.
        0.       0.        0.        0.          0.  9.54906731E7
 0                           IDAMP
 0                           ISTIFF
 0                           NBETAH
 0                           NFIELD

TEST14A.CFG:
 MAXSCR=1024
 ILOWHI=1
 IALTPOT=2
 IRR=0
 ISOLVE=1
 IQUADI=4
 IQUADO=3
 KSPLIN=3
 NUMHDR=1
 IALTFRC=2
 IPERIO=2
 USERID_PATH=\WAMITv6   (directory for *.exe, *.dll, and userid.wam)
```

A.15 SEMI-SUB – TEST15

The subroutine SEMISUB (IGDEF=-10) is used to generate a Semi-submersible with the dimensions specified in TEST15.GDF. There are five columns on each pontoon, as shown in the figures below. For this structure a total of 10 patches are required. If NPATCH=11, extensions of the pontoons can be included as explained in the subroutine comments.





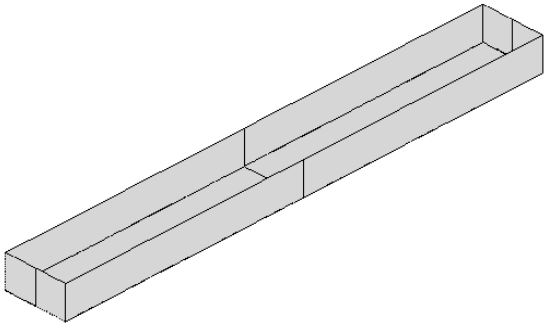
```
TEST15.GDF:
TEST15  Semi-sub, NCOL=5, IGDEF=-10
1.  9.80665  ULEN GRAV
1   1        ISX  ISY
10 -10       NPATCH IGDEF
2            NLINES
260. 20. 40. -30. -20.            XL, Y1, Y2, Z1, Z2
60. 8. 5                          DCOL, RCOL, NCOL

TEST15.SPL:
TEST15  Semi-sub, NCOL=5, IGDEF=-10
9  2        NU NV                 (patch 10+32 - pontoon bottom)
32 1        NU NV                 (patch 9+32 - pontoon side)
2  2        NU NV                 (patch 1+32 - column 3)
2  1        NU NV                 (patch 2+32 - annulus 3)
5  2        NU NV                 (patch 3+32 - between annulus 3&4)
4  2        NU NV                 (patch 4+32 - column 4)
4  1        NU NV                 (patch 5+32 - annulus 4)
5  2        NU NV                 (patch 6+32 - between annulus 4&5)
4  2        NU NV                 (patch 7+32 - column 5)
4  1        NU NV                 (patch 8+32 - annulus 5)

TEST15.POT:
TEST15  Semi-sub with five columns on each pontoon
 -1.
  0           0               IRAD, IDIFF
 1                           NPER (array PER follows)
 18.0
 1                           NBETA (array BETA follows)
 180.
 1                            NBODY
 test15.gdf
 0. 0. 0. 0.                  XBODY
 0  0  1  0  1  0             IMODE(1-6)
 0                           NEWMDS

TEST15.FRC:
TEST15  Semi-sub with five columns on each pontoon
            1    1    1   1     0     0     0     0     0
       0.0000
       20.0      0.0        0.0
       0.        60.0        0.0
       0         0.         60.0
       0
```

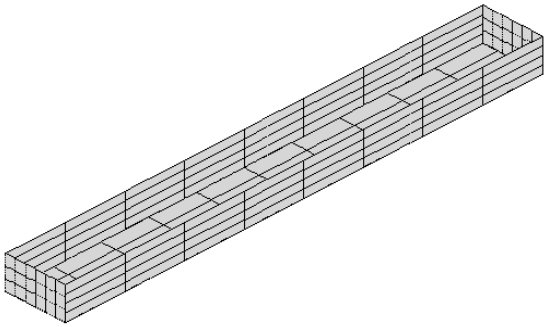
```
       0

test15.cfg:
 ISOLVE=1
 IPERIO=1
 NUMHDR=1
 KSPLIN=3
 IQUADI=4
 IQUADO=3
 ILOWHI=1
 IALTPOT=2
 IRR=0
 USERID_PATH=\WAMITv6   (directory for *.exe, *.dll, and userid.wam)
```

A.16 BARGE WITH BENDING MODES – TEST16

The test runs TEST16 and TEST16a analyze the structural response of a rectangular barge with total length 80m, beam 10m, and draft 5m. Eight free-free beam modes are included to analyze the elastic deformation of the barge. These mode shapes are defined in the subroutines NEWMODES and DEFINE, as explained in Chapter 8. The response amplitude of each mode is included in the RAO's.

In TEST16 the subroutine BARGE (IGDEF=-5) is used. The half-length, half-beam and draft are specified in TEST16.GDF. In TEST16a the option IGDEF=0 is used, with the vertices of the patches specified in TEST16a.GDF in the same format as for low-order panels.





```
TEST16.GDF:
TEST16 elastic barge
1. 9.80665  ULEN GRAV
1  1        ISX  ISY
3  -5       NPATCH  IGDEF
1           NLINES
40.0   5.0  5.0         half-length, half-beam, draft

TEST16.SPL:
TEST16 elastic barge
3  3        NU NV   (end)
4  4        KU KV
5  2                (side)
4  4
5  2                (bottom)
4  4
            IQUO IQVO are not specified IQUADO=3 in config.wam
            IQUI IQVI are not specified IQUADI=4 in config.wam

TEST16.POT:
TEST16 elastic barge with 8 beam modes
 -1.
 0           0               IRAD, IDIFF
 2                            NPER (array PER follows)
 7. 8.
 1                             NBETA (array BETA follows)
 180.
 1                            NBODY
test16.gdf
 0. 0. 0. 0.                  XBODY
 1  0  1  0  1  0             IMODE(1-6)
 8                            NEWMDS

TEST16.FRC:
TEST16 elastic barge with 8 beam modes
 1  1  1  1  0  0  0  0  0
 1000.
 0. 0. 0.
 1
  4.00000E+06 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0.
  0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0.
  0. 0. 4.E+06 0. 0. 0. 0. 0. 0. 0. 0. 0. 0.
  0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0.
  0. 0. 0. 0. 2.13333E+09 0. 0. 0. 0. 0. 0. 0. 0.
```

```
  0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0.
  0. 0. 0. 0. 0. 0. 1.E+06 0. 0. 0. 0. 0. 0. 0.
  0. 0. 0. 0. 0. 0. 0. 1.E+06 0. 0. 0. 0. 0. 0.
  0. 0. 0. 0. 0. 0. 0. 0. 1.E+06 0. 0. 0. 0. 0.
  0. 0. 0. 0. 0. 0. 0. 0. 0. 1.E+06 0. 0. 0. 0.
  0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 1.E+06 0. 0. 0.
  0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 1.E+06 0. 0.
  0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 1.E+06 0.
  0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 1.E+06
 0
 1
  0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0.
  0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0.
  0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0.
  0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0.
  0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0.
  0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0.
  0. 0. 0. 0. 0. 0. 6.25705E+06 0. 0. 0. 0. 0. 0. 0.
  0. 0. 0. 0. 0. 0. 0. 4.75441E+07 0. 0. 0. 0. 0. 0.
  0. 0. 0. 0. 0. 0. 0. 0. 1.82720E+08 0. 0. 0. 0. 0.
  0. 0. 0. 0. 0. 0. 0. 0. 0. 4.99297E+08 0. 0. 0. 0.
  0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 1.11419E+09 0. 0. 0.
  0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 2.17352E+09 0. 0.
  0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 3.85260E+09 0.
  0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 6.35602E+09
0
0

test16.cfg:
 ILOWHI=1
 IALTPOT=2
 IALTFRC=2
 IRR=0
 ISOLVE=1
 IQUADI=5
 IQUADO=4
 IPERIO=1
 MONITR=0
 NUMHDR=1
 ILOG=0
 IGENMDS=16
 USERID_PATH=\WAMITv6   (directory for *.exe, *.dll, and userid.wam)
```

```
TEST16A.GDF: (lines 1-8 only):
TEST16a elastic barge with 8 beam modes - igdef=0 (3 flat panels)
     1.0000        9.80665
           1           1
           3           0          NPATCH, IGDEF
    40.0000       0.000000       -5.00000
    40.0000        5.00000       -5.00000
    40.0000        5.00000       0.000000
    40.0000       0.000000       0.000000   (end)

TEST16A.SPL:
TEST16A elastic barge with igdef=0 (patches defined by flat panels)
3  3        NU NV       end
4  4        KU KV
5  2                    side
4  4
5  2                    bottom
4  4

TEST16A.POT:
TEST16a elastic barge with 8 beam modes - igdef=0
 -1.
 0           0              IRAD, IDIFF
 2                           NPER (array PER follows)
 7. 8.
 1                            NBETA (array BETA follows)
 180.
 1                           NBODY
test16a.gdf
 0. 0. 0. 0.                 XBODY
 1  0  1  0  1  0            IMODE(1-6)
 8                           NEWMDS

TEST16A.FRC:
TEST16a elastic barge with 8 beam modes (igdef=0)
 1  1  1  1  0  0  0  0  0
 1000.
 0. 0. 0.
 1
  4.00000E+06 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0.
  0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0.
  0. 0. 4.E+06 0. 0. 0. 0. 0. 0. 0. 0. 0. 0.
  0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0.
  0. 0. 0. 0. 2.13333E+09 0. 0. 0. 0. 0. 0. 0. 0.
```

```
  0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0.
  0. 0. 0. 0. 0. 0. 1.E+06 0. 0. 0. 0. 0. 0. 0.
  0. 0. 0. 0. 0. 0. 0. 1.E+06 0. 0. 0. 0. 0. 0.
  0. 0. 0. 0. 0. 0. 0. 0. 1.E+06 0. 0. 0. 0. 0.
  0. 0. 0. 0. 0. 0. 0. 0. 0. 1.E+06 0. 0. 0. 0.
  0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 1.E+06 0. 0. 0.
  0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 1.E+06 0. 0.
  0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 1.E+06 0.
  0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 1.E+06
 0
 1
  0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0.
  0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0.
  0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0.
  0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0.
  0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0.
  0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0.
  0. 0. 0. 0. 0. 0. 6.25705E+06 0. 0. 0. 0. 0. 0. 0.
  0. 0. 0. 0. 0. 0. 0. 4.75441E+07 0. 0. 0. 0. 0. 0.
  0. 0. 0. 0. 0. 0. 0. 0. 1.82720E+08 0. 0. 0. 0. 0.
  0. 0. 0. 0. 0. 0. 0. 0. 0. 4.99297E+08 0. 0. 0. 0.
  0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 1.11419E+09 0. 0. 0.
  0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 2.17352E+09 0. 0.
  0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 3.85260E+09 0.
  0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 6.35602E+09
0
0

test16A.cfg:
 ILOWHI=1
 IALTPOT=2
 IALTFRC=2
 IRR=0
 ISOLVE=1
 IQUADI=5
 IQUADO=4
 IPERIO=1
 MONITR=0
 NUMHDR=1
 ILOG=0
 IGENMDS=16
 USERID_PATH=\WAMITv6   (directory for *.exe, *.dll, and userid.wam)
```

A.17 CYLINDER WITH MOONPOOL – TEST17

This test run illustrates two alternative methods for analyzing bodies with moonpools. The geometry used is the circular cylinder with a concentric fluid chamber, as shown in the figure. The inner chamber of fluid, referred to as a 'moonpool', is open at the bottom of the cylinder to the external fluid domain. The top of the moonpool is a free surface with atmospheric pressure. One of the practical aspects of this problem is the existence of highly tuned resonant frequencies of the motion at the moonpool free surface. If the draft is comparable or large compared to the horizontal dimensions of the moonpool, the principal resonance is a 'pumping mode' which occurs when KT, the product of the wavenumber K and draft T is slightly less than one. Additional resonances occur in 'sloshing modes' at higher frequencies, corresponding approximately to standing waves inside the moonpool.

A cylinder with draft 1m is used, with the outer radius RADIUS=0.5m and the inner radius RADMP=0.25m. The geometry in all cases is represented analytically by the subroutine CYLMP (IGDEF=-7). To clarify the behavior near resonance, the wavenumber K is input in the POT file with the corresponding option IPERIO=3 specified in the TEST17.cfg file. Seven values of K are input in the range $0.7 \leq K \leq 1.0$ to focus on the regime including the pumping mode. The computed hydrodynamic parameters include the force coefficients, RAO's, and the elevation of the free surface at the center of the moonpool.

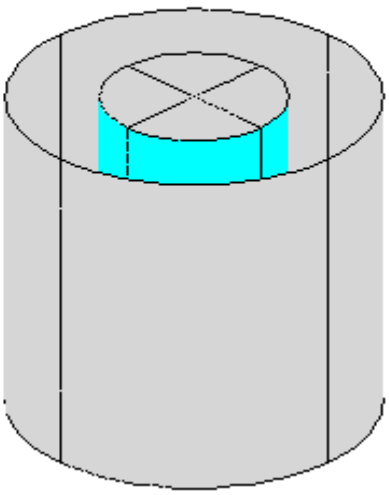
In TEST17 three patches are used to represent the outer surface r=RADIUS, the annular bottom z=-DRAFT, and the inner vertical surface r=RADMP. The free surface inside the moonpool is part of the physical free surface, and the appropriate free-surface boundary condition is satisfied automatically by the Green function as described in Chapter 12. The outputs display singular features characteristic of the resonant pumping mode. This includes large amplitudes of the heave damping and exciting force, and negative added mass in the vicinity of $KT = 0.85$. The heave RAO exceeds 8, and the free-surface elevation in the moonpool exceeds 6, relative to the incident wave amplitude. These very large responses are non-physical, and their existence in the computations can be attributed primarily to the neglect of viscous damping associated with flow separation at the outer and inner corners of the cylinder. This damping is only important when the vertical motions of the cylinder and/or moonpool are large. Experiments suggest that typical resonant amplitudes are on the order of 2 or 3 times the incident wave amplitude.

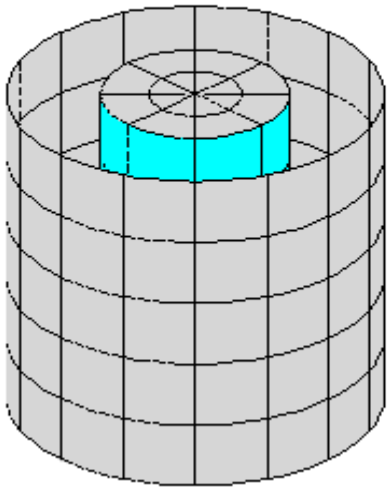
In order to damp the moonpool response and heave motions separately, a different physical problem is considered where a 'lid' is placed on the free surface of the moonpool. This lid is considered to be an extension of the body surface, and represented by an additional patch. Thus NPATCH=4 is assigned in TEST17a.GDF (and used also in TEST17b) and the subroutine CYLMP assigns the patch number 4 to be the circular disc of radius RADMP in the plane Z=0. However allowance must be made for the motions of the actual free surface relative to the body. This is done by defining appropriate generalized modes, which are nonzero only on patch 4. The most important mode is a vertical translation, assigned here in the subroutine file NEWMODES.F with the index $j = 7$. A more complete expansion can be introduced, but at the wavenumbers considered here and

for head-sea incidence angle it suffices to consider only a pitch rotation of the lid ($j = 8$). These two generalized modes, physically analogous to pitch and heave but defined relative to the body, are introduced via the subroutine MOONPOOL_FS in NEWMODES.F.

In test run TEST17a, the lid is assumed to be free with no external force or moment acting on it. The IALTFRC=2 option is employed, and the only external force matrix that is included in TEST17.FRC is the mass matrix of the body. This mass matrix is equivalent to the radii of gyration specified in TEST17.FRC. It can be confirmed by comparison of the outputs that the motions of the body (RAO) are virtually identical to TEST17, confirming that the representation of the moonpool free surface in this manner is legitimate. A comparison can also be made between the moonpool free surface elevation (numeric output file TEST17.6) and the response of the lid in mode 7 (RAO(7) in the numeric output file TEST17a.4), but in this comparison account must be made for the fact that RAO(7) is relative to the body motions, and thus it is necessary to compare the (complex) sum RAO(3)+RAO(7) in TEST17a with the moonpool free surface elevation in TEST17.

Finally, in TEST17b, empirical damping is introduced via the external damping matrix in TEST17b.FRC. Since this is the only difference between TEST17a and TEST17b, it is not necessary to re-run POTEN and the same TEST17a.P2F file is used for TEST17b. Thus TEST17a.pot is specified in FNAMES.17b, and IPOTEN=0 in the TEST17b.cfg file. The only nonzero elements of the external damping matrix are for heave ($j = 3$) and the lid vertical motion ($j = 7$). With these empirical damping coefficients added, more appropriate RAO's are obtained. This general approach can be refined based on experimental data. Experience with similar problems suggests that relatively crude estimates based on the observed response at resonance are sufficient to correct the response over a broad range of wave periods.





```
TEST17.GDF:
TEST17 cylinder with moonpool
1. 9.80665  ULEN GRAV
1  1        ISX  ISY
3  -7       NPATCH  IGDEF
1           NLINES
0.5 1.0 0.25     radius, draft, moonpool radius

TEST17.POT:
TEST17 cylinder with moonpool, NPATCH=3
 -1.
 0           0               IRAD, IDIFF
 7                            NPER (array PER follows)
 .7 .75 .8 .85 .9 .95 1.0
 1                             NBETA (array BETA follows)
 180.
 1                            NBODY
test17.gdf
 0. 0. 0. 0.                  XBODY
 1  0  1  0  1  0             IMODE(1-6)
 0                            NEWMDS

TEST17.FRC:
TEST17.FRC  Cylinder with moonpool
 1    1    1    1    0    1  0  0 0     IOPTN(1-9)
 0.000000                                 VCG
 1.000000      .0000000      .0000000
 .0000000      1.000000      .0000000
 .0000000      .0000000      1.000000     XPRDCT
 0                                        NBETAH
 1                                        NFIELD
0.0 0.0 0.0

test17.cfg:
 ILOWHI=1
 IALTPOT=2
 IALTFRC=1
 IRR=0
 ISOLVE=1
 PANEL_SIZE = 0.2       (use default .spl parameters)
 IPERIO = 3             (input wavenumber)
 NUMHDR=1
 USERID_PATH=\WAMITv6   (directory for *.exe, *.dll, and userid.wam)
```

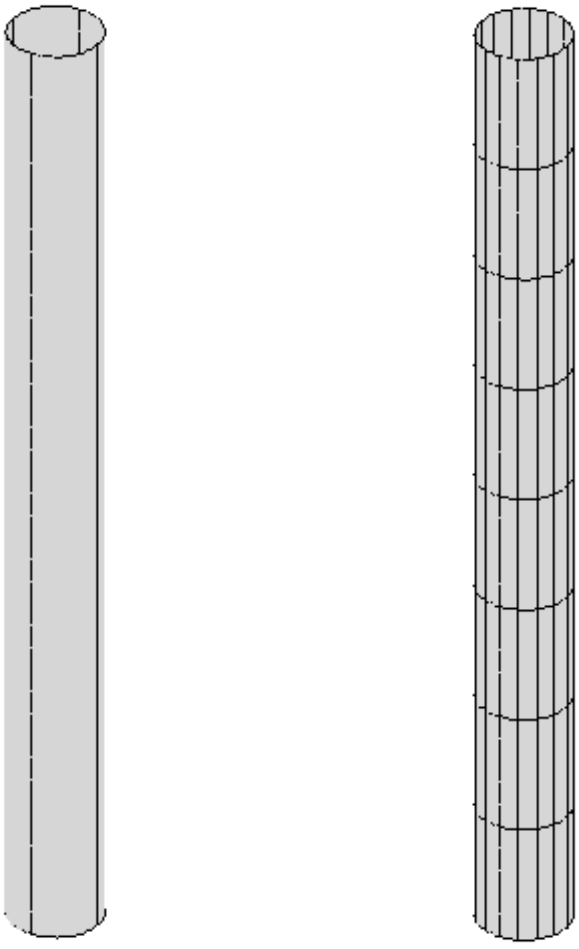
```
TEST17A.GDF: (lines 1-8 only):
TEST17a cylinder with moonpool -- undamped patch on free surface
1. 9.80665  ULEN GRAV
1  1        ISX  ISY
4  -7       NPATCH  IGDEF
1           NLINES
0.5 1.0 0.25     radius, draft, moonpool radius

TEST17A.POT:
TEST17 cylinder with moonpool, NPATCH=3
 -1.
 0           0               IRAD, IDIFF
 7                            NPER (array PER follows)
 .7 .75 .8 .85 .9 .95 1.0
 1                             NBETA (array BETA follows)
 180.
 1                            NBODY
test17a.gdf
 0. 0. 0. 0.                  XBODY
 1  0  1  0  1  0             IMODE(1-6)
 2                            NEWMDS

TEST17A.FRC:
TEST17a moonpool with generalized modes for free surface - no damping
 1  1  1  1  0  0  0  0  0
 1.
 0. 0. 0.
    1  imass (mass matrix of body)
    0.589049 0.0      0.0      0.0      0.0      0.0      0.0      0.0
    0.0      0.589049 0.0      0.0      0.0      0.0      0.0      0.0
    0.0      0.0      0.589049 0.0      0.0      0.0      0.0      0.0
    0.0      0.0      0.0      0.589    0.0      0.0      0.0      0.0
    0.0      0.0      0.0      0.0      0.589    0.0      0.0      0.0
    0.0      0.0      0.0      0.0      0.0      0.589    0.0      0.0
    0.0      0.0      0.0      0.0      0.0      0.0      0.0      0.0
    0.0      0.0      0.0      0.0      0.0      0.0      0.0      0.0
    0  idamp
    0  istif
 0
 0

test17A.cfg:
 ILOWHI=1
 IALTPOT=2
```

```
IALTFRC=2
ISOLVE=1
PANEL_SIZE = 0.2       (use default .spl parameters)
IPERIO = 3             (input wavenumber)
IRR=0
ILOG=1
NUMHDR=1
IGENMDS=17
USERID_PATH=\WAMITv6   (directory for *.exe, *.dll, and userid.wam)
```

```
TEST17B.FRC:
TEST17b cylinder+moonpool, generalized modes, damping b33=.4, b77=.1
 1  1  1  1  0  0  0  0  0
 1.
 0. 0. 0.
    1  imass (mass matrix of body)
    0.589049 0.0      0.0      0.0      0.0      0.0      0.0      0.0
    0.0      0.589049 0.0      0.0      0.0      0.0      0.0      0.0
    0.0      0.0      0.589049 0.0      0.0      0.0      0.0      0.0
    0.0      0.0      0.0      0.589    0.0      0.0      0.0      0.0
    0.0      0.0      0.0      0.0      0.589    0.0      0.0      0.0
    0.0      0.0      0.0      0.0      0.0      0.589    0.0      0.0
    0.0      0.0      0.0      0.0      0.0      0.0      0.0      0.0
    0.0      0.0      0.0      0.0      0.0      0.0      0.0      0.0
    1  idamp
    0.0      0.0      0.0      0.0      0.0      0.0      0.0      0.0
    0.0      0.0      0.0      0.0      0.0      0.0      0.0      0.0
    0.0      0.0      0.4      0.0      0.0      0.0      0.0      0.0
    0.0      0.0      0.0      0.0      0.0      0.0      0.0      0.0
    0.0      0.0      0.0      0.0      0.0      0.0      0.0      0.0
    0.0      0.0      0.0      0.0      0.0      0.0      0.0      0.0
    0.0      0.0      0.0      0.0      0.0      0.0      0.1      0.0
    0.0      0.0      0.0      0.0      0.0      0.0      0.0      0.0
    0  istif
 0
 0

test17B.cfg:
 ILOWHI=1
 IPOTEN=0               (skip POTEN subprogram, use TEST17a.p2f)
 IALTPOT=2
 IALTFRC=2
 ISOLVE=1
 PANEL_SIZE = 0.2       (use default .spl parameters)
 IPERIO = 3             (input wavenumber)
 IRR=0
 ILOG=1
 NUMHDR=1
 IGENMDS=17
 USERID_PATH=\WAMITv6   (directory for *.exe, *.dll, and userid.wam)
```

A.18 ELASTIC COLUMN – TEST18

The same inputs are used as in the low-order test run TEST08, except for the GDF file. The circular column is represented by the subroutine CIRCCYL (IGDEF=-1). Since the cylinder is bottom-mounted, NPATCH=1 and the patch on the bottom of the cylinder is omitted. The draft is set equal to the fluid depth. IGENMDS=18 is assigned in CFG file, and in NEWMODES this results in a call to subroutine DEFINE for the four shifted Jacobi polynomials.

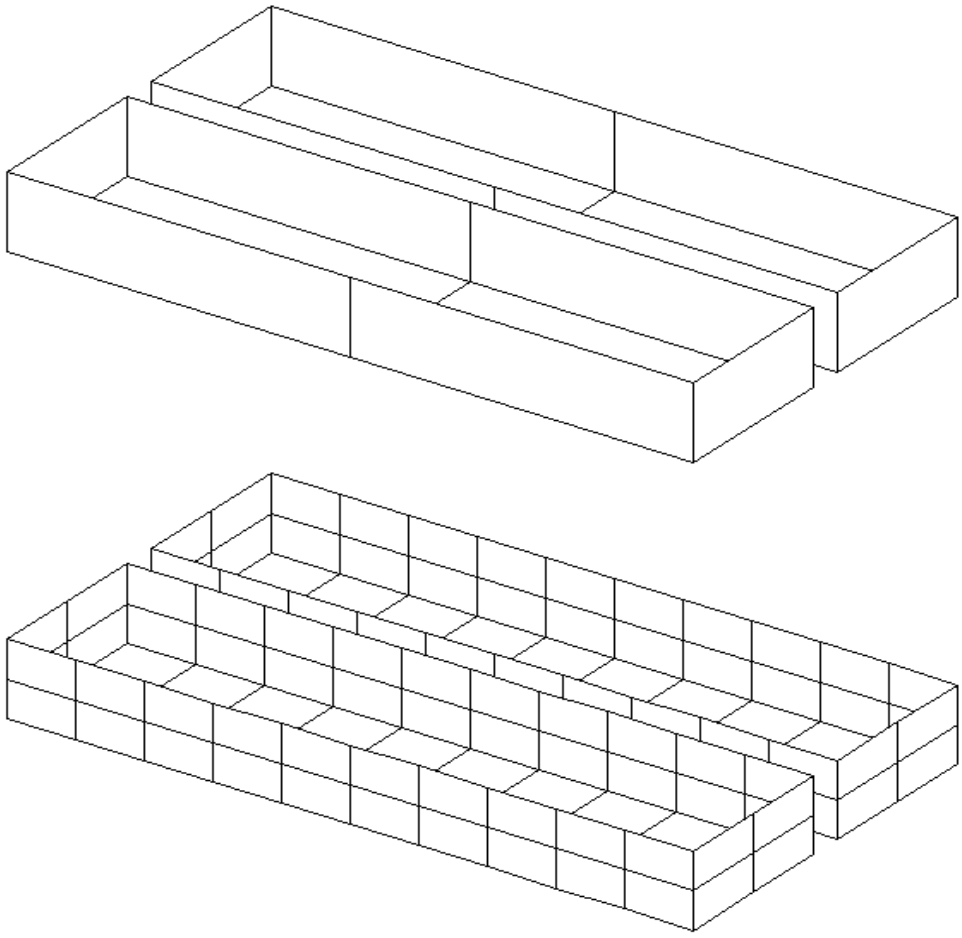


```
TEST18.GDF:
TEST18.GDF vertical cylinder, bottom mounted
    1.0000  9.80665     ulen, grav
    1   1               isx,isy
    1  -1               npatch, igdef
    2                   nlines
    10.0000  200.000    radius, draft
    0                   INONUMAP (uniform mapping)

TEST18.SPL:
TEST18.spl - bottom-mounted cylinder R=10 T=200 -- (npatch=1)
4  8        NU NV (Patch 1, side   u azimuthal v vertical)

TEST18.POT:
TEST18.POT -- bending of vertical column at resonance, 200m depth
      200.0  0.0  0.0  0.0  0.0
           0  0  0
           0   0
       0   0   0   0   0   0
           1
           6.5
           1
           0.0

TEST18.FRC:
TEST08.FRC file, vertical column with 4 bending modes
         1    1    1    1    0    0    0    0   0
        1.0
        .0000000      .0000000      1.000000
    1
  0.   0.   0.   0.   0.   0.        0.        0.        0.        0.
  0.   0.   0.   0.   0.   0.        0.        0.        0.        0.
  0.   0.   0.   0.   0.   0.        0.        0.        0.        0.
  0.   0.   0.   0.   0.   0.        0.        0.        0.        0.
  0.   0.   0.   0.   0.   0.        0.        0.        0.        0.
  0.   0.   0.   0.   0.   0.        0.        0.        0.        0.
  0.   0.   0.   0.   0.   0.    69115.    62832.    62832.    62832.
  0.   0.   0.   0.   0.   0.    62832.    67320.    62832.    62832.
  0.   0.   0.   0.   0.   0.    62832.    62832.    66323.    62832.
  0.   0.   0.   0.   0.   0.    62832.    62832.    62832.    65688.
    0
    1
  0.   0.   0.   0.   0.   0.        0.        0.        0.         0.
  0.   0.   0.   0.   0.   0.        0.        0.        0.         0.
```

```
  0.   0.   0.   0.   0.   0.        0.        0.        0.         0.
  0.   0.   0.   0.   0.   0.        0.        0.        0.         0.
  0.   0.   0.   0.   0.   0.        0.        0.        0.         0.
  0.   0.   0.   0.   0.   0.        0.        0.        0.         0.
  0.   0.   0.   0.   0.   0.   103044.   412177.   824354.   1339575.
  0.   0.   0.   0.   0.   0.   412177.  4430902.  9789203.  16487078.
  0.   0.   0.   0.   0.   0.   824354.  9789203. 37899671.  64382041.
  0.   0.   0.   0.   0.   0.  1339575. 16487078. 64382041. 162406554.
    0
    0

test18.cfg:
 MAXSCR=1024
 ISOLVE=1
 NUMHDR=1
 NUMNAM=0
 ISOR=0
 IRR=0
 NEWMDS=4
 ILOWHI=1
 IALTPOT=1
 IALTFRC=2
 KSPLIN=3
 IQUADO=3
 IQUADI=4
 IGENMDS=18
 USERID_PATH=\WAMITv6   (directory for *.exe, *.dll, and userid.wam)
```

A.19 CATAMARAN BARGE – TEST19

The geometrical configuration is the same as the barge near a wall (TEST04). Since only head seas are considered, the hydrodynamic outputs correspond to TEST04 except for the different definition of the incident wave amplitude which applies for a body near a wall. In TEST19 IGDEF=0 is used, with four patches specified in the GDF file corresponding to one quadrant of the catamaran configuration. Since there are two hulls in this case, the forces acting on both hulls are two times the corresponding forces in TEST04, but since the incident wave amplitude in TEST04 is increased by a factor of two, the exciting force coefficients and RAO's are the same in both test runs, except for small differences in accuracy. Note that in TEST19 two planes of symmetry can be utilized, unlike TEST04 where reflection about the plane $x = 0$ is required by the program. The comparisons of cross-coupling coefficients and Haskind/Diffraction exciting forces implies that the results of TEST19 are more accurate, with less computational cost.



Generalized modes can be used to extend the analysis of this configuration to include two independent bodies. In this case each of the rigid-body modes of the catamaran must be supplemented by a corresponding generalized mode which has the same normal velocity on one barge, and the opposite phase on the other. The separate modes of each independent body are then evaluated by combining the corresponding symmetric and antisymmetric modes for the catamaran. It is simpler to use the option NBODY=2 for this purpose, but the number of unknowns is increased by a factor of four, resulting in a substantial increase of the run time. For the more efficient approach used in TEST19 it is necessary to represent the entire forward half of one barge, as shown in the patch figure. The subroutine BARGE (IGDEF=-5) is not suitable, since this only represents one quadrant of one barge. On the other hand, BARGE can be used in the alternative NBODY=2 approach.

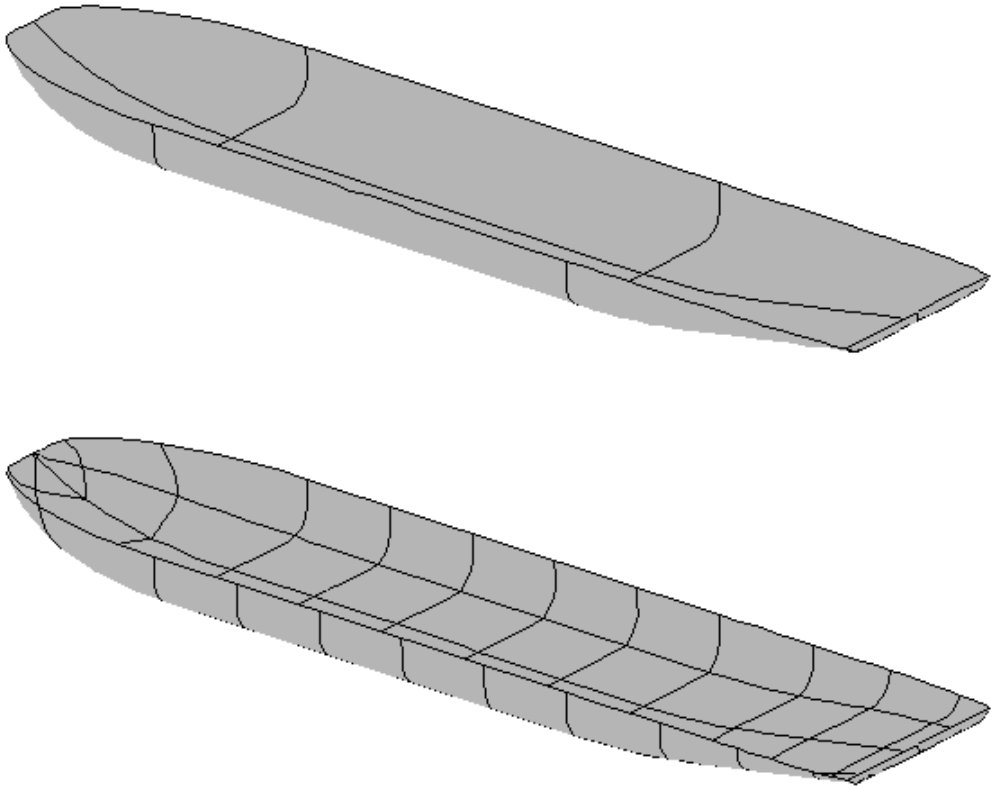
```
TEST19.GDF:
TEST19 one quadrant of catamaran barge configuration
40. 9.80665  ULEN GRAV
1  1        ISX  ISY
4  0        NPATCH  IGDEF
    40.0000       2.000000      -10.00000
    40.0000       22.00000      -10.00000
    40.0000       22.00000       0.000000
    40.0000       2.000000       0.000000   (end)
    40.0000       22.00000      -10.00000
   0.000000       22.00000      -10.00000
   0.000000       22.00000       0.000000
    40.0000       22.00000       0.000000   (outside)
    40.0000       2.000000      -10.00000
   0.000000       2.000000      -10.00000
   0.000000       22.00000      -10.00000
    40.0000       22.00000      -10.00000   (bottom)
    40.0000        2.00000       0.000000
   0.000000        2.00000       0.000000
   0.000000        2.00000      -10.00000
    40.0000        2.00000      -10.00000   (inside)

TEST19.SPL:
TEST19 catamaran barge
2  2        NU NV   (end)
4  4        KU KV
5  2                (outside)
4  4
5  2                (bottom)
4  4
5  2                (inside)
4  4
            IQUO IQVO are not specified IQUADO=3 in config.wam
            IQUI IQVI are not specified IQUADI=4 in config.wam

TEST19.POT:
TEST19.POT -- Catamaran barge, same geometry as TEST04
 -1. 0. 0. 0. 0.
 0  0   IQUAD,IDIAG must be read in IALTPOT, not used
 0  0   IRAD,IDIFF
 1  0  1  0  1  0
 3
 6.  7.  8.
 1
```

```
 0.0

TEST19.FRC:
TEST19.FRC -- Catamarn barge, ILOWHI=1 (same as TEST04.FRC)
 1    1    1    1    0    0    0    0    1
 3.0
 20.00000      0.000000      0.000000
 0.000000      5.000000      0.000000
 0.000000      0.000000      20.00000
 0
 0


test19.cfg:
 ILOWHI=1
 IALTPOT=1
 IALTFRC=1
 IRR=0
 ISOLVE=1
 IQUADI=5
 IQUADO=4
 IPERIO=1
 MONITR=0
 NUMHDR=1
 ILOG=0
 isor=0                  (must include since ialtpot=1)
 USERID_PATH=\WAMITv6    (directory for *.exe, *.dll, and userid.wam)
```

A.20 MULTISURF BARGE – TEST20

This example illustrates the use of a MultiSurf geometry representation with IGDEF=2. The barge has a length of 100m, beam 20m, and draft 4.8m with one plane of symmetry ($y = 0$). The origin of the body coordinate system is at the intersection of the baseline and midship section. Thus XBODY=(0.0, 0.0, -4.8, 0.0) is input in the POT file. The format of the GDF file is as explained in Section 6.7 and Appendix 2. Four patches are used on one side of the body to represent the forebody, parallel middlebody, afterbody, and transom. Reference 24 includes results for a multiple-body configuration including two barge hulls identical to this model.

The body pressure is evaluated at the points specified in the input file test20.bpi, as explained in Section 4.11. The parameter IPNLBPT=1 in the .cfg file is used to specify this option with the input points specified in the body coordinate system.

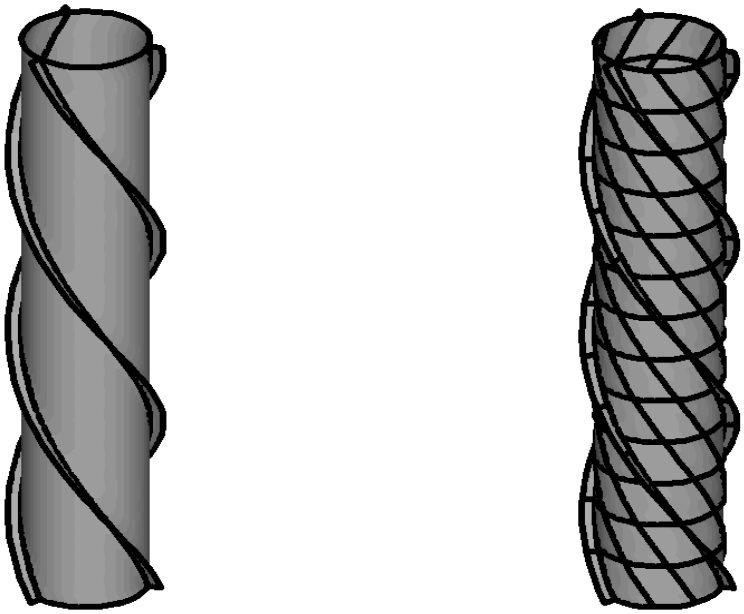


```
test20.pot:
single barge based on MultiSurf model (igdef=2)
 -1.
  1     1                      IRAD, IDIFF
   3                           NPER (array PER follows)
  6.0 9.0 12.0
 3                            NBETA (array BETA follows)
 180. 135. 90.
 1                            NBODY
  test20.gdf
 0. 0. -4.8  0.0              XBODY
 1  1  1  1  1  1             IMODE(1-6)
 0                            NEWMDS

test20.gdf:
Test run for barge modelled with MultiSurf
 1.000000  9.806600 ULEN, GRAV
  0  1  ISX, ISY
  4   2  NPATCH, IGDEF
  3  NLINES
test20.ms2
wetted_surfs
0  0  0 FAST,DivMult, outward normals

test20.ms2: (lines 1-8 only)
MultiSurf 1.23
// Barge model for WAMIT
// J. S. Letcher,  AeroHydro, Inc. 1/10/2002
Units: m MT
Symmetry:  y
Extents:  -50.000 -10.000 0.000 50.000 10.000 8.000
View:  -30.00 60.00 0
Places: 3

test20.frc:
test20.frc igdef=2
 1    1    1    1    1    0  0  1 1     IOPTN(1-9)
   0.0                                      VCG
 10.00000      .0000000      .0000000
 .0000000      25.00000      .0000000
 .0000000      .0000000      25.00000        XPRDCT
 0                                           NBETAH
 0                                           NFIELD
```

```
test20.bpi: (lines 1-8 only)
 bpi input file for test20, body pressure points for MultiSurf barge
         556
   -44.7760     0.0000     3.7926
   -42.9236     0.0000     2.9037
   -45.0675     0.2891     3.9523
   -43.5608     0.6874     3.1868
   -45.3541     0.5548     4.1152
   -44.1782     1.2902     3.4831

test20.cfg:
 MAXSCR=1024
 ILOWHI=1
 IALTFRC=1
 IALTPOT=2
 IRR=0
 ISOLVE=1
 NUMHDR=1
 USERID_PATH=\WAMITv6   (directory for *.exe, *.dll, and userid.wam)
 panel_size= 10.
 IPNLBPT=1
```

A.21 SPAR WITH THREE STRAKES – TEST21

The subroutine SPAR (IGDEF=-12) is used to generate the SPAR with three strakes with the dimensions specified in TEST21.GDF. Except for the geometry, the inputs correspond to the low-order test runs TEST09.



```
TEST21.GDF:
TEST21 SPAR2 with three strakes IGDEF=-12
18. 9.80665  ULEN GRAV
0  0        ISX  ISY
7  -12       NPATCH  IGDEF
npatch_dipole = 3
ipatch_dipole = 2 4 6
5
18. 200.              RADIUS, DRAFT
3.7 0. 1. 3           WIDTH, THICKNESS, TWIST, NSTRAKE
0                     IRRFRQ
0  0.                 IMOONPOOL, RADIUSMP
0                     IMPGEN


TEST21.POT:
TEST21.POT  SPAR with three strakes igdef=-12 (TEST21.GDF)
 -1.
 1           1               IRAD, IDIFF
 3                            NPER (array PER follows)
 0.1 0.5 1.
 2                            NBETA (array BETA follows)
 0. 120.
 1                            NBODY
 test21.gdf
 0. 0. 0. 0.                  XBODY
 1  1  1  1  1  1             IMODE(1-6)
 0                            NEWMDS


TEST21.FRC:
TEST21.FRC  SPAR with three strakes igdef=-12
 1 1 1 1 0 1 1 2 0    IOPTN(1-9)
 0.000000                                    VCG
 100.000000      .0000000      .0000000
 .0000000      100.000000      .0000000
 .0000000      .0000000      10.000000        XPRDCT
 0                                           NBETAH
 2                                           NFIELD
23. 0.  0.
15. 15. -0.5                                 (end of file)

TEST21.CFG:
 ipltdat=4
```

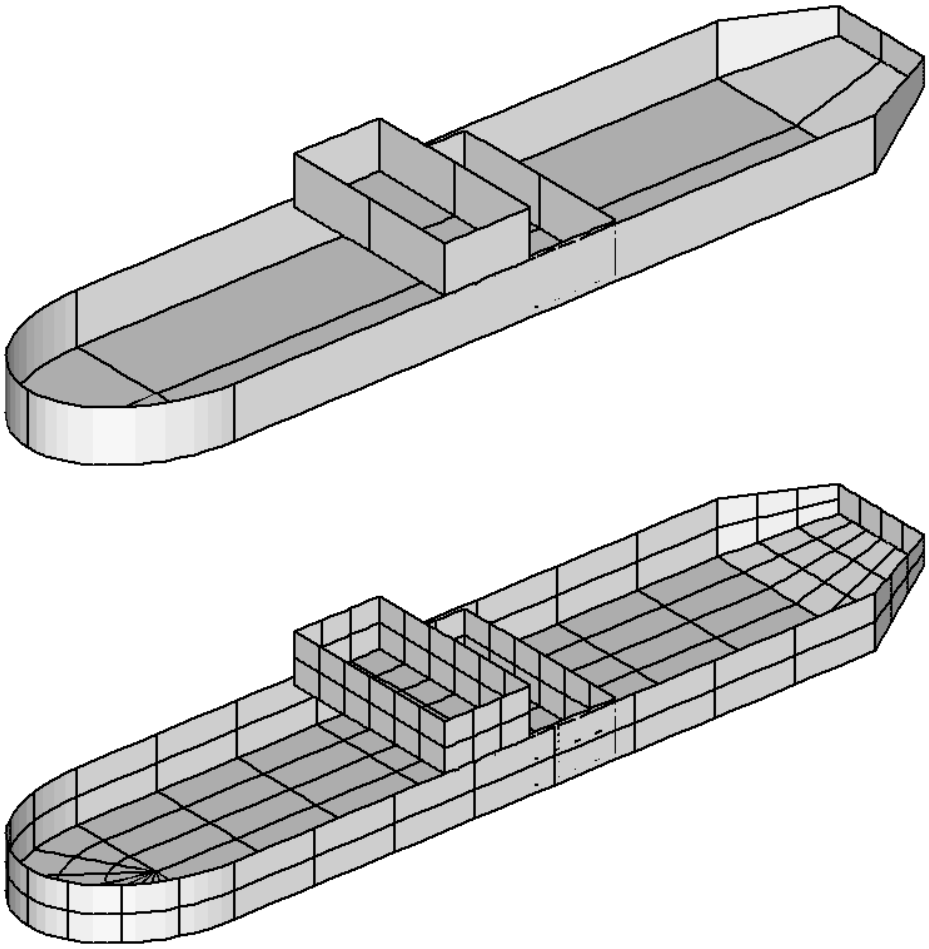
```
ilowgdf=4
ILOWHI=1
IALTPOT=2
IRR=0
ISOLVE=1
KSPLIN=3
IQUADO=3
IQUADI=4
IPERIO=3
MONITR=0
NUMHDR=1
NOOUT= 1 1 1 1 0 1 1 1 1
USERID_PATH=\WAMITv6   (directory for *.exe, *.dll, and userid.wam)
PANEL_SIZE=18
IPOTEN=1
ILOG=1
```

A.22 FPSO WITH TWO INTERNAL TANKS – TEST22

The subroutine FPSOINT (IGDEF=-21) is used to generate the FPSO with two internal tanks with the dimensions specified in TEST22.GDF. One plane of symmetry is specified, about $y = 0$. The tanks are rectangular, and the vertices of each patch are specified in TEST22.GDF. Both tanks have the same length (2m), breadth (4.2m), and depth (1.1m). The aft side of tank 1 and the forward side of tank 2 are in the same plane $x = 0.0$. The free surface of tank 1 is at $z = 1$, 1m above the plane of the exterior free surface. The free surface of tank 2 is at $z = 0.0$. The first and last patches of each tank are assigned by the parameter NPTANK. Both tanks contain fluid of relative density 1.0, as specified in TEST22.CFG. The parameter ITANKFPT=1 is used so that the field points can be assigned in each tank, on the last two lines of TEST22.FRC.

The option is used to evaluate the mean drift force and moment using a control surface, following the procedure described in Chapter 14. The parameter ICTRSURF=1 is assigned in the CFG file. The control surface surrounding the FPSO is a simple rectangular box, defined by the input file TEST22.csf. This control surface is generated using three rectangular patches (ICDEF=0) with the vertices specified in the CSF file, and with two planes of symmetry. (Note that the control surface is symmetric about the plane $x = 0$, although the body is not symmetric about this plane.) The parameter PSZCSF is negative, indicating that the subdivision of the control surface is determined by the parameters in the file TEST22.CSP.

Since the portion of the free surface between the body and outer control surface is not included, the results for the horizontal drift force and vertical moment are correctly evaluated but the vertical force and horizontal components of the moment are not correct. The corresponding output for the mean drift force and moment is contained in the file TEST13.9c. Comparison of the results for the mean drift force in the sway direction from the files TEST22.9 (direct pressure integration) and TEST22.9c (control surface), with the momentum drift force data in TEST22.8, confirms that the control surface gives a more accurate result compared to direct pressure integration for this body.



TEST22.GDF (complete file):

```
TEST22.GDF -- fpso with 2 tanks, one raised, joined at x=0
1. 9.80665  ULEN GRAV
0  1        ISX  ISY
 15 -21      NPATCH  IGDEF
36          NLINES  4+16*2
 3.   15.   2.     XBOW, XMID, XAFT
 2.2   1.2    HBEAM, HTRANSOM
 1.2  0.6   DRAFT, DTRANSOM
 0 2 0      INONUMAP, NTANKS, IRR
   2.000000      0.0000000E+00  1.0000000E+00
   2.000000       2.100000      1.0000000E+00
   2.000000       2.100000      -0.100000
   2.000000      0.0000000E+00  -0.100000
   2.000000       2.100000      1.0000000E+00
   0.000000E+00   2.100000      1.0000000E+00
```

```
  0.000000E+00   2.100000      -0.100000
  2.000000       2.100000      -0.100000
  0.000000E+00  0.0000000E+00  -0.100000
  2.000000      0.0000000E+00  -0.100000
  2.000000       2.100000      -0.100000
 -0.000000E+00   2.100000      -0.100000
 -0.000000      0.0000000E+00  -0.100000
 -0.000000       2.100000      -0.100000
 -0.000000       2.100000      1.0000000E+00
 -0.000000      0.0000000E+00  1.0000000E+00
  0.000000      0.0000000E+00  0.0000000E+00
  0.000000       2.100000      0.0000000E+00
  0.000000       2.100000      -1.100000
  0.000000      0.0000000E+00  -1.100000
  0.000000       2.100000      0.0000000E+00
 -2.000000E+00   2.100000      0.0000000E+00
 -2.000000E+00   2.100000      -1.100000
  0.000000       2.100000      -1.100000
 -2.000000E+00  0.0000000E+00  -1.100000
  0.000000      0.0000000E+00  -1.100000
  0.000000       2.100000      -1.100000
 -2.000000E+00   2.100000      -1.100000
 -2.000000      0.0000000E+00  -1.100000
 -2.000000       2.100000      -1.100000
 -2.000000       2.100000      0.0000000E+00
 -2.000000      0.0000000E+00  0.0000000E+00

TEST22.POT:
TEST22.POT  fpso with 2 interior tanks
 -1.0
   1         1               IRAD, IDIFF
     3
 2.0 2.5 3.0
 1                            NBETA (array BETA follows)
 90.
 1                            NBODY
 test22.gdf
 0. 0. 0. 0.                  XBODY
 1  1  1  1  1  1             IMODE(1-6)
 0                            NEWMDS
```

TEST22.SPL:

```
 TEST22.SPL  FPSO with two tanks
      4   3
      4   2
      8   3
      8   2
      2   1
      3   3
      3   2
      3   2
      3   2
      3   3
      3   2
      3   2
      3   2
      3   3
      3   2
```

TEST22.FRC:

```
TEST22.FRC fpso with 2 tanks, one field point on free surface in each tank
 1    1    1    1    0    1  1  1 1     IOPTN(1-9)
 0.000000                                   VCG
 1.000000      .0000000      .0000000
 .0000000      1.000000      .0000000
 .0000000      .0000000      1.000000      XPRDCT
 0                                        NBETAH
 2                                        NFIELD
1  1.0 1.0 1.0
2 -1.0 1.0 0.0
```

TEST22.CSF:

```
Test22 control surface box
1   ILOWHICSF
1  1
3  0  -2.
 12. 3.  0.   12. 3. -2.  0. 3. -2.  0. 3.  0.
 12. 0.  0.   12. 0. -2.   12. 3. -2.   12. 3.  0.
 12. 0. -2.  0. 0. -2.  0. 3. -2.   12. 3. -2.
```

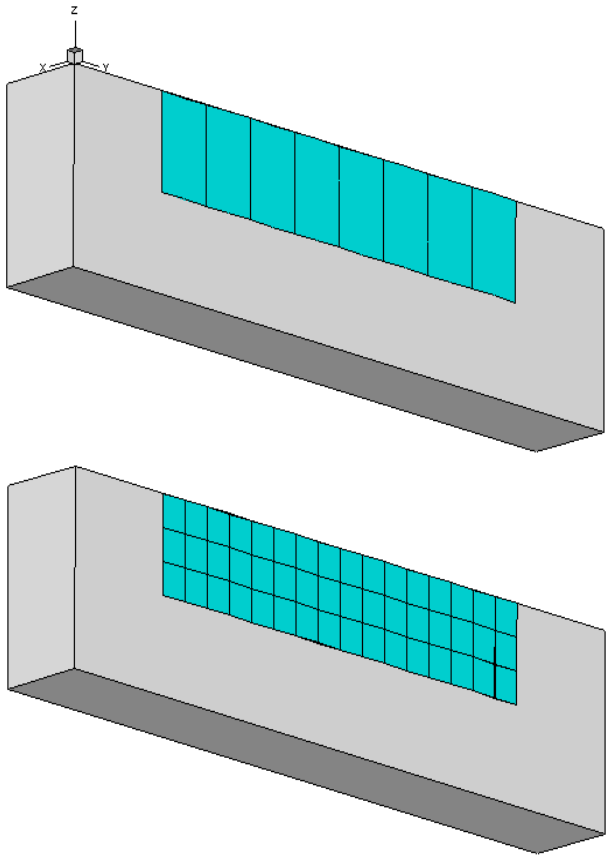
TEST22.CSP:

```
Test22.CSP control surface spline file
1 6
1 2
6 2
```

```
TEST22.CFG:
 ipltdat=4
 ILOWHI=1
 IALTPOT=2
 IRR=0
 ILOG=1
 ISOLVE=1
 KSPLIN=3
 IQUADO=3
 IQUADI=4
 MONITR=0
 NUMHDR=1
 NOOUT= 0 0 0 0 0 0 0 0 0
 USERID_PATH=\WAMITv6   (directory for *.exe, *.dll, and userid.wam)
 NPTANK=(8-11)  (12-15)
 RHOTANK= 1.0  1.0       (relative densities of tank fluids)
 ITANKFPT=1              (tank field points are in .frc file)
 ICTRSURF=1              Evaluate control surface drift forces
```

A.23 RADIATED WAVE FIELD FROM A BANK OF WAVEMAKERS – TEST23

The option ISOLVE=-1 is used to compute the radiated waves from a bank of 'paddle' wavemakers, following the procedure in Section 10.8. The wavemakers are in the plane $x = 0$ of a rectangular tank, as shown below. The tank has a reflecting wall at $y = 0$. The tank depth is 4m. Each wavemaker is represented by one rectangular patch, using IGDEF=0, with the vertices listed in TEST23.GDF. The motion of each wavemaker is rotational about its lower edge, at the same depth below the free surface, represented by a generalized mode with the same distribution of normal velocity and with symmetry prescribed about the walls $x = 0$ and $y = 0$. These generalized modes are defined in the subroutine WAVEMAKER, in the DLL file NEWMODES.F, designated by the parameter IGENMDS=21 in TEST23.CFG. This subroutine reads the depth of the lower edge of the wavemaker, ZHINGE=-2m, from the file WAVEMAKER_DEPTH.DAT. Wave elevations are evaluated at a square array of 64 field points defined in TEST23.FRC, using the uniform field point array option in Section 3.10.



```
test23.cfg:
 ILOWHI=1
 IALTPOT=2
 IALTFRC=2
 IRR=0
 ISOLVE=-1              (skip POTEN solutions for wavemakers in walls)
 MONITR=0
 NUMHDR=1
 USERID_PATH=\WAMITv6   (directory for *.exe, *.dll, and userid.wam)
 IGENMDS=21             (use NEWMODES subroutine WAVEMAKER)
 PANEL_SIZE=1.          (use default .spl parameters)
 INUMOPT6=1             (output separate radiation modes in .6 file)
 IFIELD_ARRAYS=1        (field points input in array format in .frc file)

test23.pot:
TEST23.POT -- 8 wavemaker segments in wall x=0
 4. fluid depth
 0          -1              IRAD, IDIFF
 2                          NPER (array PER follows)
 2. 4.
 1                          NBETA (array BETA follows)
 0.0
  1                          NBODY
test23.gdf
 0. 0. 0. 0.                XBODY
 0  0  0  0  0  0           IMODE(1-6)
 8                          NEWMDS

test23.gdf:
TEST23.GDF wavemaker, 8 segments in wall x=0 2<y<10m, ISY=1
 1. 9.80665  ULEN GRAV
 1  1        ISX  ISY
 8  0        NPATCH  IGDEF
     0.0000      2.000000      -2.00000
     0.0000       3.00000      -2.00000
     0.0000       3.00000       0.000000
     0.0000      2.000000       0.000000   (end of Patch 1)
     0.0000      3.000000      -2.00000
     0.0000       4.00000      -2.00000
     0.0000       4.00000       0.000000
     0.0000      3.000000       0.000000   (end of Patch 2)
     0.0000      4.000000      -2.00000
     0.0000       5.00000      -2.00000
```

```
     0.0000       5.00000       0.000000
     0.0000      4.000000       0.000000   (end of Patch 3)
     0.0000      5.000000      -2.00000
     0.0000       6.00000      -2.00000
     0.0000       6.00000       0.000000
     0.0000      5.000000       0.000000   (end of Patch 4)
     0.0000      6.000000      -2.00000
     0.0000       7.00000      -2.00000
     0.0000       7.00000       0.000000
     0.0000      6.000000       0.000000   (end of Patch 5)
     0.0000      7.000000      -2.00000
     0.0000       8.00000      -2.00000
     0.0000       8.00000       0.000000
     0.0000      7.000000       0.000000   (end of Patch 6)
     0.0000      8.000000      -2.00000
     0.0000       9.00000      -2.00000
     0.0000       9.00000       0.000000
     0.0000      8.000000       0.000000   (end of Patch 7)
     0.0000      9.000000      -2.00000
     0.0000       10.00000      -2.00000
     0.0000       10.00000       0.000000
     0.0000      9.000000       0.000000   (end of Patch 8)

WAVEMAKER_DEPTH.DAT for vertical coordinate of hinge of the wavemaker
-2.0    ZHINGE

test23.frc:
TEST23.FRC (field point wave elevations, IALTFRC=2, no external forces)
 0  0  0  0  0  1  0  0  0        (IOPTN(1-9)
 1.                               (RHO -- fluid density)
 0. 0. 0.                         (XCG)
 0                                (IMASS)
 0                                (IDAMP)
 0                                (ISTIF)
 0                                (NBETAH)
 0                                (NFIELD -- no individual field points)
 1                                (NFIELD_ARRAYS -- number of arrays)
 0                                (Array is in exterior fluid domain)
 8   2.5   1.0                    (NFX, X1, DELX)
 8   2.5   1.0                    (NFY, Y1, DELY)
 1   0.0   0.0                    (NFZ, Z1, DELZ)
```

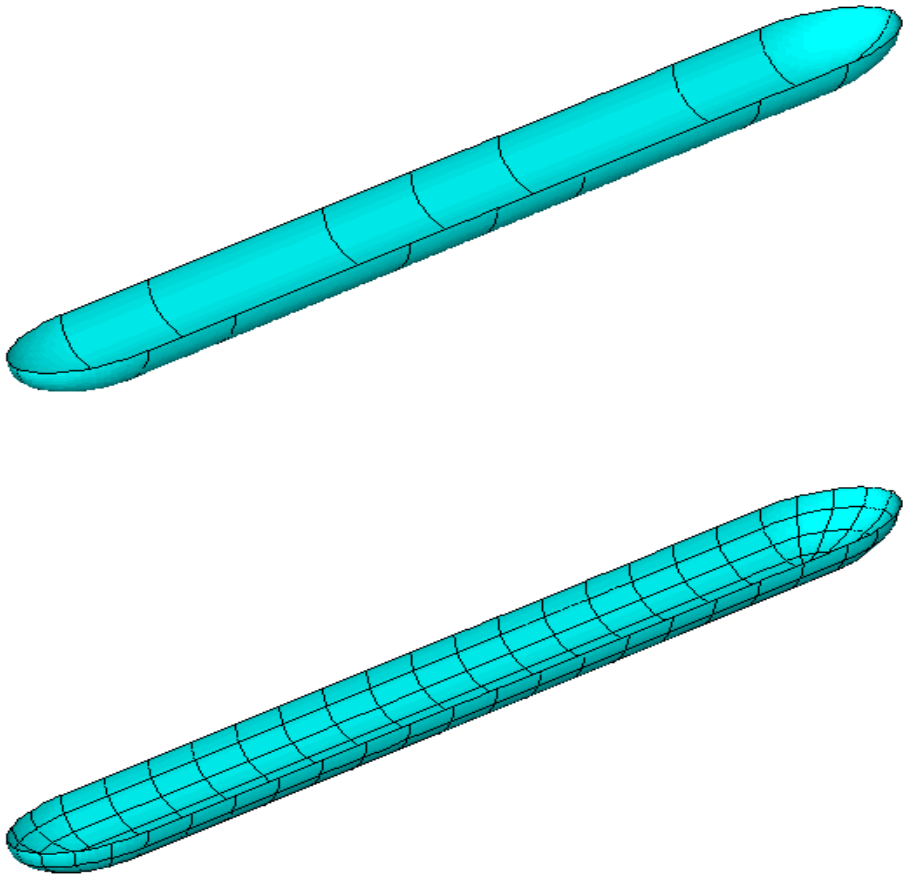
A.24 MOTIONS OF A HINGED VESSEL – TEST24

The subroutine CCYLHSP (IGDEF=-32) is used to generate a horizontal circular cylinder, with spheroidal ends, as shown below. The dimensions are specified in TEST24.GDF. Two planes of symmetry are specified. The cylinder is subdivided into five segments, to permit the analysis of a vessel with transverse hinges between the segments. Half of the middle segment and two others are in the domain $x > 0$. Four patches are required for these three elements plus the spheroidal end. The total number of segments, specified in TEST24.GDF, is used to read the x-coordinates of the boundaries between adjacent segments and also the end of the vessel. The total number of segments is equal to seven, including five cylinders plus two spheroids. Only the boundaries with coordinates x>0 are included in the last line of the GDF file, since ISX=1.

The generalized modes which represent the deflection of the hinges are defined in the subroutine HINGE_MODES in the DLL file NEWMODES.F, designated by the parameter IGENMDS=22 in TEST24.CFG. This subroutine reads the appropriate input data from the file XHINGE.DAT, which is shown below. This input file also specifies the x-coordinates of the hinges. The last cylinder and the spheroidal end are considered to be rigidly joined. Thus there are five 'active' segments corresponding to the parameter NSEG in the XHINGE.DAT file, and NEWMDS=4 is assigned in the TEST24.POT file.

In the TEST24.FRC file, the 10×10 matrix of inertia coefficients is specified. No external damping or stiffness matrices are input, corresponding to the situation where the hinges are ideal without friction or other mechanical constraints.

Further information can be found in the headers and comments of the subroutines which are used to generate the geometry and to represent the hinge modes.



```
TEST24.GDF:
TEST24 segmented vessel with 7 segments
1. 9.80665  ULEN GRAV
1  1        ISX  ISY
4  -32       NPATCH  IGDEF
3           NLINES
7     Nsegments
1.    Radius
 2. 6. 8. 10.   xseg

TEST24.SPL:
TEST24 elastic barge
TEST24 segmented vessel with 7 segments
 4 2      NU NV   mid cylinder
 4 4              next cylinder
 4 2              outer cylinder
 4 4              spheroidal end
```

```
TEST24.POT:
TEST24 segmented vessel with 7 segments and 4 hinge modes
 -1.
 0           0               IRAD, IDIFF
 2                            NPER (array PER follows)
 3. 5.
  1                             NBETA (array BETA follows)
 180.
 1                            NBODY
test24.gdf
 0. 0. 0. 0.                  XBODY
 1  0  1  0  1  0             IMODE(1-6)
 4                            NEWMDS

XHINGE.DAT file for hinge coordinates of hinged body used in TEST24
  1  5          ISX,  NSEG
 2.  6.  10. XHINGE(0:NSEG)

TEST24.FRC:
TEST24 segmented vessel with 7 segments and 4 hinge modes
 1  1  1  1  0  0  0  0  0
 1000.
 0. 0. 0.
 1
  29321.5     0.      0.    0.     0.      0.     0.     0.     0.     0.
  0.      29321.5     0.    0.     0.      0.     0.     0.     0.     0.
  0.          0.  29321.5   0.     0.      0. 12000.     0. 12000.     0.
  0.          0.      0. 1.5E4     0.      0.     0.     0.     0.     0.
  0.          0.      0.    0. 7.33E5      0.     0. 24000.     0. 72000.
  0.          0.      0.    0.     0.  7.33E5     0.     0.     0.     0.
  0.          0.  12000.    0.     0.      0. 10000.     0.  2000.     0.
  0.          0.      0.    0. 24000.      0.     0.  6000.     0.  2000.
  0.          0.  12000.    0.     0.      0.  2000.     0.  8000.     0.
  0.          0.      0.    0. 72000.      0.     0.  2000.     0.  8000.
 0
 0
 0
 0
```

```
TEST24.cfg:
 ipltdat=5
 ILOWHI=1
 IALTPOT=2
 IALTFRC=2
 IRR=0
 ILOG=0
 ISOLVE=1
 IQUADI=4
 IQUADO=3
 KSPLIN=3
 IPERIO=1
 MONITR=0
 NUMHDR=1
 IGENMDS=22
 USERID_PATH=\WAMITv6   (directory for *.exe, *.dll, and userid.wam)
```

Appendix B

DESCRIPTION OF SECOND-ORDER TEST RUNS

WAMIT V6S includes additional 8 standard test runs illustrating the evaluation of various second-order quantities. It includes 4 low-order and 4 higher-order applications.

The following table gives relevant features of each test run. In this table the first column *tst* denotes the name of the test run. All of the corresponding input/output files (except FNAMES.WAM) are assigned the filenames TEST*tst*. (For example, the input POT file for the first test run listed below is TEST101.POT.) *tst* contains three digits starting with 1 for the second-order applications, the second digit is 0 for low-order test runs (ILOWHI=0), and 1 for higher-order test runs (ILOWHI=1). Test runs which are identical except for different input options are assigned the same number with a letter suffix. Thus TEST103(TEST113) and TEST103a(TEST113a) describe the same problem using different options to evaluate the second-order solution.

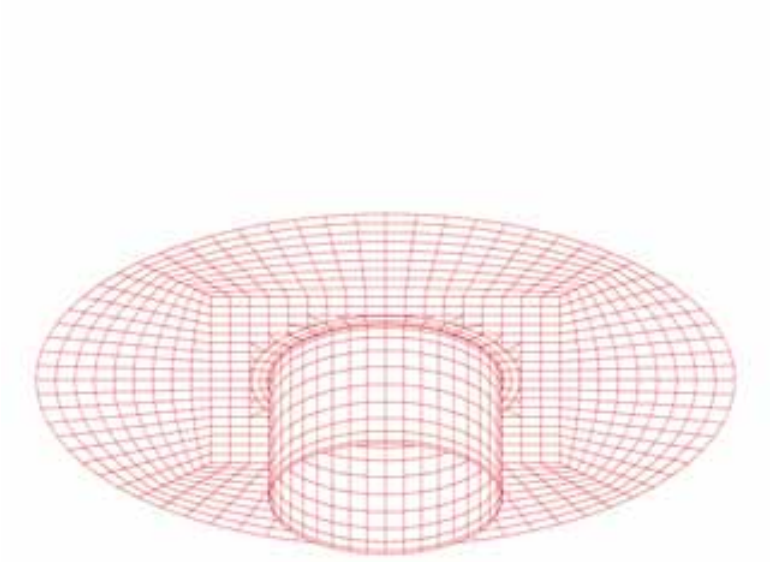
tst	geometry	ILOWHI	other parameters/comments
101	Bottom mounted cylinder	0	
102	Truncated cylinder	0	
103	Cylinder & spheroid	0	Free surface forcing ignored
103a	Cylinder & spheroid	0	
111	Bottom mounted cylinder	1	
112	Truncated cylinder	1	
113	Cylinder & spheroid	1	Free surface forcing ignored
113a	Cylinder & spheroid	1	

B.1 BOTTOM MOUNTED CYLINDER – TEST101

The first-order quantities including exciting forces, pressures and fluid velocities on the body, field pressures and wave elevation, field velocities, drift forces are evaluated. In addtion, the second-order quantities including quadratic forces, complete forces by direct and indirect methods, pressures on the body, field pressures, wave elevations are evaluated for a bottom mounted vertical circular cylinder of radius 1 meter and draft 1 meter, in finite water depth of 1 meter for two wave periods and two wave headings. Two second harmonic of unidirectional waves, a mean and a difference frequency of bidirectional waves are considered in the second-order computation (see PT2 file).

Using two planes of symmetry, only the first quadrant of the surface of the cylinder is discretized with 100 panels. 10 panels are distributed in the azimuthal and vertical directions with double cosine spacing in the latter. The characteristic length (ULEN) is set equal to the radius of the cylinder.

The free surface inside the inner circle of radius 3 meters is discretized automatically with 252 panels on the first quadrant. No intermediate region is considered and the partition radius is the same as the radius of inner circle. Perspective view of the body and the inner region of the free surface is shown below.



```
TEST101.GDF:
TEST101 Bottom mounted cylinder R=1 T=1 Cosine spacing
   1.000000       9.806650
           1           1
         100
  0.9876884      0.1564345      0.0000000E+00
   1.000000      0.0000000E+00  0.0000000E+00
   1.000000      0.0000000E+00 -2.7999997E-02
  0.9876884      0.1564345     -2.7999997E-02

TEST101.POT:
TEST101.POT  Bottom mounted Cylinder R=1, T=1, igdef=-1
 1.
 -1           1               IRAD, IDIFF
 2                            NPER (array PER follows)
0.4   0.8
 2                            NBETA (array BETA follows)
 0.  90.
 1                            NBODY
 test101.gdf
 0. 0. 0. 0.                  XBODY
 1  1  1  1  1  1             IMODE(1-6
 0                            NEWMDS

TEST101.FRC:
TEST101.FRC  Bottom mounted Cylinder R=1, T=1, igdef=-1
 0   0   1   0   3   2   2   2   2   1   1   1   1   1   1   0   (IOPTN 1-16)
 0.000000                                  VCG
 1.000000      .0000000      .0000000
 .0000000      1.000000      .0000000
 .0000000      .0000000      1.000000      XPRDCT
 0                                         NBETAH
 1
 1.044325      8.2290111E-02  0.0000000E+00

TEST101.PT2:
TEST101.PT2 -- PT2
-1  1          (diffraction for second-order force for all modes)
1 1 1 1 1 1
1 1
2
1 1 1
1 1
2 2 1
```

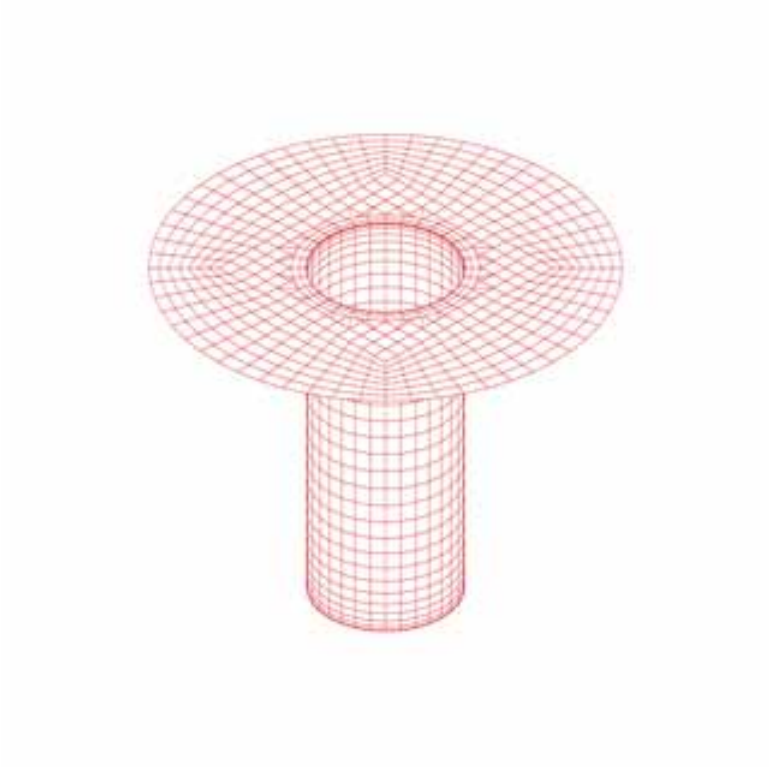
```
1 1
2
1 1 1
1 1
1 2  1
1 2

TEST101.FDF:
TEST101.FDF
  3.0000E+00    Inner radius (=Partition radius)
  -1.   1.      Auto Discretization, Scale=1.
   0            No intermediate region

TEST101.CFG:
 IPLTDAT=1
 ILOWGDF=1
 ILOWHI=0
 ISOR=1
 IALTPOT=2
 IRR=0
 ISOLVE=2
 KSPLIN=3
 IQUADO=3
 IQUADI=4
 IPERIO=3
 MONITR=0
 NUMHDR=1
 I2ND=1
 ILOG=1
 NOOUT= 1 1 1 1 0 1 1 1 1  1  1  1  0  1  1  1
 USERID_PATH=\WAMITv6   (directory for *.exe, *.dll, and userid.wam)
```

B.2 TRUNCATED CIRCULAR CYLINDER – TEST102

The first-order exciting forces, drift forces and the second-order quadratic forces and complete second-order forces by direct and indirect methods are evaluated for a vertical circular cylinder of radius 8.435 meters and draft 35 meters, in infinite water depth for two wave periods and two wave headings. All sum- and difference frequency pairs and wave heading pairs are considered for the second-order forces.

Using two planes of symmetry, only the first quadrant of the surface of the cylinder is discretized with 320 panels. 8, 28, 12 panels are distributed in the azimuthal, vertical and radial directions with double cosine spacing on the side and cosine spacing toward the corner at the bottom. The characteristic length (ULEN) is set equal to the radius of the cylinder.

The free surface inside the inner circle of radius 25.305 meters, three times of the cylinder radius, is automatically discretized with 159 panels on the first quadrant. The intermediate region consists of two annuli of width 16.87 meters, two times the radius of the cylinder. On each annulus, $2^4 + 1$ nodes and 8 nodes Gauss quadratures are applied in azimuthal and radial direction, respectively. The partition circle coincides with the outside circle of the second annulus and its radius is 58.45 meters. Perspective view of the body and the inner region of the free surface is shown below.



```
TEST102.GDF:
TEST102.GDF -- Cylinder R=8.435 T=35 meters, Cosine spacing at corner and waterline
   8.435000       9.806650
            1           1
          320
   8.272924       1.645587      0.0000000E+00
   8.435000      0.0000000E+00  0.0000000E+00
   8.435000      0.0000000E+00 -0.1307399
   8.272924       1.645587     -0.1307399

TEST102.POT:
TEST102  (Cylinder R=8.435 T=35 in infinite depth water)
-1.  0.  0.  0.  0.
 -1  1
 1  1  1  1  1  1
 2
 7.8934  5.1207
 2
 0. 90.

TEST102.FRC:
TEST102 -- Linear, mean and second-order exciting forces
   0    0    1    0    0    0    0    2   2   1    1    1    0    0    0    0
         .0000000
         1.000000      .0000000      .0000000
         .0000000      1.000000      .0000000
         .0000000      .0000000      1.000000
          0
          0

TEST102.PT2:
TEST102  (All frequency and wave heading combinations)
 -1  1      IRAD2=-1 IDIF2=1 (Exciting force only)
 1 1 1 1 1 1
 2 2

TEST102.FDF:
TEST102  Single cylinder R=8.435, Inner radius= 3*R, partition radius= 7*R
   25.30500
    -1    1.0
    2    16.87   4     8

TEST102.CFG:
 IPLTDAT=1
```

```
MAXSCR=1024
ISOR=1
IQUAD=0
ILOG=1
IDIAG=0
IRR=0
NUMHDR=1
IALTPOT = 1    !  Alternative Form 1 POT/CFG
IALTFRC = 1    !  Alternative Form 1 FRC
I2ND=1
USERID_PATH=\WAMITv6   (directory for *.exe, *.dll, and userid.wam)
```

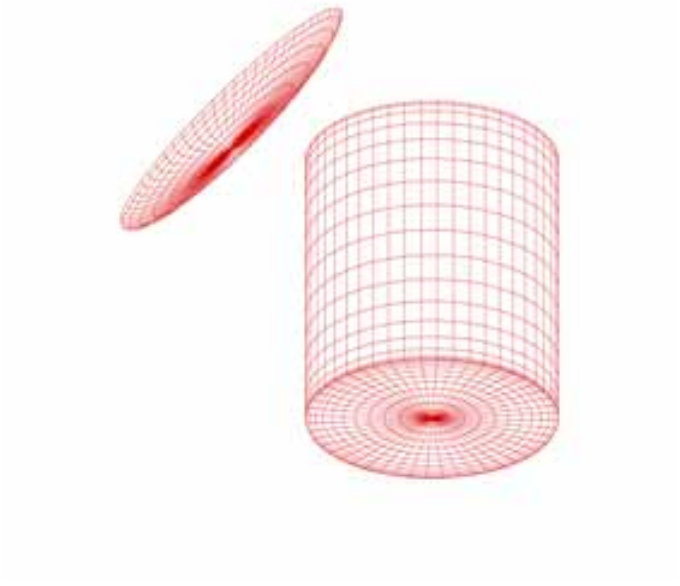
B.3 MULTIPLE BODIES (APPROXIMATION) – TEST103

The NBODY option is illustrated in this test run. The bodies are the same as TEST05. Body one is a circular cylinder of radius 1 meter and draft 2 meters. Body 2 is a spheroid of length 4 meters and maximum radius 0.25 meters. The gap between these two bodies is set equal to the beam of the spheroid (0.5 meters) and the origin of the global coordinate system is located at the mid-point of this gap. The relative locations of the two bodies and the orientation of the spheroid are specified in the GGDF file. One quadrant of the cylinder is discretized with 336 panels. 12,12 and 16 panels are distributed in the azimuthal, radial, and vertical directions using cosine spacing in radial and vertical directions. One quadrant of the spheroid is discretized with 128 panels. 16 and 8 panels are distributed in the longitudinal, and transverse directions.

The Alternative 1 input format is used for the POTEN subprogram and the Alternative 3 input format is used for FORCE. The vector IALTFRCN is included in TEST103.CFG to indicate that IALTFRC(1)=1 for the cylinder and IALTFRC(2)=2 for the spheroid in the separate FRC files.

The cylinder is fixed and the spheroid is free to move. The added-mass and damping coefficients and the first-order exciting forces, motions, wave elevations, field pressures, field velocities and mean drift forces are evaluated in infinite water depth for two wave periods and one wave heading (POT file). The second-order forces, wave elevations and field pressures are evaluated for all difference frequency pairs (PT2 file).

The forcing on the body due to the second-order incident wave and due to the first-order body motion of the spheriod are considered but the forcing on the free surface is ignored (FDF file).



```
TEST103.GDF:
TEST103.GDF -- GGDF file for use with IALTPOT=1 (Cylinder + Spheroid)
 -1. 9.80665
 2
 test103c.gdf
 1.25 0.0 0.0 0.0
 0   0   0   0   0   0    fixed
 test103s.gdf
 -0.5 0.0 0.0 90.0
 1   1   1   1   1   1    all modes are free

TEST103C.GDF: (first 8 lines only)
TEST103 & TEST103A    Cylinder R=1 T=2 Double cosines side, cosine bottom
   1.000000       9.806650
            1           1
          336
  0.0000000E+00  0.0000000E+00  -2.000000
  0.0000000E+00  0.0000000E+00  -2.000000
  0.1294095      1.7037088E-02  -2.000000
  0.1305262      0.0000000E+00  -2.000000

TEST103S.GDF: (first 8 lines only)
TEST103 TEST103A Spheriod L=2 R1=R2=0.25
   1.000000       9.806650
            1           1
          128
   1.990369      2.4504285E-02 -1.0927847E-08
   2.000000      0.0000000E+00 -1.0927847E-08
   1.961571      0.0000000E+00 -4.8772592E-02
   1.952125      2.4033442E-02 -4.8772592E-02

TEST103.POT:
TEST103.POT --  CYLINDER + SPHEROID, ILOWHI=0
 -1.0 0.0 0.0 0.0 0.0
 0   1                          Dummy (check TEST103.gdf)
 0  0  0  0  0  0               Dummy
 2
 8.971402  2.006403
 1
 0.0

TEST103.FRC:
TEST103.FRC -- Cylinder SPHEROID  , ILOWHI=0
  1  1  1  1  0  1  1  1  1  1  1  1  1  1  1  0
```

```
 1.0
test103C.frc
test103S.frc
 0
 1
 0.0001 0.0001 0.
```

TEST103C.FRC:

```
TEST 103 Cylinder
 0  0  0  0  0  0  0  0  0  0  0  0  0  0  0  0
 0.000000
 1.000000      .0000000      .0000000
 .0000000      1.000000      .0000000
 .0000000      .0000000      1.000000
 0
 0
```

TEST103S.FRC:

```
TEST103S Floating Spheroid
 0  0  0  0  0  0  0  0  0  0  0  0  0  0  0  0
 1.        rho (water density)
 0. 0. 0.  XCG (center of grativity w.r.t. body coordinates)
 1         IMASS (radii of gyration are assumed 1)
 0.257621  0.        0.        0.        0.        0.
 0.        0.257621  0.        0.        0.        0.
 0.        0.        0.257621  0.        0.        0.
 0.        0.        0.        0.257621  0.        0.
 0.        0.        0.        0.        0.257621  0.
 0.        0.        0.        0.        0.        0.257621
 0     IDAMP (if damped change it to 1 and add 6 x 6 damping forces)
 0     ISITF (if restoring force applied change it to 1 and add 6 x 6 elements)
 0
 0
```

TEST103.PT2:

```
TEST103 -- PT2
-1  1          (diffraction for second-order force for all modes)
1 1 1 1 1 1
-1  1          (diffraction for second-order force for all modes)
1 1 1 1 1 1
0 2            (all difference frequency pairs)
```

TEST103.FDF:

```
TEST103  -- Free suface forcing not included (NPF,NTCL,NAL=0)
```

```
   5.    PARTR (irrelavant because no foricing considered)
   0        0    NPF, NTCL
   0             NAL

TEST103.CFG
 IPLTDAT=1
 MAXSCR=2000
 ISOR=1
 IQUAD=0
 ILOG=1
 IDIAG=0
 IRR=0
 NUMHDR=1
 NOOUT=0  1  1  1  0  1  1  1  1  1  1  0  1  1  0
 IALTPOT = 1    !  GDF names in GGDF file
 IALTFRC = 3    !  Alternative Form 3 FRC for GFRC TEST33.FRC
 IALTFRCN= 1 2  !  form 1 TEST103c.FRC and Form 2 spheriod TEST103s.FRC
 I2ND=1
 USERID_PATH=\WAMITv6   (directory for *.exe, *.dll, and userid.wam)
```

B.3A MULTIPLE BODIES – TEST103A

This test is the same as TEST103 except the free surface forcing is not ignored in the second-order computation.

The cylinder is fixed and the spheroid is free to move. The added-mass and damping coefficients and the first-order exciting forces, motions, wave elevations, field pressures, field velocities and mean drift forces are evaluated in infinite water depth for two wave periods and one wave heading.

The second-order forces, wave elevations and field pressures are evaluated for all difference frequency pairs (see PT2 file).

The free surface inside the inner circle of radius 5 meters is discretized automatically with 2052 panels over the entire inner region. No intermediate region is considered and the partition radius is the same as the radius of inner circle. Perspective view of the body and the inner region of the free surface is shown below.

```
TEST103A.GDF:
TEST103A.GDF -- GGDF file for use with IALTPOT=0 (Cylinder + Spheroid)
 -1. 9.80665
 2
 test103c.gdf
 1.25 0.0 0.0 0.0
 0   0   0   0   0   0    fixed
 test103s.gdf
 -0.5 0.0 0.0 90.0
 1   1   1   1   1   1    all modes are free

TEST103C.GDF: (first 8 lines only)
TEST103 & TEST103A    Cylinder R=1 T=2 Double cosines side, cosine bottom
   1.000000       9.806650
            1           1
          336
  0.0000000E+00  0.0000000E+00  -2.000000
  0.0000000E+00  0.0000000E+00  -2.000000
  0.1294095      1.7037088E-02  -2.000000
  0.1305262      0.0000000E+00  -2.000000

TEST103S.GDF: (first 8 lines only)
TEST103 TEST103A Spheriod L=2 R1=R2=0.25
   1.000000       9.806650
            1           1
          128
   1.990369      2.4504285E-02 -1.0927847E-08
   2.000000      0.0000000E+00 -1.0927847E-08
   1.961571      0.0000000E+00 -4.8772592E-02
   1.952125      2.4033442E-02 -4.8772592E-02

TEST103A.POT:
TEST103A.POT --  CYLINDER + SPHEROID, ILOWHI=0
 -1.0 0.0 0.0 0.0 0.0
 0   1                          Dummy (check TEST103.gdf)
 0  0  0  0  0  0               Dummy
 2
 8.971402  2.006403
 1
 0.0

TEST103A.FRC:
TEST103A.FRC -- Cylinder SPHEROID  , ILOWHI=0
  1  1  1  1  0  1  1  1  1  1  0  1  0  1  1  0
```

```
 1.0
test103C.frc
test103S.frc
 0
 1
 0.0001 0.0001 0.
```

TEST103C.FRC:

```
TEST 103 Cylinder
 0  0  0  0  0  0  0  0  0  0  0  0  0  0  0  0
 0.000000
 1.000000      .0000000      .0000000
 .0000000      1.000000      .0000000
 .0000000      .0000000      1.000000
 0
 0
```

TEST103S.FRC:

```
TEST103S Floating Spheroid
 0  0  0  0  0  0  0  0  0  0  0  0  0  0  0  0
 1.        rho (water density)
 0. 0. 0.  XCG (center of grativity w.r.t. body coordinates)
 1         IMASS (radii of gyration are assumed 1)
 0.257621  0.        0.        0.        0.        0.
 0.        0.257621  0.        0.        0.        0.
 0.        0.        0.257621  0.        0.        0.
 0.        0.        0.        0.257621  0.        0.
 0.        0.        0.        0.        0.257621  0.
 0.        0.        0.        0.        0.        0.257621
 0     IDAMP (if damped change it to 1 and add 6 x 6 damping forces)
 0     ISITF (if restoring force applied change it to 1 and add 6 x 6 elements)
 0
 0
```

TEST103A.PT2:

```
TEST103 -- PT2
-1  1          (diffraction for second-order force for all modes)
1 1 1 1 1 1
-1  1          (diffraction for second-order force for all modes)
1 1 1 1 1 1
0 2            (all difference frequency pairs)
```

TEST103A.FDF:

```
TEST103A AND 113A
```

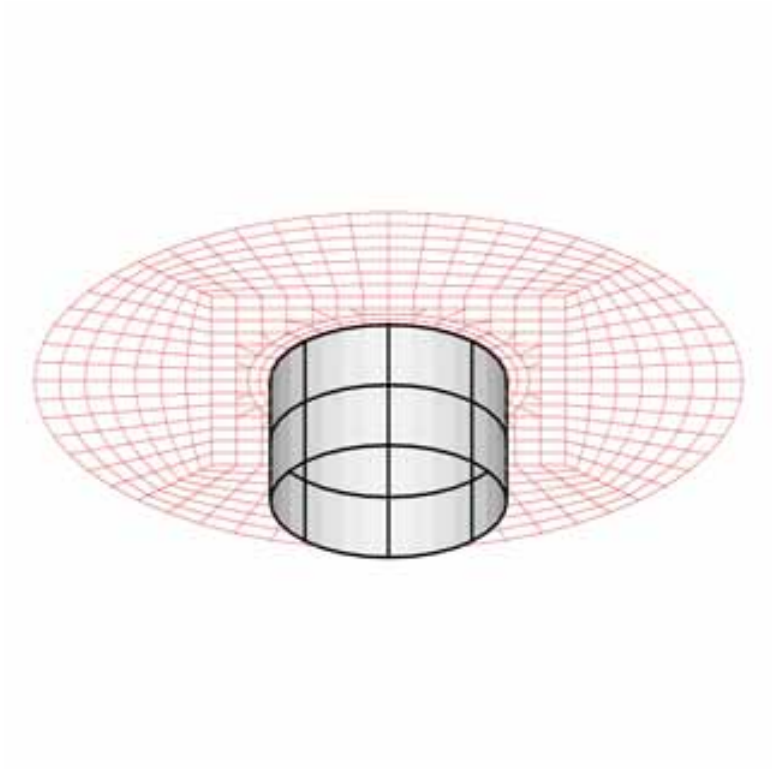
```
   5.    PARTR
   -1       1.5    AUTO-DISCRETIZATION, SCALE=1.5
   0             NAL

TEST103A.CFG
 IPLTDAT=1
 MAXSCR=2000
 ISOR=1
 IQUAD=0
 ILOG=1
 IDIAG=0
 IRR=0
 NUMHDR=1
 NOOUT=0  1  1  1  0  1  1  1  1  1  1  0  1  1  0
 IALTPOT = 1    !  GDF names in GGDF file
 IALTFRC = 3    !  Alternative Form 3 FRC for GFRC TEST33.FRC
 IALTFRCN= 1 2  !  form 1 TEST103c.FRC and Form 2 spheriod TEST103s.FRC
 I2ND=1
 USERID_PATH=\WAMITv6   (directory for *.exe, *.dll, and userid.wam)
```

B.11 BOTTOM MOUNTED CYLINDER – TEST111

The test uses the same cylinder as in the low-order TEST101. The first-order quantities including exciting forces, pressures and fluid velocities on the body, field pressures and wave elevation, field velocities, drift forces are evaluated. In addtion, the second-order quantities including quadratic forces, complete forces by direct and indirect methods, pressures on the body, field pressures, wave elevations are evaluated for a bottom mounted vertical circular cylinder of radius 1 meter and draft 1 meter, in finite water depth of 1 meter for two wave periods and two wave headings. Two second harmonic of unidirectional waves, a mean and a difference frequency of bidirectional waves are considered in the second-order computation (see PT2 file).

Using two planes of symmetry, only the first quadrant of the surface of the cylinder is divided into 4 higher-order panels as shown below. The order of B-spline is set KSPLIN=3 in CFG file. Uniform mapping is used as the pressure and velocity are finite on the body surface.

The free surface inside the inner circle of radius 3 meters is discretized automatically with 159 panels on the first quadrant. No intermediate region is considered and the partition radius is the same as the radius of inner circle. Perspective view of the body and the inner region of the free surface is shown below.



```
TEST111.GDF:
TEST111 Bottom mounted cylinder R=1 T=1 -- analytic geometry (npatch=1)
1. 9.80665  ULEN GRAV
1  1        ISX  ISY
1  -1       NPATCH  IGDEF
2
1.0 1.     RADIUS, DRAFT
0          Uniform mapping

TEST111.SPL:
TEST111.spl - cylinder R=1 T=1. -- analytic geometry (npatch=1)
2  2        NU NV (Patch 1, side   u azimuthal v vertical)

TEST111.POT
 TEST111.POT  Cylinder R=1, T=1, igdef=-1
 1.
 -1           1               IRAD, IDIFF
 2                            NPER (array PER follows)
0.4 0.8
 2                          NBETA (array BETA follows)
 0. 90.
 1                            NBODY
 test111.gdf
 0. 0. 0  0.                  XBODY
 1  0  0  0  0  0             IMODE(1-6)
 0                            NEWMDS

TEST111.FRC:
TEST111.FRC  Bottom mounted Cylinder R=1, T=1.0, igdef=-1
 0   0   1   0   3   1   1   2   2   1   1   1   1   1   1   0   IOPTN(1-16)
 0.000000                                  VCG
 1.000000      .0000000      .0000000
 .0000000      1.000000      .0000000
 .0000000      .0000000      1.000000      XPRDCT
 0                                         NBETAH
 1                                         NFIELD
 1.044325      8.2290111E-02  0.0000000E+00  XFILED

TEST111.PT2:
TEST111.PT2 -- PT2
-1  1          (diffraction for second-order force for all modes)
1 1 1 1 1 1
1 1
```

```
2
1 1 1
1 1
2 2 1
1 1
2
1 1 1
1 1
1 2  1
1 2
```

```
TEST111.FDF:
TEST111.FDF
  3.0000E+00    Inner radius (=Partition radius)
  -1.   1.      Auto Fdf, Scale=1.
   0            No intermediate region
```

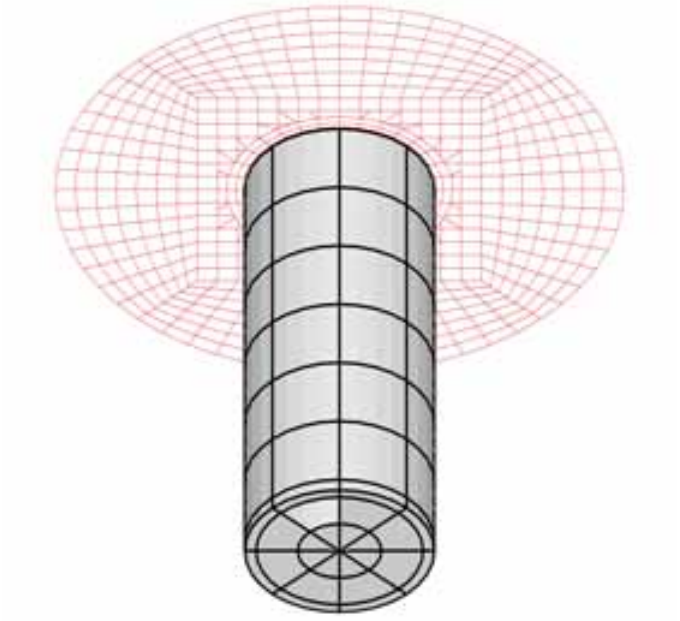
```
TEST111.CFG:
 IPLTDAT=4
 ILOWGDF=4
 ILOWHI=1
 IALTPOT=2
 IRR=0
 ISOLVE=1
 KSPLIN=3
 IQUADO=3
 IQUADI=4
 IPERIO=3
 MONITR=0
 NUMHDR=1
 I2ND=1
 IPOTEN=1
 ILOG=1
 NOOUT= 1 1 1 1 0 1 1 1 1  1  1  1  0  1  1  1
 USERID_PATH=\WAMITv6   (directory for *.exe, *.dll, and userid.wam)
```

B.12 TRUNCATED CIRCULAR CYLINDER – TEST112

The test uses the same cylinder as in the low-order TEST102. The first-order exciting forces, drift forces and the second-order quadratic forces and complete second-order forces by direct and indirect methods are evaluated for a vertical circular cylinder of radius 8.435 meters and draft 35 meters, in infinite water depth for two wave periods and two wave headings. All sum- and difference frequency pairs and wave heading pairs are considered for the second-order forces.

Using two planes of symmetry, the first quadrant of the surface of the cylinder is represented with 4 patches including two narrow patches near the corner, one on the side and the other on the bottom. The width of these patches is 1 meter. Nonuniform mapping corresponding to the corner flow is used on these patches. The rest of each surface on the side and bottom is represented be a patch with unform mapping. 20 higher-order panels are specified on the first quadrant of the surface as shown in the following figure.

The free surface inside the inner circle of radius 25.305 meters, three times of the cylinder radius, is automatically discretized with 159 panels on the first quadrant. The intermediate region consists of two annuli with width of 16.87 meters, two times the radius of the cylinder. On each annulus, $2^4 + 1$ nodes and 8 nodes Gauss quadratures are applied in azimuthal and radial directions, respectively. The partition circle coincides with the outside circle of the second annulus and its radius is 58.45 meters. Perspective view of the body and the inner region of the free surface is shown below.



```
TEST112.GDF:
TEST112.GDF -- Cylinder R=8.435m D=35m, nonuniform mapping 1m from corner
     8.435       9.80665
     1           1
     4  -101
     1
     8.435 35. 1.  RADIUS, DRAFT, WIDTH OF CORNER PATCHES

TEST112.SPL:
TEST112.SPL -- 2 azimuthal, 6 side vertical, 2 bottom radial, 1 corner patches
2 6   Side above corner patch  (2 azimutal, 6 vertical)
2 1   Side corner patch        (2           1 vertical)
2 2   Bottom inside patch      (2           2 radial  )
2 1   Bottom corner patch      (2           1 radial

TEST112.POT:
TEST112.POT infinite depth, diffraction only
-1  0.  0.  0.  0.
 -1  1
 1  1  1  1  1  1
 2
 7.8934  5.1207
 2
 0. 90.

TEST112.FRC:
TEST112.FRC -- Linear, mean and second-order exciting forces
   0    0    1    0    0    0    0    2   2   1    1    1    0    0    0    0
        .0000000
        1.000000      .0000000      .0000000
        .0000000      1.000000      .0000000
        .0000000      .0000000      1.000000
         0
         0


TEST112.PT2:
TEST112  (All frequency and wave heading combinations)
 -1  1      IRAD2=-1 IDIF2=1 (Exciting force only)
 1 1 1 1 1 1
 2 2
```

```
TEST112.FDF
TEST112  Single cylinder R=8.435, Inner radius= 3*R, partition radius= 7*R
   25.30500
    -1    1.0
    2    16.87   4     8

TEST112.CFG:
 MAXSCR=1000
 IPLTDAT=4
 IQUAD=0
 ILOG=1
 IDIAG=0
 IRR=0
 ISOR=0
 NUMHDR=1
 IALTPOT = 1    !  Alternative Form 1 POT/CFG
 IALTFRC = 1    !  Alternative Form 1 FRC
 I2ND=1
 ILOWHI=1
 KSPLIN=3
 IQUADO=3
 IQUADI=4
 ISOLVE=1
 USERID_PATH=\WAMITv6   (directory for *.exe, *.dll, and userid.wam)
```

B.13 MULTIPLE BODIES (APPROXIMATION) – TEST113

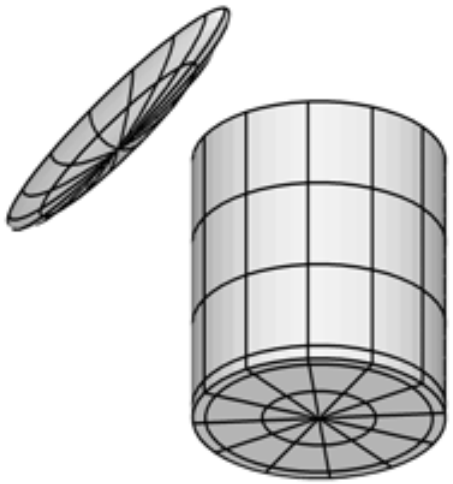
The bodies are the same as TEST103. One quadrant of the cylinder is represented with 4 patches. Nonuniform mapping corresponding to corner flows is used on the patches of width 0.1 meter at the corner. Uniform mapping is used on the other patches on the side and bottom. The first quadrant of the cylinder is represented by 21 higher order panels and that of the spheroid by 8 panels.

The Alternative 2 input format is used for the POTEN subprogram and the Alternative 3 input format is used for FORCE. The vector IALTFRCN is included in TEST113.CFG to indicate that IALTFRC(1)=1 for the cylinder and IALTFRC(2)=2 for the spheroid in the separate FRC files.

The cylinder is fixed and the spheroid is free to move. The added-mass and damping coefficients and the first-order exciting forces, motions, wave elevations, field pressures, field velocities and mean drift forces are evaluated in infinite water depth for two wave periods and one wave heading.

The second-order forces, wave elevations and field pressures are evaluated for all difference frequency pairs (see PT2 file). The forcing on the free surface is ignored. The forcing on the body due to the second-order incident wave and due to the first-order body motion of the spheriod are considered but that on the free surface is ignored (see FDF file).

Perspective view of the body and the inner region of the free surface is shown below.



```
TEST113C.GDF:
TEST113C cylinder R=1 T=2 -- analytic geometry (npatch=4)
1. 9.80665  ULEN GRAV
1  1        ISX  ISY
4  -13       NPATCH  IGDEF
1           NLINES
1.0 2. 0.1     RADIUS, DRAFT


TEST113S.GDF:
TEST113S spheroid  a=2, b=c=0.25 -- igdef=-4
1. 9.80665  ULEN GRAV
1  1        ISX  ISY
1  -4       NPATCH  IGDEF
1
2.0 0.25 0.25    A, B, C

TEST113.POT:
 TEST113.POT -- Cylinder + spheroid, ILOWHI=1
 -1.
 0           1               IRAD, IDIFF
 2                            NPER (array PER follows)
 8.971402  2.006403
 1                           NBETA (array BETA follows)
 0.
 2                            NBODY
 test113c.gdf
 1.25 0.0 0.0 0.0             XBODY
 0  0  0  0  0  0             IMODE(1-6)
 0                            NEWMDS
 test113s.gdf
 -0.5 0.0 0.0 90.0            XBODY
 1  1  1  1  1  1             IMODE(1-6)
 0                            NEWMDS

TEST113.FRC:
TEST113.FRC -- Cylinder SPHEROID
  1  1  1  1  0  1  1  1  1  1  1  1  1  1  1  0
  1.0
 test113C.frc
 test113S.frc
  0
  1
```

```
  0.0001 0.0001 0.

TEST113C.FRC:
 TESTS 103 and 113 Fixed Cylinder
 0  0  0  0  0  0  0  0  0  0  0  0  0  0  0  0
 0.000000
 1.000000      .0000000      .0000000
 .0000000      1.000000      .0000000
 .0000000      .0000000      1.000000
 0
 0

TEST113S.FRC:
 TESTS 103 and 113 Floating Spheroid
 0  0  0  0  0  0  0  0  0  0  0  0  0  0  0  0
 1.        rho (water density)
 0. 0. 0.  XCG (center of gravity w.r.t. body coordinates)
 1         IMASS (radii of gyration are assumed 1)
 0.257621  0.        0.        0.        0.        0.
 0.        0.257621  0.        0.        0.        0.
 0.        0.        0.257621  0.        0.        0.
 0.        0.        0.        0.257621  0.        0.
 0.        0.        0.        0.        0.257621  0.
 0.        0.        0.        0.        0.        0.257621
 0     IDAMP (if damped change it to 1 and add 6 x 6 damping forces)
 0     ISITF (if restoring force applied change it to 1 and add 6 x 6 elements)
 0
 0

TEST113.PT2:
TEST103  AND TEST 113 -- PT2
-1  1          (diffraction for second-order force for all modes)
1 1 1 1 1 1
-1  1          (diffraction for second-order force for all modes)
1 1 1 1 1 1
0 2            (all difference frequency pairs)

TEST113.FDF:
TEST103 AND 113 -- Free suface forcing not included (NPF,NTCL,NAL=0)
   5.    PARTR (irrelavant because no foricing considered)
   0        0    NPF, NTCL
   0             NAL

TEST113.CFG:
```

```
ILOWGDF=4
IPLTDAT=4
MAXSCR=1000
ILOWHI=1
IALTPOT=2
IRR=0
ISOLVE=2
KSPLIN=3
IQUADO=3
IQUADI=4
IPERIO=1
NUMHDR=1
NOOUT=  0  1  1  1  0  1  1  1  1  1  1   1  0  1  1  0
IALTFRC = 3    !  Alternative Form 3 FRC
IALTFRCN= 1 2
I2ND=1
ILOG=1
USERID_PATH=\WAMITv6   (directory for *.exe, *.dll, and userid.wam)
```

B.13A MULTIPLE BODIES – TEST113A

This test is the same as TEST113 except the free surface forcing is not ignored in the second-order computation.

The cylinder is fixed and the spheroid is free to move. The added-mass and damping coefficients and the first-order exciting forces, motions, wave elevations, field pressures, field velocities and mean drift forces are evaluated in infinite water depth for two wave periods and one wave heading.

The second-order forces, wave elevations and field pressures are evaluated for all difference frequency pairs (see PT2 file).

The free surface inside the inner circle of radius 5 meters is discretized automatically with 2052 panels over the entire inner region. No intermediate region is considered and the partition radius is the same as the radius of inner circle. Perspective view of the body and the inner region of the free surface is shown below.

```
TEST113C.GDF:
TEST113C cylinder R=1 T=2 -- analytic geometry (npatch=4)
1. 9.80665  ULEN GRAV
1  1        ISX  ISY
4  -13       NPATCH  IGDEF
1           NLINES
1.0 2. 0.1     RADIUS, DRAFT, WIDTH OF CONNER PATCHES


TEST113S.GDF:
TEST113S spheroid  a=2, b=c=0.25 -- igdef=-4
1. 9.80665  ULEN GRAV
1  1        ISX  ISY
1  -4       NPATCH  IGDEF
1
2.0 0.25 0.25    A, B, C

TEST113A.POT:
 TEST113A.POT -- Cylinder + spheroid, ILOWHI=1
 -1.
 0           1               IRAD, IDIFF
 2                            NPER (array PER follows)
 8.971402  2.006403
 1                           NBETA (array BETA follows)
 0.
 2                            NBODY
 test113c.gdf
 1.25 0.0 0.0 0.0             XBODY
 0  0  0  0  0  0             IMODE(1-6)
 0                            NEWMDS
 test113s.gdf
 -0.5 0.0 0.0 90.0            XBODY
 1  1  1  1  1  1             IMODE(1-6)
 0                            NEWMDS

TEST113A.FRC:
TEST113A.FRC -- Cylinder SPHEROID
  1  1  1  1  0  1  1  1  1  1  0  1  0  1  1  0
  1.0
 test113C.frc
 test113S.frc
  0
  1
```

```
  0.0001 0.0001 0.

TEST113C.FRC:
 TESTS 103 and 113 Fixed Cylinder
 0  0  0  0  0  0  0  0  0  0  0  0  0  0  0  0
 0.000000
 1.000000      .0000000      .0000000
 .0000000      1.000000      .0000000
 .0000000      .0000000      1.000000
 0
 0

TEST113S.FRC:
 TESTS 103 and 113 Floating Spheroid
 0  0  0  0  0  0  0  0  0  0  0  0  0  0  0  0
 1.        rho (water density)
 0. 0. 0.  XCG (center of gravity w.r.t. body coordinates)
 1         IMASS (radii of gyration are assumed 1)
 0.257621  0.        0.        0.        0.        0.
 0.        0.257621  0.        0.        0.        0.
 0.        0.        0.257621  0.        0.        0.
 0.        0.        0.        0.257621  0.        0.
 0.        0.        0.        0.        0.257621  0.
 0.        0.        0.        0.        0.        0.257621
 0     IDAMP (if damped change it to 1 and add 6 x 6 damping forces)
 0     ISITF (if restoring force applied change it to 1 and add 6 x 6 elements)
 0
 0

TEST113A.PT2:
TEST103/TEST103A  AND TEST113/TEST113A -- PT2
-1  1          (diffraction for second-order force for all modes)
1 1 1 1 1 1
-1  1          (diffraction for second-order force for all modes)
1 1 1 1 1 1
0 2            (all difference frequency pairs)

TEST113A.FDF:
TEST103A AND 113A -- Free suface forcing not included (NPF,NTCL,NAL=0)
   5.    PARTR
   -1        1    NPF, NTCL
   0             NAL

TEST113.CFG:
```

```
ILOWGDF=4
IPLTDAT=4
MAXSCR=1000
ILOWHI=1
IALTPOT=2
IRR=0
ISOLVE=2
KSPLIN=3
IQUADO=3
IQUADI=4
IPERIO=1
NUMHDR=1
NOOUT=  0  1  1  1  0  1  1  1  1  1  1   1  0  1  1  0
IALTFRC = 3    !  Alternative Form 3 FRC
IALTFRCN= 1 2
I2ND=1
ILOG=1
USERID_PATH=\WAMITv6   (directory for *.exe, *.dll, and userid.wam)
```

Using the WAMIT-RGKernel Interface

J. S. Letcher, Jr.
AeroHydro, Inc.
Southwest Harbor, Maine 04679 USA
207-244-4100

Sept. 8, 2002; revised June 21, 2006

Contents

1. Summary

A joint development effort between AeroHydro, Inc. and WAMIT, Inc. has forged an intimate connection between MultiSurf and WAMIT. As a result, model geometry developed in MultiSurf's "relational geometry" (RG) framework can be directly accessed by WAMIT for all setup and analysis purposes within WAMIT's "higher-order" solution method. This feature is available in WAMIT versions 6.1 and higher.

RGKernel is the C library of mathematical, geometric and data storage functions that supports MultiSurf, SurfaceWorks and other RG applications. It is compiled as a DLL. In this project a second DLL was developed to provide the interface between WAMIT's Fortran code and RGKernel's C procedures.

The advantages of this integration are:

(1) **Accuracy.** RG geometry definitions are fundamentally exact, and in practice can be evaluated to a high degree of precision. This is in contrast to faceted and NURBS-based geometric representations, whose accuracy is commonly compromised by approximations. Accuracy of surface-surface junctions is critical in WAMIT's high-order analysis method.

(2) **Convenience.** When MultiSurf is used to develop WAMIT models for this interface, the user avoids all further effort to develop panel files for the low-order method, or B-spline approximation files as have been needed for the high-order method. The Geometric Data File (GDF) is usually 8 lines for each body defined by this method.

The purpose of this document is to provide instructions for use of this new WAMIT feature. The topics include:

Supported features and options
Required files, versions and locations
MultiSurf modeling considerations
GDF file format
Log file RGKLOG.TXT
Error conditions

2. Supported features and options

The WAMIT-RGKernel interface supports all current features of the higher-order solution option.

Multiple bodies. Each body is associated with a GDF file in the POT or FNAMES file. IGDEF = 2 is reserved to mean "geometry specified by an .MS2 file." The different bodies can have various IGDEF types.

XBODY positioning. Each body has an associated XBODY transformation to position it in the global coordinate system. The MultiSurf model of a body can be constructed in any convenient position, e.g. aligned and centered in the body coordinate system. Alternatively, one MultiSurf model can be constructed with two or more bodies in their final global positions; this allows visualization of the bodies' relative and absolute positioning.

Mirror symmetry. If the body has mirror symmetry with respect to one vertical plane, only one half of the body needs to be constructed. If the body has mirror symmetry with respect to two orthogonal vertical planes, only one quarter needs to be constructed. These symmetry options are represented similarly in MultiSurf and WAMIT.

Rotational symmetry. Alternatively, rotational symmetry is supported. A MultiSurf model can have rotational symmetry about any one of the coordinate axes, with any number of copies (NCOPIES), at an angular spacing of 360/NCOPIES degrees. This

feature greatly simplifies modeling structures that have rotational symmetry; only 1/NCOPIES of the total model needs to be explicitly constructed.

Units. WAMIT is flexible with regard to units, but some of its inputs are quantities with units of length, and the global length unit is implicit in the GRAV parameter. MultiSurf supports meters, centimeters, millimeters, feet and inches as length units, and this choice is specified in the model file. The interface code compares the model and global units, and transparently performs any needed units conversions.

Parameters. The GDF file permits overriding floating-point values in the definition of any object in the model. This allows a single model (parametrically constructed) to be analyzed in a wide variety of configurations without opening the model and making any changes in MultiSurf.

Variables and formulas. Use of numeric variables and formulas in RG models is an advanced feature supported by MultiSurf versions 4.8 and higher. This represents a further powerful capability for constructing parametric model families. A variable can serve as a parameter in the GDF file.

Irregular frequency removal. Interior free-surface patches can be constructed in MultiSurf, and included in the data available to WAMIT for use with its IRR = 1 option.

3. Required files, versions and file locations

RG2WAMIT.DLL contains the Fortran-C interface code.

RGKERNEL.DLL is the relational geometry C library. RGKERNEL.DLL is a component of any MultiSurf or SurfaceWorks installation, however please NOTE: **The WAMIT-RGKernel interface may use a different RGKernel version from MultiSurf.** These DLL versions are not necessarily interchangeable, and should be kept in separate directories. Although they have the same filename, different DLL versions can be distinguished by different sizes and different dates.

(This said, it has been our uniform experience to date that a newer version of RGKERNEL.DLL can be safely substituted for an older one; RG2WAMIT.DLL does NOT have to be recompiled in this event, and new modeling features incorporated in the newer RGKERNEL.DLL become available for use with WAMIT.)

RG2WAMIT.DLL and RGKERNEL.DLL must be located in the same directory as WAMIT.EXE. Also, the GDF files and any model files named in IGDEF=2 GDF's must be located in this same directory.

MultiSurf does not need to be running when the WAMIT run takes place. WAMIT does not need to be running when the model is constructed in MultiSurf. Communication between MultiSurf and WAMIT is solely through the .MS2 model file created in MultiSurf and accessed by RG2WAMIT.DLL.

4. MultiSurf modeling considerations

4.1 TrimSurfs are excluded

Trimmed surfaces (TrimSurf and TrimSurf2 entities) cannot be handled by the RG2WAMIT facility. This is because WAMIT needs a complete parametric square over which to distribute its tensor-product description of the unknown potential, while a trimmed surface is by definition an arbitrary bounded *portion* of a parametric surface. RG2WAMIT checks for the presence of TrimSurfs in the wetted-surface ObjectList, and returns an error when one is encountered.

SubSurfs, on the other hand, are perfectly legal for RG2WAMIT, because they have (like all other RG surface types except TrimSurf) 4-sided topology and a full [0,1]x[0,1] parameter space. One or more SubSurfs can always cover a complex-shaped region of a base surface, where a single TrimSurf might otherwise be used.

4.2 Optional wetted-surface ObjectList (Entity List)

In order to be read and utilized through the WAMIT-RGKernel interface, an MS2 file ("model file") may include an ObjectList naming the wetted surfaces. When this option is used, WAMIT needs to be given the name of this ObjectList (in the GDF file, see below).

Note: In MultiSurf 6, an ObjectList is called an "Entity List". This is consistent with the following general terminology conventions:

MultiSurf 4 & 5		Surface Works and MultiSurf 6
"object"	=	"entity"
"entity"	=	"entity type"
"support"	=	"parent"
"dependent"	=	"child"

A simple alternative is to specify '*' for the ObjectList. This will signify "all visible surfaces in the model". To be visible, a surface has to have a positive visibility index, and be on an enabled layer. Surfaces that are not to be included in the hydrodynamic analysis can either be hidden (by editing their visibility attribute), or located on a layer that is disabled in the MS2 file (unchecked in Settings/Layers dialog).

ObjectList is an entity type that has been supported since MultiSurf 4.0. It is just a list of other objects, of any type. It is a non-graphical entity; you may or may not be able to see the objects in the list – that depends on whether each object is made from a graphical entity type, whether it is visible, and whether it is on a turned-on layer. But, the ObjectList itself is never drawn – it is just a list.

In MultiSurf 5, to create an ObjectList, use Create/ObjectList. (In MultiSurf 6, Insert/ Entity List.) You can preselect the "supports", i.e. the objects to be in the list. You can add or subtract objects just as you edit the multiple supports of any other object. Select/

By Name is often the best way to select an ObjectList – or find it in the Selection Set view Availables list, <F6>. (In MultiSurf 6, find it in the Entities Manager, under Entity Lists.)

This list must contain each wetted surface, and only the wetted surfaces, in your model (or in the half, quarter or sector you are modeling explicitly when using symmetry image options). If you leave a wetted surface out of the list, include a surface twice, or include a construction surface that should not be included in the hydrodynamic solution, you will very likely find volume discrepancies in WAMIT. The interface does not currently produce a warning in these situations.

You can have two or more ObjectLists in your model. They (like all other objects) will be distinguished by their unique names. By changing the ObjectList name specified in the GDF file, you can conveniently switch from one ObjectList to another without touching the model in MultiSurf. For example, you may have two sets of wetted surfaces representing two different drafts. Some surfaces can be common between these lists (e.g., surfaces that are fully wetted at the lightest draft), or the lists might be completely disjoint. (Note that in this scenario, you will need to also change XBODY when the draft changes.)

4.3 Order of patches

The ordering of patches for the WAMIT solution will often be immaterial, but in some situations it matters. For example, if you wish to use the NPATCH parameter in the GDF file to exclude one or more patches (such as a free surface patch, in order to compare results with and without irregular frequency removal), then you have to be sure that the patches to be excluded are the last in patch order.

There are two rules governing the order of patches, corresponding to the two alternatives for the ObjectList:

(1) If you use an explicit ObjectList, the patch order will be the order in which the surfaces are listed in the ObjectList.
(2) If you use the default identifier '*' for the ObjectList, the patch order will be the order of surfaces in the MS2 file. You can find out this order by searching for "Surf" in Edit/ Model File or Notepad. In MultiSurf another alternative is to turn on only the Surfaces filter; set Select/ Visible to "on"; View/ Selection Set; and read the order in the "Availables" list box.

4.4 Surface normal orientations

For a proper solution of the potential boundary-value problem, WAMIT requires that the positive normal to each surface be directed *into* the body. A patch closing the top of the body for irregular frequency removal must have its normal directed downward (again, into the body). The positive normal is calculated as the cross product $\partial\mathbf{x}/\partial u \times \partial\mathbf{x}/\partial v$, normalized to unit length.

If normals are backwards on some surfaces, WAMIT's calculation of volumes will very likely show discrepancies between the X, Y and Z directions. If all normals are reversed,

the volumes will agree but will be negative. And the hydrodynamic solution probably won't make much sense.

When making a surface in MultiSurf, the normal orientation is determined by:

(1) the orientations of supporting objects;
(2) the order in which the supports are used, and
(3) *the value of the "orientation" flag*.

The orientation flag is handy, because if a surface comes out with its normal opposite to what you want, you don't have to rebuild the surface another way, you just have to flip its "orientation" switch.

In MultiSurf, we visualize the normal orientation by drawing an arrow at the parametric center of the surface (u=0.5, v=0.5), in the positive normal direction. This arrow is drawn whenever a surface is selected. To see normal arrows simultaneously on all wetted surfaces, just select all the wetted surfaces; for example, select the ObjectList, then Select Expand ObjectLists/ First Generation. (The orientation of non-wetted construction surfaces doesn't matter, as these are not included in the surface definitions accessed by WAMIT.)

It is generally easier to see normals that point out of the geometry than those pointing inward. For this reason, our convention when making panel models for low-order WAMIT, and B-spline surface models for HIPAN and high-order WAMIT has always been to require outward normals on all wetted surfaces, and then to reverse all normals automatically in the conversion utility.

In the WAMIT-RGKernel interface, we have made the global reversal of normals optional. The choice is signaled (on a per-body basis) by the inward-normal flag in the GDF file (see below). This results in two options for the MultiSurf user who is building a model for the WAMIT-RGKernel interface:

(1) Build the model with all unit normals pointing inward (WAMIT's convention), and set the inward-normal flag to 1; or
(2) Build the model with all unit normals pointing outward, and set the inward-normal flag to 0.

We prefer option (2), because the outward normals are easier to see. In either case, the orientation flags of the individual surfaces can be freely used in achieving the desired consistent orientations.

4.5 Base plane and waterline

In MultiSurf, there are no restrictions on the location and orientation of geometry, but there are some sensible choices that will make the process easier.

Vertical position: Most models for WAMIT analysis are built in one of three vertical positions. Either

(1) The model is built above a base plane, so the lowest Z coordinates are zero, and the design waterline is at some positive Z (call it Zwl > 0); or
(2) The model is built with Z=0 as the design waterline, so the lowest parts of the wetted surface are at a negative Z (call it Zdraft < 0); or
(3) The model is built with the point representing its center of gravity (C.G.) at the origin.

Any of these approaches works fine. In the first case, you have to specify XBODY(3) = -Zwl to correctly position the body in WAMIT; in the second case, XBODY(3) will be 0; in the third case, XBODY(3) will be the height of the C.G. above or below the waterline.

The WAMIT-RGKernel interface tests the "wetted surfaces" (i.e., the surfaces in the named ObjectList, or when * is used for the ObjectList, the visible surfaces) during its initialization, and reports (in RGKLOG.TXT) any case where part of a "wetted surface" extends above the global waterplane. This usually indicates an error, either in the surfaces specified as wetted, or in XBODY(3).

4.6 Mirror symmetry

If the body has one or two planes of complete mirror symmetry, it is very advantageous to model only the independent half or quarter, and use mirror symmetry options to get the rest.

The WAMIT-RGKernel interface checks the model's symmetry flags against the ISX, ISY flags specified in the GDF file, and issues a warning if there is any discrepancy. Sometimes this can be harmless, but you should check your model out to be sure you haven't either left surfaces out, or covered some areas twice.

4.7 Rotational symmetry

An object has N-fold rotational symmetry with respect to a particular axis if it is congruent to itself after a rotation of 360 / N degrees about that axis.

Many offshore structures have complete rotational symmetry about a vertical axis. For example, three-leg TLP's may have 3-fold rotational symmetry; spars and buoys often have rotational symmetry of some order; the Hibernia bottom-mounted platform is a 16-pointed symmetric star. (Four-leg TLP's also usually have 4-fold rotational symmetry, but this case is best treated as combined X and Y mirror symmetry, because of the large efficiencies that result from exploiting mirror symmetry during the WAMIT solution.)

In MultiSurf, when the object being modeled has complete rotational symmetry, it is very beneficial to take advantage of it, as only a fraction (1/N) of the total structure then needs to be explicitly modeled. Rotational symmetry is a model-level property. Z-axis rotational symmetry is only allowed when X- and Y-mirror symmetry flags are OFF. In the Settings/Model dialog, check X-, Y- and Z-mirror symmetry OFF, Z-axis rotational symmetry ON, and enter the number of copies, e.g. 3 for a 3-leg TLP. Then you can model only the "active" or "independent" sector of the model (e.g. 1 leg and 2 half-

pontoons of a 3-leg TLP), and the copies will be present implicitly. The symmetry images will be shown automatically in Render view, and can be toggled on and off in Wireframe view with the <F5> key.

In the GDF file for a body using rotational symmetry about the Z-axis, ISX and ISY will be 0, and NPATCH needs to include the symmetry images. For example, if a 3-leg TLP is modeled with 11 wetted surfaces in the explicitly modeled 120-degree sector, NPATCH will be 3 x 11 = 33.

Rotational symmetry about the X- or Y-axis is also supported – but only one axis at a time. (In WAMIT, this would have to be a completely submerged body, positioned vertically with XBODY(3).)

Rotational symmetry that is incomplete precludes the use of model-level rotational symmetry. In this situation, the entire geometry has to be built explicitly. But note, the RotatSurf and CopySurf entity type often provide a highly efficient way to construct the portions that are rotationally symmetric.

4.8 "Fast" vs. "Accurate" evaluation

RGKernel has two evaluation modes, denoted "Fast" and "Accurate". In "Accurate" evaluation, all curves and surfaces are evaluated recursively, using the actual math functions that provide their definitions. In "Fast" evaluation, curves and surfaces are evaluated approximately by interpolation in a stored tabulation of the curve or surface; the fineness of this tabulation is controlled by the divisions x subdivisions products.

Fast vs. accurate evaluation is specified in the GDF file for each body (see below).

Accurate evaluation is typically slower -- in some cases, orders of magnitude slower; this especially occurs when there are deep levels of dependency in the model, and/or the model involves entities requiring iterative solutions, such as ProjSnakes and IntSnakes. The divisions and subdivisions should have no effect on the accuracy under "accurate" evaluation, since the table is not involved in the calculations.

Fast evaluation is usually much faster, and always more predictable in terms of performance, but involves some compromise of accuracy due to interpolation errors. Our general experience is that with the default 8x4 divisions x subdivisions, most curves and surfaces evaluate to near single-precision accuracy under fast evaluation. The accuracy should improve as the fourth power of the divisions, provided the divisions on all supporting curves and surfaces are increased in constant proportion. The "divisions multiplier" is a simple way to make this increase uniformly across the model.

The GDF file format provides for overriding the model's divisions multiplier (see below).

Pending further investigation of the accuracy of the integrated system, our current recommendation is to use fast evaluation mode, with default divisions and a modest

divisions multiplier of 2, 3, or 4. This should have little impact on performance, and should provide WAMIT with full single-precision accuracy for the geometry.

4.9 Divisions and Subdivisions

In MultiSurf, each surface has "division" and "subdivision" attributes that control how the surface is subdivided for tabulation and display. For low-order WAMIT panelizations, we use the divisions x subdivisions to determine the mesh density, and some care with divisions was required to make neat, watertight junctions between the panels on adjacent surfaces.

Since the surface geometry is more directly accessed in the WAMIT-RGKernel interface, divisions and subdivisions are less important in general. In "Accurate" mode evaluation, they should have no bearing at all. However, in "Fast" mode, there are still accuracy benefits in coordinating divisions between adjacent surfaces: for example, use the same number of (divisions x subdivisions) on a surface and its supporting curves; where possible, use matching divisions on surfaces that share a common edge.

4.10 Irregular frequency removal

In the WAMIT 6.1 higher-order solution, irregular frequency removal is effected by providing additional surface patches that cover any "interior" portions of the plane of the free surface (IRR = 1).

Such interior free surface patches can be part of the MultiSurf model. They must be oriented with their positive normal direction consistent with the wetted patches; i.e., if your model has all outward normals, an interior free surface patch must have its normal upward. Interior free surface patches must also be included in the ObjectList of wetted surfaces, and in NPATCH. If you put them last in the ObjectList, you can easily run WAMIT with and without irregular frequency removal just by setting NPATCH appropriately in the GDF file.

4.11 Coordinate singularities

A coordinate singularity is a place on a parametric surface where the cross product $\partial\mathbf{x}/\partial u \times \partial\mathbf{x}/\partial v$ vanishes. This can occur because either $\partial\mathbf{x}/\partial u$ or $\partial\mathbf{x}/\partial v$ vanishes, or because they have the same directions. The most commonly encountered coordinate singularity is a "pole", where one or the other of $\partial\mathbf{x}/\partial u$, $\partial\mathbf{x}/\partial v$ vanishes along one whole edge of the parameter space, e.g. the type of singularity at the north and south poles of a sphere that is parameterized by latitude and longitude. There are many ways to create surfaces with poles and other coordinate singularities in MultiSurf; for example when a RuledSurf is made between a curve and a point, a pole occurs at the point.

Some kinds of coordinate singularities are associated with numerical ill-conditioning in WAMIT, and so should be avoided as much as possible in modeling for RG2WAMIT. There are situations where coordinate singularities are apparently harmless, e.g. the south pole of a half-submerged sphere, or the "south pole" of a truncated vertical

cylinder. There are other situations where a coordinate singularity is distinctly beneficial. (See the remarks below under "Cosine spacing".)

WAMIT, Inc. is recommending especially against poles on the free surface or at "chines", i.e. sharp edges that protrude into the water (such as the junction between side and bottom surfaces of a truncated vertical cylinder).

4.12 Breaklines in surfaces

The RGKernel versions used in RG2WAMIT and MultiSurf 5 and 6 recognize the presence of "breakpoints" in curves and "breaklines" in surfaces. A breakpoint is a place where a curve has a discontinuous derivative; if this is the first derivative (a "degree-1 breakpoint") the curve typically has a knuckle. A "breakline" is a u- or v-constant line across which the surface has a discontinuous derivative; if this is the first derivative, the surface typically has a knuckle line at this parameter value.

Thus one reasonable way to build a truncated vertical cylinder is to construct a meridian curve with a breakpoint at the "chine" (e.g., a type-1 BCurve or CCurve), and revolve it 90 degrees about a vertical axis, creating a single RevSurf covering both the bottom and sides. MultiSurf will be aware of the degree-1 breakline along the chine and will display the sharp edge correctly.

However, WAMIT will not be aware of the breakline and will be modeling a smooth, continuous distribution of potential across the chine.

Therefore, we recommend in general that breaklines **not** be used on surfaces intended for WAMIT consumption. In this example, modeling the side and bottom surfaces as two separate surfaces (rather than one surface with a breakline) will make it much easier to utilize cosine spacing, to better resolve the rapidly changing potential around the chine.

4.13 "Cosine spacing"

The phrase "cosine spacing" refers to systematic refinement of the mesh in regions where gradients are high, to provide better resolution in these areas. Although we don't have a mesh anymore in the usual sense of a panel file, the concept of cosine spacing is in fact still quite pertinent, and the techniques for achieving it are highly similar to those used for the low-order method.

Exterior corners ("chines") are the typical places where cosine spacing improves solution accuracy. Also, accuracy benefits are experienced with cosine spacing near the free surface. In MultiSurf there are two common techniques for achieving mesh concentration:

(1) Relabeling of lines, curves, snakes and surfaces
(2) Type-2 B-spline lofted surfaces (BLoftSurfs) with duplicate master curves at the ends; also (usually for flat rectangular patches) B-spline surfaces (BSurfs) with duplicate control points at corners and/or edges.

4.14 Parameters

As described below, the GDF file can contain any number of "parameter lines", which override specified floating-point data values in the model. The format of a parameter line is just:
object-name float-index value

For example, if AbsPoint 'draft_pt' is a point whose Z coordinate (its third floating-point data value) controls the draft of the model, the parameter line:
draft_pt 3 -35.5

sets the draft to 35.5.

Obviously, carelessly setting parameters can "break" the MultiSurf geometry. For example, the leg radius and pontoon radius for a TLP might be separate parameters. As long as the pontoon radius is smaller than the leg radius, the geometry works; but if pontoon radius is set larger than the leg radius, a projection will fail, and the pontoon surface can't be evaluated. This failure will be reported in RGKLOG.TXT. If you try the same settings in MultiSurf, you can see what goes wrong.

Even when all surfaces evaluate geometrically, there can be modeling problems that affect the WAMIT solution. When the pontoon radius is somewhat smaller than the leg radius, it's possible for the two pontoons arriving at the leg to intersect each other, which may or may not cause a recognizable error condition in WAMIT. It is thus a good idea to check all parameter combinations that you plan to use visually in MultiSurf, before starting the WAMIT solutions.

5. GDF file format

A GDF file represents geometry for a single body. WAMIT's GDF file has a new format option, with IGDEF = 2 specifying "geometry to be obtained from an MS2 model file". The first four lines of the GDF are the same as in other optional GDF formats:

header
ULEN GRAV
ISX ISY
NPATCH IGDEF

Where:
header is an identifying character string, up to 72 chars.
ULEN is a characteristic length
GRAV is acceleration due to gravity
ISX, ISY are symmetry flags (0 or 1) for X and Y mirror symmetry
NPATCH = number of patches in this body (not counting X and Y mirror symmetry images, but counting Z-axis rotational symmetry images)
IGDEF = 2, for using geometry from an .MS2 file

In an IGDEF=2 GDF file, there are always 4 more lines:

Line 5: NLINES = no. of lines to follow this line (3 plus number of parameter lines)
Line 6: Model filename, including extension .MS2 (no embedded spaces)
Line 7: Object name of ObjectList to be present in the named MS2 file, listing the wetted surfaces. (Alternative: use '*' to signify "all visible surfaces" – and make sure only the wetted surfaces are visible in your model when the .MS2 file is saved.)
Line 8: 3 integers:
Fast/accurate flag (0 = fast, 1 = accurate);
Divisions multiplier override (0 to 10). A 0 for DivMult will mean "use the value in the model file".
Inward-normal flag (0 = outward normals, 1 = inward normals)

Then there are (NLINES – 3) optional parameter lines, each with 3 items:
object-name float-index value

object-name is the name of an object in the model that has one or more floating-point numbers in its object data. For example, an AbsPoint has 3 floats -- its X, Y and Z coordinates --; an AbsBead has just one, its t parameter.

float-index is an integer from 1 to the number of floats the object has, e.g. 1 to 3 for an AbsPoint; 1 to 1 for an AbsBead.

value is the floating-point value to substitute.

Note re symmetry: The actual number of patches produced internally in WAMIT by this GDF is:
(number of wetted surfaces) x (ISX+1) x (ISY+1) x N
The interface code does not need to count the X- and Y-symmetry images in indexing the patches. All mirror-image and XBODY transformations, and mirror-image indexing, are performed inside WAMIT. However, NPATCH does need to count implicit rotational images produced by Z-axis rotational symmetry.

Note re Fortran READS: Fortran will read this GDF with (5+NLINES) READ statements, each reading a complete record. This means any of the records can have other text following the required entries, separated only by one or more spaces. This additional data has no effect in Fortran, and is likewise being ignored by C.

Example GDF:

```
Tension-leg platform example
120. 32.2  ULEN, GRAV
1  1  ISX,ISY
16  2   NPATCH, IGDEF
3   NLINES
TLP4H.MS2
wetted_surfs
```

```
0  2  0 Fast; DivMult; outward normals
```

Explanation: The first line is an identifying message. ULEN is 120 and GRAV is 32.2. This value of GRAV implies that the length unit for the global WAMIT model is feet. (If this MS2 file uses meters for its units, conversion from meters to feet will be performed in RGKINIT.) The model has mirror symmetry with respect to its X and Y planes (only one quadrant is explicitly modeled). NPATCH is 16, the number of patches in one quadrant of the body. IGDEF is 2 to signify geometry from an MS2 file. 3 for NLINES indicates 3 lines to follow. The MS2 file is TLP4H.MS2. This model file must contain an ObjectList named 'wetted_surfs' which names the wetted surfaces. The last line specifies Fast evaluation; a divisions multiplier of 2, overriding any divisions multiplier in the model file; and the model is constructed with outward normals. There are no parameter lines in this example.

6. Log file RGKLOG.TXT

During operation of the WAMIT-RGKernel interface, a log file is opened and written to in the same directory as WAMIT.EXE. This file, named RGKLOG.TXT, contains an echo of XBODY and GDF data, a reflection of some information extracted from the MS2 model file(s), brief reports on some aspects of interface initialization and utilization, and possible warning and error messages.

RGKLOG.TXT also includes a summary tabulation of surfaces vs. global, body and patch indices, which will give the correct order of surfaces for constructing an SPL file, if desired.

Example:

```
RGKLOG.TXT -- log file for RG2WAMIT.DLL
=======================================================
Starting procedure RGKINIT at 25-Nov-2001  13:59:49
NBODY: 1
-------------------------------------------------------
Data for body no. 1
XBODY:    0.000   0.000   0.000   0.000
Truncated cylinder example 1.0 m radius, 0.5 m draft
ULEN:   1.000  GRAV:  9.80660
Global length unit (deduced from GRAV) is m.
ISX: 1  ISY: 1
NPATCH: 2  IGDEF: 2
MS2 filename: TRCYL2.MS2
ObjectList of wetted surfaces: wetted_surfs
Model will be evaluated in FAST mode.
Unit normals are outward.
Opened RG model file: TRCYL2.MS2
Body 1 patch 1 surface name: side_surf
Body 1 patch 2 surface name: bottom_surf
-------------------------------------------------------
index  body  igdef  patch  ior  surface name
    1     1      2      1    1  side_surf
```

```
    2     1      2      2    1  bottom_surf
-------------------------------------------------
Exiting RGKINIT at 25-Nov-2001  13:59:49, err = 0
=================================================
There were 1 RGKEVAL calls.
Executing RGKDONE at 25-Nov-2001  13:59:49
```

7. Synopsis of operation

The user generally will not need to know more about the functioning of the RG2WAMIT interface than has been explained in the operating instructions above. However, a little more perspective may be useful in case error conditions are encountered or suspected.

The sequence of communication between the programs is as follows:

1. During WAMIT's initialization process, it opens and writes to an ASCII text file named RGKINIT.TXT. This file is located in the same directory as WAMIT.EXE. It is created for each WAMIT run, whether or not the run uses IGDEF = 2 GDF's. RGKINIT.TXT contains, for each body, XBODY(1:4) plus the first 4 lines of any GDF; in addition it contains the remainder of the GDF file whenever IGDEF = 2.
2. WAMIT calls the procedure RGKINIT in RG2WAMIT.DLL. This initializes the interface, by using information read from RGKINIT.TXT. All specified model files are opened and prepared for evaluation calls.
3. WAMIT makes as many calls as it needs, to the procedure RGKEVAL in RG2WAMIT.DLL. Each such call evaluates one 3-D point at a specified u,v parameter location on one of the wetted surfaces furnished by the RG model.
4. When finished with its RGKEVAL calls, WAMIT makes one call to the procedure RGKDONE in RG2WAMIT.DLL. This frees all memory that was allocated during the RGKINIT procedure, storing data about the open models.

8. Error Conditions

There are a number of error conditions that can occur during the initialization phase of RG2WAMIT operation. If one of these occurs, there will be a message at the end of RGKLOG.TXT, and the WAMIT run will abort. The contents of RGKLOG.TXT will often be helpful in identifying the problem.

(Certain errors are also trapped during the WAMIT equation setup phase, but this error handling is primarily for debugging purposes; in our opinion such errors should not occur once the initialization phase has completed successfully.)

RGKINIT error codes

0. No error

1. Can't open RGKINIT.TXT.

This probably indicates that RG2WAMIT.DLL and/or RGKERNEL.DLL are in the wrong directories. They must both be installed in the same directory as WAMIT.EXE.

2. **Unexpected end-of-file in RGKINIT.TXT.**
   The initialization file did not contain all required information. RGKINIT.TXT is written by WAMIT, and should not be modified by the user. If the file is incomplete, there's likely a bug in WAMIT. If the file appears complete, there's likely a bug in RG2WAMIT.DLL.

3. **Can't open a specified model file.**
   A model file named in an IGDEF=2 GDF file was not found. Check to see that the file exists in the same directory as WAMIT.EXE. Check to see if the file name was spelled correctly in the GDF. Blanks are not allowed in filenames for RG2WAMIT. (NOTE: Filenames are not case sensitive in Windows.)

4. **The specified ObjectList was not found in the model file.**
   An ObjectList of wetted surfaces may be specified by name in the GDF file. Object names in RG are case sensitive and must use an underscore '_' in place of a space. Check to see that the ObjectList name is correctly spelled and correctly capitalized in the GDF. Open the MS2 file in either MultiSurf or Notepad and see that the ObjectList is there, and you have given its correct name in the GDF. You can find the ObjectList in the Selection Set view (<F6>), Availables list, or use Select/ By Name.

5. **ObjectList has an error**
   The RGKernel error code will be given. This is most likely to occur because one of the surfaces failed to evaluate, and the ObjectList got an error 284 (support failed) as a result. If the .MS2 opens in MultiSurf without error or warnings, it is very likely to open and evaluate successfully in RG2WAMIT. However, the two RGKernel.dll versions will generally be different and will have some incompatibilities, so it is possible for this error to occur through no fault of the user. This situation should be investigated by AeroHydro.

6. **Failed to build surface table**
   Open the MS2 file in MultiSurf and confirm that the surface in question evaluates without error. If the surface evaluates without error in MultiSurf, it is very unlikely to fail in RG2WAMIT. However, because of some incompatibilities between RGKernel.dll versions, this error might occur through no fault of the user. AeroHydro should investigate. (Note: a table is built for each surface in the model even when "Accurate" evaluation is specified.)

7. **Unexpected end-of-line in RGKINIT.TXT**
   A file record in RGKINIT.TXT did not contain all required data. For example, if the file line giving fast/accurate, divisions multiplier, and inward-normal settings has only one or two tokens, this error will occur. Check that your GDF file contains all required information, and that NLINES is correct.

8. **ObjectList has fewer than NPATCH surfaces**
   The GDF file specifies NPATCH, the number of patches to use in the solution for a given body; the ObjectList does not contain the full complement. Check that NPATCH is correctly specified in the GDF. Open the MS2 file in either MultiSurf or Notepad and check that all wetted surfaces and/or interior free surface patches are in the ObjectList. If the number of surfaces is large, making counting difficult, select the ObjectList in MultiSurf (usually Select/ By Name is the easiest way), edit its attributes, edit the supports ("Objects"), and the number of them will be indicated in the Selection Set view (F6); cancel the edit if no changes are needed. Visually, you can select the ObjectList and Select Support/ First Generation; all surfaces in the list will then be highlighted.

9. **ObjectList contains a non-surface object.**
   The wetted-surface ObjectList must contain all the wetted patches and/or interior free-surface patches, and no other objects. (An ObjectList, in general, can legally contain objects of any type.) Check the contents of the ObjectList and see that they are correct in number and identity.

10. **ObjectList includes a TrimSurf.**
    TrimSurfs are not eligible for use in WAMIT models, because they do not provide a complete square parameter space for distribution of the unknown potential. You can replace a TrimSurf with one or more SubSurfs.

11. **Surface object is in error.**
    The RGKernel error code will be given. This error code is compatible with the MultiSurf error system, so can be looked up in the MultiSurf reference manual or help system under "Error codes". The same error should occur when the model is opened in MultiSurf; however, this will not always be true, because of the presence of incompatibilities between the two RGKernel.dll versions. See also remarks for error 6.