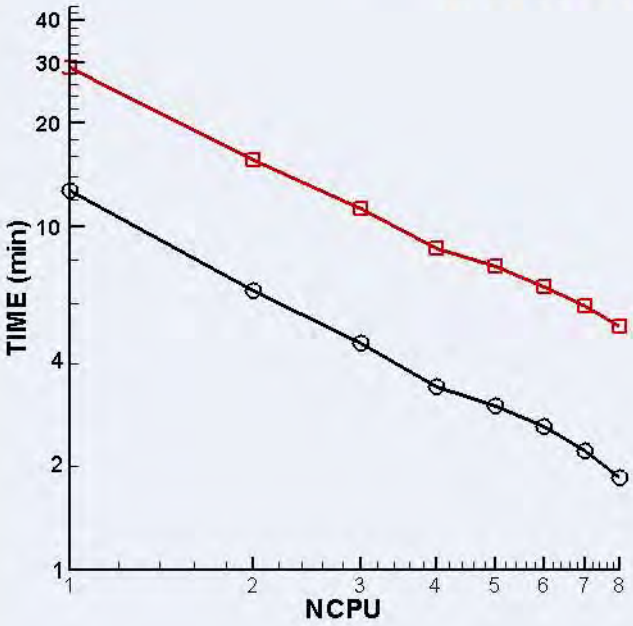
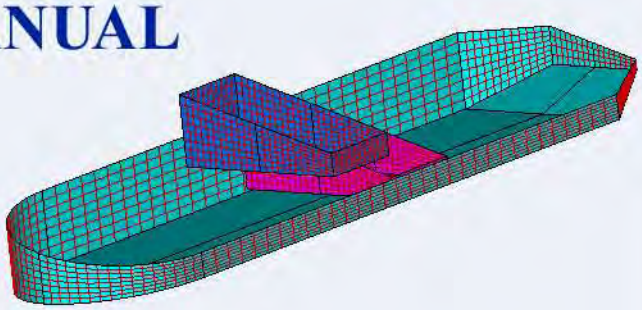
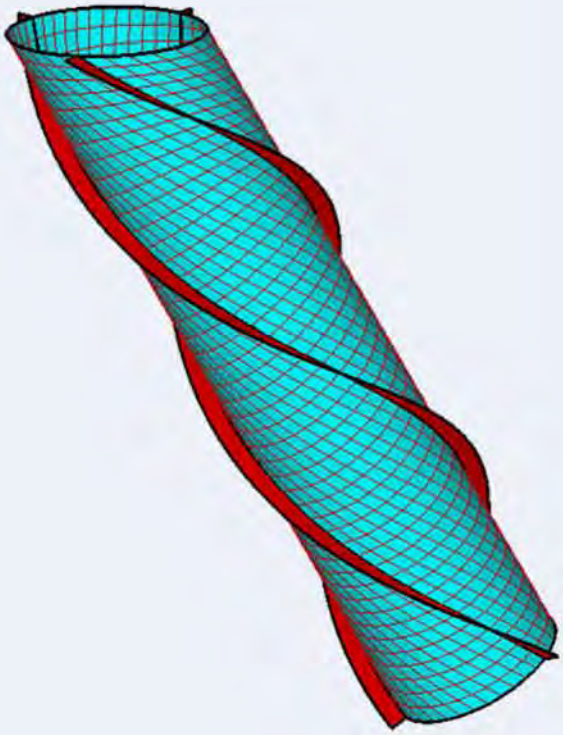
WAMIT INC

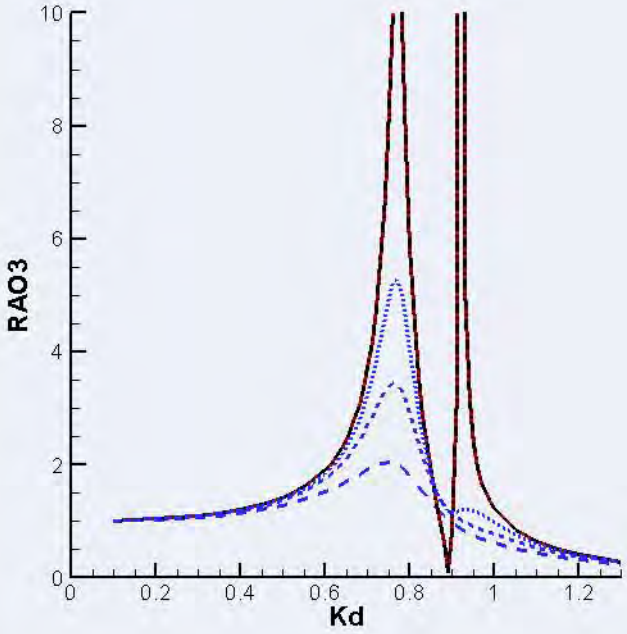
The State of the Art in Wave Interaction Analysis

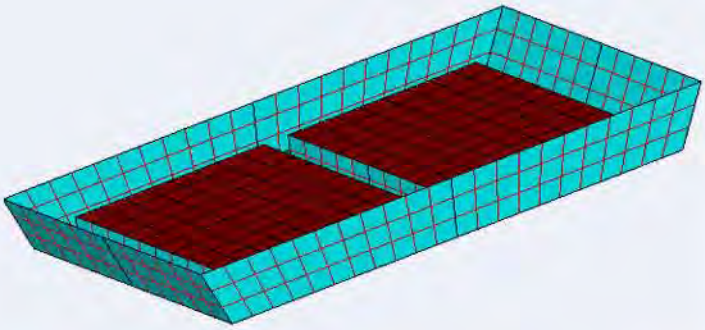
USER MANUAL











WAMIT, INC.

www.wamit.com

WAMIT®

USER MANUAL

Version 7.0

This User Manual has been prepared for users of WAMIT Version 7.0.

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.

The WAMIT software and this User Manual are Copyrighted by WAMIT, Inc., and by the Massachusetts Institute of Technology. The User Manual can be downloaded from the website without a license.

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

Copyright© 1998-2012 WAMIT, Incorporated and Massachusetts Institute of Technology. No part of the WAMIT software or documentation may be reproduced, translated, or transmitted in any form without the express written permission of WAMIT, Inc.

WAMIT is a registered trademark of WAMIT, Inc.

Intel is a registered trademark of Intel Corporation.

Windows is a registered trademark of Microsoft Corporation.

MultiSurf and RGKernel are registered trademarks of AeroHydro, Inc.

Tecplot is a registered trademark of Amtec Engineering, Inc.

Revised 1 October 2012. Corrected and updated to WAMIT Version 7.03.

TABLE OF CONTENTS

1. INTRODUCTION
   - 1.1 WAMIT Version 7
   - 1.2 Changes introduced in Version 7.0
2. GETTING STARTED
   - 2.1 Installation and setup
   - 2.2 Demonstration programs
   - 2.3 Standard test runs
   - 2.4 Running Test 01
   - 2.5 Running Test 11
   - 2.6 Other test runs
   - 2.7 Using multiple processors
   - 2.8 Memory and storage restrictions
   - 2.9 Modifying the input files
3. DEFINITION OF QUANTITIES EVALUATED BY WAMIT
   - 3.1 Hydrostatic data
   - 3.2 Added-mass and damping coefficients
   - 3.3 Exciting forces
   - 3.4 Body motions in waves
   - 3.5 Hydrodynamic pressure
   - 3.6 Free-surface elevation
   - 3.7 Velocity vector on the body and in the fluid domain
   - 3.8 Mean drift force and moment
   - 3.9 Zero and infinite wave periods
4. INPUT FILES
   - 4.1 Summary of changes in Version 7 input files
   - 4.2 The Potential Control File
   - 4.3 The Force Control File (Alternative form 1)
   - 4.4 The Force Control File (Alternative form 2)
   - 4.5 Definition of fixed or free modes
   - 4.6 Body pressure and fluid velocity at specified points
   - 4.7 Configuration files CONFIG.WAM and ∗.CFG
   - 4.8 Filenames list FNAMES.WAM
   - 4.9 File names
   - 4.10 File format
   - 4.11 Uniform arrays of field points
   - 4.12 Using the optional file BREAK.WAM
   - 4.13 Assigning RAO'S in an external file
   - 4.14 Evaluating FORCE outputs in POTEN (IFORCE=2)

5. OUTPUT FILES
5.1 The formatted output file (.out)
5.2 Numeric output files
5.3 Froude-Krylov and scattering forces
5.4 Body pressure for the higher-order method
5.5 Body pressure and velocity at specified points
5.6 Hydrostatics output file (.hst)
5.7 Auxiliary output files for the geometry
5.8 Error messages
5.9 The log file (wamitlog.txt)
5.10 The intermediate data transfer file (.p2f)
6. THE LOW-ORDER METHOD (ILOWHI=0)
6.1 The Geometric Data File (GDF)
6.2 Use of the source formulation (ISOR=1)
6.3 Bodies with thin submerged elements
7. THE HIGHER-ORDER METHOD (ILOWHI=1)
7.1 Subdivision of the body surface in patches and panels
7.2 B-Spline representation of the solution
7.3 Order of Gauss Quadratures
7.4 The Geometric Data File
7.5 Geometry represented by low-order panels (IGDEF=0)
7.6 Geometry represented by B-Splines (IGDEF=1)
7.7 Geometry represented by MultiSurf (IGDEF=2)
7.8 Analytic representation of the geometry
7.9 Modifying the DLL Subroutine GEOMXACT
7.10 Bodies with thin submerged elements
7.11 The Optional Spline Control File
7.12 The use of default values to simplify inputs
7.13 Advantages and disadvantages of the higher-order method
8. ANALYSIS OF MULTIPLE BODIES (NBODY > 1)
8.1 Input to FORCE (IALTFRC=1)
8.2 Input to FORCE (IALTFRC=2)
8.3 Input to FORCE (IALTFRC=3)
8.4 Parameters in the configuration files
8.5 Global symmetry indices
8.6 Output
9. GENERALIZED BODY MODES (NEWMDS > 0)
9.1 Input Files
9.2 Using DEFMOD with the Low-Order Method
9.3 Using the DLL Subroutine NEWMODES
9.4 Hydrostatics
9.5 NBODY Analysis
10. USE OF IRREGULAR FREQUENCY OPTION (IRR > 0)
10.1 Input parameters
10.2 Automatic free-surface discretization (IRR=2 and ILOWHI=0)

10.3 Automatic free-surface discretization (IRR=3 and ILOWHI=0)
10.4 Automatic free-surface discretization (IRR=3 and ILOWHI=1)
10.5 Assigning different values of IRR for NBODY>1

11. MEAN DRIFT FORCES USING CONTROL SURFACES
11.1 Control surface file (CSF)
11.2 Low-order control surface file
11.3 Higher-order control surface file
11.4 Combining two control surface files
11.5 Automatic control surfaces
11.6 Output

12. SPECIAL EXTENSIONS
12.1 Internal tanks
12.2 Trimmed waterlines
12.3 Radiated waves from wavemakers in tank walls
12.4 Bodies and wavemakers with vertical walls
12.5 Bodies with pressure surfaces

13. THE F2T UTILITY
13.1 Definitions of radiation and diffraction outputs
13.2 Acquiring input data for F2T with WAMIT
13.3 How to use F2T
13.4 Output files
13.5 Options 5 and 6
13.6 Theory
13.7 Dimensional input and output data

14. COMPUTATIONAL TOPICS
14.1 Number of equations (NEQN) and left-hand sides (NLHS)
14.2 Solution of the linear systems
14.3 Temporary data storage
14.4 Data stored in RAM
14.5 Data stored in scratch files
14.6 Multiple processors (NCPU>1)
14.7 Modifying DLL files
14.8 Reserved file names

15. THEORY
15.1 The boundary-value problem
15.2 Integral equations for the velocity potential
15.3 Integral equations for the source formulation
15.4 Discretization of the integral equations in the low-order method (ILOWHI=0)
15.5 Discretization of the integral equations in the higher-order method (ILOWHI=1)
15.6 Removal of irregular frequencies
15.7 Integral equations for bodies with thin submerged elements
15.8 Mean drift forces based on pressure integration
15.9 Mean drift forces using control surfaces
15.10 Internal tank effects
15.11 Pressure surfaces

REFERENCES

APPENDIX A – DESCRIPTION OF TEST RUNS

A.1 Truncated vertical cylinder (TEST01)
A.2 Irregular-frequency removal (TEST02)
A.3 Local pressure drift force (TEST03)
A.4 Body near a wall (TEST04)
A.5 Multiple bodies (TEST05)
A.6 The ISSC Tension-Leg Platform (TEST06)
A.7 The ISSC Tension-Leg Platform (TEST07)
A.8 Elastic column with generalized modes (TEST08)
A.9 Spar with three strakes (TEST09)
A.11 Higher-order analysis of circular cylinder (TEST11)
A.12 Irregular-frequency removal (TEST12)
A.13 Multiple bodies (TEST13)
A.14 ISSC TLP (TEST14)
A.15 Semi-Sub (TEST15)
A.16 Barge with bending modes (TEST16)
A.17 Cylinder with moonpool (TEST17)
A.18 Elastic column (TEST18)
A.19 Catamaran barge (TEST19)
A.20 MultiSurf Barge (TEST20)
A.21 Spar with three strakes (TEST21)
A.22 FPSO with two internal tanks (TEST22)
A.23 Radiated wave field from a bank of wavemakers (TEST23)
A.24 Motions of a hinged vessel (TEST24)
A.25 Air-cushion vessel with pressure chambers (TEST25)

APPENDIX B – FILE CONVERSION USING THE UTILITY `V6V7inp`

APPENDIX C – USING THE WAMIT-RGKERNEL INTERFACE (By J. S. Letcher, Jr.)

Chapter 1

INTRODUCTION

WAMIT Version 7 is a radiation/diffraction panel program developed for the linear analysis of the interaction of surface waves with various types of floating and submerged structures. WAMIT Version 6.4S, which is described in a separate User Manual [29][1], performs the extended analysis for the second-order solution in bichromatic and bidirectional waves, including sum- and difference-frequency components.

Version 7 has been developed to exploit important features of contemporary computing systems, especially in the PC environment. For systems with relatively large random-access memory (RAM) and with multiple processors (CPUs, also known as 'cores'), Version 7 is developed to take advantage of these features with substantial reductions of the computing time in many applications. Another important development in this context is the 64-bit operating system, which is essential for data access with large RAM.

The remainder of this Chapter gives a general description of WAMIT Version 7, and changes made from earlier versions. Users of earlier versions should refer particularly to Section 1.2 which lists the changes introduced in Version 7.0. A mark in the left margin, as on this line, is used throughout this User Manual to call attention to changes in Version 7.0.

WAMIT includes options to use either the traditional low-order panel method or a more versatile higher-order method based on B-splines. 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 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 users get started using WAMIT in the PC/Windows environment. The examples described in Chapter 2 are for the simplest context of a single body.

[1] Throughout this User Manual numbers in square brackets refer to the references listed after Chapter 15

Chapter 3 defines the various quantities which can be evaluated by WAMIT, and which are contained in the output files.

Chapter 4 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 files `fnames.wam`, `config.wam` and `break.wam`, which are useful to specify input filenames and various parameters or options.

Users of Version 6 should note that some changes have been made which are intended to make the input data more consistent. As a result, Version 7 may require modifications of old input files, as explained in Section 4.1. Appendix B describes the utility `v6v7inp`, which has been prepared to automate the conversion of old input files.

Chapter 5 describes the output files, which contain the principal data computed by the program as well as log files, error files, and auxiliary files which provide useful information regarding the geometry of the structures.

Chapter 6 describes topics which are specific to the low-order method. These include the low-order 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, and the analysis of bodies with thin elements such as damping plates or strakes.

Chapter 7 describes topics which are specific 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.

Chapters 8-12 describe several extended features in WAMIT. These include the analysis of multiple interacting bodies (Chapter 8), the use of generalized modes of body motion which can be used to describe structural deformations, motions of hinged bodies, etc. (Chapter 9), and the use of a method to remove the effect of the irregular frequencies (Chapter 10). Chapter 11 describes the procedure for evaluating the mean drift forces and moments by integration of the momentum flux on a control surface which surrounds the body in the fluid. Chapter 12 describes additional extensions to include the dynamics of fluid in internal tanks, trimmed waterlines, radiated waves from wavemakers, interactions of bodies and wavemakers with vertical walls, and applications where part or all of the body surface consists of free surfaces with oscillatory pressures.

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

Chapter 14 describes various computational topics including temporary data storage, numbers of unknowns, and input parameters, to provide a qualitative basis for estimating the requirements for RAM and hard disk storage, and for estimating run times. Instructions are provided for using multiple processors and extended RAM to reduce run times. Section

14.7 outlines the procedure for modification and use of .dll files to describe the geometry and generalized modes. Section 14.8 lists the reserved filenames used by the program.

A brief outline for the theoretical basis of WAMIT is presented in Chapter 15. Reference 26 contains a more complete review of the pertinent theory.

A list of relevant references is included after the final chapter. Appendix A includes descriptions of the standard test runs. Appendix B documents the use of the utility `v6v7inp` for converting Version 6 input files. Appendix C describes the use of the interface between WAMIT and the MultiSurf kernel.

1.1 WAMIT Version 7

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 15. The water depth can be infinite or finite. 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. In addition to the conventional case where the bodies are rigid, and moving with six modes of rigid-body motion, WAMIT permits the analysis of 'generalized modes' to represent structural deflections, motions of hinged vessels, devices for wave-energy conversion, etc. Part or all of the boundary surface can be defined as a free surface with an oscillatory pressure, as in the case of an air-cushion vehicle or oscillating water column.

The flow is assumed to be ideal and time-harmonic. The free-surface condition is linearized (except in Version 6.4S 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.

The first versions of WAMIT (up to and including Version 5) were based entirely on the low-order panel method. There 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. Starting with Version 6 WAMIT was extended to include as an alternative option a higher-order panel method based on a continuous B-spline representation for the velocity potential, and several alternative schemes for defining the geometry of the body surface. 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* refer to the method for representation of the body surface and

solution.

The following quantities can be evaluated by WAMIT:

- Hydrostatic coefficients
- Added-mass and damping coefficients for all modes
- 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
- All components of the drift force and moment by momentum integration over a control surface
- Horizontal drift forces and mean yaw moment by momentum integration in the far-field
- All components of the drift force and moment by local pressure integration over the body surface
- Drift force and moment in bidirectional waves

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.

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 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. Version 7 has been developed to exploit the additional features of multiple processors, 64-bit operating systems, and optimum use of available RAM. In combination these result in a fast, versatile, and robust code capable of analyzing a wide variety of offshore structures.

WAMIT 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 multiple runs with FORCE, varying the requested parameters to be output, without re-running POTEN in each instance. (The evaluation of drift forces using a control surface is an exception where the computational burden in FORCE may be greater than that in POTEN.)

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, and 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.

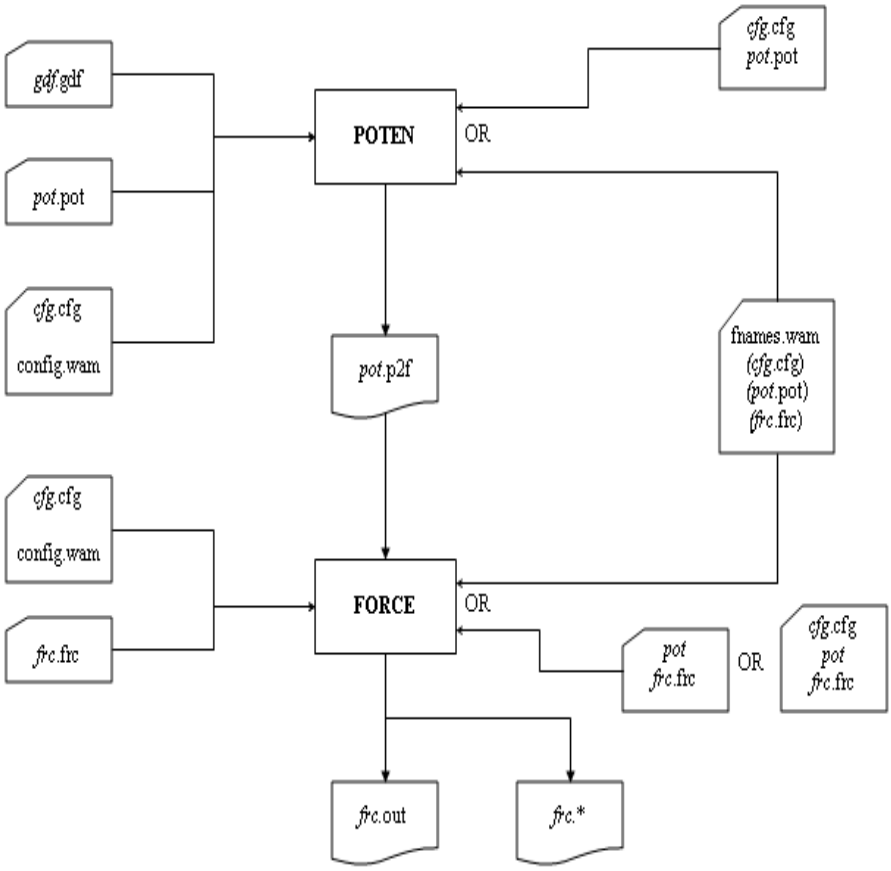


Figure 1.1: Flow chart of WAMIT showing the subprograms POTEN and FORCE with their associated input and output files. Filenames in italics are specified by the user. The four primary input files and optional configuration files described in Chapters 4, 6, and 7 are indicated in the left-hand column. 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. Additional input files which may be required, depending on the options used, are listed in the Introduction to Chapter 4. Note that the P2F file output from POTEN is given the same filename as the input control file, with the extension `p2f`. The P2F file may be saved and reused for various applications of the FORCE module where the same velocity potentials apply. The output file from FORCE is given the same filename as the force control file, with the extension `out`. Asterisks ($*$) denote the extensions corresponding to each option in the numeric output files, as listed in the table in Section 4.3.

1.2 CHANGES INTRODUCED IN Version 7.0

New features which are included starting in Version 7.0 are outlined below.

- For runs where NBODY>1 global planes of symmetry can be used if the body geometry is suitable (see Section 8.6).
- When the higher-order method (ILOWHI=1) is used, patch data are output in the `wamitlog.txt` file if NPER=0, to assist in checking the geometry.
- Part or all of the boundary surface associated with the bodies can be defined as a free surface with oscillatory pressure distribution (see Section 12.5).
- The mean drift forces can be evaluated using control surfaces without evaluating the same forces from pressure integration (see Chapter 11).
- When the mean drift forces are evaluated from pressure integration, points where the velocity on the body surface exceeds a specified limit can be output in the file `wamitlog.txt`, using the parameter VMAXOPT9 in the CFG file (see Section 4.7).
- The configuration parameter `TOLGAPWL` can be used to adjust the tolerance for gaps between adjacent elements of the body waterline (see Section 4.7).
- The configuration parameter RAMGBMAX can be used to take maximum advantage of the computer's available RAM for storage of temporary scratch files. Depending on the input parameters of the run and the hardware, this can be used to achieve substantial savings in run time. (See Sections 4.7 and 14.3).
- Version 7.0 is compiled with the Intel Fortran Compiler Version 12.1, using special directives to provide parallel processing on systems with multiple processors. This can result in significant reductions of run times with a single processor, and dramatic reductions with multiple processors. (See Section 14.6).
- The separate Froude-Krylov and scattering components of the exciting force and moment can be evaluated by using the extended options OPTN(2:3)=2. (See Sections 4.3 and 5.3.)
- Both configuration files `config.wam` and *.cfg can be used to input the configuration parameters. (See Section 4.7).
- New algorithms are used to evaluate the Rankine and log singularities in the low-order method
- The option IFORCE=2 can be used to run FORCE and POTEN in the same period loop and obtain portions of the numeric output files before the run is completed (See Section 4.14).
- The number of processors used and estimated RAM required for the run are output in the file `wamitlog.txt`.

- Several changes have been introduced to simplify the input files and to use a more consistent notation for the numeric output files (See Section 4.1).
- The configuration parameter IPERIO has been replaced by two parameters IPERIN, IPEROUT to control the definitions of both the input and output of the wave period array (See Section 4.7).

Chapter 2

GETTING STARTED

In this Chapter instructions are provided for installing WAMIT and making simple test runs in the PC/Windows environment. 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 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 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.

Three additional input files should also be used: (1) Licensed users of WAMIT V7PC must utilize a unique input file `userid.wam`, which identifies the license; (2) the optional input file `fnames.wam` is recommended to specify the input filenames; and (3) the configuration files (`config.wam` and/or `*.cfg`) specify parameters and options which are required by the program. Samples of the `fnames.wam` and configuration files are included with the test runs; for example the files `test01.wam` and `test01.cfg` are intended for use with TEST01.

By default in WAMIT, the file `fnames.wam` lists the filenames of the POT, CFG, and FRC files. The GDF filenames are included in the POT file.

In Version 7 it is possible to use both the `config.wam` and `*.cfg` files together, as explained in Section 4.7. It is recommended to use `config.wam` to specify inputs which are the same for most or all runs, and to use `*.cfg` with an appropriate filename to specify inputs which are specific to that particular run. Thus the file `config.wam` delivered with the

standard test runs includes the parameters RAMGBMAX, NCPU, and USERID_PATH, which are the same for all test runs; the other parameters required for each run are input in the separate files `test*.cfg`.

2.1 INSTALLATION AND SETUP

The WAMIT software includes the main executable program `wamit.exe`, the DLL files listed below, a complete set of input files for the standard test runs TESTn (n=01,02, ...), and the formatted output files TESTn.OUT for these test runs.

The PC-executable Version 7PC includes additional dynamic link library files (DLL) which must be installed in the same directory (i.e. 'folder') as the executable file `wamit.exe`, or in a different folder which is included in the system path. If these DLL files are missing the program will not run, regardless of the inputs and options specified. The required DLL files include special WAMIT DLL files and also special Intel or Microsoft DLL files. The three required WAMIT DLL files are

```
geomxact.dll
newmodes.dll
rg2wamit.dll
```

The extended Version including the capability to input MultiSurf models requires one additional DLL file, `rgkernel.dll`, and an extended version of `rg2wamit.dll`, as explained in Section 7.7.

Version 7PC is compiled using Intel Visual Fortran (IVF), Version 12.1. The following DLL files are required to run executable code compiled with IVF:

```
libifcoremd.dll
libmmd.dll
libiomp5md.dll
msvcr100.dll
svml_dispmd.dll
```

These DLL files are distributed by Intel and may be redistributed to all users. Copies of these files are included with each distribution of WAMIT. Users may encounter a problem on some older PC systems, indicated by the runtime error message 'The system cannot execute the specified program'. This problem can be overcome by downloading and installing the Microsoft Visual C++ 2005 Redistributable Package, which is available from Microsoft.

Before installing the software a new main directory (folder) should be made. The recommended name is `c:\wamitv7`, but the user may prefer to use a different drive or directory name, with a maximum length of 40 characters for the string. The software must be unzipped using a local utility, and all unzipped files should be saved in the directory `c:\wamitv7` and in subdirectories corresponding to those in the zip file. If a different drive or directory name is used, this must be specified as USERID_PATH in the configuration file, as explained in Section 4.7. 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 purposes without obtaining an end-user license, subject to the conditions stated in the website. The installation and use of this software are 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 some 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 available 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 Appendix A.

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:\wamitv7`. All required input files for the standard test runs will be copied to the subdirectory `c:\wamitv7\testruns`. Benchmark versions of the output files `test*.out` will be copied to the subdirectory `c:\wamitv7\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 \wamitv7\testruns`.

Since the executable file `wamit.exe` is resident in the directory `c:\wamitv7`, the appropriate command to execute WAMIT is '`c:\wamitv7\wamit`'. There are three alternative

shortcuts which may be used to simplify this command: (1) add `c:\wamitv7` 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 Section A.1 in Appendix A.

The corresponding input files `test01.gdf`, `test01.cfg`, `test01.pot`, and `test01.frc` are included in the subdirectory `c:\wamitv7\testruns`. In order to specify the appropriate filenames during the run, first copy the file `test01.wam` 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 test01.wam 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 for each wave period a new line of information is displayed including the wave period, time, and the maximum number of iterations required for the radiation and diffraction solutions. After the first wave period the computational time required for each subsequent period is reduced, since the integration of the Rankine components of the source potential are evaluated initially and saved for reuse. After the last period the file `test01.p2f` is saved 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 for TEST01 should be a few seconds.

2.5 RUNNING TEST11

TEST11 is intended to complement TEST01, 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 the solution for the velocity potential are represented in a more accurate, continuous manner by B-splines, as explained in Chapter 7.

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

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 `test*.wam` where '*' 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. The display shown on the monitor gives some indication of the progress through the set of runs from test01 to test25. The entire set of runs may take several minutes, depending on the computer.

2.7 USING MULTIPLE PROCESSORS

In Version 7 parallel processing can be performed on systems with multiple processors (CPU's, also known as 'cores'). Depending on the inputs and hardware, the total run time can be reduced substantially. If the system includes more than one CPU, open the file `config.wam` with a text editor. The default settings in this file are as follows:

```
NCPU=1
RAMGBMAX=0.5
USERID_PATH=\wamitv7
```

Edit the first line to show the actual number of CPU's on the system, and save the edited file with the same name. With this change the total run time for all of the standard tests should be substantially reduced. Test14a is a particularly good example, as noted in Appendix A.14.

Instructions for finding the number of CPU's on the system are in Section 14.6.

Depending on the size of the system RAM it may be appropriate to change the parameter RAMGBMAX, following the guidelines in Section 14.3. The value 0.5 is sufficient for all of the test runs unless NCPU is greater than 16. For older systems with less than 1Mb of RAM the parameter RAMGBMAX should be reduced to about half of the available RAM.

2.8 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.

2.9 MODIFYING THE INPUT FILES

A text editor can be used to edit the input files. Users should refer to Chapter 4 for a complete explanation of the data in these files. A few simple modifications will be outlined here for tutorial purposes, in the context of Test Run 01. Before proceeding further check that the file `fnames.wam` corresponds to this test run, or re-copy `test01.wam` to `fnames.wam` 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 `test01.cfg` (See Section 4.7, and also the file `test17b.cfg` which includes the same line). After modifications are made to the file `test01.frc` it is advisable to save the modified file with a 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 this new filename.

Since it is necessary to re-run POTEN in this case, delete the line IPOTEN=0 if this was added to `test01.cfg`. During the run, if the original filename `test01.p2f` 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

DEFINITION OF QUANTITIES EVALUATED BY WAMIT

Cartesian coordinates are used to define the body geometry, forces, and other hydrodynamic quantities evaluated by WAMIT. These are output in nondimensional forms, 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 notation and definitions of physical quantities here correspond with those in Reference [3], except that in the latter reference the y axis is vertical.

The body geometry, motions and forces are defined in relation to the *body* coordinates (x, y, z), which can be different for each body if multiple bodies are analyzed. The z-axis must be vertical, and positive upwards. If planes of symmetry are defined for the body, the origin must be on these planes of symmetry. The *global* coordinates (X, Y, Z) are defined with $Z = 0$ in the plane of the undisturbed free surface, and the Z-axis positive upwards. The body coordinates for each body are related to the global coordinates by the array XBODY defined in Section 4.2.

The incident-wave system is defined relative to the global coordinate system, and the phases of the exciting forces, motions, pressure and fluid velocity are defined relative to the incident-wave elevation at $X = Y = 0$. These outputs are defined in the general form

$$\mathrm{Re}\left((U + iV)e^{i\omega t}\right) = W\cos(\omega t + \delta). \tag{3.1}$$

where $W = |U + iV|$ is the modulus and δ is the phase. With reference to equations (15.2) and (15.4), the incident-wave elevation at $X = Y = 0$ is equal to

$$\mathrm{Re}(Ae^{i\omega t}) = A\cos(\omega t). \tag{3.2}$$

If $\delta > 0$ the output leads the phase of the incident wave, and if $\delta < 0$ the output lags the incident wave.

For field data (pressure, velocity, and free-surface elevation) and mean drift forces the definitions given below in Sections 3.5-3.8 apply to the complete solution for the combined

radiation and diffraction problems. The components of the field data can be evaluated separately for either the radiation or diffraction problems using the configuration parameters INUMOPT5 and INUMOPT6, as explained in Sections 4.7 and 5.2.

For the sake of simplicity, the definitions which follow in this Section assume that the origin of the body coordinate system is located on the free surface. Special definitions apply to some quantities if vertical walls are defined, as explained in Section 12.4.

3.1 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.

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 where they are denoted by (VOLX, VOLY, VOLZ) respectively. The median volume of the three volumes 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, VOLZ differs from the correct submerged volume by the product of the bottom area and depth.

Coordinates of center of buoyancy:

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

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

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

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 4.3) the body mass is computed from the relation $m = \rho\forall$. When Alternative form 2 is used for the FRC file (Section 4.4) the body mass is defined by EXMASS(3,3).

3.2 ADDED-MASS AND DAMPING COEFFICIENTS

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

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

Here

k	$=$	3	for	$(i, j = 1, 2, 3)$
k	$=$	4	for	$(i = 1, 2, 3,\ j = 4, 5, 6)$ or $(i = 4, 5, 6,\ j = 1, 2, 3)$
k	$=$	5	for	$(i, j = 4, 5, 6)$

3.3 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 $i = 4, 5, 6$.

The separate Froude-Krylov and scattering components of the exciting forces can be evaluated, using the options IOPTN(2)=2 and IOPTN(3)=2 as described in Section 4.3. The Froude-Krylov component is defined as the contribution from the incident-wave potential φ_0 and the scattering component is the remainder. Using the Haskind relations, these two components correspond respectively to the contributions from the first and second terms in parenthesis in the equation above. Using direct integration they correspond to the components of the total diffraction potential φ_D in equation 15.8.

3.4 BODY MOTIONS IN WAVES

Two alternative procedures are followed to evaluate the body motions in waves, corresponding respectively to the Alternative 1 (Section 4.3) and Alternative 2 (Section 4.4) 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.

$$M = \begin{pmatrix} m & 0 & 0 & 0 & mz_g & -my_g \\ 0 & m & 0 & -mz_g & 0 & mx_g \\ 0 & 0 & m & my_g & -mx_g & 0 \\ 0 & -mz_g & my_g & I_{11} & I_{12} & I_{13} \\ mz_g & 0 & -mx_g & I_{21} & I_{22} & I_{23} \\ -my_g & mx_g & 0 & I_{31} & I_{32} & I_{33} \end{pmatrix}. \tag{3.3}$$

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 $i = 4, 5, 6$. The rotational motions (ξ_4, ξ_5, ξ_6) are measured in radians.

3.5 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 15.1. The nondimensional velocity potential and hydrodynamic pressure are defined as follows:

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

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 $j = 4, 5, 6$.

The body pressure can be evaluated separately for the diffraction or radiation problems by assigning the configuration parameters INUMOPT5=1 and/or INUMOPT6=1, as explained in Section 4.7. When the radiation components are output separately, the nondimensional pressure due to jth mode is defined by

$$\bar{p}_j = \frac{p}{\rho g \xi_j L^n} = KL\bar{\varphi}_j$$

Special definitions are required in the limits of zero and infinite wave periods, as explained in Section 3.9.

3.6 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 nondimensional 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 3.5. (The nondimensional hydrodynamic pressure and wave elevation are equal to the nondimensional velocity potential at the respective positions.)

These parameters can be evaluated separately for the diffraction or radiation problems by assigning the configuration parameter INUMOPT6=1, as explained in Section 4.7. When the radiation components are output separately, the nondimensional free-surface elevation due to jth mode is defined by

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

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.

3.7 VELOCITY VECTOR ON THE BODY AND IN THE FLUID DOMAIN

The nondimensional 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 nondimensional gradient operator.

These parameters can be evaluated separately for the diffraction or radiation problems by assigning the configuration parameters INUMOPT5=1 and/or INUMOPT6=1, as explained in Section 4.7.

The evaluation of the velocity requires special caution close to the body surface, in the same manner as the pressure and free-surface elevation.

When the radiation components are output separately, the nondimensional velocity due to jth mode is defined by

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

Here $V = v_j\nabla\varphi_j$ is the dimensional fluid velocity, and $v_j = i\omega\xi_j$ 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 $j = 4, 5, 6$.

Special definitions are required in the limits of zero and infinite wave periods, as explained in Section 3.9.

3.8 MEAN DRIFT FORCE AND MOMENT

The definition of the nondimensional 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 7 the drift force and moment are evaluated based on the momentum flux across a control surface as described in Chapter 11, using equations (15.57-60).

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. 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. A warning message is displayed in the event that this convergence criterion is not satisfied. See Section 5.8 for further information regarding the interpretation and control of this warning message.

Often the warning message is issued because the length scale parameter ULEN is much smaller than the relevant length scale of the body. Since the drift force increases in proportion to length, and the moment in proportion to (length)^2, relatively small differences between large values may not be significant. In this case the warning message can be avoided by increasing ULEN to a value more representative of the length.

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.

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 15.7.

In Options 7 and 9, the mean drift force and moment evaluated from pressure integration and from momentum flux on a control surface are defined with respect to the body coordinate system. Conversely, the mean drift force and moment evaluated from momentum conservation in Option 8 are defined with respect to the global coordinate system.

3.9 ZERO AND INFINITE WAVE PERIODS

It is possible to evaluate the added mass (Option 1), and also the pressure and fluid velocity (Options 5 and 6) in the limiting cases of zero and infinite period (or equivalently, infinite and zero frequency), by inputting PER=0.0 and PER<0.0 in the POT file (see Section 4.2). This extension is particularly important in the context of evaluating the corresponding time-domain impulse response functions, as explained in Chapter 13.

The definition of the added-mass coefficients is the same as shown in Section 3.2. All other force coefficients are zero in these limits, with the exception of the heave exciting force and horizontal exciting moments, which tend to nonzero hydrostatic limits at zero frequency.

Special definitions are applied to the radiation pressure and velocity in the case of zero frequency (infinite wave period), which is identified in the output files by a negative value of the parameter PER, and also in the case of infinite frequency (zero wave period), which is identified by PER=0. In general, for nonzero finite values of the frequency, the nondimensional outputs for the radiation pressure and velocity are as defined in Sections 3.5 and 3.7. 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 and 6. The following table summarizes these definitions:

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

Chapter 4

INPUT FILES

A typical application of the WAMIT program 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 6 and 7 respectively. To simplify the presentation this Chapter will describe the required input files for a basic application involving the analysis of a single body. Further information is given in Chapters 8-10 for the appropriate modifications of the input files for specific purposes, including the analysis of multiple bodies, generalized modes of body motion, and the removal of 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 4.7. In the default case (IPOTEN=1, IFORCE=1) both subprograms are executed sequentially in the same run.

The input files `fnames.wam`, `config.wam`, and `break.wam` use reserved filenames with the extension `.wam`. All of the other input files are identified by three user-defined filenames *gdf*, *pot*, and *frc*. These are respectively the filenames used for the Geometry Data File (GDF), Potential Control File (POT), and Force Control File (FRC). Other input/output files are assigned the same filenames, depending on their context, and with different extensions. Thus *gdf* is used for files which relate primarily to a specific body geometry, *pot* to output files from POTEN which are associated with a specific set of inputs in the POT file, and *frc* to output files from FORCE which are associated with a specific set of inputs in the FRC file. Some input files are used only by POTEN or FORCE, whereas other input files are used by both. In this User Manual the font `fnames.wam` indicates a specific filename and extension, usually in lower-case letters. Upper case letters such as GDF are used more generally, to abbreviate the type of file. Filenames in italics, e.g. *pot*.p2f and *frc*.out refer to the user-defined filenames of the POT and FRC files, from which the output files derive their own filenames.

The following table lists all of the input files which may be prepared by the user, and indicates the relevant subprogram(s):

Filename	Usage	Description
pot.pot	POTEN	Potential Control File (Sections 4.2)
gdf.gdf	POTEN	Geometric Data File (Chapters 6,7)
frc.frc	FORCE	Force Control File (Sections 4.3-4)
gdf.spl	POTEN	Spline Control File (Section 7.11)
`fnames.wam`	POTEN/FORCE	Filenames list (Section 4.8)
`break.wam`	POTEN	Optional file for runtime breakpoints (Section 4.12)
`config.wam`	POTEN/FORCE	Configuration file (Section 4.7)
∗.cfg	POTEN/FORCE	Configuration file (Section 4.7)
gdf.ms2	POTEN	MultiSurf geometry file (Section 7.7)
gdf.csf	FORCE	Control surface geometry file (Section 11.1)
gdf.bpi	FORCE	Specified points for body pressure (Section 4.6)
frc.rao	FORCE	External RAO file (Section 4.13)

The first three input files are required in all cases. The others are required in some cases or are optional, as explained below.

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

Filename	Program	Description
pot.p2f	POTEN	P2F File (binary data for transfer to FORCE)
`errorp.log`	POTEN	Error Log File (Section 5.8)
`errorf.log`	FORCE	Error Log File (Section 5.8)
`wamitlog.txt`	POTEN/FORCE	Log file of inputs (Section 5.9)
frc.out	FORCE	Formatted output file (Section 5.1)
frc.num	FORCE	Numeric output files (Section 5.2)
*gdf*_pan.dat	POTEN	Panel data file (Section 5.7)
*gdf*_pat.dat	POTEN	Patch data file (Section 5.7)
gdf.pnl	FORCE	Panel data file (Section 5.7)
gdf.hst	FORCE	Hydrostatic data file (Section 5.6)
`rgklog.txt`	POTEN/FORCE	MultiSurf log file (Appendix C)
frc.fpt	FORCE	List of field points (Section 5.2)
gdf.idf	POTEN	Interior free-surface panels (Section 10.1)
gdf.bpo	FORCE	Specified points for body pressure (Section 5.5)
*gdf*_low.gdf	POTEN	Low-order GDF file (Section 5.7)
*gdf*_csf.dat	FORCE	Control surface data file (Section 11.6)
*gdf*_low.csf	FORCE	Low-order control surface file (Section 11.6)

The structure of input and output files is illustrated in the flow chart shown in Figure 1.1. The filenames assigned to the various output files are intended to correspond logically with the pertinent inputs, and to simplify file maintenance.

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 4.10.

Following the implicit FORTRAN convention, parameter names beginning with I,J,K,L, M or N denote integers, and all other numeric parameters are represented by real (decimal format) numbers. It is good practice to represent real numbers including a decimal point, and not to use a decimal point for integer parameters.

Additional input files may be used to select various options, and to optimize the computations for specific applications. The file `fnames.wam` is used to specify the names of the CFG, POT, and FRC input files to avoid interactive input of these names (see Section 4.8). The configuration files `config.wam` and *pot*.cfg are used to configure WAMIT and to specify various options, as described in Section 4.7. The optional input file `break.wam` may be used to set break points in the execution of POTEN, as described in Section 4.12. The optional Spline Control File *gdf*.spl may be used in the higher-order method, as described in Section 7.11.

The input file `userid.wam` is read by both POTEN and FORCE, to identify the licensee 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 one of the configuration files, as explained in Sections 2.1 and 4.7.

Two alternative formats for the FRC files are described separately in Sections 4.3-4. For a rigid body which is freely floating, and not subject to external constraints, Alternative Form 1 (Section 4.3) 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 4.4) 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 8.5.)

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 4.3.

- Additional numeric output files are generated for the exciting forces using Option 2 or Option 3, to provide data for the separate Froude-Krylov and scattering components of these forces, as explained in Section 5.3.

  Two additional numeric files are generated when the FRC file specifies either Option 5 (pressure and fluid velocity on the body surface) or Option 6 (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 Option 6 the 'field point' file with the extension `.fpt` specifies the coordinates of the field points in the fluid.

4.1 SUMMARY OF CHANGES IN VERSION 7 INPUT FILES

Several changes have been introduced in WAMIT Version 7 to simplify the input files and to use a more consistent notation for the numeric output files. These include the following:

- The POT file designated as Alternative Form 1 in Version 6 is not supported. The POT file designated as Alternative Form 2 in Version 6 must be used, with two exceptions: (1) the option to include the parameter IRR on line 2 is not supported, and (2) the parameter NEWMDS is removed from the POT file. If IRR and/or NEWMDS is nonzero the value(s) must be input in the configuration file.
- The parameters IQUAD and IDIAG, which controlled the accuracy of integration over panels in the low-order method, are no longer used. (In Version 7 the integration is performed in the same manner as for IQUAD=0, IDIAG=1 in older versions.)
- Options 6 (formerly field pressure) and 7 (formerly field velocity) are modified in the FRC file and in the corresponding numeric output files. Option 6 is used for both the field pressure and velocity, in a similar manner as for Option 5 (body pressure and velocity). The numeric output files for these are designated by the extensions (.6p, .6vx, .6vy, .6vz), replacing the old extensions (.6, .7x, .7y, .7z).
- Option 7 is used for the mean drift force and moment using a control surface, replacing the old CFG parameter ICTRSURF. The corresponding numeric output file extension is .7 (replacing the old extension .9c).
- Dipole panels or patches must be identified in the configuration file, following the same format as in Version 6. The option to identify these elements in the GDF file is not supported.
- Walls must be identified with the parameters IWALLX0, IWALLY0 in the CFG file, as explained in Section 12.4. The older alternative to assign negative values of ISX,ISY in the low-order GDF file is not supported.
- The option IALTFRC=2, used to specify the format and data in the FRC file, must be input in the configuration files. The older alternative to add an extra line in the FRC file is not supported.
- Two optional configuration files can be used, as explained in Section 4.7. Several configuration parameters which are no longer used have been removed, including IALTPOT, ICTRSURF, IDIAG, IQUAD, MAXSCR, IPERIO. Several new parameters are included, as explained in Section 4.7. The array XBODY is required in the POT file and must not be in the CFG file.
- The F2T utility described in Chapter 13 is modified in accordance with the new extensions of the numeric output files.

The utility program `v6v7inp.exe` is supplied with Version 7.0, to facilitate the conversion of old input files from Version 6 to Version 7. Special instructions for using this

program are in Appendix B. This program is intended to work for all input files which were valid in Version 6. However it is impossible to test the program with all possible combinations of inputs. Users should verify the changes made by the utility, especially if error messages occur running the new files.

4.2 THE POTENTIAL CONTROL FILE

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

In Version 7 the following format is used for the data in this file:

```
header
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)
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)
.
.
.
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)
```

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 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, with a maximum length of 72 characters (including leading blanks).

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

$$10^{-5} < \text{HBOT} \times \omega^2/g < 10^5$$

where ω is the radian frequency of the incident waves and g=GRAV is the gravitational

acceleration constant. For each run the value of GRAV is input in the GDF file, as described in Chapters 6 and 7. All dimensional inputs with the units of length, including HBOT, must be consistent with the input GRAV. Typically, either GRAV=9.80665 is used for the metric system or GRAV=32.174 is used if the length scale is in feet.

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.

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

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

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

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 IPERIN. Normally the values of PER must be positive. By using the optional parameter IPERIN 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 IPERIN:

IPERIN	Input	Definition
1	Period in seconds	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 IPERIN=1 is assumed if IPERIN is not specified in the configuration file. The corresponding parameter IPEROUT controls the same perameters in the output files, as explained in Section 4.7.

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. Likewise, 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 3.9). 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 IPERIN and the corresponding interpretation of the positive elements of the array PER. For example, the effect of the parameter IPERIN=2 and the array PER with the four inputs 0.0, 1.0, 2.0, -1.0 is identical to the default case IPERIN=1 with the array PER equal to 0.0, 2π, π, -1.0. If the irregular-frequency removal option is used (IRR$\geq$1) the computations for PER$\leq$0 are performed with 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 IPERIN 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 IPERIN=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. NBETA must be an integer. The following alternatives can be used:

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

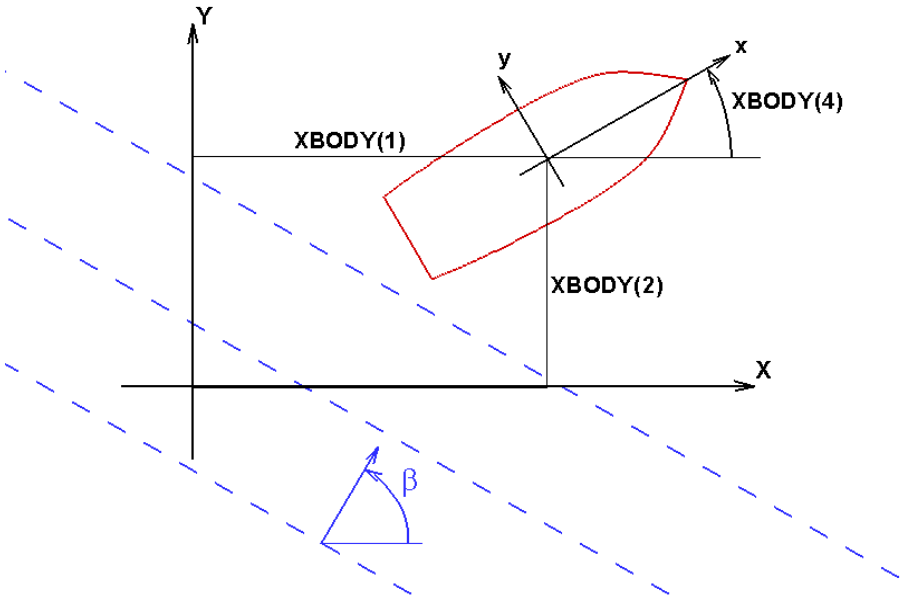


Figure 4.1: Sketch defining the global coordinates (X, Y, Z), body coordinates (x, y, z), and wave-heading angle β. The vertical Z and z axes are positive upwards, with Z=0 the plane of the undisturbed free surface. The origin of the body-coordinate system is at the point X=XBODY(1), Y=XBODY(2), Z=XBODY(3). A separate set of body coordinates is defined for each body if NBODY>1.

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 4.1. 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. If IDIFF= -1 NBETA should be set equal to 0 and 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)
```

NBODY is the number of bodies. Except for the analysis of multiple bodies (Chapter 8), 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, as shown in Figure 4.1, in the same units of length as ULEN. The global coordinate system is required when walls are present (Section 12.4) and when multiple bodies are analyzed (Chapter 8). 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 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 defined 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, defined as positive in the counterclockwise sense about the vertical axis, as shown in Figure 4.1.

MODE is a six-element array of indices, assigned for each 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 around the same axes, respectively. Each of these six elements 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 radiation problem will be solved. To understand the significance of the MODE array in the diffraction solution, 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.

4.3 THE FORCE CONTROL FILE (Alternative form 1)

The FRC file 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. 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 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, with a maximum length of 72 characters (including leading blanks).

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 filenames of the numeric output files 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)
6p	Pressure/ free-surface elevation at field points	*frc*.6p
6v	Fluid velocity vector at field points	*frc* .(6vx,6vy,6vz)
7	Mean drift force and moment from control surface	*frc*.7
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 in 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 2-9 may have additional values as listed below:

IOPTN(2) and IOPTN(3)

IOPTN(I) = 0: do not output the exciting forces

IOPTN(I) = 1: output the exciting forces

IOPTN(I) = 2: output the exciting forces and also the separate Froude-Krylov and scattering components of these forces (see Sections 3.3 and 5.3)

IOPTN(4)

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

IOPTN(4) = $\pm$1: output the RAO using the Haskind exciting force

IOPTN(4) = $\pm$2: output the RAO using the 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 4.5.

IOPTN(5)

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

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

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

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

- **IOPTN(6)**

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

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

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

  IOPTN(6) = 2: output velocity in the fluid by the potential formulation

  IOPTN(6) = −2: output velocity in the fluid by the source formulation

  IOPTN(6) = 3: output both pressure and velocity in the fluid and/or free-surface elevation by the potential formulation

  IOPTN(6) = −3: output both pressure and velocity in the fluid and/or free-surface elevation by the source formulation

- **IOPTN(7)**

  IOPTN(7) = 0: do not output the mean force and moment from control-surface integration

  IOPTN(7) = 1: output the mean force and moment from control-surface integration only for unidirectional waves

  IOPTN(7) = 2: output the mean force and moment from control-surface integration for all combinations of wave headings

IOPTN(8)

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

IOPTN(8) = 1: output the mean force and moment from momentum integration only for unidirectional waves

IOPTN(8) = 2: output the mean force and moment from momentum integration for all combinations of wave headings

IOPTN(9)

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

IOPTN(9) = 1: output the mean force and moment from pressure integration only for unidirectional waves

IOPTN(9) = 2: output the mean force and moment from pressure integration for all combinations of wave headings

The options IOPTN(6)< 0 apply only for the low-order method (ILOWHI=0), and require the source formulation (ISOR=1). If the low-order method is used the options IOPTN(5)>1 and IOPTN(9)>0 require the source formulation (ISOR=1).

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 motion are specified by the MODE indices in the POT file. The diffraction index IDIFF should be set equal to -1. In the FRC file the indices IOTPTN(2:4) should be set equal to zero. 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. However this requires a separate run of WAMIT for each mode. It is possible to output separate results for options 5,6 for each mode of forced motion in a single run, using the configuration parameters INUMOPT5, INUMOPT6, as explained in Section 4.7.)

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.)

Body motions in incident waves. In this case 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.

For bodies with one or two planes of symmetry (ISX=1 and/or ISY=1), the solution of the diffraction problem is decomposed into symmetric and antisymmetric components. If IDIFF=1 all of these components are computed, but if IDIFF=0 only the modes needed to evaluate the exciting forces corresponding to nonzero elements of the index IMODE in the POT file are computed. Special attention is necessary when computing the pressure, fluid velocity, and drift forces (Options 5-9), since these generally require all components of the diffraction solution. To ensure a complete solution, IDIFF=1 should be used in the POT file whenever Options 5-9 are computed. As explained in Section 5.8, a warning message is issued if IDIFF=0 and IOPTN(I)>0 for I=5-9, stating that the solution is non-physical. In special cases, where the heading angle coincides with a plane of symmetry, the complete

diffraction solution is symmetric and it is not necessary to compute the antisymmetric component. In such cases the warning message can be ignored. This 'shortcut' is illustrated in Tests 06, 07, and 14, where a TLP with two planes of symmetry is analyzed only in head seas (BETA=0), with IDIFF=0 and IMODE=0 for sway, roll and yaw; in this case the complete diffraction solution is symmetric about the plane Y=0, and the above inputs give correct results for Options 5-9.

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 $m_{ij} = m \times \text{XPRDCT(i} - 3, \text{j} - 3) \times |\text{XPRDCT}(i - 3, j - 3)|$. 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. This is a useful feature when the computational time is significant, since 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 × 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, as explained in Sections 4.7 and 12.1. When a large number of uniformly-spaced field points are required it is convenient to use the alternative inputs described in Section 4.11.

4.4 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 4.3).

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 header, IOPTN array, and all lines beginning with the variable NBETAH, are identical to the data in the Alternative form 1 FRC file. The data which differ in Form 2

are described below.

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) × (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) × (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 Section 3.1. (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 $\times$ 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 procedure to designate Alternative Form 2 is to assign the parameter IALTFRC=2 in the configuration file, as described in Section 4.7, and use the formats above. In early

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. This variant of the FRC file format was supported in Version 6. If a file with this variant is input, Version 7 issues an error message.

4.5 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, if (IOPTN(4)>0, the body motions are computed for all degrees of freedom. 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. 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 8), with no generalized modes, NDFR=6*NBODY. If generalized modes are analyzed (cf. Chapter 9) 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.

4.6 BODY PRESSURE AND FLUID VELOCITY AT SPECIFIED POINTS

If IOPTN(5)> 0 the points on the body surface where the pressure and fluid velocity are evaluated depend on the parameter IPNLBPT in the configuration file. In the default case IPNLBPT=0, these points are 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)
```

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 procedure used 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 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.

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 parametric coordinates (U, V) of the point on this patch which is closest to the input point. The pressure and velocity are evaluated at the corresponding point (U, V). 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 Jaco-

bian of the parametric transformation at the center of the patch.) A warning message is generated if nonconvergence occurs for one or more input points, showing the total number of unconverged points. (See also Section 5.5.)

The pressure and velocity are output in the .5p and .5v∗ files as explained in Section 5.5. Data related to the coordinates of the evaluation points are tabulated in the supplementary output file *gdf*.bpo which is explained in the same Section.

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.

4.7 CONFIGURATION FILES 'CONFIG.WAM' AND '*.CFG'

Two optional configuration files may be used to specify various parameters and options in WAMIT. The first configuration file is assigned the reserved name `config.wam`. The second filename is specified by the user, with the extension '.cfg'. Both files are opened and read, if they exist, and the configuration parameters defined below can be included in either the first or second file. It is recommended to use the first file for parameters which are the same for most or all applications, and to use the second file for parameters which depend on the specific run. If the same parameter is defined in both files, the input from the second is used for the run. If the second configuration file is used its complete filename including the extension '.cfg' must be included in the input file `fnames.wam`. The test runs listed in Appendix A show examples of these conventions.

The inputs which may be specified in the configuration files are as follows:

Parameter	Default value
IALTCSF	1
IALTFRC	1
IBODYW	0
IFIELD_ARRAYS	0
IFORCE	1
IGENMDS	0
ILOG	0
ILOWGDF	0
ILOWHI	0
IMODESFSP	
INUMOPT5	0
INUMOPT6	0
IPERIN	1
IPEROUT	1
IPLTDAT	0
IPNLBPT	0
IPOTEN	1
IQUADI†	4
IQUADO†	3
IREADRAO	0
IRR	0
ISCATT	0
ISOLVE	0
ISOR	0
ITANKFPT	0
ITRIMWL	0
IWALLX0	0
IWALLY0	0
KSPLIN†	3

MAXITT	35
MAXMIT	8
MODLST	0
MONITR	0
NCPU	1
NEWMDS	0
NMODESFSP	0
NOOUT	1
NPDIPOLE	
NPFSP	
NPTANK	
NUMHDR	0
NUMNAM	0
PANEL_SIZE	-1.0
RAMGBMAX	0.5
RHOTANK	
SCRATCH_PATH	current directory
TOLGAPWL	1.E-3
USERID_PATH	current directory
VMAXOPT9	-1.0
XTRIM	
ZTANKFS	

These parameters only need to be included when they are required. Parameters where a specified default value is shown are assigned this value when the parameter is not included explicitly. The parameters marked † are spline control parameters defined in Sections 7.11-12. These can be input either in the .cfg or .spl file.

When multiple bodies are analyzed (NBODY>1) there are additional parameters, and some of the parameters in the list above must be assigned with separate values for each body (See Section 8.6).

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

IALTCSF is an integer specifying the alternative methods of evaluating the mean drift force and moment using a control surface, as explained in Chapter 11 and Section 15.9.

IALTCSF= 1: use Alternative 1

IALTCSF= 2: use Alternative 2

The default value is IALTCSF=1.

IALTFRC is an integer specifying the alternative forms of the FRC file as explained in Sections 4.3, 4.4, and 8.3-5.

IALTFRC= 1: use the Alternative Form 1 FRC format shown in Section 4.3,

IALTFRC= 2: use the Alternative Form 2 FRC format shown in Section 4.4,

IALTFRC= 3: use the Alternative Form 3 FRC format shown in Section 8.5.

The default value is IALTFRC=1.

IBODYW is an integer specifying the body number of the first body which represents a wavemaker in a wall (see Section 12.3).

The default value is IBODYW= 0: No wavemakers are present in walls.

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 4.11.

IFIELD_ARRAYS= 0: Field point data is assigned only as shown in the conventional FRC files, as explained in Sections 4.3 and 4.4.

IFIELD_ARRAYS= 1: Additional field point data is assigned in the conventional FRC files, as explained in Section 4.11.

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: Execute FORCE.

IFORCE= 2: Execute FORCE and POTEN simultaneously for each wave period (see Section 4.14).

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 9.

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$\neq$ 0: 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.

ILOG is an integer parameter specifying the option to integrate the logarithmic singularity in the Green function separately, with a more accurate numerical integration scheme.

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.)

The default value is ILOG= 0.

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.

ILOWGDF is an integer parameter specifying the option to generate a low-order GDF output file based on the input geometry (see Section 5.7). 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. If ILOWGDF$>$ 1, the panels are subdivided into ILOWGDF$\times$ILOWGDF sub-panels.

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

ILOWGDF$\geq$ 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 6)

ILOWHI= 1: Higher-order panel method (Chapter 7)

- **IMODESFSP** is an integer parameter specifying the subroutine used to define pressure distributions on interior free surfaces of the body (see Section 12.5).

INUMOPT5 is an integer parameter specifying the option to evaluate the separate radiation and diffraction components of the body pressure and velocity (see Sections 3.5 and 3.7).

INUMOPT5= 0: output the total body pressure and velocity.

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 and velocity at field points (see Sections 3.5 and 3.7).

INUMOPT6= 0: output the total field pressure and velocity.

INUMOPT6= 1: output the total field pressure and velocity in the formatted output file (.out). Output the separate components in the numeric output files (.6p, .6vx, .6vy, .6vz).

- **IPERIN** is an integer parameter specifying the input data for PER in the POT file (see Section 4.2).

IPERIN= 1: Input periods in seconds

IPERIN= 2: Input radian frequencies

IPERIN= 3: Input infinite-depth wavenumbers

IPERIN= 4: Input finite-depth wavenumbers

The default value is IPERIN= 1.

(The use of IPERIN is identical to the parameter IPERIO in Version 6. The name of this parameter is changed in Version 7 to correspond with the parameter IPEROUT.)

IPEROUT is an integer parameter specifying the output data for PER. The same four optional values IPEROUT= $1, 2, 3, 4$ are used as defined above for the corresponding input parameter IPERIN. The default value is IPEROUT= 1. IPEROUT controls the first column of data in the numeric output files, and also the first column of data in the header of the OUT file showing the runtimes and iterations for the POTEN run, as well as the same runtime outputs to the monitor during the POTEN run.

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 5.7).

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.

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

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 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 4.6 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: Execute POTEN.

The default value is IPOTEN= 1.

IQUADI is an integer parameter specifying the order of the inner Gauss quadrature parameter IQUI for all patches when the higher-order method is used (ILOWHI=1). The default value is IQUADI=4. (See Sections 7.11-12.)

IQUADO is an integer parameter specifying the order of the outer Gauss quadrature

parameter IQUO for all patches when the higher-order method is used (ILOWHI=1). The default value is IQUADO=3. (See Sections 7.11-12 for further information.)

IREADRAO is an integer parameter specifying if the RAO's are to be input from an external file, as explained in Section 4.3.

IREADRAO= 0: Use RAO's computed by WAMIT in Option 4 to evaluate Options 5-9

IREADRAO= 1: Evaluate only Options 1,2,3,4

IREADRAO= 2: Read RAO's from an external file to evaluate Options 5-9

The default value is IREADRAO=0.

IRR is the integer used to specify whether the irregular frequencies are removed. The use of IRR is described in Chapter 9.

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

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

IRR= 2: Remove the effect of the irregular frequencies. (Program projects the body panels on the interior free surface.) This option is restricted to the low-order method (ILOWHI=0).

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

The default value is IRR=0.

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 (15.12) which we define as the diffraction problem. Alternatively in the scattering problem, the scattered potential is solved by (15.13) and the diffraction potential is obtained from equation (15.8): This option is only available in the low-order method (ILOWHI=0).

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. These three alternative methods are described in Section 14.2.

ISOLVE=0: Use the iterative solver

ISOLVE= 1: Use the direct solver

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

ISOLVE= -1 Special value used for wavemakers in planes of symmetry (See Section 12.3)

The default value is ISOLVE=0.

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 default value is ISOR=0.

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)<0) and the mean drift force and moment from pressure integration (IOPTN(9)=1 or 2). For the control-surface drift force and moment (IOPTN(7)=1 or 2), if ISOR=0, only the horizontal drift force and vertical moment can be evaluated using Alternative 1 (IALTCSF=1); ISOR=1 is required for the other components using Alternative 1, and for all components using Alternative 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.

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 internal tanks. The default value ITANKFPT=0 should be used except in cases where tanks are included with field points inside the tanks.

ITANKFPT=0: All field points are in the exterior domain. Field points are input as specified in Sections 4.3 and 4.4, with the coordinates of each field point on one line, or using the array format described in Section 4.11.

ITANKFPT= 1: Field points are listed in the .frc control file with the tank number preceding the coordinates. Thus, in place of the XFIELD lines shown in Section 4.3, the correponding data are input as follows:

```
ITANK(n) XFIELD(1,n) XFIELD(2,n) XFIELD(3,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.

ITRIMWL is the integer parameter used to specify the trimmed-waterline option (See Section 12.2). When ITRIMWL=1 the array XTRIM must be included for each body.

The default value is ITRIMWL=0.

IWALLX0 is the integer parameter used to specify that the plane X=0 is a reflecting wall (See Section 12.4). When IWALLX0=1 the fluid is bounded by a reflecting wall at X=0.

The default value is IWALLX0=0.

IWALLY0 is the integer parameter used to specify that the plane Y=0 is a reflecting wall (See Section 12.4). When IWALLY0=1 the fluid is bounded by a reflecting wall at

Y=0.

The default value is IWALLY0=0.

KSPLIN is an integer parameter specifying the value of the B-spline orders KU,KV for all patches when the higher-order method is used (ILOWHI=1). The default value is KSPLIN=3. (See Sections 7.11-12 for further information.)

MAXITT is an integer parameter used to specify the maximum number of iterations in the iterative solver (See Section 14.2).

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 2^{MAXMIT}.) The default value MAXMIT=8 is assigned unless a different value is input in the CFG file; this value is recommended for general use.

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. 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.

- **NCPU** is an integer which specifies the number of parallel processors (CPU's, also known as 'cores') which are available on the system. The default value NCPU=1 is applicable for most older PC's. The use of NCPU>1, when this is available on the system, permits substantial reduction in run times (See Section 14.6).

NEWMDS is an integer parameter specifying the number of generalized modes for each body. The default value is NEWMDS=0. (See Chapter 9.)

- **NMODESFSP** is an integer array used to specify the number of modes used to define the pressure distribution on free-surface pressure surfaces. Further details are given in Section 12.5.

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.)

NPDIPOLE is an integer array used to specify the panel or patch indices of zero-thickness elements represented by dipoles. Multiple lines of input can be used, but each line must begin with 'NPDIPOLE ='. The data on these lines may specify either individual indices of each dipole panel/patch, or ranges of consecutive indices which are indicated by enclosing a pair of indices in parenthesis. The following three examples are equivalent ways of specifying that the indices 2, 4, 5, 6, 8 are dipole panels or patches:

NPDIPOLE= 2 4 5 6 8

NPDIPOLE= 2 (4 6) 8

NPDIPOLE= (2 2) (4 6) (8 8)

Note that parenthesis must be used to denote the lower and upper limits of a range of consecutive indices, and parentheses must not be used for any other purpose. Other ASCII characters may be included on these lines, and are ignored. It is not necessary for the individual indices or ranges to be in ascending order, except for the first and last indices of each range enclosed in parentheses.

- The Version 6 option to define dipole panels or patches in the GDF file is not supported in Version 7.

- **NPFSP** is an integer array used to specify the panel or patch indices of free-surface pressure surfaces. The data in this array are in pairs, denoting the first and last index for each surface. An even number of indices must be included on each line. Each pair of indices must be enclosed in parentheses, as shown in the input file `test17c.cfg`. More than one line can be used for multiple surfaces, and/or multiple surfaces can be defined on the same line. Only integer data and parentheses are read for the array NPFSP, with spaces or other characters separating each index. Other ASCII characters may be used in addition to the integers and parenthesis, but integers and parenthesis must be used only for the inputs above. Further details are given in Section 12.5.

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. Each pair of tank indices must be enclosed in parentheses, as shown in the input file `test22.cfg`. More than one line can be used for multiple tanks, and/or multiple tanks can be defined on the same line. Only integer data and parentheses are read for the array NPTANK, with spaces or other characters separating each index. Other ASCII characters may be used in addition to the integers and parenthesis, but integers and parenthesis must be used only for the inputs above. Note that all of the panels or patches defining the interior of a tank must be contiguous, and specified by a single pair of indices, since this is the way in which each separate tank is defined. Further details and examples are given in Section 12.1.

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 7.

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.

The default value is PANEL_SIZE = -1.0

RAMGBMAX is the number of Gigabytes of RAM available for use as scratch memory. Instructions for using this parameter are given in Section 14.3. The default value is RAMGBMAX= 0.5. This parameter replaces the parameter MAXSCR in Version 6.

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. 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 12.1.

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 maximum length of the pathname is 40 characters. Spaces cannot be used in the pathname. The example below illustrates this option. (This option was more useful for older systems with limited memory and disk size. On contemporary systems it is less useful, especially if the size of RAM is sufficiently large to store all of the temporary arrays, as discussed in Chapter 14.)

TOLGAPWL is the tolerance used to neglect gaps between waterline panels or patches, when tracing the waterline(s) in conjunction with automatic representation of the interior free surface (Chapter 10) and automatic representation of control surfaces (Section 11.4). The default value is 10^{-3}.

USERID_PATH designates the directory (folder) where the input file `userid.wam` is stored. (This input file is required for users of V7PC.) It is convenient to store `userid.wam`, the executable `wamit.exe`, and the dynamic link libraries (DLL) listed in Section 2.1 together in one directory, as explained in Section 2.1. In this case USERID_PATH should be specified as in the example below. The maximum length of the pathname is 40 characters. Spaces cannot be used in the pathname. The default value of the pathname is the current directory.

VMAXOPT9 is a real parameter which can be used to output a warning message when the nondimensional fluid velocity on the body surface exceeds this value, during the evaluation of the mean drift force and moment from pressure integration on the body surface. If VMAXOPT9$\geq$0 at any point(s) used in the integration, the body number, panel or patch number, coordinates of the point and magnitude of the velocity are output in the file `wamitlog.txt`, as explained in Section 5.6. This procedure can be used to identify points on the body surface where the representation of the body geometry may be deficient in some respect. The default value is -1.0.

XTRIM is the real array used to specify the trimmed-waterline option. XTRIM includes three real numbers, specifying the vertical displacement, pitch, and roll of the body in the trimmed condition, as explained in Section 12.2.

ZTANKFS is the real array used to specify the free-surface elevations in internal tanks. The data in the array ZTANKFS 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 'ZTANKFS='. If the array ZTANKFS is included, it must include one real number for each tank. If the array ZTANKFS is not included, the waterline of each tank is derived from the highest vertex of the GDF inputs. ZTANKFS is only used when the waterline trimming parameter ITRIMWL=1. Further information is given in Section 12.1.

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 NOOUT(9) and XBODY(4) should be displayed in order on one line, for the sake of readability. Lines which start with any characters other than the explicit parameters shown below are ignored. A convenient way to remove parameters temporarily is to add a 'comment' character such as '!' before the name of the parameter. (In the .cfg files shown in Appendix A this is done to emphasize that the header on line one is a comment.)

The following example of a configuration file illustrates all of the possible input parameters for NBODY=1. Section 8.6 shows additional inputs for NBODY>1. For clarity the parameters are arranged in alphabetic order, but their actual order is arbitrary.

IALTCSF=1 (use Alternative 1 to evaluate the control-surface drift force)
IALTFRC=1 (use FRC format in Section 4.3)
IBODYW =0 (No wavemakers are present in walls)
IFIELD_ARRAYS=1 (arrays of field point data is assigned, as in Section 4.11)
IFORCE=1 (run the FORCE subprogram)
IGENMDS=1 (run DEFINE subroutine in .dll file for generalized modes)
ILOG=1 (integrate the log singularity separately)
ILOWGDF=0 (do not output low-order GDF from higher-order geometry)
ILOWHI=0 (use low-order method)
IMODESFSP=1 (specifies NEWMODES subroutine to define FSP modes)
INUMOPT5=1 (output the separate components of the body pressure/velocity)
INUMOPT6=1 (output separate components of the field pressure and velocity)
IPERIN=1 (POT file contains wave periods in seconds)
IPEROUT=1 (Output files contain 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)
IQUADI=4 (inner Gauss quadrature order in higher-order method)
IQUADO=3 (outer Gauss quadrature order in higher-order method)
IREADRAO=2 (input modified RAO's from external file)
IRR=0 do not use irregular-frequency removal
ISCATT=1 (Solve for scattered potential)
ISOLVE=2 (Use two blocks in iterative solver)

ISOR = 1 (solve for source strength)
ITANKFPT=1 (input field points in internal tanks with special format)
ITRIMWL =1 (Trim waterlines as specified by XTRIM)
IWALLX0 =1 (the plane X=0 is a reflecting wall)
IWALLY0 =1 (the plane Y=0 is a reflecting wall)
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)
MODLST =1 (Outputs in same order as left-hand-sides)
MONITR=1 (display all FORCE output to monitor)
NCPU=2 (Use 2 dual-core processors in parallel)
NEWMDS=0 (No generalized modes)
NMODESFSP=1 (Use one FSP mode)
NOOUT=1 1 1 1 0 1 1 1 1 (output all but body pressure/velocity data)
NPDIPOLE= 2 4 6 (dipole panels/patches)
NPTANK= (8-11) (12-15) (range of panels/patches on two internal tanks)
NPFSP= (22-24) (range of panels/patches on FSP surfaces)
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)
RAMGBMAX = 1.5 (Gigabytes of RAM available for scratch memory)
RHOTANK = 0.6 1.0 (relative fluid densities in the two tanks)
SCRATCH_PATH=d:\temp (use temp directory on drive D for scratch files)
TOLGAPWL= 0.01 (tolerance for waterline gaps)
USERID_PATH=c:\wamitv7 (directory where `userid.wam` is stored)
VMAXOPT9=10.0 (output points on the body where the fluid velocity >10)
XTRIM = 1. 2. 0. (Trim 1 unit vertically and 2 degrees pitch angle)
ZTANKFS = 1. 2. (Elevations of tank free surfaces above the plane Z=0.)

4.8 FILENAMES LIST 'FNAMES.WAM'

An optional input file may be used to specify the filenames of the primary input files POT, FRC, and GDF. Use of this optional file is recommended. The optional file must be named `fnames.wam`. This file includes a list of the input filenames above, including their respective extensions '.cfg', '.pot', and '.frc'. If this file does not exist, or if it is incomplete, the user is prompted to supply the missing filenames interactively. The number and order of these filenames is arbitrary. (In Version 7, the GDF filename(s) are specified in the POT file and it is not necessary to specify any GDF filename in the FNAMES file.)

The only lines which are significant in the `fnames.wam` file are those which contain an ASCII string ending in '.cfg', '.pot', or '.frc'. Thus comment lines can be inserted, as long as they do not include one of these three strings.

4.9 FILE NAMES

The names of the input data files must not exceed 20 characters in length including extensions, and including one period ('.') separating the filename and extension. Filenames cannot include intermediate blank spaces (white spaces).

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 `test01.frc` is retained as described in Section 2.9, instead of a modified name such as `test01x.frc`, the user is prompted at the start of the FORCE subprogram with the choice of either overwriting the old file `test01.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 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 Windows PC environment filenames are not case sensitive. A distinction must be made for systems such as UNIX and LINUX, which are case sensitive. Most filenames and extensions which are assigned by WAMIT are specified in lower-case letters (via assignments of type CHARACTER in the source file modulesc.f. 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. Input parameters which are read by the program, such as the parameters in the configuration files, are not case sensitive.

The extensions ‘.gdf’, ‘.pot’, and ‘.frc’ are required, unless the corresponding assignments are changed in the source code.

A list of all reserved filenames is in Section 14.8.

4.10 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 in Appendix A. 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 can 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.

All lines of data in input files which contain ASCII text must be restricted to columns 1-80.

4.11 UNIFORM ARRAYS OF FIELD POINTS

In some applications large numbers of field points are required, with a spacing on a rectangular grid. This can be facilitated by assigning the parameter IFIELD_ARRAYS= 1 in the CFG file, as explained in Section 4.7. 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 4.3:

```
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 12.1. If 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 inSection 4.3. 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.

4.12 USING THE OPTIONAL FILE BREAK.WAM

In most cases the principal computational time required in WAMIT runs is in POTEN, to set up and solve the linear system of equations for the velocity potential at each wave period. This starts with the first wave period (JPER=1) and continues in sequence to the last period (JPER=NPER), where NPER is the number of wave periods specified in the POT file. When the computations are completed for each wave period a one-line display is shown on the monitor including the wave period, clock time, and numbers of iterations for the radiation and diffraction solutions. (The same line is reproduced in the header of the OUT file.) This display during the run permits the user to monitor the progress, and estimate the total time to complete the run.

The optional input file `break.wam` can be used to activate break points during the POTEN run. This makes it possible to break a run, when the computational time is excessive, without losing data that has already been computed. The use of `break.wam` is optional, and if `break.wam` does not exist in the default directory the run will continue normally without any breaks. It is possible, as described below, to input `break.wam` after the WAMIT run has started, in order to break a run which is taking more time than was expected at the start of the run.

If the input file `break.wam` exists, and can be opened without an error, the user is prompted at two break points within the loop over NPER wave periods. The first break point is at the beginning of the period loop, before setting up the linear system, and the second break point is after setup is completed but before solving the linear system. Since the relative balance of time required for setup and solution depends on the inputs, two opportunities to break the run are provided.

When a break occurs, the monitor displays a message to identify the break point, followed by a menu of options. The two messages for the first and second break points are as follows:

```
' Break at start of period loop, period JPER='
' Break before CALL SOLVE, period JPER='
```

The value displayed for JPER is the index of the current wave period. If the run is terminated with JPER=1 there will be no outputs (except for the header information and hydrostatics). To minimize the amount of wasted computations it is most efficient to break the run at the start of the period loop, with JPER>1, instead of after the setup of the linear system is completed.

Following the above message, the menu prompts the user to select one of three options, with the following display on the monitor:

```
   File BREAK.WAM exists.  Select option (b,c,d):

b: Break run and continue with reduced NPER
c: Continue run and keep BREAK.WAM
```

```
d: Delete BREAK.WAM and continue run
                                Enter b,c, or d:
```

The results of these three options are as follows:

b The execution of POTEN is terminated, and NPER=JPER-1 is assigned for the remainder of the run

c The execution of POTEN is continued to the next break point

d The file BREAK.WAM is deleted and execution of POTEN is continued without further breaks

The inputs (b,c,d) are not case sensitive; either lower-case or upper-case letters may be used. Inputting any other characters will result in a repeat of the above menu of options, until either b,c, or d is input.

Since the file `break.wam` is not read, the data in this file is arbitrary. One convenient possibility, when break points are requested, is to copy a small existing file such as `fnames.wam` to `break.wam`. The file `break.wam` can be established either before execution of WAMIT or during the run. To establish the file during runtime, in the PC environment, it is necessary either to use a second DOS (Command Prompt) window, or to establish the file using a Windows Edit command.

When multiple processors are used (NCPU>1) the `break.wam` file is ignored and it is not possible to interrupt the execution of POTEN until the loop over all wave periods is completed.

4.13 ASSIGNING RAO'S IN AN EXTERNAL FILE

It is possible to input RAO's from an external file, referred to as the external RAO file. This permits users to modify RAO's to take into account physical effects which are not included in the original WAMIT computations, and to evaluate other hydrodynamic parameters including the drift forces, pressures and velocities based on the modified values of the RAO's. This option is controlled by the parameter IREADRAO in the CFG file, as explained in Section 4.7. If IREADRAO=0 (default), the RAO's computed by WAMIT in Option 4 are used to evaluate the other hydrodynamic parameters (Options 5-9).

In a typical application where the RAO's are to be modified, two separate WAMIT runs are executed. These are summarized in the two following paragraphs.

If IREADRAO=1 is assigned in the CFG file, the WAMIT run is executed in the same manner as in the default case, except that Options 5-9 are not evaluated. (The results are the same as if IOPTN(5:9)=0 are assigned in the FRC file.) This provides the necessary added-mass, damping, exciting-force and hydrostatic coefficients for use in modifying the RAO's. A normal RAO numeric output file is also generated, to serve as a guide for the format and contents of the external RAO file. At this stage the user can prepare the external RAO file, which must be named *frc*.rao and correspond to the contents of the WAMIT output file *frc*.4 with the same definitions of the data (cf. Section 5.2).

If IREADRAO=2 and IPOTEN=0 are assigned in the CFG file, the program skips the POTEN computations and reads the solution from a previous run, saved in the intermediate binary file *pot*.p2f (the filename *pot* of the POT input file must be retained for this purpose). Then the program reads the RAO's from the external file *frc*.rao and evaluates the hydrodynamic outputs for Options 5-9 using these values of the RAO's.

Important rules which must be followed in preparing the external RAO file are as follows:

- If header lines are included in the numeric output files (NUMHDR=1) a header line must be included in *frc*.rao Conversely, if header lines are not included in the numeric output files (NUMHDR=0) a header line must not be included in *frc*.rao
- For each wave period, heading, and mode index a separate line (or successive lines) must be included in *frc*.rao with the same set of data including four real (decimal) numbers which define the modulus, phase, real and imaginary components of each RAO as shown in Section 5.2 for OPTN.4. (Only the real and imaginary components in the last two columns are used as inputs; thus the values assigned to the modulus and phase can be arbitrary real numbers.)
- The wave periods, heading angles, and mode indices must be the same as in *frc*.rao and these must be in the same order with respect to period and heading angles (the order of mode indices is arbitrary).

The precise format of the data in *frc*.rao is not important and does not need to be identical to *frc*.4 with respect to column widths, spaces, or number of decimal digits. This file is read with FORTRAN free-format read statements, so that the data can be input in integer, fixed, or floating-point formats. It is recommended to use only integer format for

the mode index and fixed or floating-point format for all other data. If the data cannot be read or is not consistent with the file frc.4 an error message is issued and the program execution is terminated.

There are two exceptions where the data in frc.rao do not correspond with the data in frc.4. If the parameter IBODYW>0 is assigned in the CFG file to define one or more wavemakers, the modes for these wavemakers must not be included in the external RAO file. Similarly, if fixed modes of the body are defined in the FRC file, as explained in Section 4.5, and designated by negative mode indices in frc.4, these modes must not be included in the external RAO file. Users are advised not to use fixed modes in combination with the option to define RAO's in the external RAO file, since the modified RAO's cannot be used to evaluate the fixed mode loads. A warning message is issued in this case.

It is not necessary to use IREADRAO=1 in place of IREADRAO=0, but this avoids unnecessary computations for Options 5-9 when the RAO's will be modified by the user.

4.14 EVALUATING FORCE OUTPUTS IN POTEN (IFORCE=2)

In the default case where IPOTEN=1 and IFORCE=1 WAMIT first executes the POTEN subroutines to evaluate the potential for all wave periods, and then the FORCE subroutines to evaluate the output parameters specified in the FRC control file, as shown in the flow chart in Figure 1.1. In that case the output parameters are included in the frc.OUT file, and also in the separate numeric output files which use either the filename frc in the default case (NUMNAM=0) or alternatively the filename 'optn' if NUMNAM=1.

In long runs where the execution time for POTEN is large, no FORCE outputs are available until after a substantial time. This may be inconvenient for the user, especially if it is desired to check a few outputs for the initial wave periods prior to the completion of the run. Also, if the run is interrupted prior to completion all of the computed data is lost unless the special procedure described in Section 4.12 is used.

If IFORCE=2 and NCPU=1, the computations of POTEN and FORCE are executed in sequence for each wave period, and the outputs from the intermediate numeric output files of FORCE are available as soon as each wave period is completed. The 'optn' filename is used for these intermediate outputs in POTEN. After completion of the POTEN period loop, the same data are written to the numeric output files (with the filenames frc or 'optn', depending on the configuration parameter NUMNAM), and to the file frc.out. The optn files will be overwritten in the next WAMIT run, without warning. During a long POTEN run the data which has been evaluated in the optn files can be read with a text editor, but should not be re-saved with the same names to avoid conflicts during the WAMIT run.

When multiple processors are used (NCPU>1) with IFORCE=2, POTEN and FORCE are executed in the same manner as described above. However the data in the optn files are not output until after the period loop is completed, and thus it is not possible to access these outputs during the execution of POTEN.

Chapter 5

OUTPUT FILES

Several output files are generated during a typical WAMIT run. These are intended to facilitate visual examination of the results in tabular form, and post-processing by other software.

The formatted output file is described in Section 5.1. This summarizes the hydrodynamic outputs in a single file with appropriate identifying text. The header which includes a summary of the inputs and hydrostatic computations. The numeric output files, described in Sections 5.2-5, are intended to tabulate the hydrodynamic outputs in a more concise form for post-processing. The principal numeric output files are described in Section 5.2. Section 5.3 explains the additional files used to output the Froude-Krylov and scattering components of the exciting forces. Section 5.4 describes the output file used for the B-spline coefficients of the body pressure when the higher-order method is used. Section 5.5 explains the special output files which are generated when the pressure and velocity are evaluated at user-specified points on the body.

Section 5.6 describes the output file for the hydrostatic restoring coefficients which are computed by WAMIT and used in the equations of motion. Section 5.7 describes the auxiliary files which output geometric data. Section 5.8 describes the outputs of warning and error messages. The log file `wamitlog.txt` described in Section 5.9 is intended to provide useful archival information including a condensed summary of all input files and error or warning messages which are generated during the run. The intermediate data transfer file, which is described briefly in Section 5.10, is a binary file used internally in WAMIT to transfer data from the POTEN to FORCE.

5.1 THE FORMATTED OUTPUT FILE (OUT)

The formatted output file *frc*.out is written in the sub-program FORCE to summarize inputs and outputs. The principal inputs are in the header portion of the file, including licensing information, input filenames, run times and dates, and hydrostatic data. The hydrodynamic outputs are written in sequence for each wave period, heading angle, and parameter. Appropriate text labels are included to identify the data.

If an old file exists with the same name, in the same directory, the user is interrogated with options to over-write the old OUT file or to assign a different name to the new file. This interruption can be avoided by renaming or deleting the old file. Often, when different options or variants of the inputs are being compared, it is convenient to assign a new name which is related to the old name in a logical manner. This situation can be avoided by using different filenames for the corresponding FRC input files, even if the files are the same.

5.2 NUMERIC OUTPUT FILES

Separate output files of numeric data are generated for each of the nine options of the FORCE subprogram listed in Section 4.3. 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.6P:	PER	BETA	L		Mod($\bar{p}$)	Pha($\bar{p}$)	Re($\bar{p}$)	Im($\bar{p}$)	
OPTN.6VX:	PER	BETA	L		Mod($\bar{V}_x$)	Pha($\bar{V}_x$)	Re($\bar{V}_x$)	Im($\bar{V}_x$)	
OPTN.6VY:	PER	BETA	L		Mod($\bar{V}_y$)	Pha($\bar{V}_y$)	Re($\bar{V}_y$)	Im($\bar{V}_y$)	
OPTN.6VZ:	PER	BETA	L		Mod($\bar{V}_z$)	Pha($\bar{V}_z$)	Re($\bar{V}_z$)	Im($\bar{V}_z$)	
OPTN.7:	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($\bar{F}_{io}$)]
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 are replaced by *frc*, as explained in Section 4.7.

All output quantities are nondimensionalized as defined in Sections 3.2-8. 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.

If Option 5 is specified and INUMOPT5=1, 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.

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 the numeric output files .6p, or .6vx, .6vy and 6vz contain the separate components of the radiation and diffraction pressure and velocity in the following modified format:

OPTN.6p:	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.6vx, OPTN.6vy, OPTN.6vz.

If Option 6 is specified the supplementary output file *frc*.fpt is created with 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

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 5.5)

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 Options 7 and 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 $\geq$ 0, 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 15-2. In all cases the components of the vector force and moment are defined with respect to the inertial (mean) coordinate system.

If NBODY>1, the panels of all bodies are merged with a common index K, following the same order as the body parameters in the POT file (See Chapter 8).

5.3 FROUDE-KRYLOV AND SCATTERING FORCES

The separate Froude-Krylov and scattering components of the exciting force and moment can be evaluated by setting the index IOPTN(2)=2 and/or IOPTN(3)=2 in the FRC file, as explained in Section 4.3. In this case additional numeric output files are generated by the program, with the extensions .2fk and .2sc for the Haskind forces and .3fk and .3sc for the diffraction forces. These files contain the Froude-Krylov and scattering components respectively, in the same format as shown above for the OPTN.2 and OPTN.3 files. These are defined in Section 3.3. The Froude-Krylov outputs in the .2fk and .3fk files are identical.

5.4 BODY PRESSURE 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 7). 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 5.2. 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 5.2. (If IPNLBPT$\neq$0 is assigned in the configuration file the alternative option is utilized with the points on the body surface specified by the user, as described in Sections 4.6 and 5.5.)

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 3.5) these three components are related by the equation

$$\varphi = \varphi_D + KL \sum_j \xi_j \varphi_j$$

Here 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 in all cases. 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,WVNFIN,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 wavenumber, infinite-depth wavenumber, (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)

5.5 BODY PRESSURE AND VELOCITY AT SPECIFIED POINTS

As explained in Section 4.6, the body pressure and velocity can be evaluated at specified points on the body using the special input file *gdf*.bpi to input the coordinates of these points. In this case 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 5.2, 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)$\geq$2.

In addition to these hydrodynamic outputs, a supplementary file *gdf*.bpo (Body Point Output) is output to provide information about the actual points where these evaluations are made.

If the low-order method is used (ILOWHI=0) the format of the BPO file is as follows:

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). NNEAR is equal to the absolute value of the configuration parameter IPNLBPT. In the example shown above NNEAR=4.

In the higher-order method (ILOWHI=1), where the solution for the velocity potential and pressure is represented by continuous B-splines on each patch, the program searches iteratively for the patch index and (U, V) coordinates of the point closest to the input point, as explained in Section 4.6. In this case 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. I is the number of iterations. A maximum of 16 iterations are used in this search, and if I=17 this indicates noncovergence of the search. 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.

5.6 HYDROSTATICS OUTPUT FILE (HST)

The supplementary output file *frc*.hst is created in the following format, to output values of the hydrostatic matrix C_{ij}:

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

This matrix is defined in Section 3.1.

5.7 AUXILIARY OUTPUT FILES FOR THE GEOMETRY

Options exist to generate three auxiliary geometry output files: *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 is used.)

The files *gdf*_pan.dat and *gdf*_pat.dat contain the Cartesian coordinates of panels or patches in formats suitable for perspective plotting with programs such as Tecplot. This facilitates the use of perspective plots to illustrate and check the GDF inputs. Examples of these plots are included in Appendix A for each test run.

The file *gdf*_low.gdf is a low-oder GDF file, with 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 6.1). 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 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 contains 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 A.5 and A.13 illustrate this feature.

These 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 using the Tecplot finite-element format FEPOINT. The integer parameter IPLTDAT in the configuration files 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 configuration files. 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 7. These files are in the Tecplot ordered-list format POINT. The integer parameter IPLTDAT in the configuration files 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 number of panel subdivisions on each patch is determined by the parameters NU and NV in the SPL file, as explained in Section 7.11. 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 A.11-19.)

In the higher-order method (ILOWHI=1) the optional output file *gdf*_LOW.GDF is controlled by the integer parameter ILOWGDF in the configuration files. 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 7.5), B-splines (Section 7.6), and geometries which are defined in the subroutine GEOMXACT (Section 7.8). 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.

5.8 ERROR MESSAGES

Numerous checks are made in WAMIT for consistency of the input data. 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. These files are 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 fluid pressures, velocities, and mean drift forces (Options 5-9) 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 outputs will be correctly evaluated. For further discussion of this 'shortcut' see the discussion of MODE in Section 4.2 and related discussion in Section 4.3.

The same messages are displayed on the monitor during the run, and in the log file `wamitlog.txt`. 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.

Two particular warning messages which occur relatively frequently are the following:

- 'Number of subdivisions exceeds MAXSQR'
- 'WARNING – no convergence in momentum Dx/Dy/Mz for headings ...'

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. Usually this indicates either an error in the geometry definition, or specification of a field point too close to the body surface.

The convergence test for the momentum drift force and moment is used to ensure accurate integration of the momentum flux in the far field. This integration is performed recursively, increasing the number of azimuthal integration points by factors of 2. Convergence is achieved when the difference between two successive iterations, in each of the three components Dx,Dy,Mz, is less than a prescribed tolerance TOL=10^{-4}. If the component is less than one in absolute value, absolute differences are used, otherwise relative differences are used. The maximum number of iterations is controlled by the parameter MAXMIT in the configuration files. (With the default value MAXMIT=8 the maximum number of integration ordinates is $2^8 = 256$.) When necessary this parameter can be increased, but it should be noted that this increases the computational time exponentially when the mean drift force and moment are evaluated from Option 8.

Since the components of the mean drift force are nondimensionalized by ULEN, and the moment by ULEN^2, convergence can also be affected by the choice of ULEN in the GDF file. If ULEN is much smaller than the physical length scale of the body it will not affect the convergence tests, and vice versa. Another point to note is that some components of

the force and moment may be relatively small, and of little practical importance, whereas they may affect the convergence test. When in doubt about situations where the warning message occurs, it may be advisable to increase MAXMIT by 1 or 2 units and compare the resulting outputs manually.

- Starting in Version 7.0, an option can be used to issue a warning message and to output in the file WAMITLOG.TXT all points on the body surface where the magnitude of the nondimensional fluid velocity is greater than VMAXOPT9, during the evaluation of the mean drift force and moment from pressure integration on the body surface. In this case, if VMAXOPT9≥0, data is output including the body index, panel or patch index, location of the point in dimensional body coordinates, and magnitude of the fluid velocity. The parameter VMAXOPT9 may be included in the configuration file (cfg), as explained in Section 3.7. The default value VMAXOPT9=-1.0 is assigned if there is no input, with the result that the above warning and outputs are not included. This option can only be used if IOPTN(9)>0 in the Force Control File. This option can be used to identify points where the representation of the body geometry is deficient in such a way that the evaluation of the fluid velocity is non-physical. In general the fluid velocity should be of order of magnitude one, or smaller, on the body surface, and values which are much greater than this may be due to either sharp corners (which are physically correct) or defects in the representation of the body geometry (which are not physically correct). This option should be used with care, to avoid excessive outputs. If VMAXOPT9=0.0, every integration point will be output, for all wave periods, heading angles, and symmetry planes of the body.

5.9 THE LOG FILE (`wamitlog.txt`)

The file `wamitlog.txt` is output during the run to provide an archival record. 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, only the first 10 lines of the GDF file are copied in these cases. 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. If it is appropriate to save this file it should be renamed, or moved to a directory where it is protected.

When the input data in the FRC file is in the Alternative form 1, as described in Section 4.3, the nondimensional inertia matrix for each body is included in the file `wamitlog.txt`. This is 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 described in Section 4.4, these normalizing factors must be included in the inputs EXMASS when these are derived from the nondimensional inertia matrix.

5.10 THE INTERMEDIATE DATA TRANSFER FILE (P2F)

The file *pot*.p2f is written by POTEN and read by FORCE. The filename *pot* is the same as the POT input file. The P2F file contains the solutions of the linear systems of equations for the velocity potential (and source strength) on the body surface, and also some inputs to POTEN which are required by FORCE, e.g. the wave periods and heading angles specified in the POT file. To facilitate data transfer the P2F file is a binary file, and it cannot be used for purposes other than as input to FORCE.

The P2F file can be used for multiple runs of FORCE, in situations where the outputs from POTEN are the same. TEST17b is an example of this situation, where the only change from TEST17a is to apply an external damping force on the 'lid' to attenuate the moonpool resonance (See Section A.17). In this case IPOTEN=0 is assigned in the configuration file and the POTEN run is skipped, with considerable savings of time. Increasing the number of Haskind wave heading angles, adding options in FRC which were omitted in the original run, and using an external RAO file (Section 4.13) are examples of other situations where it is useful to save the original P2F file and avoid the extra computational time required to repeat the POTEN run.

If another run is made using the same POT file, with IPOTEN=1 (default) and with an old P2F file in the same directory with the same filename, the user is interrogated with options to over-write the old P2F file or to assign a different name to the new file. This interruption can be avoided by renaming or deleting the old P2F file.

The P2F files can be relatively large, depending on the parameters of the run. Unless future use is anticipated it may be best to erase or over-write old files.

Chapter 6

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 6.1. The syntax for data in this file follows the same requirements outlined for the generic input files in Chapter 4.

When the low-order method is used there are two different solutions which can be used, referred to as the potential source formulations. The potential formulation, which is always evaluated, represents the velocity potential in terms of surface distributions of sources and normal dipoles (See Section 15.2). This is used to evaluate hydrodynamic quantities including the first-order pressure, force coefficients and drift forces based on momentum conservation (Option 8). For the evaluation of these outputs the potential formulation is more general and efficient.

The source formulation is optional, depending on the configuration parameter ISOR (See Section 4.7). In the source formulation the potential is represented by a surface distribution of sources only, as explained in Section 15.3. The source formulation must be used if the mean drift force and moment are evaluated by pressure integration, also in some cases where the drift forces are evaluated using control surfaces (Chapter 11), and more generally if the fluid velocity is required on the body surface. The procedure for including the source formulation is used is described in Section 6.2.

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]. WAMIT permits the user to specify a set of dipole panels, as described in Section 6.3. 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.

6.1 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'.

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

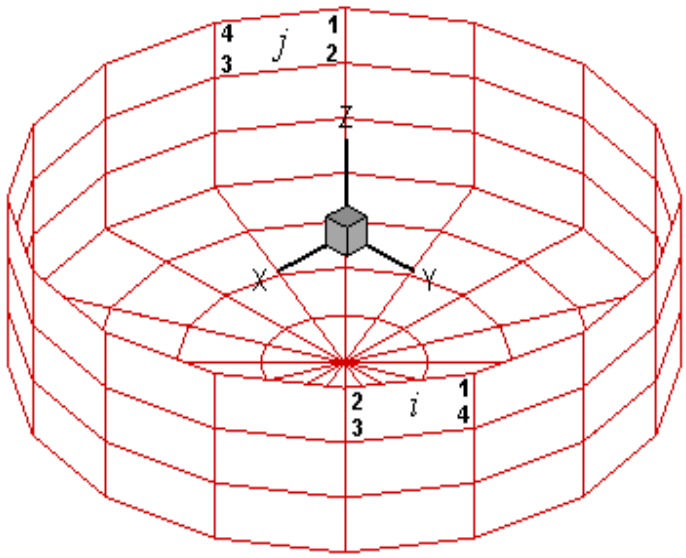


Figure 6.1: Discretization of a circular cylinder showing the convention for panel vertex numbering. The perspective view is from above the free surface, showing portions of the exterior and interior of the cylinder (lower and upper portions of the figure, respectively). The view of panel i is from the '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.*

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 $0, +1$. 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 6.1. 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 10. 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 Chapter 10.

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 with zero draft, 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 6.1. 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 3.1. 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.

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)

6.2 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. A brief description of the theory is provided in Section 15.4. (Further details are given in [10] and [26].)

Setting the parameter ISOR=1 in the configuration files specifies that the source-distribution integral equation is solved in addition to the velocity-potential integral equation. This is required in some cases for Options 5,6,7 and 9 in the FRC file, as noted in Section 4.3.

Values of the drift force and moment can be compared with the corresponding outputs evaluated using momentum conservation (Option 8), and with the drift forces evaluated using the control surface (Option 7). 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.

6.3 BODIES WITH THIN SUBMERGED ELEMENTS

Bodies which consist partially (or completely) of elements with small or zero thickness can be analysed by defining these elements as 'dipole panels'. The geometry of these elements is represented by panels in the same manner as conventional body panels (Section 6.1). Figure 6.2 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).

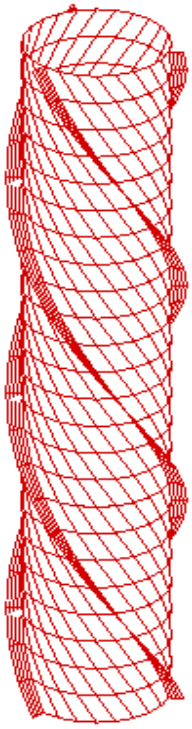


Figure 6.2: Perspective view of a floating spar with three helical strakes, showing the conventional panels on the spar and dipole panels on the strakes.

The velocity potential on the dipole panels is represented by dipoles alone, with no corresponding sources. The unknown is the difference of the velocity potential on the two sides, which is proportional to the pressure jump across the panel. Since both sides of the dipole panels adjoin the fluid, the direction of the normal vector is irrelevant. A positive difference of the velocity potential is defined to act in the normal direction to the surface from the side on which the vertices are in the counter-clockwise direction to the opposite side. The order of the vertices in the GDF file is arbitrary, as long as they are in a logical sequence to form a closed quadrilateral with contiguous sides.

- The indices of the dipole panels are defined in the CFG file by including one or more lines starting with 'NPDIPOLE=', followed by the indices or ranges of indices of the dipole panels, as explained in Section 4.7. In this case the format of the GDF file is as explained for the case without dipole panels in Section 6.1, and the parameter NPAN is the total number of panels including both conventional and dipole types. The dipole panels may be located arbitrarily within the array of all panels. It is possible to analyze bodies which consist entirely of zero-thickness elements, by including the line 'NPDIPOLE= (1 nn)' in the CFG file, where nn=NPAN is the same integer value input in the GDF file.

  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 or control-surface methods (Options 7 and 8).

  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 7

THE HIGHER-ORDER METHOD (ILOWHI=1)

The higher-order method is fundamentally different from the low-order panel method described in Chapter 6. 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 is 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.

A brief outline of the method is provided in Sections 7.1-7.3, 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 7.4 describes the data in the Geometric Data File (GDF) which is common to all applications of the higher-order method. Sections 7.5-7.8 describe the four different options for describing the body geometry, and the corresponding inputs. Section 7.9 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 analogous case of low-order panels described in Chapter 6. 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]. The procedure for defining dipole patches is described in Section 7.10. In Version 7 the dipole patches must be defined in the configuration files. The alternative option in Version 6 to define the dipole patches in the GDF file is no longer supported.

Section 7.11 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 files, 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 7.12 describes this procedure, which permits users to exploit the flexibility and efficiency of the higher-order method with a minimum of inputs.

Section 7.13 compares the advantages of the higher-order and low-order methods.

7.1 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 7.1. (The same cylinder is shown in Figure 6.1 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 7.1. 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).

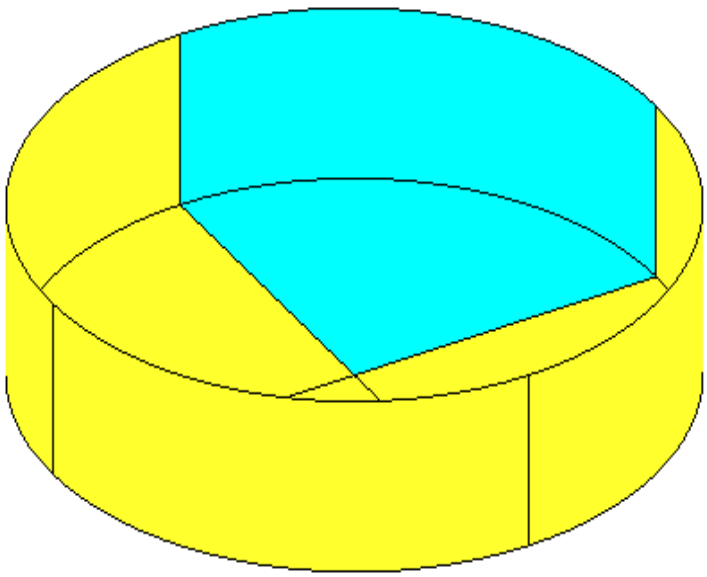


Figure 7.1: Representation of the circular cylinder by two patches on one quadrant, shown by the shaded portion, with reflections about the two planes of symmetry.

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 7.1, 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

$$u = \frac{4\theta}{\pi} - 1, \qquad v = 1 - 2\frac{r}{R} \tag{7.1}$$

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

$$u = \frac{4\theta}{\pi} - 1, \qquad v = -2\frac{z}{D} - 1 \tag{7.2}$$

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*

$$x = X(u, v), \quad y = Y(u, v), \quad z = Z(u, v) \tag{7.3}$$

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 7.5-7.8.

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 7.2. 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 7.2 shows the example where the side and bottom of the shaded quadrant in Figure 7.1 are each subdivided into four panels.

In addition to the requirement of geometric continuity within the domain of each patch, it is also necessary that the hydrodynamic solution should be continuous in the same domain. For this reason, if discontinous generalized modes are used, as in Test24 described in Appendix A.24, the modal discontinuities should coincide with boundaries between adjacent patches.

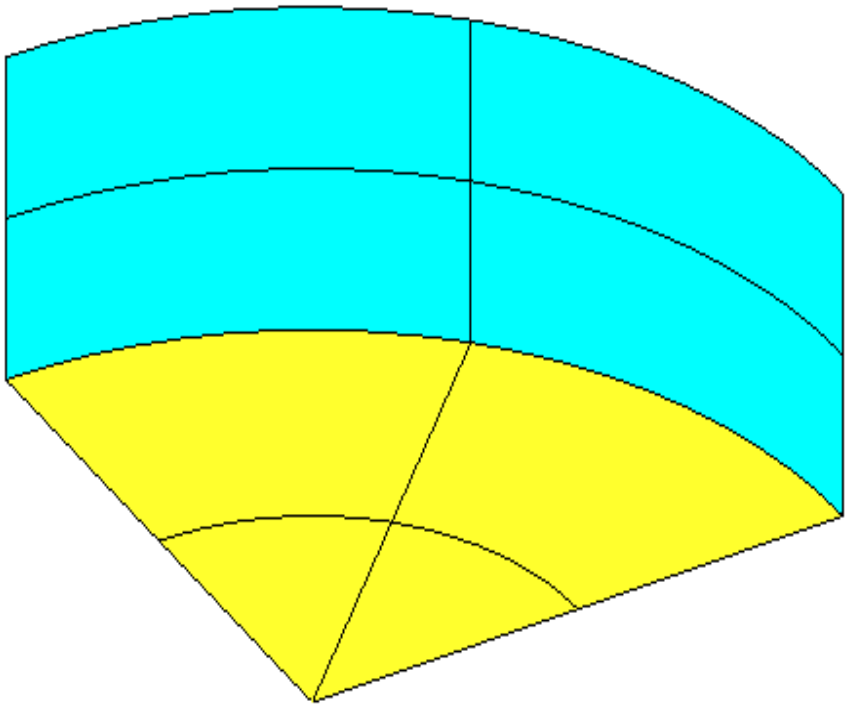


Figure 7.2: Subdivision of one quadrant of the cylinder shown in Figure 6,1 into panels. In this case $N_u = N_v = 2$ on both patches.

7.2 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

$$\phi(u,v) = \sum_{j=1}^{M_v} \sum_{i=1}^{M_u} \phi_{ij} U_i(u) V_j(v) \tag{7.4}$$

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 15. 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:

$$M_u = N_u + K_u - 1 \quad M_v = N_v + K_v - 1 \tag{7.5}$$

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.

7.3 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 15, 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 $(K_u + 1,\, K_v + 1)$.

7.4 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 7.5-7.8.)

The data on the first three lines are identical to the low-order method as described in Section 6.1. 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 7.1. 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 7.5, 7.6, 7.7 and 7.8:

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

7.5 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 6.1 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 7.4. Note that IGDEF=0 is assigned on line 4. The patch vertices ($X1, Y1, Z1$, ... , $X4, Y4, Z4$) are defined in precisely the same manner as the panel vertices in Section 6.1. The convention defined in Figure 6.1 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). 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.

7.6 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 7.2). 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. The domain of the parameters of the B-splines representing the geometry is not 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 (7.3) is represented on each patch in the tensor-product form

$$\mathbf{X}(u, v) = \sum_{j=1}^{M_v^{(g)}} \sum_{i=1}^{M_u^{(g)}} \mathbf{X}_{ij} U_i(u) V_j(v) \tag{7.6}$$

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 7.2.) As in (7.5),

$$M_u^{(g)} = N_u^{(g)} + K_u^{(g)} - 1 \quad M_v^{(g)} = N_v^{(g)} + K_v^{(g)} - 1 \tag{7.7}$$

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 (7.6). 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.

7.7 GEOMETRY REPRESENTED BY MULTISURF (IGDEF=2)

WAMIT 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 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 eight .dll files. 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	7Kb
real	RG2WAMIT.DLL	233Kb

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 7.4. 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.

NPATCH can be specified as 0; use of this option is recommended, to avoid errors in counting surfaces or patches. In this case the number of patches is evaluated from the MultiSurf model, and the user does not need to input NPATCH separately. If NPATCH>0 is input by the user, the number of MultiSurf *surfaces* used in the solution will be limited to NPATCH; thus input files for earlier versions of WAMIT can still be used without modification.

An import utility (File/ Import/ WAMIT GDF) has been added to MultiSurf[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)

7.8 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=0,1,2 are reserved for the options described in Sections 7.5-7.7, and thus IGDEF$\geq$ 3 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$\geq$ 3.

Continuing with the example of the circular cylinder shown in Figures 7.1 and 7.2, 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
INONUMAP
```

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 INONUMAP is used in subroutine CIRCYL to specify either uniform (INONUMAP=0) or nonuniform (INONUMAP=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, INONUMAP, and an appropriate header. As noted in Section 4.10, 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 10. The restriction DRAFT<HBOT must be imposed if NPATCH>1. Figure 7.3 illustrates the patch numbering to achieve this flexibility.

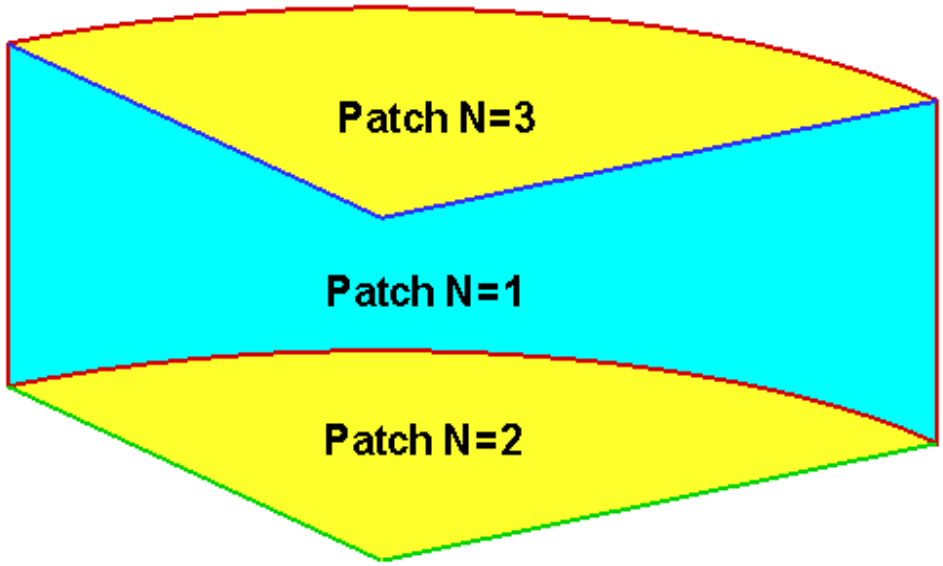


Figure 7.3: One quadrant of the cylinder shown in Figure 7.1 showing the patch numbering system which permits using the subroutine CIRCYL with NPATCH=1 (bottom-mounted caisson), NPATCH=2 (floating cylinder of finite draft), or NPATCH=3 (floating cylinder with a patch on the interior free surface to remove 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 are reserved for users. 33 body subroutines are included in the standard release of GEOMXACT.F and GEOMXACT.DLL, as 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. In addition there are 4 control-surface subroutines in the standard release of GEOMXACT, which are described in Chapter 11.

The following table lists the 33 body 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 INONUMAP
-4	ELLIPSOID	1,2	A,B,C
-5	BARGE	1,2,3,4	HALFLEN,HALFBEAM,DRAFT
-6	BARGEMP	6,7	HALFLEN,HALFBEAM,DRAFT XMP,YMP
-7	CYLMP	3,4	RADIUS,DRAFT,RADMP INONUMAP
-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
-13	AUV	2,3	RADIUS, DCYL, DTAIL
-14	SPAR2	3,4,5	RAD1,RAD2 DRAFT,SKIRT_HEIGHT
-15	SPHERXYZ	1	RADIUS,X0,Y0,Z0
-16	FPSO2	7,10	XBOW,XMID,XAFT HBEAM,HTRANSOM DRAFT,DTRANSOM INONUMAP
-17	FPSO12	7,10	XBOW,XMID,XAFT HBEAM,HTRANSOM DRAFT,DTRANSOM INONUMAP
-18	TORUS_ELLIP	1,2	RCIRC,RAXIS,DRAFT
-19	TORUS2	2	RCIRC1,RAXIS1,DRAFT1 RCIRC2,RAXIS2,DRAFT2
-20	CIRCCYLH	2,3	RADIUS,HALFLEN

IGDEF	SUBROUTINE	NPATCH	GDF INPUTS
-21	FPSOINT	-	XBOW,XMID,XAFT HBEAM,HTRANSOM DRAFT,DTRANSOM INONUMAP,NTANKS XVER
-22	CIRCCYL_ARRAY		RADIUS,DRAFT,ASPACE NX,NY,INONUMAP
-23	ELLIPINT	-	A,B,C NTANKS XVER
-24	GAPLID	1	X1,X2,GAP INONUMAP
-25	CYLFIN	2,3,4	RADIUS,DRAFT WIDTH INONUMAP
-26	CYLFIN4	4,6,9	RADIUS,DRAFT WIDTH INONUMAP
-27	SKEW_SPHERE	1,2	RADIUS,SKEW
-28	CIRCCYL_NOSYM	1,2,3	RADIUS,DRAFT INONUMAP XS,YS,ZS
-29	ELLIPSOID_NOSYM_TANK	-	A,B,C XS,YS,ZS XL,XB,XD,SL,SB,SD
-30	BARGE_INT	3	HALFLEN,HALFBEAM,DRAFT
-31	BARGENUC	12,13	HALFLEN,HALFBEAM,DRAFT,STRIP
-32	CCYLHSP		NSEG RADIUS XSEG
-33	CIRCCYL_MULTI		NCYL,INONUMAP, IBOT,IFS RADIUS(1),DRAFT(1),XC(1),YC(1) RADIUS(2),DRAFT(2),XC(2),YC(2) .... RADIUS(NCYL),DRAFT(NCYL), XC(NCYL),YC(NCYL)

The last column indicates the dimensions and other input parameters to be included in the GDF file. Where two or more lines of inputs are shown in the table the GDF file should follow the same format, as illustrated in the test runs.

Brief descriptions of each subroutine are given below. More specific information is included in the comments of each subroutine. These bodies can be combined for multiple-body analysis, as described in Chapter 8, 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). The optional parameter INONUMAP can be included to specify either uniform (INONUMAP=0) or nonuniform (INONUMAP=1) mapping. Uniform mapping is the default, and it is not necessary to include INONUMAP in this case. If INONUMAP=1 is specified the mapping in the azimuthal direction on the hemisphere is quadratic, to give a finer discretization close to the waterline. Similarly, if IRR=1, INONUMAP=1 defines a quadratic radial mapping on the interior free surface with finer discretization close to the waterline so that the interior and exterior discretizations are similar at this point.

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. In the simplest case NPATCH=3 patches are used to represent the end, side, and bottom on one quadrant. If NPATCH=1 and DRAFT=0.0 only the bottom is represented, corresponding to a rectangular lid in the free surface. 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. The optional parameter INONUMAP can be input in the GDF file to control nonuniform mapping (INONUMAP=1 or INONUMAP=2) or to replace the

flat bottom by a semi-circular profile (INONUMAP=3). The latter option is useful to avoid sharp corners.

TORUS defines a floating or submerged torus, as illustrated in Figure 7.4. 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$\times\sqrt{2}$ 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$\times\sqrt{2}$, 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.

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)

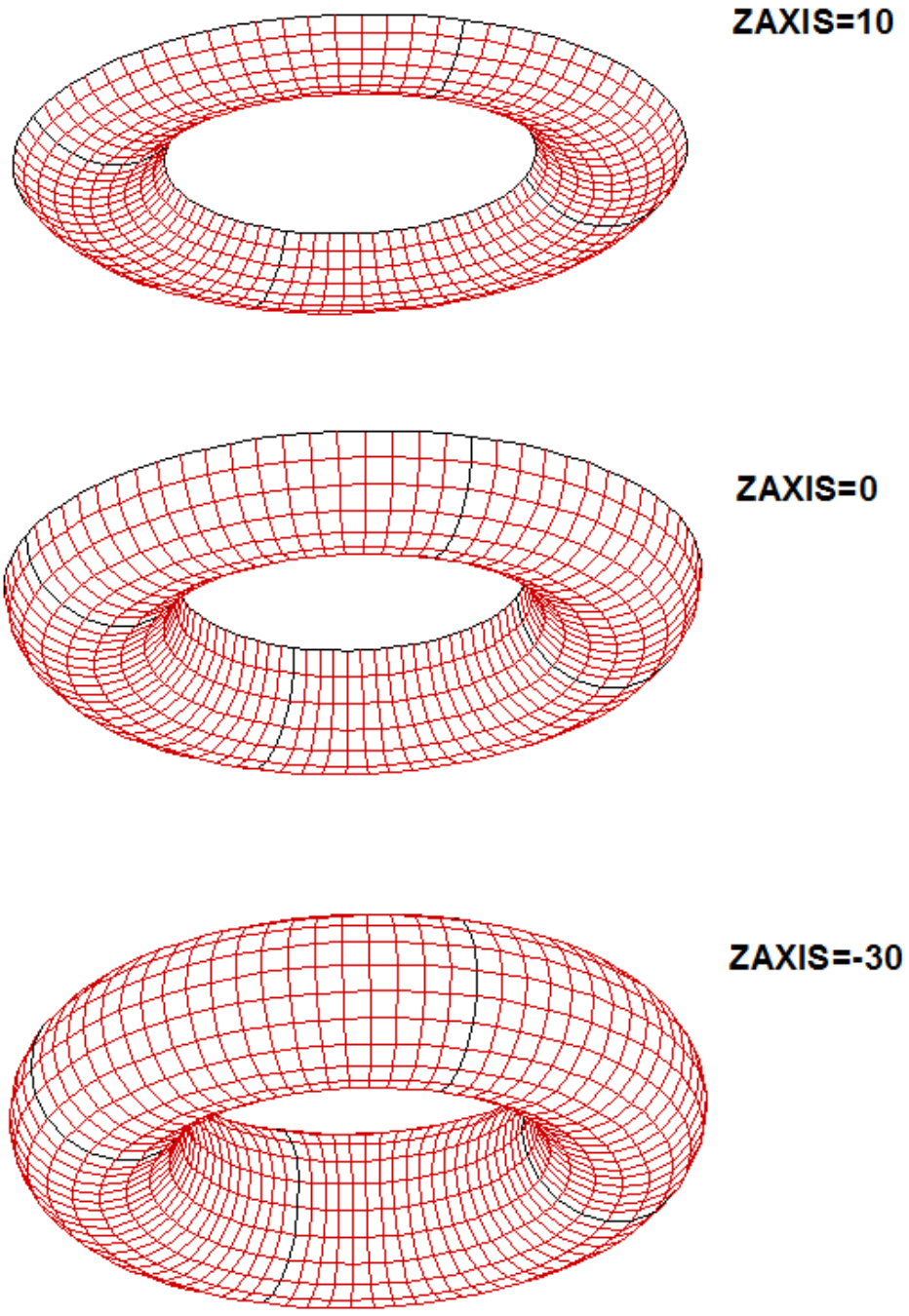


Figure 7.4: Perspective views of the torus, with RCIRC=20, RAXIS=60, and three different values of ZAXIS as shown. ZAXIS>0 in the top figure corresponds to the axis above the free surface. In the middle figure the axis is in the plane of the free surface, and the sections are semi-circles. In these two 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.

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.

AUV defines one quadrant of an axisymmetric submerged body with vertical axis. The body is defined by a hemispherical bow of radius RADIUS, conical tail of length DTAIL, and optional cylindrical midbody of length DCYL. The origin is at the center of the cylindrical midbody. If NPATCH=2 and DCYL=0.0 the midbody is omitted.

SPAR2 defines the first quadrant of a circular cylinder of radius RAD1 with a circular damping 'skirt' of radius RAD2 on the lower part of the cylinder. The lower surface of the skirt is in the plane of the bottom of the cylinder, at Z=-DRAFT, and SKIRT_HEIGHT is the height of the skirt. NPATCH=4 is the conventional case, NPATCH=5 includes the interior free surface for use with IRR=1, and NPATCH=3 can be used for a bottom-mounted structure.

SPHERXYZ defines a sphere of radius RADIUS, with its center at X0,Y0,Z0. If (-RADIUS < Z0 < RADIUS) the sphere is partially submerged, and if (Z0 < -RADIUS) it is completely submerged. If X0=0 ISX=1, and *vice versa*. If Y0=0 ISY=1, and *vice versa*.

FPSO2 defines an FPSO with one extra patch on the bottom in the bow to provide a more uniform mapping of the bottom relative to the subroutine FPSO described above. If NPATCH=10 the interior free surface is included for use with IRR=1. INONUMAP=0 gives a uniform mapping on all patches; INONUMAP=1 gives a nonuniform mapping with finer discretization near the chines; and INONUMAP=2 gives a nonuniform mapping with finer discretization near both the chines and waterline. Uniform mapping is used for the prismatic stern in all cases.

FPSO12 defines NBODY=2 FPSO'S with different dimensions. This subroutine illustrates the use of one subroutine to define multiple bodies of the same type with different dimensions. In all other respects it is the same as FPSO2 described above. The same value of INONUMAP must be used for both bodies.

TORUS_ELLIP Torus with elliptical sections. This subroutine is the same as TORUS, described above, except that the generating sections are elliptical with their centers in the free surface. The horizontal semi-axis of the ellipses is equal to RCIRC and the vertical semi-axis is equal to DRAFT. It is required that RAXIS>RCIRC, i.e. there is a free surface

in the center of the torus.

TORUS2 Two concentric toroids with elliptical sections. The dimensions of each toroid are as defined in subroutine TORUS_ELLIP above. It is required that RAXIS1>RCIRC1 and RAXIS2>(RAXIS1+RCIRC1+RCIRC2), i.e. there is a circular free surface in the center of the inner torus, and also an annular free surface between the toroids.

CIRCCYLH First quadrant of a circular cylinder with a horizontal axis in the free surface. RADIUS and HALFLEN are the radius and half-length of the cylinder. If NPATCH=3 the interior free surface is included for use with IRR=1.

FPSOINT FPSO with internal tanks of rectangular shape, as illustrated in TEST22. The dimensions and representation of the hull are the same as in subroutine FPSO2 described above. NTANKS is the number of tanks, which must be consistent with the inputs for NPTANK in the CFG file. XVER is the array of vertex coordinates for each tank, input in the same format as in Section 7.5. The FPSO and tanks are symmetric about the Y=0 plane (ISX=0,ISY=1). Patches 1-7 represent the hull and 8-10 represent the interior free surface if IRR=1, as in FPSO2. One half of each tank is represented by 4 additional patches.

CIRCCYL_ARRAY First quadrant of a rectangular array of circular cylinders, with radius RADIUS, draft DRAFT, and horizontal spacing ASPACE between the centers in the X- and Y- directions. The array is symmetric about X=0 and Y=0. NX*NY is the total number of cylinders in all four quadrants. NX must be even (no cylinders are in the plane X=0). NY may be odd or even. If NY is odd the middle row of cylinders is centered on the X-axis and only the upper half of each cylinder is represented. There are two patches for each cylinder, representing the side and bottom. If INONUMAP=1 nonuniform mapping is used with finer discretization near the corners and waterlines.

ELLIPINT defines one side of an ellipsoid with internal tanks. A,B,C are the semi-axes of the ellipsoid. ISX=0 and ISY=1 are used to permit the tanks to be assymmetrical about X=0. The tank vertices are defined by the array XVER as described above for FPSOINT. If IRR=1 the internal free surface is represented by patch 2.

GAPLID defines a rectangular 'lid' in the free surface with one patch. The lid extends from X=X1 to X=X2 and between Y=-GAP/2 and Y=+GAP/2. The entire surface of the lid is represented (ISX=ISY=0). Nonuniform discretization is used in the X-direction if INONUMAP=1, in the Y-direction if INONUMAP=2, and in both directions if INONUMAP=3. This subroutine can be used with appropriate generalized modes to establish an artifical damping lid on the free surface between two vessels.

CYLFIN defines the first 1 or 2 quadrants of a circular cylinder with symmetric fins in the plane x=0. RADIUS is the cylinder radius and WIDTH is the width of the fins. Patch 1 is the side of the cylinder in quadrant 1 and patch 2 is the fin (represented by a dipole patch) on the positive Y-axis. If NPATCH=3 the side of the cylinder in quadrant 2 is also represented. If RADIUS=0 and NPATCH=1 the cylinder is omitted and the subroutine defines the upper half of a single fin extending from Y=-WIDTH to Y=+WIDTH. INONUMAP is optional with default value 0. If INONUMAP=1 is input the discretization on the fins is finer near the outer ends, using a cosine-spacing transformation.

CYLFIN4 defines a circular cylinder with 4 symmetric fins represented by dipole patches in the planes X=0 and Y=0. DRAFT is the draft. The other dimensions and parameter INONUMAP are as defined for CYLFIN above. Arbitrary combinations of ISX and ISY can be specified. The number of patches is equal to 4 with two planes of symmetry, 6 with one plane of symmetry, and 9 with no planes of symmetry. The last patch is on the bottom, and can be omitted if the cylinder is bottom-mounted.

SKEW_SPHERE defines two quadrants of a floating skewed hemisphere. The center-plane of the body is inclined, at the position X=SKEW*Z. Patch 1 represents the body surface and patch 2 can be used to represent the interior free surface if IRR=1. ISX=0 and ISY=1.

CIRCCYL_NOSYM defines the entire surface of a circular cylinder. Patch 1 represents the side and patch 2 represents the bottom of the cylinder. If IRR=1 the internal free surface is represented by patch 3. If NPATCH=1 and DRAFT≥1.E-8 the cylinder is considered to be bottom-mounted and DRAFT must be equal to the parameter HBOT in the POT file. If NPATCH=1 and DRAFT<1.E-8 the cylinder is considered to be of zero draft and patch 1 represents the bottom. If INONUMAP=1 nonuniform mapping is used on the side and bottom, with finer discretization near the corner and waterline. The center of the waterplane of the cylinder is located at the position XS,YS,ZS relative to the body coordinate system.

ELLIPSOID_NOSYM_TANK 4 quadrants of ellipsoidal body with one tank, no planes of symmetry. A,B,C are the semi-axes of the ellipsoid. The center of the ellipsoid is at (XS,YS,ZS) with respect to the body coordinate system. XL,XB,XD are the length, width and depth of the tank. The center of the tank free surface is at the position (SL,SB,SD) relative to the center of the ellipsoid. The center of the ellipsoid must be in the plane of the free surface (only the lower half is represented).

BARGE_INT defines the first quadrant of a rectangular tank, equivalent to the interior surface of a rectangular barge. HALFLEN is half the length, HALFBEAM is half the width, and DRAFT is the tank depth.

BARGENUC defines the first quadrant of a rectangular barge with extra nonuniform patches near the corners at the ends. The inputs are the same as for subroutine BARGE as defined above, except for the additional parameter STRIP. Extra patches are added within strips of width STRIP on the end, side and bottom which adjoin the corners at the end. The mapping is nonuniform in this strip to give a finer discretization near the corners. There are four patches on the end, 4 patches on the side, and 4 patches on the bottom. The interior free surface is represented by patch 13 if IRR=1.

CCYLHSP defines the first quadrant of a vessel with semi-circular sections and horizontal axis. The vessel can be sub-divided into separate segments, to permit the analysis of a hinged structure using generalized modes. The ends of the vessel are spheroidal. The vessel has two planes of symmetry (ISX=ISY=1). NSEG is the number of segments, including cylindrical elements and the two spheroidal ends. The array XSEG defines the X-coordinate of the end of each segment (location of joints between adjacent segments). The array XSEG must be of dimension (NSEG+1)/2. If NPATCH=(NSEG+1)/2 only the submerged portion of the body is represented, with one patch for each segment; if

NPATCH=(NSEG+1)/2 + 2 the interior free surface is also represented by the last two patches (the first is a rectangle covering the interior of all the cylinders, and the second is a semi-ellipse covering the interior of the spheroidal end). (Note that, in accordance with FORTRAN convention, (NSEG+1)/2 is defined as the integer part of this fraction.)

- **CIRCCYL_MULTI** defines an array of circular cylinders with the radius, draft, and coordinates (XC,YC) of the axis input separately for each cylinder. The GDF parameter NCYL is the total number of cylinders defined in the GDF file. INONUMAP=0 or 1 specifies uniform or nonuniform mapping on the side and bottom of each cylinder. The parameters IBOT and IFS specify if a patch is to be included on the bottom (IBOT=1) or on the interior free surface (IFS=1). IFS=1 should be used if the irregular-frequency option IRR=1 is used. IBOT=0 can be used in the case of bottom-mounted cylinders, in which case the draft of each cylinder must be equal to the fluid depth. The inputs RADIUS,DRAFT,XC,YC must be included on a separate line for each cylinder. The number of patches is NPATCH=NCYL*(1+IBOT+IFS).

7.9 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. A more extensive library of subroutines is available for downloading from www.wamit.com.

Users of WAMIT 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. (A similar facility exists for the subroutine NEWMODES, to define generalized modes, as described in Chapter 9.)

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. The symmetry indices ISX,ISY, irregular-frequency parameter IRR and NPATCH have been added to the argument list of GEOMXACT, to permit use of these inputs in special cases.
- 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 procedure for doing this is similar to that described in Section 9.3 for NEWMODES.DLL. 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 14.7.

7.10 BODIES WITH THIN SUBMERGED ELEMENTS

The higher-order method can be used to analyze bodies which consist partially (or completely) of elements with zero thickness, as in the analogous extension of the low-order method described in Section 6.3.

In the higher-order method the patches representing these elements are referred to as 'dipole patches'. Dipole patches are represented in the same manner as the conventional body surface. 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. A positive difference is defined to act in the direction of the normal vector.

As an example, the floating spar shown in Figure 6.2 is analyzed 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 indices of the patches on the side are 1, 3, and 5, the strakes are 2,4, and 6, and the bottom is patch number 7.

When dipole patches are used, the mean drift force/moment can be evaluated by the momentum method (Option 8), and in some cases by the use of a control surface (Option 7), as described in Chapter 11. The direct pressure method (Option 9) cannot be used, and a warning message is output when this option 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

The following discussion applies in all cases except IGDEF= 2, *i.e.* exclusive of the option to use MultiSurf geometry as described in Section 7.7. The recommended procedure to define dipole patches with MultiSurf is explained in Appendix C. In that case the dipole patches are identified in the *gdf*.ms2 file output from MultiSurf and input to WAMIT. No reference to the dipole patches should be included in the CFG file.

To analyze bodies with zero-thickness elements, for IGDEF$\neq$ 2, the corresponding dipole patches are identified in the configuration files using the parameter NPDIPOLE. The indices of the dipole patches are defined by including one or more lines starting with 'NPDIPOLE=', followed by the indices or ranges of indices of the dipole patches, as explained in Section 4.7.

In earlier versions of WAMIT it was possible to define dipole patches in an alternative manner, by specifying extra data in the GDF file. That alternative is not supported in Version 7, as explained in Section 4.1. The utility `v6v7inp` described in Appendix B converts old GDF files using that alternative to the format required in Version 7, and adds the appropriate data for NPDIPOLE in the CFG file.

7.11 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 configuration files (See Section 4.7). 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 configuration files. 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 configuration files 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.

7.12 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 configuration files 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 configuration files. **It is important not to specify the same parameters in both the SPL and configuration 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.

7.13 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.

4. The higher-order method usually gives a more accurate evaluation of the free-surface elevation (runup) at the body waterline. This is particularly important when the mean drift forces are evaluated using a control surface, as described in Chapter 14.

Disadvantages:

1. The linear system which must be solved for the velocity potentials is not as well conditioned in the higher-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 8

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 9.) 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 15. 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, with their filenames listed in the POT file as shown in Sections 4.2. 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.

- Except for the special cases described in Section 8.5, 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.

When $N > 1$ walls can be defined using the procedure described in Section 12.4.

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.

The GEOMXACT subroutines described in Section 7.8 can be used for multiple bodies, but special attention is required unless different subroutines are used for each body, or 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. 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 8.1, 8.2, and 8.3.

The POT file is as shown in Section 4.2, with the last three lines of data repeated in sequence for each body.

8.1 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 4.3), is that the VCG and 3×3 matrix of each body's radii of gyration are entered in succession.

8.2 INPUT TO FORCE (IALTFRC=2)

If IALTFRC=2 the format of the FRC file is the same as described in Section 4.4 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 4.4, 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.

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 4.4.

8.3 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.

8.4 PARAMETERS IN THE CONFIGURATION FILES

The configuration files described in Section 4.7 include several inputs that must be assigned separate values for each body. These include IALTFRCN, IGENMDS, IRR, NEWMDS, NPTANK, XBODY and XTRIM. The procedure for doing this when NBODY>1 is to indicate the index of the corresponding body in parenthesis, as shown below. If IALTFRC=3 the array IALTFRCN must also be specified with values for each body, unless the default values 1 are applicable for all bodies. 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 8.3)
IALTFRCN=2 1 2  (Form of FRC file for each body)
IGENMDS(1)=-1  (Body 1: IGENMDS=-1)
IGENMDS(2)=1  (Body 2: IGENMDS=1)
IGENMDS(3)=2  (Body 3: IGENMDS=2)
IRR(1)=2  (Body 1: IRR=2)
IRR(2)=1  (Body 2: IRR=1)
IRR(3)=1  (Body 3: IRR=1)
ITRIMWL=1  Trim waterlines for at least one body
NPTANK(1)=(8-11) (12-15)  (Body 1: tank patches or panels)
XTRIM(1) = 1.0 15. 0.  (Body 1 trim coordinates)
XTRIM(2) = 0.0 0.0 0.0  (Body 2 trim coordinates)
XTRIM(3) = 5.0 0.0 10.0  (Body 3 trim coordinates)
```

Some of these inputs are illustrated in the test runs TEST05, TEST13 and TEST13a described in Appendix A.

- The array IALTFRCN specifies the value of IALTFRC for each separate body. The value of each element in this array must be either 1 or 2, as in the example above. If the number of elements in the configuration files is less than NBODY, the remaining elements are assigned the same value as the last input. Thus if NBODY is large, it is not necessary to define each element of the array explicitly. If IALTFRC is the same for all bodies, only the first body needs to be specified in IALTFRCN. The following alternative inputs are equivalent, for the case NBODY=8, to illustrate this rule in a situation where IALTFRC is not the same for all bodies:

  ```
  IALTFRCN=2 1 2 2 2 2 2 2
  IALTFRCN=2 1 2
  ```

  Different values of the irregular-frequency parameter IRR can be assigned for each body, as indicated in the example above, by including the body index in parenthesis. Alternatively, the same value of IRR can be input for all bodies (See Section 10.5).

  The parameter NPTANK identifies the internal tanks which are associated with the corresponding body. Thus the input NPTANK(1) defines the tanks to be in body 1. The associated inputs RHOTANK and ZTANK are identified by the number of each tank, and not by the body number. The number of each tank is defined by the order of the inputs NPTANK in the configuration files.

8.5 GLOBAL SYMMETRY INDICES

The geometric symmetry of each body is defined by the symmetry indices ISX,ISY in the corresponding GDF file, as explained in Sections 6.1 and 7.4. In solving the hydrodynamic problem for multiple bodies, including body interactions, it is necessary to consider the global symmetry, which is affected by the position and orientation of each body. Global symmetry exists only in cases where the combination of all bodies is symmetric about one or both planes.

In WAMIT Version 6 it was assumed for NBODY>1 that there are no planes of global symmetry. If one or more bodies are defined with planes of symmetry, the geometric inputs for these bodies are reflected about the original planes of symmetry by the program, to provide a complete description of the entire surface.

Starting in Version 7.0, global symmetry can be used to reduce the computational time, in special cases where the body symmetries and their orientations are suitable. These cases are described below. In all other cases, as in prior Versions, the global symmetry indices are set to (0 0) when NBODY>1. When reflection of the original body geometry is required, this is performed by the program.

We define ISG(i) as the vector of the global indices (i=1,2), and ISB(i,n) as the vector of body symmetries for each body (n=1,2,...,NBODY). XBODY(j,n) is the array specifying the global coordinates of the origin of each body (j=1,2,3), and its rotation (j=4) about the vertical axis, as explained in Section 4.2. The following rules are applied to determine the global symmetry indices:

- If XBODY(4,n) is nonzero for any body, ISG=(0 0)
- If ISB(1,n)=1 and XBODY(1,n)=0 and XBODY(4,n)=0 for all bodies, ISG(1)=1
- If ISB(2,n)=1 and XBODY(2,n)=0 and XBODY(4,n)=0 for all bodies, ISG(2)=1

To illustrate these rules, Test05 in Appendix A includes a cylinder and spheroid, both with two planes of geometric symmetry and located at different positions on the global x-axis. However in TEST05 (also in TEST13) the spheroid is rotated 90° about the vertical axis, and this prevents the use of global symmetry. In the modified Test05a, the two bodies have the same relative orientations (since the cylinder is axisymmetric) but their centers are on the global y-axis to avoid rotation of the spheroid about its vertical axis. Thus in TEST05a the global symmetry indices are (1 0) and computational times are reduced substantially.

When NBODY>1 both the global symmetry and body symmetry for each body are output in the .OUT file.

8.6 OUTPUT

The nondimensionalizations given in Chapter 3 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 9

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 9.5.

Each generalized mode is defined by specifying the normal velocity in the form

$$\varphi_{jn} = n_j = u_j n_x + v_j n_y + w_j n_z \tag{9.1}$$

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 3.2 and 3.3 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 examples are intended to illustrate applications. (The first four 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:

$$u_7 = 0, \quad v_7 = P_2(q), \quad w_7 = 0 \tag{9.2}$$

$$u_8 = 0, \quad v_8 = 0, \quad w_8 = P_2(q) \tag{9.3}$$

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:

$$u_7 = q^2, \quad v_7 = 0, \quad w_7 = 0 \tag{9.4}$$

$$u_8 = 6q^3 - 5q^2, \quad v_8 = 0, \quad w_8 = 0 \tag{9.5}$$

$$u_9 = 28q^4 - 42q^3 + 15q^2, \quad v_9 = 0, \quad w_9 = 0 \tag{9.6}$$

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

$$u_7 = zn_x\mathrm{sgn}(x), \quad v_7 = 0, \quad w_7 = |x|n_z \tag{9.7}$$

with NEWMDS=1. Here $\mathrm{sgn}(x)$ is equal to ± 1 according as $x > 0$ or $x < 0$. Test 24, described in Appendix A.24, is a more complicated example with multiple hinges.

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

$$u_7 = 1 \tag{9.8}$$

$$w_8 = 1 \tag{9.9}$$

$$u_9 = z \quad w_9 = -x \tag{9.10}$$

(With these definitions the exciting force coefficients can be evaluated either from the Haskind relations or direct pressure integrals as defined in Section 3.3, with the surface integrals extended over the ensemble of the central body and its images.)

5. A ship with a discontinuous shearing mode at the midship section, which can be used in conjunction with the fixed mode option (described in Section 4.5) to evaluate the shear force acting on the hull.

In cases such as examples 3 and 5 above where the modes are discontinuous, it is important when the higher-order method is used (ILOWHI=1) for the mode discontinuities to coincide with the boundaries between patches and not to occur within the interior of patches, as explained in Section 7.1.

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 can only 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 7.9).

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 9.1 the input files are described for performing the generalized mode analysis for a single body. Section 9.2 describes the use of DEFMOD, and Section 9.3 describes the alternative use of the DLL file NEWMODES. The definitions of hydrostatic restoring coefficients are described in Section 9.4 and the analysis of multiple bodies (NBODY>1) is described in Section 9.5.

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. TEST24 analyses the motions of a vessel with five separate segments connected by hinges. Further information is contained in the Appendix.

9.1 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. 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 9.2. 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 9.3.

Different values of IGENMDS may be assigned for each body if NBODY>1, and the body numbers must be included as shown in Section 8.4.

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 4.2). 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 7). 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 7 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 7 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.

9.2 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 can 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 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 (9.1) 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 9.1. 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 (9.12).

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.

9.3 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. Users can 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 7.9.

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.

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 for each generalized mode in an appropriate subroutine, 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. (Note

that (sway/roll) and (surge/pitch) have the same symmetries. Thus the symmetry indices (2/4) and (1/5) are redundant, and either value can be assigned.)

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. The 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 (9.1), 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. IPP is the *local* panel or patch number of the body, as listed in the corresponding GDF file.

When NEWMODES is used to define generalized modes, the input IGENMDS must be specified in the configuration files with a nonzero value. 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 14.7.

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. The following arguments have been added to NEWMODES to facilitate this procedure:

IFILEDLL is the unit number assigned by WAMIT to open and read the special data file. This unit number should be used in all cases so that there is no conflict with other files used by WAMIT for input and output.

FILENDLL is the filename *gdf* of the GDF input file *gdf*.GDF for the same body. This may be used optionally to distinguish between different special input data files. Examples are explained below for TEST23 and TEST24.

IERROR is the unit number assigned by WAMIT for the log files ERRORP.LOG and ERRORF.LOG, which are described in Section 10.1. 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.

IWAMLOG is the unit number assigned by WAMIT for the log file WAMITLOG.TXT. This unit number can be used to copy the special input data file to the log file, as is done for other input files by WAMIT.

The use of this procedure, with a special input data file, is illustrated in TEST23 and TEST24 which make use of the subroutines WAVEMAKER (IGENMDS=21) and HINGE_MODES (IGENMDS=22), respectively. In TEST23 the subroutine WAVEMAKER inputs the depth of the hinge axis from a special file TEST23_WMKRHINGE.DAT. In TEST24 the subroutine HINGE_MODES inputs data from the file TEST24_XHINGE.DAT. Users who modify the DLL subroutines or make new DLL subroutines can follow the code in these two subroutines for guidance in inputting data from an external file.

9.4 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)

$$c_{ij} = \rho g \iint_{S_b} n_j (w_i + z D_i) dS \tag{9.11}$$

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

$$c_{ij} = \rho g \iint_{S_b} n_j w_i dS \tag{9.12}$$

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 (9.11) 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 5.6. (Here *out* denotes the filename of the .out file for the run.)

Note that the hydrostatic coefficients associated with generalized modes do not include gravitational restoring due to the internal mass of the body. Nonzero restoring effects due to gravity should be included as 'external' force components in the stiffness matrix of the FRC file. The coupling between the generalized modes, and between the generalized and rigid-body modes, should be included. The gravitational force and moment associated with the rigid-body modes ($i, j \leq 6$) should not be included since these components are evaluated in the program.

It is not possible for the program to evaluate the hydrostatic matrix in a unique manner which applies to all possible situations, including the effects of the gravitational force associated with the internal mass of the body. The coefficients evaluated and used by the program are specifically those defined by the matrix of hydrostatic and gravitational restoring coefficients listed in Section 3.1 for the rigid-body modes and by the hydrostatic matrix (9.12) for the generalized modes. Users should input additional restoring coefficients as necessary using the EXSTIF matrix.

9.5 NBODY ANALYSIS

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 and IGENMDS must be specified for each body. Both parameters are defined in the configuration file using separate lines for each body with the syntax 'NEWMDS(m) = n' 'IGENMDS(m) = i'. Here m is the body index, n is the number of new modes for that body, and i is the value of IGENMDS appropriate for the same body. For example, if there are three bodies, and body 1 has 3 free beam bending modes defined by NEWMODES.DLL, body 2 has no generalized modes, and body 3 has 2 moonpool free-surface lids, four lines should be added to the configuration file as follows:

NEWMDS(1)=3
IGENMDS(1)=16
NEWMDS(3)=2
IGENMDS(3)=17

(It is not necessary to include the additional lines NEWMDS(2)=0 and IGENMDS(2)=0, since zero is assigned by default, but these explicit inputs may be added for clarity.)

In the approach described in Section 9.2, 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 9.2 for a single body.

In the approach described in Section 9.3, 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 ($j = 16, 17, ..., 23$).

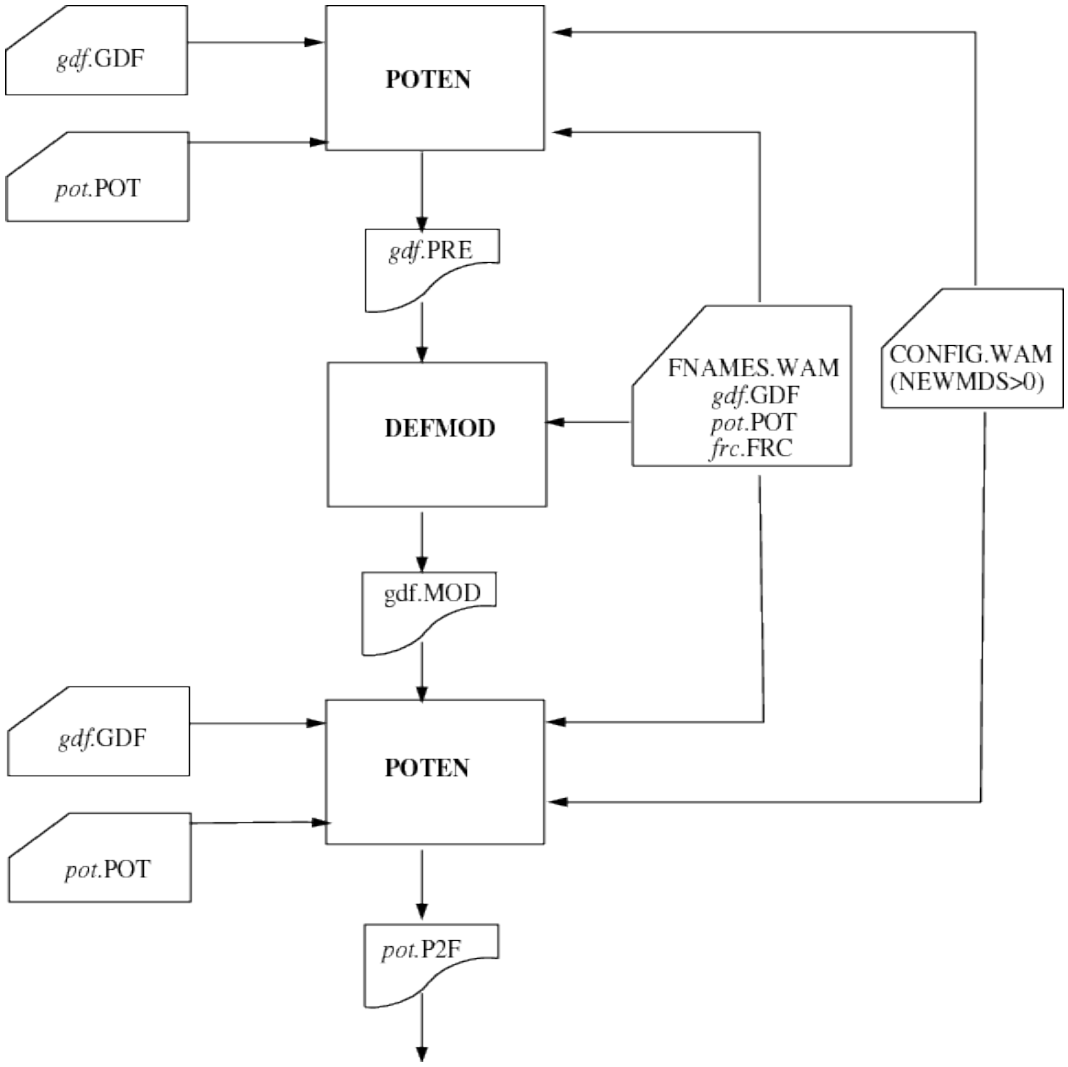


Figure 9.1: Flow chart showing the use of WAMIT and DEFMOD to define and analyze generalized modes in the low-order method (ILOWHI=0). The number of generalized modes NEWMDS is specified in the configuration file. On the first run of WAMIT the centroid coordinates, area, normal vector $\mathbf{n}$, and 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 10

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 15.7, 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 both the body surface and the interior free surface. The integer parameter IRR controls the removal of the effect of irregular frequencies. 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.

10.1 INPUT PARAMETERS

The parameter IRR is specified in the the configuration file as described in Section 4.7, with the default value zero. 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) can be used in the low-order method (ILOWHI=0). Only the values IRR=(0,1,3) can be used in the higher-order method (ILOWHI=1). The definitions of each value are listed below, with details given in the following sections.

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

IRR= 1: Remove the effect of the irregular frequencies. The geometrical description of the interior free surface is included in the GDF file.

IRR> 1: Remove the effect of the irregular frequencies. The geometrical description of the interior free surface is provided automatically by the program. These options are described in Sections 10.2-4 below.

- When NBODY>1 different values of IRR may be assigned for each body, following the format in Section 8.4. More specific information is given in Section 10.5.

  If IRR= 1 in the low-order method (ILOWHI=0) the user must discretize the free surface. The coordinates of these panel vertices are included in 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 panels, the parameter NPAN in the GDF file, is the sum of the number of panels on the wetted surface of the body and the interior free surface. The order of the indices for the body panels and free-surface panels is not restricted in Version 7.

  If IRR= 1 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 7.1, this is equivalent to requiring that the parametric coordinates (u, v) for the free-surface patch are defined such that the positive normal vector in the (u, v) plane points downward from the free surface into the body. The order of the indices for the free-surface patches and body patches is not restricted.

- The subroutine CIRCCYL in GEOMXACT.F is an example where the interior free surface is represented analytically as one extra patch. Several other subroutines in GEOMXACT.F include similar options, with one or more patches on the interior free surface, as explained in the headers of these subroutines.

  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 generation of free-surface patches can generally be facilitated by using MultiSurf, as described in Appendix C.

  In the low-order method, 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 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.

10.2 AUTOMATIC FREE-SURFACE DISCRETIZATION (IRR=2 and ILOWHI=0)

If IRR=2 is input in the low-order method (ILOWHI=0) the program projects the body panels onto the free surface to generate panels on the free surface. The GDF file contains only the body panels (it is the same as the GDF file used with IRR= 0).

IRR=2 should not be specified for a body such as a Tension Leg Platform or semi-sub with pontoons, or more generally for any body where a vertical line intersects the body surface more than once and thus the projection of two body panels overlap on the free surface. A warning message is issued for any panel where the inward normal vector slopes downward, away from the free surface, unless the panel is entirely in the plane of the free surface. Panels which are entirely in the plane of the free surface are ignored. Dipole panels are ignored.

This simple procedure is effective for bodies such as the circular cylinder used in Test01, consisting of flat horizontal panels on the bottom (which are projected up to the interior free surface), and vertical panels on the sides (which are ignored in defining the interior free surface). Since dipole panels are ignored, this procedure can be applied for the spar with helical strakes used in Test09.

10.3 AUTOMATIC FREE-SURFACE DISCRETIZATION (IRR=3 and ILOWHI=0)

If IRR=3 is input using the low-order method, 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. 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.

When ISOR=0 the interior free surface is discretized with triangular panels, based on the algorithm described in [8]. The sides of the triangular panels are similar to the average length of the waterline segments. An example is shown in the left figure of Figure 9.1.

When ISOR=1 the interior free surface is discretized in a regular pattern of quadrilateral panels, as shown in the right figure of Figure 9.1. Since the aspect ratio of the panels is $O(1)$, the number of panels on the interior free surface is not so large as in the case ISOR=0. Connectivity between the free surface panels and the waterline segments is important in this case, as 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

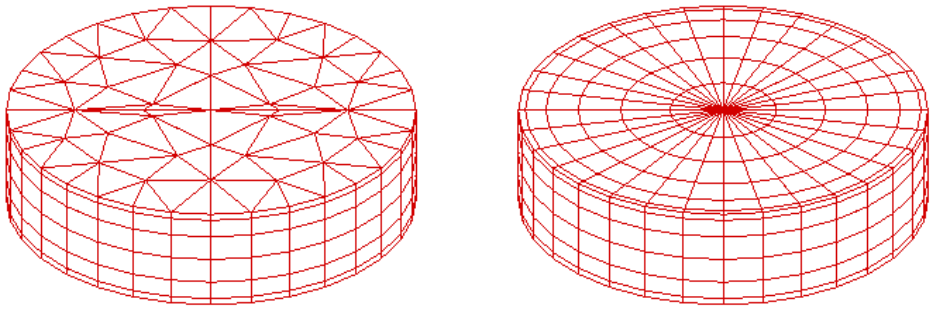


Figure 10.1: Automatic discretizations on the interior free surface of a circular cylinder. The figure on the left shows the result of the discretization algorithm used when ISOR=0. The figure on the right shows the result of the special algorithm used when ISOR=1 to ensure continuity of the panels adjacent to the waterline.

waterline segments to the centroid with more dense 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 option IRR=1 should be used.

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.

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) 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'. The parameter TOL=TOLGAPWL can be modified in the configuration files, as explained in Section 4.7.

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 TOLGAPWL. Modification of this parameter is potentially dangerous, if there are relatively short 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.

10.4 AUTOMATIC FREE-SURFACE DISCRETIZATION (IRR=3 and ILOWHI=1)

WAMIT includes the option to define the interior free surface automatically when the higher-order method is used (ILOWHI=1). This option can be used for multiple bodies, for bodies with trimmed waterlines, and for bodies with multiple waterlines such as TLP's and Semi-Subs. It cannot be used for bodies with internal waterlines, such as moonpools. This option also cannot be used for some bodies with abnormal waterlines. It is strongly recommended to visualize the interior free surface, using the auxiliary output files described in Section 5.7, to ensure that the representation of this surface is appropriate.

When IRR=3, no other changes in the inputs are required relative to IRR=0. The GDF file should specify the number of exterior body patches NPATCH, not including the patches on the interior free surface. The optional SPL file should also refer only to the exterior body patches. After reading the GDF data the program searches for all patches which have one side in the free surface. These sides are connected into closed waterlines, or waterlines which end on one or two symmetry planes. The waterplane interior to each waterline is divided into patches which are bounded by the vertices of the exterior patches on the waterline, and by an interior axis on or near the center of the waterplane. The interior patches are mapped into parametric (u, v) coordinates with $u = 1$ on the axis, $u = -1$ on the waterline, and v increasing from -1 to +1 as one moves along the waterline

[1] The cumulative length is defined here as the length of the vector sum of the waterline segments

- in the positive direction (counterclockwise when viewed from above the free surface). The configuration parameter TOLGAPWL can be modified if necessary to define the allowable gaps between adjacent patches on the waterline.

  Two alternative mappings are used to relate the Cartesian body coordinates x, y and parametric coordinates u, v, depending on the shape of the waterline along the outer edge of the patch. If the slope of this waterline segment changes sufficiently (greater than .5 radians over half of the segment) a polar mapping is used. Otherwise a ruled mapping is used, spanning the half-width between the axis and the waterline. When ruled mapping is used the axis must be parallel to the plane $y = 0$, as in the conventional choice of coordinates of ship-like hulls. Thus, when such bodies are to be analyzed, the longitudinal axis of the body should be parallel to the $x - axis$.

  In general there is one interior patch for each exterior waterline patch. However if one or both ends of the waterline are straight lines within 0.1 radians of being perpendicular to the mapping axis, this segment is defined as the side v=-1 or v=+1 of the adjacent interior patch. Various examples of interior patches and panels are shown in Figure 10.2, corresponding to several of the standard test runs in the Appendix.

  When the interior free surface patches are defined by the program, the spline control parameters assigned for each patch are the same as those input for the associated external patch, except that the parameter NU is increased or decreased based on the relative widths of the two patches. The parameter NV is the same as the corresponding exterior parameter (either NU or NV, depending on the orientation of the exterior patch), so the panel subdivisions are the same along the waterline. The spline control parameters for the interior patches are displayed in the header of the OUT file, when NBODY=1, following the panels of the exterior body surface. However when NBODY>1 these parameters are not output.

  When the program connects adjacent sides of patches in the waterline, the patches are identified based on the coincidence or close proximity of their vertices. The parameter TOLGAPWL is used for this purpose, to allow for small gaps (or overlaps) between adjacent patches at the waterline. The default value TOLGAPWL=10^{-3} is used unless a different value of this parameter is defined in the CFG file, as explained in Section 3.7. In the test for adjacent patches the nondimensional Cartesian coordinates of the adjacent patch corners are evaluated, and the distance between these points is computed. The patches are assumed to be connected if this distance is less than either TOLGAPWL or the product of TOLGAPWL and the maximum length of one of the patch sides. The latter value is introduced to allow for cases where the size of the structure is much larger than the characteristic length ULEN. The default value is recommended in general. If the program is unable to close a waterline using this value, an error message is displayed stating that the waterline is not closed. In that case a larger value of TOLGAPWL should be input in the CFG file.

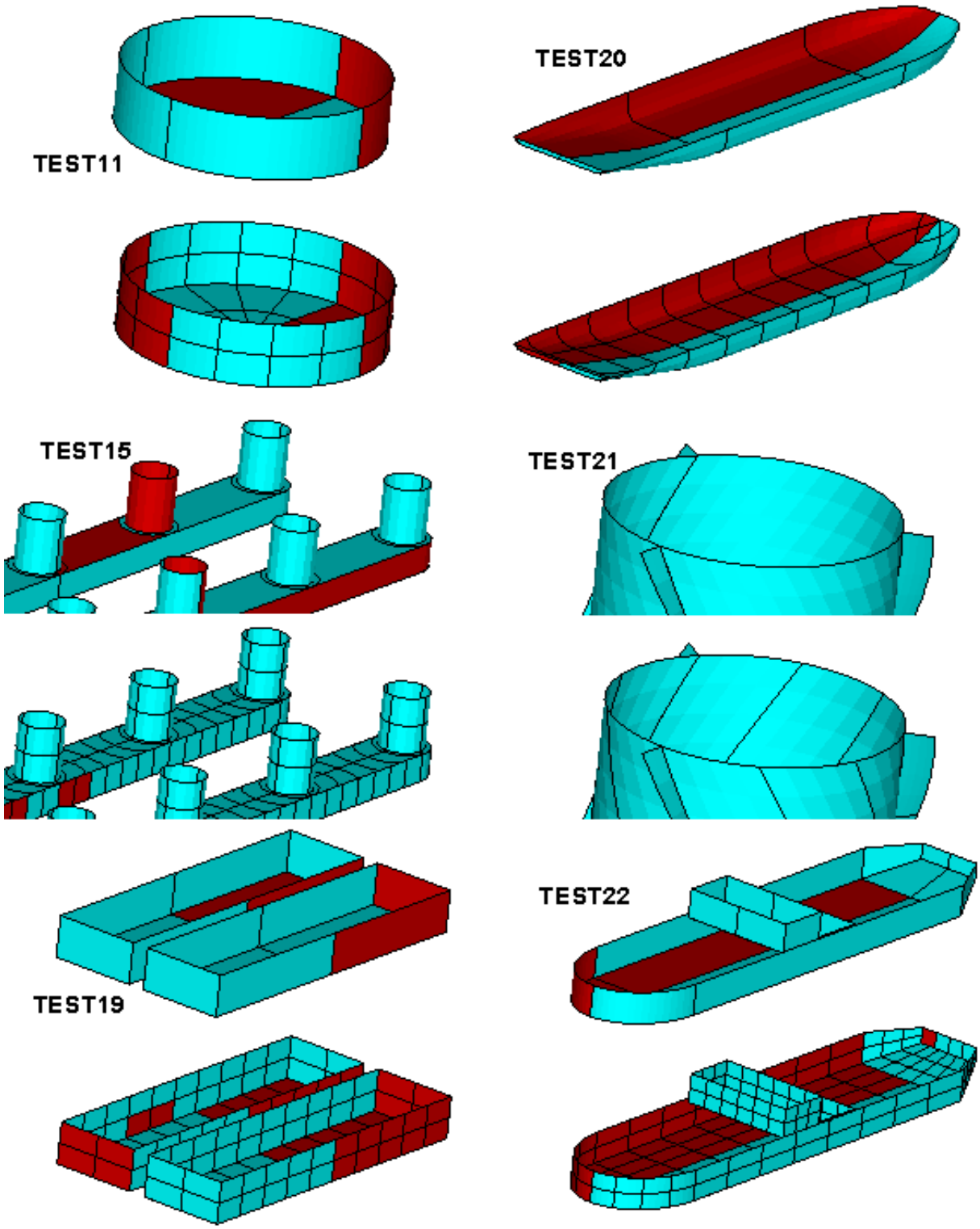


Figure 10.2: Automatic discretizations on the interior free surface for various higher-oder test runs, corresponding to the inputs described in Appendix A but with IRR=3. See also the perspective figures for Test13a in Appendix A. In Test15 two patches are required on the middle columns, due to the symmetry plane $x = 0$, whereas the other interior free surfaces are covered by a single patch. In Test22 the interior free surface covers the entire interior of the waterline, but it is obscured by the forward tank.

10.5 ASSIGNING DIFFERENT VALUES OF IRR FOR NBODY>1

It is possible to use different values of IRR for each body in a multibody analysis. The definition of IRR for each body is the same as in Sections 10.1-10.4. The format for inputting different values of IRR for each body, in the CFG file, is shown in Section 8.4. If the same value of IRR is used for all bodies it is not necessary to use the notation 'IRR(n)=', and one scalar value with the notation 'IRR=' can be input.

If the special notation 'IRR(n)=' is used in the CFG file, it is recommended to input values for each body explicitly, even if IRR=0 for some bodies. (The order of these inputs is arbitrary.) However this procedure is not required. If less than NBODY values of IRR(n) are included in the CFG file, the rules are as follows:

- Explicit lines of input must be included for IRR(n) with $n = 1, 2, ..., N$, where $N \leq$NBODY.
- If $N <$NBODY, the remaining elements for $N < n \leq$NBODY are assigned with the same value as IRR(N)

For example, if NBODY=3 the inputs IRR(1)=1 and IRR(2)=3 will result in IRR(3)=3, whereas if IRR(1)=1 is the only input then all three elements are assigned IRR=1. Assigning only IRR(2) or IRR(3) will result in an error message and halt excution of the program.

Test13a illustrates this scheme in the case where the cylinder is trimmed, and the spheroid is untrimmed. The interior free surface of the spheroid is generated by the subroutine ELLIPSOID in GEOMXACT.F. Since the cylinder is trimmed it cannot use the standard definition of the interior free surface generated by the subroutine CIRCYL. As shown in Appendix A.13, the GDF and SPL inputs for the spheroid are the same as one would used with IRR=1, whereas the inputs for the cylinder specify IRR=3 and do not include the interior free surface. In this case the interior free surface of the cylinder is elliptical, due to the trim angle, and different values of NU are assigned on the deep and shallow sides due to the corresponding depths of the exterior patches.

Chapter 11

MEAN DRIFT FORCES USING CONTROL SURFACES

- If IOPTN(7)>0 in the FRC file the mean drift force and moment are evaluated from the control-surface momentum-flux method, with output in the OPTN.7 or $frc.7$ file. (In WAMIT Version 6 this Option was designated as Option 9c.) 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 force and moment, on a single body or on each one of multiple bodies, are evaluated as in the pressure integration method, 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 force and moment using a control surface are evaluated by one of two alternative methods. These two alternatives are analytically equivalent. In Alternative 1, based on equations (15.57) and (15.58) the contribution from the integral on the waterline (WL) is transferred 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 the waterline integral. However, Alternative 2, based on equations (15.59) and (15.60) must be used when CL is very close to the body. This is because the evaluation of the momentum flux (which involves the pressure and/or velocity) is not accurate very close to the body. An example for which the Alternative 2 may be required is when the gap between two adjacent bodies is very small. The evaluations of the vertical drift force and horizontal components of the drift moment are identical in these two alternative methods.

- The option to use Alternative 1 or Alternative 2 is controlled by the parameter IALTCSF

in the configuration file, as explained in Section 4.7. The default value is IALTCSF=1.

It is possible to use two separate control surface files to represent the inner free surface and the remaining outer portion of the control surface. This procedure is described in Section 10.4.

It is also possible to define the control surface automatically, as described in Section 10.5. When this is possible it avoids most of the effort required to define the control surface.

To evaluate the mean drift forces and moments using this method, the Control Surface File (CSF) 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. This simplification is illustrated in Test22, as described in Appendix A.22.

If thin submerged elements are represented by dipole panels or patches, the mean drift force and moment cannot be evaluated by direct pressure integration on the body. The alternative method using a control surface is valid in this case, with some exceptions. If the dipole elements are entirely below the free surface, both Alternatives 1 and 2 can be used. If the dipole elements intersect the free surface, as in the case of the spar with helical strakes shown in Appendix A.21, Alternative 1 must be used and only the horizontal drift force and vertical drift moment can be evaluated correctly.

The drift force and moment evaluated using a control surface are defined in terms of the body coordinates, as in the case of direct pressure integration.

The control surface can be defined using MultiSurf, as explained in Appendix C.

- The mean drift forces can be evaluated using control surfaces without evaluating the mean drift forces using pressure integration (Option 9). If the low-order method is used (ILOWHI=0), it is not necessary to use the source formulation. However the potential formulation (ISOR=0) may give inaccurate results for the vertical components of the drift force and horizontal components of the drift moment, since these require evaluations of the fluid velocity close to and on the body waterline. Thus the option to use ISOR=0 with ILOWHI=0 to evaluate the Option 7 drift forces should be restricted to applications where only the horizontal drift force and vertical moment are required.

11.1 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 represented by quadrilateral panels in the same manner as described for the body in Chapter 6. In the higher-order method, ILOWHICSF=1 is assigned on line 2 of the CSF file and the control surface is represented in the same manner as described for the body in Chapter 7, using any of the options available for higher-order representation of the body surface including flat panels, B-splines, MS2 files and 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 coordinate system, using the same unit of length. The normal vector is defined to point into the interior of the fluid domain between the control surface and the body, i.e. toward 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 third-order Gauss quadratures. The subdivision of each patch into panels is controlled by the parameters NUC,NVC in the same way as the parameters NU,NV are used on the body surface (See Chapter 7).

The input files for the test runs 5, 13, and 22 in Appendix A illustrate the different methods for defining the control surfaces. 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, and different values of

ILOWHICSF can be used for multiple bodies, as illustrated in TEST05.

One or two planes of symmetry can be used to simplify the definition of the control surface, as specified by the symmetry indices ISXCSF and ISYCSF in the CSF file. These are analogous to the indices ISX and ISY in the body GDF file. In most cases the same values must be used in the two files (ISXCSF=ISX and ISYCSF=ISY). Certain exceptions should be noted:

- If the CSF is defined by the user, as described in Sections 11.2 and 11.3 below, and if the intermediate free surface is not required (as explained above), a symmetric CSF can be used regardless of the body symmetries.
- If the CSF is defined by the user, as described in Sections 11.3 and 11.4 below, and if the body is submerged (or in unusual cases where the body is not symmetric but the waterline is symmetric), a symmetric CSF can be used regardless of the body symmetries.

If a control surface is defined with one or two planes of symmetry, and the flow is not symmetric about these planes (for example due to multiple-body interactions), the program reflects the control surface.

11.2 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 the vertex coordinates of the panels, in the same format as in the GDF file.

The data in the CSF file is 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 6.1.

11.3 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 7.5-7.8.)

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:

ICDEF = 0: The geometry of each patch is a flat quadrilateral, with vertices listed in the CSF file (cf. Section 7.5).

ICDEF = 1: The geometry of each patch is represented by B-splines, with the corresponding data in the CSF file (cf. Section 7.6).

ICDEF = 2: The geometry is defined by inputs from a MultiSurf .ms2 file (cf. Section 7.7).

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 7.8).

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 7). 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 NUC,NVC in the file *gdf*.CSP, in an analogous manner to the use of the parameters NU,NV in the spline control file *gdf*.SPL (Section 7.11). TEST22 is an example where the latter procedure is used.

Four special CSF subroutines are included in the standard GEOMXACT.F and GEOMXACT.DLL files to define control surfaces including the intermediate free surface for bodies with circular or elliptical waterlines. The following table lists these subroutines, which are described in more detail below:

ICDEF	SUBROUTINE	NPATCH	CSF INPUTS
-1001	CCYL_CS	3	RADIUS,DRAFT,RADIUSI
-1002	CCYL_CS_NOSYSM	3	RADIUS,DRAFT,RADIUSI
			XS,YS,ZS
-1003	ELLIPSOID_CS	2	A,B,C
			AI,BI
-1004	ELLIPSOID_CS_NOSYM	2	A,B,C
			AI,BI
			XS,YS,ZS

The last column indicates the dimensions and other input parameters to be included in the GDF file. Where two or more lines of inputs is shown in the table the CSF file should follow the same format, as illustrated in the test runs.

Brief descriptions of each subroutine are given below. More specific information is included in the comments of each subroutine.

CCYL_CS defines one quadrant of a circular cylindrical control surface of outer radius RADIUS and depth DRAFT. The inner boundary of the free surface is bounded by a circular body waterline of radius RADIUSI. ISXCSF=1 and ISYCSF=1 should be specified in the CSF file. This subroutine is used for the circular cylinder in TEST13.

CCYL_CS_NOSYSM defines the entire curface of a circular cylindrical control surface as described above for subroutine CCYL_CS. The center of the waterplane is shifted by XS,YS,ZS relative to the body coordinate system. ISXCSF=0 and ISYCSF=0 should be specified in the CSF file.

ELLIPSOID_CS defines one quadrant of an ellipsoidal outer control surface with semi-axes A,B,C, plus an elliptical annular free surface with interior boundary an ellipse with semi-axes AI,BI. The center of the ellipsoid is at the origin of the body coordinate system. ISXCSF=1 and ISYCSF=1 should be specified in the CSF file. This subroutine is used for the spheroid in TEST13.

ELLIPSOID_CS_NOSYM defines a complete ellipsoidal outer control surface and annular free surface. The center of the waterplane is shifted by XS,YS,ZS relative to the body coordinate system. ISXCSF=0 and ISYCSF=0 should be specified in the CSF file.

In these subroutines the parametric coordinate U is related to the polar angle about the vertical axis, and the parameteric coordinate V is related to the radial coordinates on the free surface and bottom and the vertical coordinate on the side of the cylinder.

11.4 COMBINING TWO CONTROL SURFACE FILES

In some cases it is convenient to use separate CSF files to represent the 'inner' part on the free surface and the 'outer' part which forms the remainder of the closed control surface below the free surface. For example, a low-order surface is generally used when the body surface is defined by low-order panels, in order to provide a good fit at the waterline. On the other hand, a higher-order outer surface has the advantage of more efficient integration using Gauss quadratures, to reduce the computational time. Thus, in this situation, it may be advantageous to use ILOWHICSF=0 for the inner free surface and ILOWHICSF=1 for the outer part of the control surface.

In this case the file *gdf*.csf is input in the following form:

```
header
2          ILOWHICSF
fname1.csf
fname2.csf
```

Here `fname1` and `fname2` are the filenames of the two separate CSF files. Note that the special value ILOWHICSF=2 is used for this purpose. No other data should be included in this file. The two separate CSF files are as described in Sections 11.2 and 11.3, except that one file should represent only the inner free surface, and the other should represent only the outer surface together with the submerged part of the control surface. Both surfaces should meet on the common outer waterline. The order of the two files is arbitrary, e.g. `fname1` and `fname2` can correspond to the inner and outer surfaces, or vice versa.

When two CSF files are used to represent one control surface, the symmetry indices ISXCSF and ISYCSF should be the same in both files. If ILOWHICSF=1 is used for one or both files, the parameter PSZCSF may be assigned independently in each file. If PSZCSF is negative, the optional .csp file should be used to assign the parameters NUC,NVC. Note that the filename(s) of the .csp file(s) should be the same as `fname1` or `fname2`.

The output files described in Section 11.6 for visualization of the control surfaces and for low-order control surfaces correspond to the complete control surface, with the same filename as the gdf file. Thus these output files are the same as in the case where only one CSF file is used.

11.5 AUTOMATIC CONTROL SURFACES

For bodies with 'regular' waterlines it is possible to define the control surface automatically, during the run. The higher-order option ILOWHICSF=1 must be specified for the control surface, but either the low-order (ILOWHI=0) or higher-order (ILOWHI=1) options can be used for the body geometry and solution. When this procedure is used the outer part of the control surface is defined either as a circular cylinder of specified radius and depth, or by a quadrilateral 'box', usually rectangular, with specified waterline vertices and depth. The program automatically defines the intermediate free surface, between the outer boundary and the body waterline(s), by first tracing the waterlines and then establishing appropriate patches to cover the area of the free surface between the waterlines and the outer boundary.

This procedure can be used with multiple waterlines, as in the case of a TLP, semi-sub, or catamaran. In these cases it is necessary to define quadrilateral partitions which separate the waterlines and serve to define the outer boundaries of local patches surrounding each waterline. Examples of these inputs are shown below for the TLP and semi-sub. When partitions are used for this purpose, certain restrictions must be followed:

- The patches defined by the partitions and the waterline of the outer control surface must cover the free surface with no gaps or overlaps.
- Each partition, as well as the outer waterline, must obey the rule that as one progresses in the positive direction from one vertex to the next, one passes around the waterline in a counter-clockwise direction as viewed from above the free surface.
- When multiple waterlines are defined by the GDF inputs, the order of the waterlines and the order of the partitions must correspond, with the same number of each. For example in TEST15 the columns of the semi-sub are defined starting at the midship section X=0 and moving out toward the bow, following the patch definitions of the subroutine SEMISUB in GEOMXACT; in this case the partitions in the CSF file must follow the same order, as shown in Example 4 below. A proper sequence of the patches and panels must be followed in the GDF file and geometry definition: all indices of the patches or panels belonging to each waterline must be either smaller or larger than all indices of the patches or panels belonging to the other waterlines. The order of these indices defines the order of the waterlines, and the partitions defined in the CSF file must follow the same order.
- When partitions are used to separate multiple waterlines the program will extend (or contract) the outer ends of the partitions so that they are located at the intersections with the outer boundary. Thus, as shown in Figures 11.1 and 11.2, it is not necessary for the user to compute the exact coordinates of the intersections. In these two Figures the coordinates of the intersections are obvious, but in some cases extra computation would be required, and the program is intended to perform this computation automatically. However in some cases the program may not identify end points which should be moved to the outer boundary, and the user should verify that this has been done correctly by plotting the data in the auxiliary file *gdf*_`csf.dat`. This problem

can be avoided by specifying the correct vertex coordinates in the CSF file, for the outer ends of the partitions.

Automatic definition of the intermediate free surface may fail in some cases where the waterlines are irregular. Examples of 'irregular' waterlines where the automatic option may fail include: (1) locally concave waterlines, (2) moonpools, (3) bodies with thin elements (dipole patches) which intersect the free surface (e.g. TEST21), and (4) bodies with horizontal patches or panels in the plane of the free surface. It is advisable to confirm the representation of automatic control surfaces by plotting the data in the auxiliary file *gdf*_csf.dat. In cases where only the horizontal components of the drift force and vertical component of the drift moment are required, the intermediate free surface can be omitted.

The option to use two separate CSF files, described in Section 11.4 above, should not be used for automatic control surfaces.

When automatic representation of the control surface is used, special values must be assigned to the parameters NPATCSF and ICDEF, as defined below, and the complete CSF file should be of the following format:

```
header
1 (ILOWHICSF)
ISXCSF  ISYCSF
NPATCSF  ICDEF  PSZCSF
RADIUS  DEPTH
NPART
NV(1)
X(1,1) Y(1,1)
X(1,2) Y(1,2)
.
.
.
X(1,NV1) Y(1,NV1)
NV(2)
X(2,1) Y(2,1)
X(2,2) Y(2,2)
.
.
.
X(2,NV2) Y(2,NV2)
NV(NPART)
X(NPART,1) Y(NPART,1)
X(NPART,2) Y(NPART,2)
.
.
.
X(NPART,NV1) Y(NPART,NV1)
```

The symmetry indices **ISXCSF** and **ISYCSF** of the control surface must be the same as the symmetry indices ISX,ISY of the body. Here the body symmetry indices ISX,ISY are the same as the inputs in the GDF file, except in the case where the body is trimmed.

Special attention to symmetry is required if the body waterline is trimmed, since this may affect the symmetry of the body. If the trim includes a roll angle, XTRIM(3) is nonzero and ISY=0 regardless of the GDF input symmetry index. In this case ISYCSF=0 must be specified in the CSF file. Similarly, if the trim includes a pitch angle, XTRIM(2) is nonzero and ISXCSF=0 must be specified.

For cases where NBODY>1 and one or more bodies have planes of symmetry, as specified in the GDF file, and no trim angles are specified for the body, then the same planes of symmetry should be specified for the control surface (regardless of the fact that no symmetry is used for the potential solution). In this case the control surface is reflected in the same manner as the body. Tests 05 and 13 are examples of this convention.

If incorrect symmetry indices are input in the CSF file an error message is issued and the run is terminated.

NPATCSF must be equal to zero or less than zero:

NPATCSF=0: the control surface is automatic and includes the intermediate free surface

NPATCSF<0 the control surface is automatic and the intermediate free surface is omitted

ICDEF must be equal to zero:

ICDEF=0: the control surface is automatic

RADIUS is the parameter which controls the radius of a circular outer surface, or specifies that the outer surface is quadrilateral:

RADIUS>0: the outer surface is a circular cylinder of this radius

RADIUS$\leq$0: the boundary of the outer surface is quadrilateral with NV(1) vertices specified by the coordinates (X(1,n), Y(1,n)) (n=1,2,...,NV(1)).

DEPTH is the depth of the control surface. This must be a positive real number, greater than the maximum depth (draft) of the body.

NPART is an integer which specifies the number of partition boundaries. Each partition boundary includes NV vertices, defined by the coordinates X,Y. Partition boundaries are required for two possible purposes, (a) to define the outer boundary of a quadrilateral control surface, and (b) to separate multiple waterlines. If the body has only one waterline and the outer boundary is circular, NPART=0. If the body has only one waterline and the outer boundary is a quadrilateral, NPART=1. When the outer boundary is a quadrilateral its vertices must be defined by the first partition, with NV(1) vertices, and other partition boundaries (if any) must be included after this in the file.

The vertex coordinates (X,Y) must be ordered so the partition boundaries enclose the waterlines in the 'counter-clockwise' direction, as illustrated in the examples below.

In cases where there are no planes of symmetry, each waterline is a closed curve which must be surrounded by a closed partition boundary. When there is more than one waterline the partition boundaries must not overlap, nor should gaps exist between them.

In cases where the body is symmetric about X=0 and/or Y=0, and the same symmetry planes are used for the CSF, the completeness of the partition boundaries depends on whether or not the waterline intersects the planes of symmetry. If the waterline is entirely within the interior of a quadrant or half-space and does not intersect the symmetry plane(s), then it must be completely enclosed by a partition boundary. If the waterline intersects a symmetry plane then the partition boundary should not include that plane, since it will be closed by reflection about the plane.

For patches on the free surface, the parametric coordinates are defined with U=+1 on the body waterline and U=-1 on the outer partition boundary; V is positive in the counter-clockwise direction around body waterline. When a circular cylindrical outer surface is used (IPARTR=1) U is in the azimuthal direction and V is in the vertical direction on the side and radial direction on the bottom.

Several examples of .csf files are included below to illustrate the use of partition boundaries. (Header lines are omitted for brevity.)

Example 1: single waterline with circular outer boundary, as in TEST05c and TEST13c:

```
1          ILOWHICSF
1  1       ISX  ISY
0  0  1.   NPATCSF  ICDEF  PSZCSF
1.2 2.2    RADIUS, DEPTH
0          NPART
```

Example 2: single waterline with quadrilateral outer boundary, as in TEST05s and TEST13s:

```
1             ILOWHICSF
1  1          ISX  ISY
0  0  0.5    NPATCSF  ICDEF  PSZCSF
0.0   0.3    RADIUS, DEPTH
1             NPART
3             NV(1)
2.2  0.0
2.2  0.3
0.0  0.3    (X,Y coordinates of three vertices)
```

Note in Example 2 the partition boundary is specified only in quadrant one, since ISX=1 and ISY=1. The first vertex is on the $+x-$axis, the second vertex is above the first, and the third vertex is on the $+y-$axis to close the partition and surround the body waterline. The order of the vertices is such that they follow a counter-clockwise progression around the body with increasing polar angle relative to a point inside the body.

Example 3: TLP or semi-sub with four columns and two planes of symmetry with a circular outer boundary, as in TEST07 and TEST14:

```
1   ILOWHICSF
1  1        ISX  ISY
0   0  10. NPATCSF  ICDEF  PSZCSF
85.0  40.0    RADIUS, DEPTH
 1   NPART
 3   nv1
  0.0  50.0
  0.0  0.0
 30.0  0.0  (X,Y coordinates of vertices)
```

In this case, since the waterline is a closed curve in the interior of quadrant one, the partition boundary is required to separate the waterline in quadrant one from the other three waterlines of the TLP. The partition boundary is down, along the $+y-$axis, and then to the right along the $+x-$axis, in accordance with the 'counter-clockwise rule'. This example illustrates a useful feature that the outermost points do not need to intersect the outer boundary; the program extends or reduces the first and last segments automatically, to intersect the outer boundary (the inputs 50.0 and 30.0 could be replaced by any positive numbers).

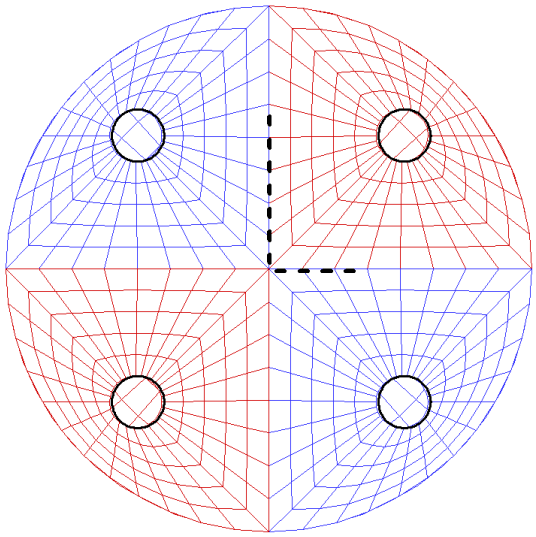


Figure 11.1: View from above the free surface showing four quadrants of the TLP control surface with a circular outer boundary. The partition boundary is represented by the heavy dashed line, as input in the CSF file shown above for Example 3, before it is extended by the program to intersect the outer boundary.

Example 4: ten waterlines, two and a half in each quadrant, with rectangular outer boundary, as in TEST15 (semi-sub with a total of ten columns):

```
1            ILOWHICSF
1  1         ISX  ISY
0   0  10.   NPATCSF  ICDEF  PSZCSF
0.0  40.0    RADIUS, DEPTH
 4   NPART
 3   NV(1)
 150.0   0.0
 150.0  60.0
   0.0  60.0   end of partition 1 (outer boundary of control surface)
 3   NV(2)
  0.0  0.0
 30.0  0.0
 30.0  50.0   end of partition 2 (encloses middle half-column)
 4   NV(3)
 30.0  50.0
 30.0  0.0
 90.0  0.0
 90.0  50.0   end of partition 3 (encloses second column)
 3   NV(4)
 90.0  50.0
 90.0  0.0
 150.0  0.0   end of partition 4 (encloses third column)
```

In this case the first partition boundary defines the outer rectangular boundary of the control surface, and the other three are required to separate the waterlines in quadrant one. Note that partition 2 starts at the origin in this case, without a segment along the $y-$axis, since only half of the middle column is in the first quadrant and the images of both the body and partition form a closed waterline and partition. As in Example 3 the outermost points do not need to intersect the outer boundary; the program extends or reduces the first and last segments automatically, to intersect the outer boundary (the inputs 50.0 are replaced by 60.0). For the last vertex (150, 0) the X-coordinate could be replaced by any value greater than 90.0. All four partitions obey the 'counter-clockwise rule' with respect to their domains. Also note that they completely define the interior free surface, without gaps or overlap. In this example, if only the horizontal drift forces and vertical drift moment were required, one could assign NPATCSF=-1 and NPART=1, and omit all but the first partition.

The view of this control surface from above the free surface is shown in Figure 11.2.

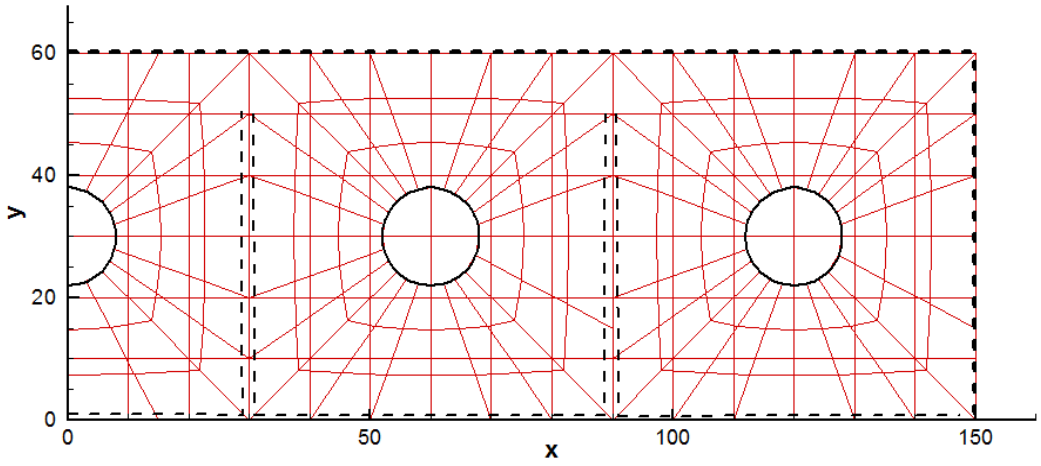


Figure 11.2: View from above the free surface showing one quadrant of the control surface described in Example 4. The rectangular outer boundary and partition boundaries are represented by the heavy dashed lines. Gaps between the partition boundaries are included for clarity, and are not present in the CSF file.

Example 5: Monohull with one plane of symmetry, as in TEST22 (FPSO with internal tanks) using a rectangular outer control surface:

```
1            ILOWHICSF
0  1         ISX  ISY
0   0  2.    NPATCSF  ICDEF  PSZCSF   (1st two indicate this is automatic)
 1   NPART
 4   nv0
 12.0   0.0
 12.0   3.0
-12.0   3.0
-12.0   0.0
```

In this case the outer rectangular boundary has four vertices, starting on the $+x$ axis and ending on the $-x$ axis.

When automatic representation of the CSF is implemented, the program traces the body waterline(s) and establishes extra patches or panels on the free surface between these waterlines and the partition boundaries. When the program connects adjacent sides of patches in the waterline, the patches or panels are identified based on the coincidence or close proximity of their vertices. The parameter TOLGAPWL is used for this purpose, to allow for small gaps (or overlaps) between adjacent patches at the waterline. The default value TOLGAPWL=10^{-3} is used unless a different value of this parameter is defined in the CFG file, as explained in Section 4.7. In the test for adjacent patches the nondimensional Cartesian coordinates of the adjacent patch corners are evaluated, and the distance between these points is computed. The patches are assumed to be connected if this distance is less than either TOLGAPWL or the product of TOLGAPWL and the maximum length of one of the patch sides. The latter value is introduced to allow for cases where the size of the structure is much larger than the characteristic length ULEN. The default value is recommended in general. If the program is unable to close a waterline using this value, an error message is displayed stating that the waterline is not closed. The default value is recommended in general. If the program is unable to close a waterline using this value, an error message is displayed stating that the waterline is not closed. In that case a larger value of TOLGAPWL should be input in the CFG file.

The automatic CSF option should not be used for a body which is totally submerged. This case can be handled more simply, by using one of the GEOMXACT subroutines which represent closed bodies with an additional patch on the interior free surface, as one would do to remove irregular frequencies (IRR=1). This option is included in the GEOMXACT subroutines CIRCCYL, ELLIPCYL, SPHERE, ELLIPSOID, or BARGE, as explained in Section 7.8. Alternatively, one can use ICDEF=0 as explained above in Section 11.3, and include one or more quadrilateral patches to represent the interior free surface.

11.6 OUTPUT

The mean forces and moments using a control surface are output in the OUT file and in the numeric output file `optn.7` or *frc*`.7` in the same format as used for pressure integration (Option 9).

When IPLTDAT> 0 in the CFG file, the auxiliary file *gdf*_`csf.dat` is output. This can be used for visualization of the control surface, using TecPlot or other similar programs. As in the case of the corresponding data files for visualizing the body surface the data in this file are defined with respect to the global coordinate system, and when NBODY>1 there is only one output file with the filename associated to the first body.

When ILOWHICSF= 1 and ILOWGDF> 0 in the CFG file, the file *gdf*_`low.csf` is output. This contains the data of the control surface in the low-order form described in Section 11.2.

TEST05, TEST13 and TEST22 show examples of using the control surface in the evaluation of the mean forces and moments.

Chapter 12

SPECIAL EXTENSIONS

12.1 INTERNAL TANKS

WAMIT includes 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 is 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 or 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 6.1 (ILOWHI=0) and Section 7.1 (ILOWHI=1). In the context of a tank, this means that the positive normal vector points out of the tank and into the space exterior to the tank. 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 7.5-7.9. In both cases the tank panels or patches are identified by their starting and ending indices, which must be listed in the configuration files using the parameter NPTANK as explained in Section 4.7.

TEST22 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.

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. Two alternative options can be used to specify the elevations of the tank free surfaces, depending on the parameters ZTANKFS and ITRIMWL in the CFG file.

If ITRIMWL=0, or if ZTANKFS is not included in the CFG file, then for each tank the free surface is defined to coincide with the highest point of the panel vertices or patch corners defining that tank. This option is illustrated in TEST 22, where 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 by 1m above this plane.

If ITRIMWL=1, and the array ZTANKFS is included in the CFG file, then for each tank the free surface is defined to be at the elevation above $Z = 0$ specified by the corresponding element of ZTANKFS. This option can be used to vary the filling ratio of tanks, without changing the GDF inputs, and also to trim the waterlines of the tanks so that the free surfaces remain level when the vessel is trimmed. Further information is given in Section 12.2. This option is illustrated in TEST22A, where the vessel is given a heel (static roll) angle of 15 degrees. When this option is used, the geometry of the internal tank surfaces must be defined up to (or above) the trimmed free surface of the tank.

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 4.7:

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, and each pair must be enclosed in parenthesis. 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 numbers for each body containing tanks must be appended to the parameter name. Only integer data and parenthesis are read for the array NPTANK. Other ASCII characters may be included on these lines, and are ignored when the input data is read.

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 3 and Section 4.4. 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.

The following are equivalent formats for the required lines in the file TEST22.CFG:

```
NPTANK= (8-11) (12-15) (patch/panel indices for two tanks)
RHOTANK= 0.6 0.6 (fluid densities for tanks one and two)


NPTANK= (8 11)    (patch/panel indices for tank one)
NPTANK= (12 15)   (patch/panel 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.

The real array **ZTANKFS** can be included in the configuration files, as explained in Section 4.7, to specify the free-surface elevations in internal tanks. The data in this array define the elevations of the tank free surfaces, above the plane Z= 0 of the exterior free surface. The data in the array ZTANKFS 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 'ZTANKFS='. If the array ZTANKFS is included, it must include one real number for each tank. If the array ZTANKFS is not included, the waterline of each tank is derived from the highest vertex of the GDF inputs. ZTANKFS is only used when the waterline trimming parameter ITRIMWL=1, as

explained in Section 12.2. The data in ZTANKFS are dimensional, with the same units of length as are used in the GDF file.

Special inputs are required when pressures and/or velocities are evaluated for field points inside the tanks (Option 6). In this case, when the field point coordinates XFIELD are input as explained in Section 4.3, the parameter ITANKFPT=1 must be specified in the .cfg file, as explained in Section 4.7, 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 4.11, 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.

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 ZTANKFS, 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. When ITRIMWL=0 or ZTANKFS is not included in the CFG file, the values of ZTANKFS displayed in the header of the .out file correspond to the tank free-surface elevations as used in the program, derived from the upper boundary of the tank surfaces.

Hydrodynamic parameters which are physically relevant for the tanks alone (Options 1,5,6,9) 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 5,6,9 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 conservation 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 15.10. The hydrostatic matrix, which is output in the *out* and *hst* files, includes the effects of the tanks. For example the hydrostatic coefficient C_{33} in the output files is equal to the total area of the waterplane inside the waterline of the body, minus 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

When planes of symmetry are specified in the GDF file (ISX=1 and/or ISY=1) the tank geometry is reflected in the same manner as the hull. This procedure should only be used if all of the tanks intersect the symmetry plane (with half of the tank on each side of the plane). For example in the case of the FPSO shown in Appendix A.22, where both tanks

extend across the plane of symmetry Y=0, it is appropriate to use ISY=1 as is done in TEST22. On the other hand if there are separate tanks on each side of the symmetry plane (e.g. wing tanks) the hydrostatic and hydrodnamic coefficients for the tanks will not be physically correct. This situation can be avoided by defining both sides of the hull and both tanks separately in the GDF file, and setting ISY=0.

12.2 TRIMMED WATERLINES

WAMIT includes the option to specify a 'trimmed waterline' for each body. In the trimmed condition, the static orientation of the body is shifted relative to the horizontal plane of the free surface, first by a prescribed vertical elevation, then by a pitch angle (often referred to as the 'trim angle'), and then by a roll angle ('heel'). For a given body geometry, different planes of flotation can be analyzed without redefining the geometry in the GDF file. A necessary condition for this procedure to be implemented is that the definition of the body geometry in the GDF file must include all surface elements which are submerged when the body is trimmed. Thus it is necessary to define a sufficient portion of the body surface above the untrimmed waterplane.

Trimming of the waterline is supported in both the low-order (ILOWHI=0) and higher-order (ILOWHI=1) methods. There are some practical restrictions that must be considered when the higher-order option is used, as described below.

The trimmed condition of the body is specified by two parameters in the configuration files, ITRIMWL and XTRIM, as defined in Section 4.7. ITRIMWL=0 is the default, indicating that the body is not trimmed. ITRIMWL=1 specifies that the body is trimmed, and XTRIM(1:3) specifies the vertical elevation and angles of rotation of the body, as explained below. If ITRIMWL=0, XTRIM is ignored. Note that if ITRIMWL=0 (or ITRIMWL is not included in the inputs), then the GDF file must only define the submerged portion of the body as in previous versions of WAMIT; in this case the program checks to ensure that no elements of the body surface are above the plane of the free surface, and an error stop occurs if such elements are detected. Conversely, when ITRIMWL=1, elements of the body surface above Z=0 are permitted, and there is no check or error stop. It is possible to suppress the error stop, without trimming, with the inputs ITRIMWL=1 and (XTRIM=0.0 0.0 0.0).

Appendix A includes four test runs with trimmed waterlines: TEST01A, TEST09A, TEST13A, and TEST22A. The perspective plots which accompany these descriptions illustrate the trimmed conditions of the structures.

When multiple bodies are analyzed, the vector XTRIM must be input separately for each body, following the same format as for XBODY in Section 8.4. TEST13A illustrates this procedure for NBODY=2.

The following coordinate systems and input parameters should be considered in defining the process of trimming waterlines:

(X,Y,Z) are global coordinates with Z=0 in the plane of the free surface.

(x,y,z) are conventional body coordinates, which define the submerged body surface in WAMIT.

(ξ, η, ζ), referred to hereafter as 'GDF coordinates', are the coordinates used to define the body geometry in the GDF file.

In the low-order method (ILOWHI=0), when the panel vertices are input from the GDF file, these are converted from GDF coordinates to body coordinates, using the translation and rotations defined by XTRIM. When the higher-order method is used (ILOWHI=1),

the same coordinate transformations are performed from the subroutine outputs each time that the subroutine defining the body surface is called. The practical effect, in both cases, is that the original definition of the body geometry in the GDF file is replaced during the WAMIT computations by a new transformed description in (x,y,z) coordinates, which represents only the submerged portion of the body in the trimmed condition. This submerged surface can be plotted and visualized, in the same manner as for untrimmed bodies, using the parameter IPLTDAT as explained in Section 4.7.

The transformation from GDF coordinates to body coordinates is defined by the following relations:

$$\begin{aligned} x = &\quad \xi c_2 + \eta s_2 s_3 + \zeta s_2 c_3 \\ y = &\quad \eta c_3 - \zeta s_3 \\ z = &\quad -\xi s_2 + \eta c_2 s_3 + \zeta c_2 c_3 + \mathrm{XTRIM}(1) \end{aligned}$$

where

$$\begin{aligned} c_2 = &\quad \cos(\mathrm{XTRIM}(2)), \quad s_2 = \quad \sin(\mathrm{XTRIM}(2)) \\ c_3 = &\quad \cos(\mathrm{XTRIM}(3)), \quad s_3 = \quad \sin(\mathrm{XTRIM}(3)) \end{aligned}$$

Except for the GDF coordinates, the definitions of coordinate systems, etc., are as defined in earlier chapters. In particular, the body motions and forces are defined with respect to the conventional body coordinates (x,y,z). Thus, for example, surge is in the horizontal direction and heave is in the vertical direction, relative to the free surface. Rotations and moments are defined with respect to the origin of this coordinate system. XBODY(1), XBODY(2), XBODY(3) are the (X,Y,Z) coordinates of the origin of the body coordinate system relative to the global coordinate system, and XBODY(4) is the angle in degrees of the x-axis relative to the X-axis of the global system in the counterclockwise sense, as shown in Figure 6.2.

The input parameter XTRIM(1), which defines the vertical coordinate z of the origin of the GDF coordinates, is in the same units as the dimensional GDF coordinates, corresponding to the parameter GRAV in the GDF file. The angles XTRIM(2) (pitch) and XTRIM(3) (roll) are in degrees. XTRIM(2:3) are Euler angles, with the convention that the body is first pitched about the transverse η-axis, and then rolled about the longitudinal ξ-axis. (Yaw can be included via XBODY(4). With this convention the projection of the ξ-axis on the plane Z=0 is at the same angle XBODY(4) as the projection of the x-axis.)

In the low-order method (ILOWHI=0), the input panels from the GDF file are tested for their positions relative to the plane of the free surface (Z=0). 'Dry' panels, which are entirely above the free surface, are removed from the array of panel coordinates within the program. 'Wet' panels, which are entirely submerged, are retained without modification. Panels which intersect the waterline, referred to as 'waterline panels', are trimmed at the waterline. In some cases, where waterline panels have three vertices below the free surface and one vertex above the free surface, the resulting trimmed panel is a polygon with five sides; in these cases the panel is subdivided to form two separate panels (one quadrilateral and one triangular). The number of panels is increased by one for each subdivided panel. Converesely, when dry panels are removed, the number of panels is decreased. Examples

of subdivided panels can be seen in the perspective plot corresponding to TEST01A in Appendix A.

In the higher-order method (ILOWHI=1), an analogous procedure is followed for each patch of the body, based on the vertical positions of the four corners of each patch. If the patch intersects the waterline, an iterative procedure is used to remap the submerged portion onto a square domain in parametric space. There are situations where this procedure breaks down, and thus it must be used with caution. One example is where the vertices of a patch are all submerged, but a portion of one side maps in physical space above the free surface.

Body symmetry can be affected by trimming. When it is necessary, WAMIT automatically reflects the body geometry and resets the symmetry indices. For example in TEST01A the GDF file specifies ISX=ISY=1, but the trimmed body has only one plane of symmetry (x=0). Similarly in TEST22A the GDF file specifies ISY=1, but the trimmed body has no planes of symmetry due to the heel angle.

Dipole panels are trimmed in the same manner as conventional panels, as illustrated in TEST09A.

Special attention is required for bodies with internal tanks. Two alternative options are included, as explained in Section 12.1, depending on the parameter array ZBODYFS in the CFG file. If ZBODYFS is not included in the CFG file (or if ITRIMWL=0), the geometry of internal tanks is *not* trimmed, and the tank free surface is always defined by the highest points of the specified panels or patches. If trimming includes a pitch or roll displacement of the body, this approach requires the tank geometry to be redefined so that its upper boundary coincides with the correct plane of the free surface inside the tank. If ZBODYFS is included in the CFG file (and if ITRIMWL=1), the geometry of internal tanks is trimmed using the same algorithms as for the exterior surface. In this case it is necessary to define the tank geometry up to the level of the trimmed free surface, as in TEST22A.

Special attention is required also when irregular-frequency removal is used (IRR>0). If the interior free surface is defined by the user as part of the overall geometry of the body (IRR=1) it is necessary to ensure that the interior free surface coincides with the Z=0 plane after trimming. If automatic discretization of the interior free surface is used (IRR=3), the interior free surface is defined correctly in the plane of the trimmed free surface. The option (IRR=2) cannot be used when ITRIMWL>0 (See Section 10.2).

It is strongly recommended to plot and visualize the geometry after trimming, using the data files which are defined by the parameter IPLTDAT in the CFG file, to ensure that the actual trimmed structure is correct.

12.3 RADIATED WAVES FROM WAVEMAKERS IN TANK WALLS

Different options are available for the analysis of problems where walls are present in the plane(s) of symmetry $x = 0$ and/or $y = 0$. The analysis of wave radiation by wavemakers in the plane(s) of the walls is discussed in this Section. A more general approach applicable to wavemakers in the planes of the walls and/or bodies in the interior domain of the fluid is described in Section 12.4. These two approaches require somewhat different inputs.

Either one tank wall of infinite width, or two semi-infinite walls at right angles, can be considered. These correspond to one or both of the symmetry planes $x = 0$ and/or $y = 0$. Wavemakers with prescribed normal velocities are located in the walls. The opposite walls of the wave tank are assumed to have absorbing beaches, represented here by open domains extending to infinity. 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 7.5.

The parameter ISOLVE=-1 is used in the configuration files to indicate that the wavemakers are in planes of symmetry, and that the solution of the integral equation is not required. Suitable generalized modes must be defined, as explained in Chapter 9, 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 the potential and fluid velocity at specified field points (Option 6). If the field point is on the free surface the potential is equivalent to the wave elevation, as explained in Chapter 3. No other Options in the FRC file 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 5.2. A post-processor should be used to combine these outputs, for arbitrary combinations of simultaneous motion of all wavemakers.

The subroutine WAVEMAKER is included in the NEWMODES DLL file, to analyze one or more wavemakers. If IGENMDS=21 is specified in the configuration files the wavemakers

are hinged with pitching motions about a horizontal axis. The depth of this axis is specified in the input file *gdf*_`wmkerhinge.dat` where *gdf* is the filename of the GDF file. 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.

The subroutine WAVEMAKER includes two additional options, specified by IGENMDS=211 and IGENMDS=212. If IGENMDS=211 each wavemaker is like a piston, with constant normal velocity on its surface. If IGENMDS=212 a bank of contiguous wavemakers are joined by vertical hinges, and represented by tent functions with constant normal velocity in the vertical direction. Further information is given in the header of this subroutine.

TEST23, described in Appendix A.23, illustrates the use of the option IGENMDS=21 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 `test23_wmkerhinge.dat`.

12.4 BODIES AND WAVEMAKERS WITH 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. The presence of walls is specified by the optional integer inputs IWALLX0 and IWALLY0 in the configuration files. In the default case these inputs are assigned the values zero, signifying that there are no walls. The inputs IWALLX0=1 and/or IWALLY0=1 signify that there are walls in the planes $x = 0$ and/or $y = 0$, respectively. All other inputs are unchanged from the case without walls. In particular the GDF files for one or more bodies should specify the symmetry indices ISX,ISY corresponding to the planes of symmetry of the body, with values 0 or 1, just as in the case without walls.

Figure 12-1 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 4.2 must coincide with the wall coordinate system as defined in Figure 12.1.

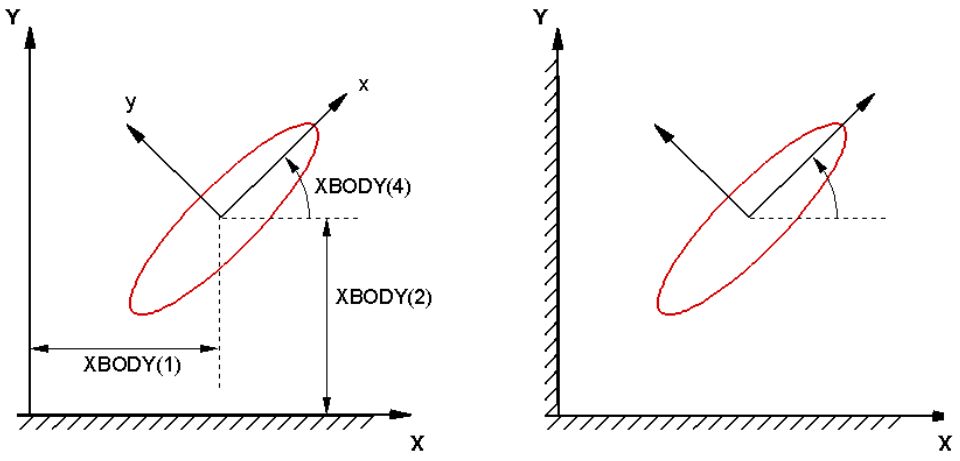


Figure 12.1: Definition sketch of coordinates

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 12.1. 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. If only one wall is present then, 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). If two walls are present, the resulting wave motion after reflection from both walls will have a maximum elevation of $4A$.

In the Force Control File the array IOPTN is unchanged from the definitions in Section 4.3. 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. IOPTN(7) can be used provided the control surface is within the fluid domain. The Haskind wave heading angles **BETAH** are defined with respect to the walls in the same manner as BETA. The coordinates of field points **XFIELD** where the pressure, wave elevation, and velocity are evaluated, are defined as in Section 4.3 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 3 are unchanged.

Wavemakers may be included in the walls, following a similar procedure as described in Section 12.3. When bodies and wavemakers are both present, ISOLVE≥0 must be specified. The wavemakers are treated as one or more separate bodies. In the simplest case, where one body is in the fluid domain and all wavemakers are defined within another GDF file, NBODY=2 is used and the parameter IBODYW=2 is defined in the CFG file, indicating that body 2 consists of wavemakers in the walls. The order of the bodies is specified by the order of the corresponding inputs in the POT file. There can be one or more physical bodies, and one or more wavemaker bodies. The physical bodies must precede the wavemaker bodies in all cases. Thus if K=1,2,...,NBDYP are the indices for NBDYP physical bodies, and K=NBDYP+1,NBDYP+2,...,NBDYP+NBDYW are the indices for NBDYW wavemakers, NBODY=NBDYP+NBDYW is the total number of bodies identified in the input files and IBODYW=NBDYP+1 is the index of the first wavemaker body, input in the configuration files.

Only the radiation problems are solved when wavemakers are present, including radiation from the wavemakers as generalized modes. Outputs with wavemakers include the added mass and damping matrices (Option 1), a surrogate for the Haskind exciting force to be explained below (Option 2), RAO's in response to the Option 2 exciting force and moment (Option 4), as well as Options 5,6,7.

The 'surrogate' exciting force is the force and moment acting on the body due to the motions of the wavemakers, computed from the cross-coupling added mass and damping coefficients where one mode is for the body and the other mode is for the wavemaker. The

outputs for the exciting force (Option 2) and RAO (Option 4) are the same as for incident waves, except that the index of the wavemaker is output in place of the wave incidence angle BETA. These 'wavemaker RAO's' cannot be compared directly with the conventional incident-wave RAO's since the amplitude of the waves generated by the wavemaker modes are not used to normalize the RAO's. It is possible to make a separate run without the bodies to measure the wave elevations at the locations of the bodies using Option 6, as one might do to calibrate the wavemakers in a physical wavetank. If the wavemaker RAO's are normalized by these calculations of the wave elevations the results are directly comparable with the conventional RAO's in incident waves of unit amplitude.

When walls are specified by the parameters IWALLX0=1 and/or IWALLY0=1, the program automatically reflects the body and wavemaker geometry about the walls. The planes of symmetry for each body or wavemaker, defined by the parameters ISX and ISY in the GDF file, refer specifically to the local planes of symmetry of the body or wavemaker. When IWALLX0=1 and wavemakers are defined in the wall X=0, the symmetry index ISX=0 should be used for the wavemakers; similarly, for wavemakers in the wall Y=0, ISY=0. (Except for special cases where the GDF file only defines one half of the wavemaker, both wavemaker symmetry indices should be zero.) The symmetry indices for bodies are the same as without walls.

Since the wavemakers correspond to one or more bodies, it is necessary to include the usual inputs for these bodies in one or more FRC files. However, the dynamic characteristics of the wavemakers are ignored, since the RAO's are not evaluated for the wavemakers. Thus dummy values of the inputs for the dynamic characteristics of the wavemakers (VCG and XPRDCT, or XCT and external force matrics) must be included, but their values are not relevant.

12.5 BODIES WITH PRESSURE SURFACES

Starting in Version 7.0 it is possible to analyze problems where part of the body surface consists of one or more free surfaces, on which the pressure distribution is specified instead of the normal velocity. These are referred to as 'free-surface pressure' surfaces (FSP). Practical examples of bodies with FSP surfaces include oscillating water columns used for wave-energy conversion and air-cushion vehicles which are supported by positive mean pressure acting on the free surface below the body.

FSP surfaces can be either in the same plane ($Z = 0$) as the exterior free surface, or submerged ($Z < 0$) below the level of the exterior free surface (corresponding to a positive static pressure imposed on the surface as in the case of an air-cushion vessel). All FSP surfaces must be in horizontal planes, corresponding to constant values of the static pressure on each surface. The theory for this extension is described in Section 15.11. Examples of the use of FSP surfaces are included in Appendix A, TEST17c and TEST25.

The geometric definition of FSP surfaces is the same as for the other parts of the body surface, defined within the same GDF input file(s). The panels or patches which represent FSP surfaces are specified using the parameter NPFSP in the configuration files to identify the panel or patch indices. The inputs NPFSP must be in pairs, denoting the first and last values of the index (See Section 4.7). Multiple lines containing NPFSP=(N1 N2) can be used to indicate all of the FSP panels or patches for the body.

The oscillatory pressure distributions acting on the FSP surfaces are defined in a manner similar to the representation of generalized modes, using subroutines in the file `newmodes.dll`, as described in Chapter 9. In the simplest case the pressure distribution is constant, and only one mode is required for each FSP surface. In more general cases the pressure distribution can be represented by appropriate sets of modes such as Fourier modes. The parameters IMODESFSP and NMODESFSP specify for each body the subroutine used to define the modes and the number of active modes, respectively. These are analogous to the parameters IGENMDS and NEWMDS used for generalized modes. Each pressure mode is assigned a mode index $j > 6$ and a complex amplitude ξ_j. In the case of one body (or body one if there are multiple bodies), the pressure distribution is defined by the equation

$$p_0(x, y) = -\rho g \sum_{j=7}^{6+M_p} \xi_j n_j(x, y),$$

where M_p=NMODESFSP is the number of pressure modes, and the spatial distribution of each mode is described by the real function $n_j(x, y)$. The standard distribution version of `newmodes.dll` includes the subroutine PRESSURE_FS which permits the representation of a constant pressure distribution on a surface. Symmetric and antisymmetric modes can be used to analyze cases such as that described in Appendix A25, with appropriate planes of symmetry and with independent pressures acting on separate surfaces. Users should refer to the comments in this subroutine in the source file `newmodes.f` for further information.

For bodies with submerged FSP surfaces the volumes and center of buoyancy are computed in the same manner as if these surfaces were part of the body. Thus the displaced volume includes the volume between the submerged FSP surface and the exterior free surface,

which ensures that the body is in hydrostatic equilibrium if the hydrostatic force on the FSP surface is balanced by an equal and opposite force on the body. The hydrostatic restoring coefficients are computed based only on the waterplane of the body, and not including the FSP surfaces. Special attention is required to analyze body motions in waves if the pressure acting on the FSP surface results in a static restoring force on the body. This will occur if the pressure is contained within a chamber, such as an air-cushion vehicle, exerting a force and moment on the upper surface of the chamber. Since the pressure in the chamber is related to the volume, the restoring force will depend on the aerodynamic characteristics of the problem. This can be included as an external restoring force.

When FSP surfaces are included the mean drift force and moment based on momentum (Option 8) and based on the use of a control surface (Option 7) includes the effects of the pressure acting on the FSP surfaces. However the mean drift force and moment based on pressure integration (Option 9) only includes the contribution from integration over the wetted surface of the body. When Option 9 is used with FSP surfaces a warning message is issued. When Option 7 is used, as described in Chapter 11, the control surface should include the free surface exterior to the body, if this is necessary, and should not include the FSP surfaces.

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 1-6 (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.

Section 6.6 of Reference [26] contains additional information including an outline of the numerical method and 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 (Option 6) 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.

An important difference 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.

- The principal radiation IRFs correspond to the added-mass and damping coefficients, evaluated by WAMIT in Option 1. To displace the body in calm water in mode j with time-varying acceleration $\ddot{\xi}_j(t)$ the component i of the force required to oppose the hydrodynamic pressure is given [29] by

$$F_i(t) = A_{ij}(\infty)\ddot{\xi}_j(t) + \int_0^\infty L_{ij}(\tau)\ddot{\xi}_j(t-\tau)\,d\tau \tag{13.1}$$

  where $A_{ij}(\infty)$ is the infinite-frequency limit of the added mass. The radiation IRF $L_{ij}(t)$ is evaluated from either the Fourier cosine or sine transforms (13.5-6). These IRFs are output from F2T in the files _IR.1, _JR.1 and _KR.1 as explained below in Section 13.4.

- The exciting forces and RAOs are of the diffraction type, evaluated by WAMIT in Options 2, 3 and 4 and defined relative to an incident wave of uniform amplitude A, propagating in the direction β. The corresponding IRFs are defined with respect to an impulsive incident wave, moving in the same direction, where the free-surface elevation $\eta(x,y,t)$ is equal to a delta-function $\delta(t)$ at the origin $x = y = 0$. Further details are given in [26] and [30]. Here we define a general output $U_i(t)$ to be either the exciting force or the RAO, with respect to the mode i. It follows that

$$U_i(t) = \int_{-\infty}^\infty K_i(\tau)\eta(0,0,t-\tau)\,d\tau \tag{13.2}$$

  The IRFs $K_i(t)$ are evaluated from the Fourier transform of the frequency-domain exciting force or RAO using (13.10). These IRFs are output from F2T in the files _IR.n and _JR.n where (n=2,3,4), as explained below in Section 13.4.

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). When radiation IRFs are evaluated it is necessary to also evaluate the corresponding frequency-domain outputs for $\omega_0 = 0$ and $\omega_{N+1} = \infty$, using the inputs PER<0.0 and PER=0.0 respectively (See Section 4.2). In the WAMIT run this is done most easily by setting the parameter IPERIN=2 in the configuration files (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 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 IPERIN=2, 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 TEST14a described in Appendix A:

```
header line for inputs.f2t control file, TLP example
TEST14a
0 0 1 (IRAD IDIFF NUMHDR)
0 0 2 (INUMOPT5 INUMOPT6 IPEROUT)
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 TEST14a.FRC control file was used as in the standard WAMIT test runs, Options 1,2,3,4 are 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 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 here 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 and INUMOPT6 must be specified here with the value 0 (default) or the separate-components values 1. IPEROUT=1 or 2 must be specified, to distinguish wave periods and frequencies in the WAMIT output files

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 value IOUTFCFS=0 is recommended except in cases involving a large number of radiation modes, where the number of columns in the _JR1 output files may be excessive.

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 or 6 are included, and when the radiation outputs are specified, as explained in Section 13.5. 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 IOUTFCFS 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.

- The program F2T has been updated to function with output files from WAMIT Version 7. To ensure compatibility, users should verify that (1) WAMIT Version 7.0 or later has been used to generate the WAMIT numeric output files, and (b) the file f2t.exe is dated after 31 December 2011. It is possible to mix earlier versions, subject to the following restrictions:

  - If output files from WAMIT Version 6 are used as inputs to the updated f2t.exe, set the input parameter IPEROUT=1. Post-processing of field pressures and velocities is not possible unless these output files are renamed with extensions (.6p, .6vx, .6vy, .6vz).
  - If older versions of f2t.exe are used with output files from WAMIT Version 7, the default value IPEROUT=1 must be used for the WAMIT runs. Post-processing of field pressures and velocities is not possible unless these output files are renamed with extensions (.6, .7x, .7y, .7z).

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 13.3 and 13.4 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 extra output file is produced with the extension _KR.1, containing the impulse-response functions K_{ij} which are defined in Reference [30]. These alternative IRF's are

evaluated in F2T by numerical differentiation of the IRF's L_{ij}, and can be used with a similar convolution integral as in (13.1), but using the velocity $\dot{\xi}_j$ instead of the acceleration $\ddot{\xi}_j$.

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 13.2).

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 AND 6

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 may be difficult to transform accurately, due to limited or non-convergence of the Fourier transforms at high frequencies. The outputs from F2T include transforms of the WAMIT outputs for Options 5 and 6, which may be useful for special purposes. A distinction is made between radiation and diffraction components. Radiation components are output from WAMIT separately if INUMOPT5=1 and/or INUMOPT6 = 1, and IRAD = 0 or 1. All other outputs are of the diffraction type, including both the separate diffraction components of the pressure and velocity, and the total superpositions which are output when INUMOPT5=0 and/or INUMOPT6=0.

The diffraction-type pressure and velocity are transformed in the same manner as the exciting forces and RAO's. The IRFs $K_i(t)$ are evaluated using (13.10), with X_i the frequency-domain output from WAMIT for the same pressure or velocity. The free-surface elevation is equivalent to the pressure on $z = 0$, as defined in Section 3.6.

The radiation components of the pressure are not output directly by F2T. Instead, the radiation potentials $\bar{\varphi}_j$ defined in Section 3.5 are transformed, using (13.5-6) with the real and imaginary parts of the potential substituted for the added mass and damping. The radiation components of the fluid velocity are defined as the nondimensional gradients of these potentials, as in Section 3.7.

If IDIFF=$-$1 is specified in the WAMIT run, the WAMIT outputs from Options 5 and 6 are the total responses from superposition of all specified radiation modes. 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 unless the configuration parameters INUMOPT5=1 and INUMOPT6=1 are used.

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 $L_{ij}(t)$:

$$A_{ij}(\omega) - A_{ij}(\infty) = \int_0^\infty L_{ij}(t) \cos \omega t \, dt \tag{13.3}$$

$$B_{ij}(\omega) = \omega \int_0^\infty L_{ij}(t) \sin \omega t \, dt \tag{13.4}$$

The inverse-transforms of (13.1-2) give complementary relations for the impulse-response function:

$$L_{ij}(t) = \frac{2}{\pi} \int_0^\infty [A_{ij}(\omega) - A_{ij}(\infty)] \cos \omega t \, d\omega \tag{13.5}$$

$$L_{ij}(t) = \frac{2}{\pi} \int_0^\infty \frac{B_{ij}(\omega)}{\omega} \sin \omega t \, d\omega \tag{13.6}$$

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:

$$X_i(\omega) = \int_{-\infty}^\infty K_i(t) \mathrm{e}^{-i\omega t} \, dt \tag{13.7}$$

and

$$2\pi K_i(t) = \int_{-\infty}^\infty X_i(\omega) \mathrm{e}^{i\omega t} \, d\omega \tag{13.8}$$

Formally, since K_i is real, $X_i(-\omega) = X_i^*(\omega)$, and thus

$$2\pi K_i(t) = \int_0^\infty \left[X_i(\omega) \mathrm{e}^{i\omega t} + X_i^*(\omega) \mathrm{e}^{-i\omega t} \right] \, d\omega \tag{13.9}$$

or

$$K_i(t) = \frac{1}{\pi} \int_0^\infty \left[\mathrm{Re}(X_i) \cos \omega t - \mathrm{Im}(X_i) \sin \omega t \right] \, d\omega \tag{13.10}$$

The Fourier transofrms (13.5), (13.6), and (13.10) are evaluated 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 applied to (13.5). Further details regarding this procedure are given in [26].

13.7 DIMENSIONAL INPUT AND OUTPUT DATA

In the WAMIT output files all of the output data is nondimensional, except for the wave period. Definitions of the nondimensional outputs are given in Chapter 3. The wave period, defined in seconds, has the same dimension as $\sqrt{ULEN/GRAV}$, where the parameters ULEN and GRAV are input in the GDF file. Since the hydrodynamic force coefficients and other outputs from WAMIT are nondimensional, and the frequency ω has the dimension of inverse time, it follows that the impulse response functions defined by equations 13.3, 13.4 and 13.8 are all dimensional, with the dimension of inverse time. The additional impulse-response functions in the output file .KR1, defined as the time-derivatives of the IRF's L_{ij}, have the dimension of inverse time squared.

For Options 1-4 the normalized outputs from F2T are defined as follows:

$$\begin{aligned}
\text{Option 1}: \quad \bar{L}_{ij}(t) &= \frac{2}{\pi}\int_0^\infty \left[\bar{A}_{ij}(\omega) - \bar{A}_{ij}(\infty)\right]\cos\omega t\, d\omega = \frac{2}{\pi}\int_0^\infty \bar{B}_{ij}(\omega)\sin\omega t\, d\omega \\
\text{Options 2, 3}: \quad \bar{K}_i(t) &= \frac{1}{\pi}\int_0^\infty \left[\mathrm{Re}(\bar{X}_i(\omega))\cos\omega t - \mathrm{Im}(\bar{X}_i(\omega))\sin\omega t\right]\, d\omega \\
\text{Option 4}: \quad \bar{K}_i(t) &= \frac{1}{\pi}\int_0^\infty \left[\mathrm{Re}(\bar{\xi}_i(\omega))\cos\omega t - \mathrm{Im}(\bar{\xi}_i(\omega))\sin\omega t\right]\, d\omega
\end{aligned}$$

In these equations the nondimensional frequency-domain parameters are defined in Sections 3.2-4.

For Options 5 and 6 the pressure and velocities are of diffraction type when INUMOPT5,6=0. In this case the normalized outputs from F2T are defined as follows:

$$\begin{aligned}
\text{Options 5p, 6p}: \quad \bar{K}(t) &= \frac{1}{\pi}\int_0^\infty \left[\mathrm{Re}(\bar{p}(\omega))\cos\omega t - \mathrm{Im}(\bar{p}(\omega))\sin\omega t\right]\, d\omega \\
\text{Options 5v*, 6v*}: \quad \bar{\mathbf{K}}(t) &= \frac{1}{\pi}\int_0^\infty \left[\mathrm{Re}(\bar{\mathbf{V}}(\omega))\cos\omega t - \mathrm{Im}(\bar{\mathbf{V}}(\omega))\sin\omega t\right]\, d\omega
\end{aligned}$$

Here $\bar{p}$ is the nondimensional total pressure, defined in Section 3.5, and $\bar{\mathbf{V}}$ is the nondimensional total velocity vector defined in Section 3.7.

When INUMOPT5,6=1 the separate components of the diffraction and radiation potentials and velocities are output by WAMIT and transformed by F2T. In this case the normalized outputs from F2T are defined as follows:

$$\begin{aligned}
\text{Options 5p, 6p}: \quad \bar{K}_D(t) &= \frac{1}{\pi}\int_0^\infty \left[\mathrm{Re}(\bar{\varphi}_D(\omega))\cos\omega t - \mathrm{Im}(\bar{\varphi}_D(\omega))\sin\omega t\right]\, d\omega \\
\bar{K}_j(t) &= \frac{2}{\pi}\int_0^\infty \mathrm{Re}\left(\bar{\varphi}_j(\omega) - \bar{\varphi}_j(\infty)\right)\cos\omega t\, d\omega = \frac{2}{\pi}\int_0^\infty \mathrm{Im}\left(\bar{\varphi}_j(\omega)\right)\sin\omega t\, d\omega \\
\text{Options 5v*, 6v*}: \quad \bar{\mathbf{K}}_d(t) &= \frac{1}{\pi}\int_0^\infty \left[\mathrm{Re}(\bar{\nabla}\bar{\varphi}_D(\omega))\cos\omega t - \mathrm{Im}(\bar{\nabla}\bar{\varphi}_D(\omega))\sin\omega t\right]\, d\omega \\
\bar{K}_j(t) &= \frac{2}{\pi}\int_0^\infty \mathrm{Re}\left(\bar{V}_j(\omega) - \bar{V}_j(\infty)\right)\cos\omega t\, d\omega = \frac{2}{\pi}\int_0^\infty \mathrm{Im}\left(\bar{V}_j(\omega)\right)\sin\omega t\, d\omega
\end{aligned}$$

In these equations the nondimensional frequency-domain functions are defined in Sections 3.5 and 3.7. Note in the transforms $\bar{K}_j$ that the frequency-domain functions $\bar{\varphi}_j$ and

$\bar{V}_j = \bar{\nabla}\bar{\varphi}_j$ differ from the corresponding outputs in the WAMIT numeric output files by the factor $1/KL$. Thus, before applying the Fourier integrations of these functions in F2T, the WAMIT outputs are divided by $KL = \omega^2 L/g$ where L=ULEN and g=GRAV. In all other cases above the frequency-domain functions are the same as the WAMIT outputs in the corresponding numeric output files.

Chapter 14

COMPUTATIONAL TOPICS

In this Chapter various topics are covered which affect the use of WAMIT, especially for more advanced applications with complicated geometry, or multiple structures. These require a large number of panels in the low-order method, or corresponding number of unknowns in the higher-order method, to represent both the geometry and the hydrodynamic solution with a satisfactory degree of accuracy. In such cases the required storage for temporary data and the time required to set up and solve the linear systems of equations are both large.

From the computational standpoint the principal task is to set up and solve the linear systems of equations which correspond to the discretized integral equations described in Sections 15.3 and 15.5. 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 14.1.

WAMIT includes three optional methods for solving the linear systems of equations, including a direct solver which is robust but time-consuming for large values of NEQN, an iterative solver which for large systems of equations is much faster, and a block-iterative solver which combines the advantages of each to some extent. These methods are described and compared in Section 14.2.

Section 14.3 describes the required storage for the influence coefficients on the left-hand-sides of the linear systems of equations. These must be stored either in random-access-memory (RAM) or in scratch files on the hard disk. Since access to RAM is much faster, this option is advantageous from the standpoint of computing time if sufficient RAM is available. Section 14.4 describes the procedure to allocate the data storage between RAM and the hard disk in an efficient manner. Section 14.5 explains details associated with the use of scratch files on the hard disk.

The advantages and use of multiple processors are described in Section 14.6, including a comparison of computing times for two applications. This shows the dramatic reduction in computing time that can be achieved when the number of CPUs is increased. However the use of multiple CPUs requires a proportionally large size of RAM.

Section 14.7 gives instructions for users to modify the WAMIT DLL files `geomxact` and `newmodes`. Section 14.8 lists the filenames which are reserved for use by WAMIT.

14.1 NUMBER OF EQUATIONS (NEQN) AND LEFT-HAND SIDES (NLHS)

In the low-order method (ILOWHI=0) the number of equations depends on the number of panels NPAN specified in the GDF input file(s). In the higher-order method (ILOWHI=1) NEQN depends on the number of patches, and on the spline paramters NU, NV, KU, and KV. NEQN is modified if the body geometry is reflected about planes of symmetry, automatic discretization of the interior free surface is utilized (IRR>1), or waterline trimming is used (ITRIWMWL=1). The value of NEQN for each run is listed in the header of the .out output file. Typical values of NEQN are between 100 and 10,000. NEQN is always equal to the number of unknowns in the representation of the velocity potential on the body surface (and interior free surface if IRR≥1).

The body surface is defined in the .gdf file on NPAN panels in the low-order method or NPATCH patches in the higher-order method. the .gdf file also specifies the symmetry indices ISX,ISY, which define planes of symmetry ($x = 0$, $y = 0$ respectively) for the body as explained in Chapters 6 and 7. If one or two planes of symmetry are defined, only one half or one quarter of the body surface is defined in the .gdf file. The program uses these symmetries to reduce NEQN, when it is possible to do so, by defining separate solutions which are symmetric and antisymmetric with respect to each plane of symmetry. The number of the corresponding sets of influence functions, or left-hand sides, is denoted by NLHS. If there are no planes of symmetry NLHS=1, with one plane of symmetry NLHS=2, and with two planes of symmetry NLHS=4.

Before considering possible modifications which may be required, the value of NEQN is defined based on the inputs in the .gdf file. In the low-order method NEQN=NPAN is the number of panels on this surface. In the higher-order method

$$\mathrm{NEQN} = \sum_{i=1}^{N_P} (\mathrm{NU}(i) + \mathrm{KU}(i) - 1) \times (\mathrm{NV}(i) + \mathrm{KV}(i) - 1) \tag{14.1}$$

Here N_P=NPATCH is the 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, as explained in Section 7.2.

If the irregular-frequency removal option is used with IRR=1, the interior free-surface is included as a part of the surface defined in the gdf file, and the value of NEQN defined above is unchanged. But if automatic discretization of the interior free surface is utilized (IRR>1) NEQN is increased by the program to include the additional unknowns on the interior free surface. (See Chapter 10.)

If waterline trimming is used (ITRIMWL=1) the value of NEQN may change, either increasing or decreasing depending on the trimming displacement and angles. (See Section 12.2.)

Reflection is performed automatically by the program if the 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. This may occur in the following cases:

- if the input parameters XBODY(1), XBODY(2), or XBODY(4) which define the origin of the body coordinate system are nonzero in the POT file (See Section 4.2)
- for bodies near vertical walls (Section 12.4)
- for multiple bodies (NBODY>1) as explained in Chapter 8
- If the trim angle about the $x-$axis is nonzero, symmetry about the plane $y = 0$ is destroyed, and vice-versa. In these cases the body surface is reflected, NEQN is increased, and NLHS is decreased.

When reflection about one plane of symmetry is required, NEQN is increased by a factor of two and NLHS is reduced by a factor of one-half. When reflections about two planes of symmetry are required, NEQN is increased by a factor of four and NLHS is reduced by a factor of one-quarter.

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 with a factor of $+1$ or -1 for symmetric and antisymmetric flow, respectively.

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.

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 4.2). 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. A warning message is displayed in cases where the solution of the diffraction problem is incomplete.

14.2 SOLUTION OF THE LINEAR SYSTEMS

WAMIT includes three optional methods for solving the linear systems of equations, including a direct solver which is relatively robust but time-consuming, an iterative solver which for large systems of equations is much faster, and a block-iterative solver which combines the advantages of each to some extent. The parameter ISOLVE in the configuration file is used to select which method is used for the run.

With the default value ISOLVE=0, WAMIT solves the linear systems by means of a special iterative solver. The maximum number of iterations is controlled by the parameter MAXITT in the configuration files (See Section 4.7), with the default value equal to 35. If convergence is not archieved within this limit a warning message is issued, 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. The time required for this method is proportional to $(\text{NEQN})^2$ times the number of iterations. For large NEQN this is much faster than the methods described below. Another advantage of the iterative solver is that it does not require temporary storage proportional to $(\text{NEQN})^2$.

The direct solver (ISOLVE=1) is useful for cases where the iterative solver does not converge, or requires a very large number of iterations to achieve convergence. The direct solver is based on standard Gauss reduction, with partial pivoting. The LUD algorithm is employed for efficiency in solving several linear systems simultaneously, with different right-hand sides. The time required for this method is proportional to $(\text{NEQN})^3$. In cases where NEQN is relatively small the direct solver can result in reduced computing time, particularly if the number of right-hand sides is large. The direct solver requires sufficient RAM to store at least one complete set of $(\text{NEQN})^2$ influence coefficients.

The block-iterative solver (ISOLVE$\geq$2) provides increased options in the solution of the linear system. This solver is based on the same algorithm as the iterative solver, but local LU decompositions are performed for specified blocks adjacent to the main diagonal. Back substitution is performed for each block, at each stage of iteration. 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 iterative method is useful primarily for the low–order method, where NEQN is relatively large and the rate of convergence is good in most cases. Usually in the low-order method the number of iterations required to obtain convergence is in the range 10-20. In the standard test runs described in Appendicies A using the low-order method, the iterative or block-iterative solvers converge within the default number of iterations MAXITT=35, for all cases except TEST02.

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 higher-order method the linear system loses diagonal dominance as the order of the basis functions increases, as shown in the expression for d_{ik}^H in (15.32). Experience indicates that the convergence rate is reduced, and it is generally advisable to use the direct solver (ISOLVE= 1) or, if necessary in cases where NEQN is very large, the block-iterative solver (ISOLVE> 1).

Results from convergence tests using the low-order method have been published in References 5, 6, 9, 10 and 12. 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.

14.3 TEMPORARY DATA STORAGE

From the computational standpoint the principal tasks are to set up and solve the linear systems of equations for the unknown potentials and source strengths. These tasks require substantial temporary storage for most practical applications, either in RAM (random-access memory) or in scratch files on the hard disk. Generally access to RAM is much faster than to the hard disk, but the size of RAM is relatively small. The first versions of WAMIT were developed when RAM was quite small (typically measured in Kilobytes) and it was essential to use scratch files whenever possible. WAMIT Version 7 has been developed to take advantage of the much larger RAM available in contemporary systems, measured in Gigabytes (1 GB is equal to 10^9 bytes or 10^6 Kilobytes).

For most applications the dominant storage requirement is for the $(\mathrm{NEQN})^2$ influence coefficients on the left-hand-side of the linear systems of equations. Since typical values of NEQN are between 100 and 10,000, between 10^4 and 10^8 influence coefficients must be stored for each left-hand-side. Here we consider only the storage of these matrices, since other data storage requirements are negligible by comparison.

The influence coefficients are related to the free-surface Green function and its derivatives, as defined in Section 15.2. These include real components associated with the Rankine source potential $1/r$, and complex components associated with the effect of the free surface (cf. Equations 15.14-18). To avoid redundant computations the real components are evaluated only once, whereas the complex components must be evaluated separately for each wave period. Thus separate storage is required for the real and complex matrices. The estimated storage for each of these is described below.

WAMIT takes into account flow symmetries in setting up the linear systems, to minimize NEQN. When planes of symmetry exist with respect to the planes $X = 0$ and/or $Y = 0$, NEQN can be reduced by one half in each case, thus reducing the number of influence coefficients on one left-hand side by a factor of 1/4 or 1/16. In general, the number of left-

hand-sides NLHS must be increased by 2 or 4 in order to solve both the symmetric and anti-symmetric problems (e.g. heave and surge in the case of a body which is symmetric about $X = 0$). Depending on the number of relevant modes and symmetry planes, NLHS=1,2 or 4.

The minimum number of influence coefficients which must be stored is equal to the product NLHS$\times$(NEQN)^2. However additional matrices may be required, depending on the input parameters ILOG and ISOR in the configuration file. ILOG=0 or 1 in all cases. ISOR=0 or 1 for the low-order panel method, and ISOR=0 for the higher-order method. For the real components 4 bytes are required for each coefficient, and the total storage required for all matrices is $S_r = Q_r(\text{NEQN})^2$ where

$$Q_r = 4 \times \text{NLHS} \times (1 + 4 \times \text{ISOR}) \times (1 + \text{ILOG}) \tag{14.2}$$

For the complex components 8 bytes are required for each coefficient, and the total storage required for all matrices is $S_c = Q_c(\text{NEQN})^2$ where

$$Q_c = 8 \times \text{NLHS} \times (1 + \text{ISOR}) \tag{14.3}$$

These can be estimated using Figure 14.1, with the factor Q defined in (14.2-3). Note that $4 \le Q_r \le 160$ and $8 \le Q_c \le 64$. In three special cases Q_r is greater by a relatively small amount: (1) in the low-order method if the scattering parameter ISCATT=1 in the configuration files, (2) if pressure surface panels or patches are used, as described in Section 12.5, or (3) if PER=0 is assigned in the POT file corresponding to zero wave period or infinite frequency. More significantly, if multiple processors are used (NCPU>1), the factor Q_c must be multiplied by NCPU (See Section 14.6).

14.4 DATA STORAGE IN RAM

In Version 7 provisions have been made to replace scratch files on the hard disk by arrays in RAM, up to the limit of RAM that is available. This can save substantial computing time. To fully utilize this capability, the user should assign the parameter RAMGBMAX in the configuration file, as described in Section 4.7, based on the estimated amount of RAM that is not required for other purposes. Some experience may be required to determine this input. For a system that is not used concurrently for other substantial computations, a suggested estimate is one-half of the total RAM if the RAM is less than 2 Gb, or the total RAM minus 1 Gb for systems with more than 2 Gb. For systems running under Windows the total RAM is displayed by the Control Panel System option. It is important not to assign a value of RAMGBMAX that is too large, since this may result in 'paging' or transfer of data to virtual memory on the hard disk, which will slow down the computations.

The parameter RAMGBMAX is based on the available memory of the system, and is not dependent on the inputs to each run. Thus it is recommended to assign RAMGBMAX in the file `config.wam` and not in the configuration file *runid*.cfg associated with a particular run. If RAMGBMAX is not assigned in the configuration files the default value 0.5 is used. An estimate of the actual RAM used for each run is shown in the output file `wamitlog.txt`.

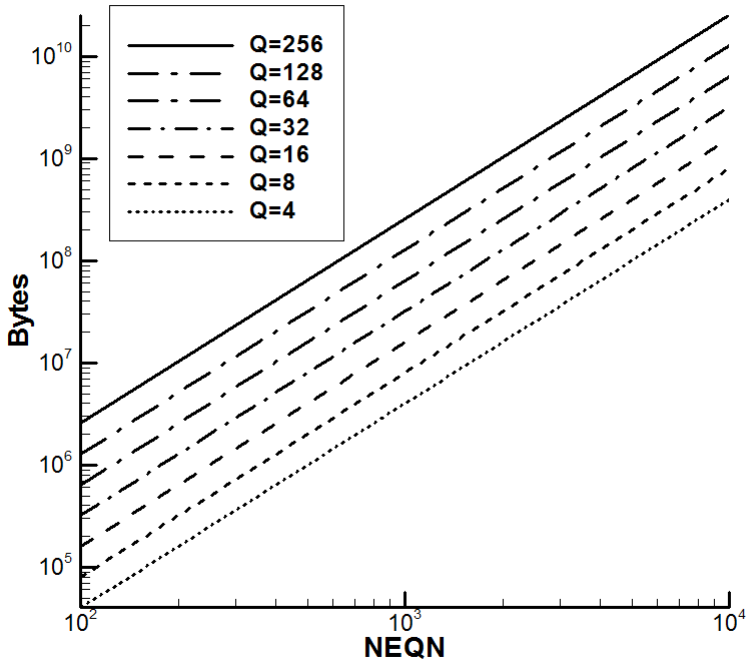


Figure 14.1: RAM required for storage of the real and complex arrays. The parameter Q is given by equations (14.2-3).

If RAMGBMAX is sufficiently large, all of the real and complex influence coefficients are stored in RAM. In this case the total RAM required is estimated from Figure 14.1 with $Q = Q_r + \text{NCPU} \times Q_c$. If this is not possible, the program will distribute the data between RAM and the hard disk using the following order of priorities:

1. real coefficients are stored on the hard disk, leaving all available RAM for the storage of complex coefficients
2. if NLHS>1, some but not all left-hand-side arrays are stored in RAM and the remainder on the hard disk
3. if the RAM can not store one complete complex left-hand-side array, a subset of the coefficients are stored in RAM and the remainder on the hard disk

If multiple processors are used (NCPU>1) all of the complex arrays must be in RAM. Thus the minimum RAM required for multiple processing is estimated from Figure 14.1 with $Q = \text{NCPU} \times Q_c$. If this is not possible, execution of the program terminates with an error message advising the user to increase RAMGBMAX or reduce NCPU.

If NCPU=1 the following minimum requirements apply for available RAM:

1. if ISOLVE=1 (direct solver) or ILOWHI=1 (higher-order method), one complete complex left-hand-side must be stored in RAM

2. if ISOLVE>1 (block-iterative solver) one complex left-hand-side must be stored in RAM with dimensions equal to the maximum block size

14.5 DATA STORAGE IN SCRATCH FILES

Two types of temporary scratch files are opened during execution of the subprogram POTEN. One group are opened formally using the FORTRAN scratch file 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 these scratch files may remain on the hard disk. In the latter case the user is advised to delete these files manually.

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. The parameter SCRATCH_PATH can be used in the configuration files to distribute storage between two disks, as explained in Section 4.7.

14.6 MULTIPLE PROCESSORS (NCPU>1)

WAMIT Version 7.0pc is compiled with the Intel Fortran Compiler Version 12, using special directives to enable parallel processing on systems with multiple processors. Depending on the inputs and hardware, the total run time can be reduced substantially by using this capability. Figure 14.2 shows two examples, using modified inputs of the test runs 07 (ISSC TLP) and 15 (Semi-sub) which are described in Appendix A, and using the low-order and higher-order options respectively. In order to provide examples for relatively large computations, the input parameters for these two runs have been modified as explained in the caption.

Users should first verify the number of CPU's and size of RAM of the system. (For Windows systems the RAM size is displayed after selecting 'Start', 'Control Panel', 'System'. The number of processors is listed under 'Advanced', 'Environment Variables') NCPU should be determined based on the number of physical processors, also referred to as 'cores', and not based on the number of 'hyper-threads'.

If the system includes more than one CPU, open the file `config.wam` with a text editor. The default settings in this file are as follows:

```
NCPU=1
RAMGBMAX=0.5
```

Change NCPU to the appropriate number for the system, and increase RAMGBMAX to the maximum value which can be used for scratch memory, following the guidelines in Section 14.4. Note that the RAM required for multiple processing is proportional to NCPU. The actual RAM required during a run is displayed in the output file wamitlog.txt.

If RAMGBMAX is not sufficiently large for the value of NCPU specified, the program stops with an error message displayed. In that case the user should reduce NCPU, or modify the other input parameters to increase RAMGBMAX or reduce the required RAM.

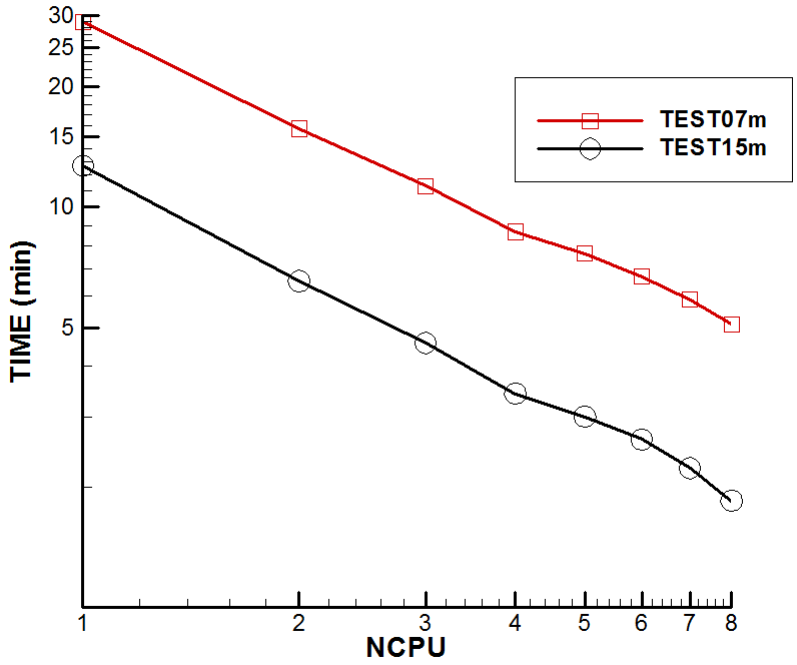


Figure 14.2: Run times for modified versions of test07 (ISSC TLP) and test15 (Semi-Sub) for values of NCPU between 1 and 8. The test07m inputs are the same as in Appendix A.7 except that the number of panels NPAN=4048 and the number of wave periods NPER=32. The test15m inputs are the same as in Appendix A.15 except that NPER=32. These computations were run on a Dell T7500 PC with 8 CPUs and 96 Gb of RAM, under the Windows XP Pro x64 operating system.

The principal advantage of multiple processing is in the computing time for loops over the wave period. During the run, when the time and number of iterations are output for each wave period on the monitor, these are displayed in groups of NCPU lines more-or-less simultaneously, with values of the wave period which usually are not in a logical sequence. (The original sequence is restored in the header of the .out output file, where it will be noted that the clock time for each wave period is not sequential.) Test14a is a useful example of this process, and of the advantage of using multiple processors for runs with a large number of wave periods (See Appendix A.14).

When NCPU>1, the value of NCPU used is shown in the `wamitlog.txt` output file. If the total number of wave periods NPER is less than the value of NCPU in the configuration files, NCPU is reduced for the loops over the wave period and the reduced value is displayed in the `wamitlog.txt` file. Maximum efficiency of the computing time is achieved when NPER is an integer multiple of NCPU.

When NCPU>1 the 'BREAK' option to interrupt the run is disabled and cannot be used

(See Section 4.12).

14.7 MODIFYING DLL FILES

The files `geomxact.f` and `newmodes.f` can be modified by users following the instructions in Sections 7.9 and 9.3. This makes it possible for users to develop special subroutines for the definitions of the body geometry and generalized modes, respectively, and to link these subroutines with WAMIT at runtime.

- WAMIT Version 7.0PC is compiled with Intel Visual Fortran (Version 12.1). The previous Version 6.4PC was compiled with Intel Visual Fortran (Version 10.1) and earlier versions were compiled with Compaq Visual Fortran. Any of these Fortran compilers can be used to compile modified versions of the files `geomxact.dll` and `newmodes.dll` for use with a single processor (NCPU=1), using the following procedure:

  - 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

  The same procedure is used for NEWMODES, except for the different filenames.

- Users who modify the DLL files for runs with multiple processors (NCPU>1) are advised to contact WAMIT Inc. for special instructions.

  It may be possible to use other FORTRAN compilers to build the DLL files, but certain conventions in calling subroutines must be consistent with those of Intel Visual Fortran. Further information is provided in [23], Chapters 8 and 18.

14.8 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`, `bpi`, `bpo`, `idf`, `rao`, `1`, `2`, `3`, `4`, `5`p, `5`vx, `5`vy, `5`vz, `6`p, `6`vx, `6`vy, `6`vz, `7`, `8`,and `9` for WAMIT input and output. Other reserved filenames are `config.wam`, `fnames.wam`, `break.wam`, `errorp.log`, `errorf.log`, `wamitlog.txt`, SCRATCH* (where *=A,B,C,...,O), as well as `wamit.exe`, `defmod.f`, `defmod.exe`, the DLL files `geomxact.dll` and `newmodes.dll`, and the Intel DLL files listed in Section 2.1 which are required to execute the program. The utility `f2t.exe` described in Chapter 13 uses the reserved file `inputs.f2t`.

Chapter 15

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.

15.1 THE BOUNDARY-VALUE PROBLEM

A three-dimensional body interacts with plane progressive waves in water of finite depth H. The objective of WAMIT is to evaluate the unsteady hydrodynamic pressure, loads and motions of the body, as well as the pressure and velocity in the fluid domain. The flow is assumed to be potential, free of separation or lifting effects. The free-surface and body-boundary conditions are linearized. A harmonic time dependence is adopted.

The Cartesian coordinate system (x, y, z) is defined as shown in Figure 4.1, fixed relative to the undisturbed positions of the free surface and body with the $z-$axis positive upwards. The body geometry input to WAMIT is defined relative to the *body* coordinate system and the incident-wave system is defined relative to the global coordinates, as explained in Chapter 3 and Section 4.2. For the sake of simplicity here it is assumed that these two coordinate systems coincide.

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

$$\nabla^2 \Phi = 0 \tag{15.1}$$

in the fluid domain. The harmonic time dependence allows the definition of a complex velocity potential φ, related to Φ by

$$\Phi = \mathrm{Re}(\varphi e^{i\omega t}), \tag{15.2}$$

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

$$\varphi_z - K\varphi = 0 \quad \text{on} \quad z = 0. \tag{15.3}$$

Here $K = \omega^2/g$ is the infinite-depth wavenumber and g is the acceleration of gravity. The velocity potential of the incident wave is defined by

$$\varphi_0 = \frac{igA}{\omega} \frac{\cosh[k(z+H)]}{\cosh kH} e^{-ikx\cos\beta - iky\sin\beta}, \tag{15.4}$$

where the wavenumber k is the real root of the dispersion relation

$$\frac{\omega^2}{g} = k \tanh kH \tag{15.5}$$

and β is the angle between the direction of propagation of the incident wave and the positive $x-$axis as defined in Figure 15.1. In the limiting case of infinite depth $k = K$. To distinguish these two parameters, they are referred to as the 'finite-depth' and 'infinite-depth' wavenumbers, respectively. The wavelength is equal to $2\pi/k$ and the wave period is equal to $2\pi/\omega$.

The linearization of the problem permits the decomposition of the velocity potential φ into the radiation and diffraction components

$$\varphi = \varphi_R + \varphi_D, \tag{15.6}$$

$$\varphi_R = i\omega \sum_{j=1}^{6} \xi_j \varphi_j, \tag{15.7}$$

$$\varphi_D = \varphi_0 + \varphi_S. \tag{15.8}$$

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_S represents the *scattered* disturbance of the incident wave by the body fixed at its undisturbed position. We will refer to the sum (15.8) as the *diffraction* potential φ_D.

On the undisturbed position of the body boundary, the radiation and diffraction potentials are subject to the conditions,

$$\varphi_{jn} = n_j, \tag{15.9}$$

$$\varphi_{Dn} = 0, \tag{15.10}$$

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 radiation condition of outgoing waves in the far field is applied to the velocity potentials φ_j, $j = 1, \ldots, 7$.

15.2 INTEGRAL EQUATIONS FOR THE VELOCITY POTENTIAL

In WAMIT the boundary value problems defined above 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

$$2\pi\varphi_j(\mathbf{x}) + \iint_{S_b} \varphi_j(\boldsymbol{\xi}) \frac{\partial G(\boldsymbol{\xi};\mathbf{x})}{\partial n_\xi} d\boldsymbol{\xi} = \iint_{S_b} n_j G(\boldsymbol{\xi};\mathbf{x}) d\boldsymbol{\xi}. \tag{15.11}$$

Here S_b denotes the wetted surface of the body, in its fixed mean position, below the plane $z = 0$.

The corresponding equation for the total diffraction velocity potential φ_D is

$$2\pi\varphi_D(\mathbf{x}) + \iint_{S_b} \varphi_D(\boldsymbol{\xi}) \frac{\partial G(\boldsymbol{\xi};\mathbf{x})}{\partial n_\xi} d\boldsymbol{\xi} = 4\pi\varphi_0(\mathbf{x}). \tag{15.12}$$

The diffraction potential may also be obtained from equation (15.8) after solving for the scattered potential φ_S. The equation for the scattered velocity potential is

$$2\pi\varphi_S(\mathbf{x}) + \iint_{S_b} \varphi_S(\boldsymbol{\xi}) \frac{\partial G(\boldsymbol{\xi};\mathbf{x})}{\partial n_\xi} d\boldsymbol{\xi} = -\iint_{S_b} \frac{\partial \varphi_0(\xi)}{\partial n} G(\boldsymbol{\xi};\mathbf{x}) d\boldsymbol{\xi}. \tag{15.13}$$

From the computational point of view, equation (15.12) has some advantages over (15.13) 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

$$G(\mathbf{x};\boldsymbol{\xi}) = \frac{1}{r} + \frac{1}{r'} + \frac{2K}{\pi} \int_0^\infty dk \, \frac{e^{k(z+\zeta)}}{k-K} J_0(kR) \tag{15.14}$$

$$r^2 = (x-\xi)^2 + (y-\eta)^2 + (z-\zeta)^2 \tag{15.15}$$

$$r'^2 = (x-\xi)^2 + (y-\eta)^2 + (z+\zeta)^2, \tag{15.16}$$

where $J_0(x)$ is the Bessel function of zero order. In finite depth, the Green function is defined by

$$G(\mathbf{x};\boldsymbol{\xi}) = \frac{1}{r} + \frac{1}{r''} + 2\int_0^\infty dk \, \frac{(k+K)\cosh k(z+H)\cosh k(\zeta+H)}{k\sinh kH - K\cosh kH} \, e^{-kH} J_0(kR) \tag{15.17}$$

$$(r'')^2 = (x-\xi)^2 + (y-\eta)^2 + (z+\zeta+2H)^2. \tag{15.18}$$

In both expressions (15.14) and (15.17) 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].

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,

$$G(\mathbf{x};\boldsymbol{\xi}) = \frac{1}{r} + \frac{1}{r'} - 2K \log K(r' + |z+\zeta|) + O(1), \tag{15.19}$$

(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 15.3.

15.3 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 6.

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 (15.11) and (15.12) 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

$$2\pi\varphi(\mathbf{x}_i) + \sum_{k=1}^{N} D_{ik}\varphi_k = \sum_{k=1}^{N} S_{ik}\left(\frac{\partial\varphi}{\partial n}\right)_k, \tag{15.20}$$

where $i = 1, \ldots, N$, N being the number of panels. For the total diffraction potential

$$2\pi\varphi(\mathbf{x}_i) + \sum_{k=1}^{N} D_{ik}\varphi_k = 4\pi\varphi_0(\mathbf{x_i}). \tag{15.21}$$

The matrices D_{ik} and S_{ik} are defined by

$$D_{ik} = \iint_{s_k} \frac{\partial G(\boldsymbol{\xi}, \mathbf{x_i})}{\partial n_\xi} d\boldsymbol{\xi}, \tag{15.22}$$

$$S_{ik} = \iint_{s_k} G(\boldsymbol{\xi}, \mathbf{x_i}) d\boldsymbol{\xi}, \tag{15.23}$$

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 analytic integration of the Rankine source potentials and their derivatives follows the theory outlined in [2]. 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 is approximated by multiplying the value of the integrand at the centroid by the panel area.

15.4 INTEGRAL EQUATIONS FOR THE SOURCE FORMULATION

In the low-order method the source formulation is used to derive the fluid velocity components on the body surface. The velocity potential is expressed by a distribution of sources only, in the form

$$\varphi(\mathbf{x}) = \iint_{S_b} \sigma(\boldsymbol{\xi}) G(\mathbf{x}; \boldsymbol{\xi}) d\boldsymbol{\xi}. \tag{15.24}$$

After discretizing the body surface with plane panels, with constant source strength on each panel, the potential is expressed by

$$\varphi(\mathbf{x}_i) = \sum_{j=1}^{N} \sigma(\mathbf{x}_j) \iint_{S_j} G(\mathbf{x}_i; \boldsymbol{\xi}) d\boldsymbol{\xi}. \tag{15.25}$$

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:

$$\varphi_n(\mathbf{x}_i) = 2\pi\sigma(\mathbf{x}_i) + \sum_{j=1}^{N} \sigma(\mathbf{x}_j) \iint_{S_j} G_n(\mathbf{x}_i;\boldsymbol{\xi}) d\boldsymbol{\xi}, \tag{15.26}$$

$$\varphi_s(\mathbf{x}_i) = \sum_{j=1}^{N} \sigma(\mathbf{x}_j) \iint_{S_j} G_s(\mathbf{x}_i;\boldsymbol{\xi}) d\boldsymbol{\xi}, \tag{15.27}$$

$$\varphi_t(\mathbf{x}_i) = \sum_{j=1}^{N} \sigma(\mathbf{x}_j) \iint_{S_j} G_t(\mathbf{x}_i;\boldsymbol{\xi}) d\boldsymbol{\xi}. \tag{15.28}$$

15.5 DISCRETIZATION OF THE INTEGRAL EQUATIONS IN THE HIGHER-ORDER METHOD (ILOWHI=1)

The mean body surface is defined by 'patches', as explained in Chapter 7. 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 $\mathbf{u} = (u, v)$. These are normalized so that they vary between ± 1 in the domain of the patch. Details of the geometric description of the body surface are provided in Chapter 7.

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 (7.4). The total number of B-spline basis functions on each patch is M_u times M_v.

Upon substituting equations (7.3) and (7.4), (15.11) can be expressed in the form

$$\begin{aligned} 2\pi \sum_{m=1}^{M_v} \sum_{l=1}^{M_u} (\phi_j)_k \mathbf{U}_k(\mathbf{u}_f) \quad &+ \quad \sum_{n=1}^{N_p} \iint d\mathbf{u} \sum_{m=1}^{M_v} \sum_{l=1}^{M_u} (\phi_j)_k \mathbf{U}_k(\mathbf{u}) \frac{\partial G(\mathbf{u};\mathbf{u_f})}{\partial n(\mathbf{u})} J(\mathbf{u}) \\ &= \sum_{n=1}^{N_p} \iint d\mathbf{u} \frac{\partial \phi_j}{\partial n}(\mathbf{u}) G(\mathbf{u};\mathbf{u_f}) J(\mathbf{u}) \end{aligned} \tag{15.29}$$

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, (15.29) is multiplied by $\mathbf{U}_i$ and integrated over each patch. This results in the linear system

$$2\pi d_{ik}^H (\phi_j)_k + \sum_{k=1}^{N} D_{ik}^H (\phi_j)_k = S_i^H \tag{15.30}$$

for the radiation potential. Similarly, from (15.12), we obtain

$$2\pi d_{ik}^H(\phi_D)_k + \sum_{k=1}^{N} D_{ik}^H(\phi_D)_k = I_i^H \tag{15.31}$$

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

$$d_{ik}^H = \iint d\mathbf{u}_f \mathbf{U}_i(\mathbf{u}_f)\mathbf{U}_k(\mathbf{u}_f) \tag{15.32}$$

$$D_{ik}^H = \iint d\mathbf{u}_f \mathbf{U}_i(\mathbf{u}_f) \iint d\mathbf{u}\mathbf{U}_k(\mathbf{u})\frac{\partial G(\mathbf{u};\mathbf{u}_f)}{\partial n(\mathbf{u})}J(\mathbf{u}) \tag{15.33}$$

$$S_i^H = \iint d\mathbf{u}_f \mathbf{U}_i(\mathbf{u}_f) \iint d\mathbf{u}\frac{\partial \phi_j}{\partial n}(\mathbf{u})G(\mathbf{u};\mathbf{u}_f)J(\mathbf{u}) \tag{15.34}$$

$$I_i^H = \iint d\mathbf{u}_f \mathbf{U}_i(\mathbf{u}_f)\varphi_0(\mathbf{u}_f). \tag{15.35}$$

As explained in Chapter 7, 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 the configuration files. The order of the basis functions are specified by the parameters KU and KV in the SPL file, or KSPLIN in the configuration files.

The *inner* integrations in $\mathbf{u}$ in equations (15.33) and (15.34) 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 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 the configuration files. Numerical tests suggest that the order of the quadratures should be one order higher than the order of the basis functions.

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 $\mathbf{u}_f$, 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. 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 the accuracy of most relevant hydrodynamic outputs are not affected significantly by this problem. In other cases the warning message may be an indication that the geometrical representation of the body surface is defective.

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 the same manner.

After an appropriate normalization of the length and use of local coordinates with the origin at (u_f, v_f), the integral takes the form

$$I = \int_{-1}^{1}\int_{-1}^{1} \frac{A(u,v)}{|\mathbf{x}(u,v)|} du dv \tag{15.36}$$

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 the 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 (15.36) can be expressed in the form

$$I = \int_{-1}^{1}\int_{-1}^{1} \frac{f(u,v)}{\sqrt{u^2+v^2}} du dv \tag{15.37}$$

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

$$I^{(1)} = \int_0^1 du \int_{-u}^{u} dv \frac{f(u,v)}{\sqrt{u^2+v^2}}. \tag{15.38}$$

After the change of variables $u = p$, $v = pq$, we have

$$I^{(1)} = \int_0^1 dp \int_{-1}^{1} dq \frac{f(p,pq)}{\sqrt{1+q^2}}. \tag{15.39}$$

After adding the contributions from other three triangles, we have

$$I = \int_{-1}^{1} dp \int_{-1}^{1} dq \frac{f(p, pq) + f(pq, p)}{\sqrt{1+q^2}}. \tag{15.40}$$

Next we remove the square-root function from (15.40). By change of the variables $q = \sinh(as)$, we have

$$I = a \int_{-1}^{1} dp \int_{-1}^{1} ds f(p, p\sinh as) + f(p\sinh as, p) \tag{15.41}$$

where $a = \sinh^{-1} 1$. The integral (15.41) 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 parameter ILOG in the configuration files 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.

15.6 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 (15.11) takes a form

$$\begin{aligned} 2\pi\varphi(\mathbf{x}) + \iint_{S_b} \varphi(\boldsymbol{\xi}) \frac{\partial G(\mathbf{x};\boldsymbol{\xi})}{\partial n_\xi} d\boldsymbol{\xi} + \iint_{S_i} \varphi'(\boldsymbol{\xi}) \frac{\partial G(\mathbf{x};\boldsymbol{\xi})}{\partial n_\xi} d\boldsymbol{\xi} \\ = \iint_{S_b} \frac{\partial \varphi(\boldsymbol{\xi})}{\partial n_\xi} G(\mathbf{x};\boldsymbol{\xi}) d\boldsymbol{\xi}, \qquad \mathbf{x} \in S_b, \end{aligned} \tag{15.42}$$

$$\begin{aligned} -4\pi\varphi'(\mathbf{x}) + \iint_{S_b} \varphi(\boldsymbol{\xi}) \frac{\partial G(\mathbf{x};\boldsymbol{\xi})}{\partial n_\xi} d\boldsymbol{\xi} + \iint_{S_i} \varphi'(\boldsymbol{\xi}) \frac{\partial G(\mathbf{x};\boldsymbol{\xi})}{\partial n_\xi} d\boldsymbol{\xi} \\ = \iint_{S_b} \frac{\partial \varphi(\boldsymbol{\xi})}{\partial n_\xi} G(\mathbf{x};\boldsymbol{\xi}) d\boldsymbol{\xi}, \qquad \mathbf{x} \in S_i. \end{aligned} \tag{15.43}$$

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 equations for the source formulation (15.26) are

$$2\pi\sigma(\mathbf{x}) + \iint_{S_b} \sigma(\boldsymbol{\xi}) \frac{\partial G(\mathbf{x};\boldsymbol{\xi})}{\partial n_x} d\boldsymbol{\xi} + \iint_{S_i} \sigma(\boldsymbol{\xi}) \frac{\partial G(\mathbf{x};\boldsymbol{\xi})}{\partial n_x} d\boldsymbol{\xi} = \frac{\partial \varphi(\boldsymbol{\xi})}{\partial n_x}, \qquad \mathbf{x} \in S_b, \tag{15.44}$$

$$-4\pi\sigma(\mathbf{x}) + \iint_{S_b} \sigma(\boldsymbol{\xi}) \frac{\partial G(\mathbf{x};\boldsymbol{\xi})}{\partial n_x} d\boldsymbol{\xi} + \iint_{S_i} \sigma(\boldsymbol{\xi}) \frac{\partial G(\mathbf{x};\boldsymbol{\xi})}{\partial n_x} d\boldsymbol{\xi} = \frac{\partial \varphi(\boldsymbol{\xi})}{\partial n_x}, \qquad \mathbf{x} \in S_i. \tag{15.45}$$

15.7 INTEGRAL EQUATIONS FOR BODIES WITH THIN SUBMERGED ELEMENTS

Green's integral equations in Section 15.2 become singular as the thickness of the body (or part 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 is zero. 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.

If the body surface S_b consists partly of thin 'dipole' elements S_d and partly of conventional 'source' elements S_s which are represented by both sources and dipoles, Green's integral equations (15.11-13) and the dipole distribution can be combined in the following form:

$$2\pi\phi(\mathbf{x}) + \iint_{S_s} \phi G_{n_\xi} dS_\xi + \iint_{S_d} \Delta\phi G_{n_\xi} dS_\xi = \iint_{S_s} \phi_{n_\xi} G dS_\xi \tag{15.46}$$

when $\mathbf{x}$ is on the conventional body surface S_s, and

$$\iint_{S_s} \phi G_{n_\xi n_x} dS_\xi + \iint_{S_d} \Delta\phi G_{n_\xi n_x} dS_\xi = -4\pi\phi_{n_x} + \iint_{S_s} \phi_{n_\xi} G_{n_x} dS_\xi \tag{15.47}$$

when $\mathbf{x}$ is on the surface of zero thickness S_d.

Instructions for using this option are in Sections 6.3 and 7.10. TEST09 and TEST21, described in Appendix A, are examples of its use.

It should be emphasized that thin dipole elements must be submerged, in contact with the fluid on both sides. A thin element which is in the plane of the free surface, and only in contact with the fluid on the lower side, should be considered as part of the conventional surface S_s.

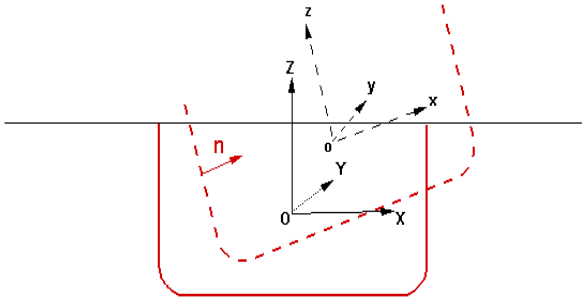


Figure 15.1: Coordinate systems fixed with respect to the body (dashed lines) and its mean position (solid lines)

15.8 MEAN DRIFT FORCES 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

$$\vec{X} = \vec{x} + \vec{\xi} + \vec{\alpha} \times \vec{x} + H\vec{x}. \tag{15.48}$$

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

$$\mathrm{H} = \frac{1}{2}\begin{pmatrix} -({\alpha_2}^2 + {\alpha_3}^2) & 0 & 0 \\ 2\alpha_1\alpha_2 & -({\alpha_1}^2 + {\alpha_3}^2) & 0 \\ 2\alpha_1\alpha_3 & 2\alpha_2\alpha_3 & -({\alpha_1}^2 + {\alpha_2}^2) \end{pmatrix}. \tag{15.49}$$

The normal vector in the inertial coordinate system can be expressed by

$$\vec{N} = \vec{n} + \vec{\alpha} \times \vec{n} + H\vec{n} \tag{15.50}$$

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 (15.48) can be expressed by

$$\begin{aligned} P = \; & - \; \rho[g(z + Z_o) + (\phi_t + g(\xi_3 + \alpha_1 y - \alpha_2 x)) \\ & + \; (\frac{1}{2}\nabla\phi \cdot \nabla\phi + (\vec{\xi} + \vec{\alpha} \times \vec{x}) \cdot \nabla\phi_t + gH\vec{x} \cdot \nabla z)] \end{aligned} \tag{15.51}$$

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:

$$\vec{F}^{(2)} = \iint_{\tilde{S}_b} \vec{N} P ds \tag{15.52}$$

$$\vec{M}^{(2)} = \iint_{\tilde{S}_b} (\vec{X} \times \vec{N}) P ds. \tag{15.53}$$

In equations (15.52) and (15.53), the integrations are over the exact body wetted surface $\tilde{S}_b$.

After substituting (15.48), (15.50), and (15.51) and integrating the hydrostatic components,

$$\begin{aligned}
\vec{F}^{(2)} &= \frac{1}{2}\rho g \int_{WL} \vec{n}'[\zeta^2 - 2\zeta(\xi_3 + \alpha_1 y - \alpha_2 x)]dl \\
&+ \frac{1}{2}\rho g \vec{k} \int_{WL} \frac{n_z}{\sqrt{1-n_z^2}}[\zeta - (\xi_3 + \alpha_1 y - \alpha_2 x)]^2 dl \\
&- \rho \iint_{S_b} \vec{n}(\frac{1}{2}\nabla\phi \cdot \nabla\phi + (\vec{\xi} + \vec{\alpha} \times \vec{x}) \cdot \nabla\phi_t) ds \\
&+ \vec{\alpha} \times (-\rho \iint_{S_b} \vec{n}\phi_t ds) \\
&+ \vec{F}_S^{(2)}
\end{aligned} \tag{15.54}$$

$$\begin{aligned}
\vec{M}^{(2)} &= \frac{1}{2}\rho g \int_{WL} (\vec{x} \times \vec{n}')[\zeta^2 - 2\zeta(\xi_3 + \alpha_1 y - \alpha_2 x)]dl \\
&+ \frac{1}{2}\rho g \int_{WL} \frac{n_z}{\sqrt{1-n_z^2}}(\vec{x} \times \vec{k})[\zeta - (\xi_3 + \alpha_1 y - \alpha_2 x)]^2 dl \\
&- \rho \iint_{S_b} (\vec{x} \times \vec{n})(\frac{1}{2}\nabla\phi \cdot \nabla\phi + (\vec{\xi} + \vec{\alpha} \times \vec{x}) \cdot \nabla\phi_t) ds \\
&- \rho\vec{\alpha} \times \iint_{S_b} (\vec{x} \times \vec{n})\phi_t ds \\
&+ \vec{\xi} \times (-\rho \iint_{S_b} \vec{n}\phi_t ds) \\
&+ \vec{M}_S^{(2)}
\end{aligned} \tag{15.55}$$

where the hydrostatic components are

$$\vec{F}_S^{(2)} = -\rho g A_{wp} \left\{ \alpha_1\alpha_3 x_f + \alpha_2\alpha_3 y_f + \tfrac{1}{2}(\alpha_1^2 + \alpha_2^2) Z_o \right\} \vec{k} \tag{15.56}$$

$$\begin{aligned}
\vec{M}_S^{(2)} = \rho g \Big\{ & [(\xi_2\alpha_2 - \xi_3\alpha_3) A_{wp} x_f - \xi_2\alpha_1 A_{wp} y_f + \alpha_1\alpha_2 A_{wp} x_f Z_o \\
& - (\tfrac{3}{2}\alpha_1^2 + \tfrac{1}{2}\alpha_2^2) A_{wp} y_f Z_o - \xi_2\xi_3 A_{wp} - \xi_3\alpha_1 A_{wp} Z_o
\end{aligned}$$

$$
\begin{aligned}
&+\alpha_2\alpha_3(L_{11}-L_{22})-2\alpha_1\alpha_3L_{12}+\alpha_1\alpha_2\forall x_b-\tfrac{1}{2}(\alpha_1^2+\alpha_3^2)\forall y_b]\,\vec{i}\\
&[-\xi_1\alpha_2A_{wp}x_f+(\xi_1\alpha_1-\xi_3\alpha_3)A_{wp}y_f+(\tfrac{1}{2}\alpha_1^2+\tfrac{3}{2}\alpha_2^2)A_{wp}x_fZ_o\\
&-\alpha_1\alpha_2A_{wp}y_fZ_o+\xi_1\xi_3A_{wp}-\xi_3\alpha_2A_{wp}Z_o+\alpha_1\alpha_3(L_{11}-L_{22})\\
&+2\alpha_2\alpha_3L_{12}+\tfrac{1}{2}(\alpha_2^2+\alpha_3^2)\forall x_b]\,\vec{j}\Big\}. \qquad (15.57)
\end{aligned}
$$

Here 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.

15.9 MEAN DRIFT FORCES USING CONTROL SURFACES

The mean drift force and moment are evaluated by one the following two alternatives depending on the value of IALTCSF in the CFG file.

IALTCSF=1:

$$\begin{aligned}
\vec{F}^{(2)} = & - \frac{1}{2}\rho g \int_{CL} \vec{n}'\zeta^2 dl - \rho g \vec{k} \int_{WL} \zeta[(\vec{\xi} + \vec{\alpha} \times \vec{x}) \cdot \vec{n}']dl \\
& + \frac{1}{2}\rho g \vec{k} \int_{WL} \frac{n_z}{\sqrt{1-n_z^2}}[\zeta - (\xi_3 + \alpha_1 y - \alpha_2 x)]^2 dl \\
& - \rho \iint_{S_c} [\nabla\phi \frac{\partial\phi}{\partial n} - \frac{1}{2}\vec{n}(\nabla\phi \cdot \nabla\phi)]ds \\
& + \rho \vec{k} \iint_{S_f} (\zeta \frac{\partial\phi_t}{\partial z} + \frac{1}{2}\nabla\phi \cdot \nabla\phi)ds + \vec{F}_S^{(2)}
\end{aligned} \tag{15.58}$$

$$\begin{aligned}
\vec{M}^{(2)} = & - \frac{1}{2}\rho g \int_{CL} (\vec{x} \times \vec{n}')\zeta^2 dl - \rho g \int_{WL} \zeta[(\vec{\xi} + \vec{\alpha} \times \vec{x}) \cdot \vec{n}'](\vec{x} \times \vec{k})dl \\
& + \frac{1}{2}\rho g \int_{WL} \frac{n_z}{\sqrt{1-n_z^2}}(\vec{x} \times \vec{k})[\zeta - (\xi_3 + \alpha_1 y - \alpha_2 x)]^2 dl \\
& - \rho \iint_{S_c} [(\vec{x} \times \nabla\phi)\frac{\partial\phi}{\partial n} - \frac{1}{2}(\vec{x} \times \vec{n})(\nabla\phi \cdot \nabla\phi)]ds \\
& + \rho \iint_{S_f} (\vec{x} \times \vec{k})(\zeta \frac{\partial\phi_t}{\partial z} + \frac{1}{2}\nabla\phi \cdot \nabla\phi)ds + \vec{M}_S^{(2)}
\end{aligned} \tag{15.59}$$

IALTCSF=2:

$$\begin{aligned}
\vec{F}^{(2)} = & \frac{1}{2}\rho g \int_{WL} \vec{n}'\zeta^2 dl - \rho g \iint_{S_f} \zeta\nabla'\zeta ds \\
& - \rho g \vec{k} \int_{WL} \zeta[(\vec{\xi} + \vec{\alpha} \times \vec{x}) \cdot \vec{n}']dl \\
& + \frac{1}{2}\rho g \vec{k} \int_{WL} \frac{n_z}{\sqrt{1-n_z^2}}[\zeta - (\xi_3 + \alpha_1 y - \alpha_2 x)]^2 dl \\
& - \rho \iint_{S_c} [\nabla\phi \frac{\partial\phi}{\partial n} - \frac{1}{2}\vec{n}(\nabla\phi \cdot \nabla\phi)]ds \\
& + \rho \vec{k} \iint_{S_f} (\zeta \frac{\partial\phi_t}{\partial z} + \frac{1}{2}\nabla\phi \cdot \nabla\phi)ds + \vec{F}_S^{(2)}
\end{aligned} \tag{15.60}$$

$$
\begin{aligned}
\vec{M}^{(2)} &= \frac{1}{2}\rho g \int_{WL} (\vec{x} \times \vec{n}')\zeta^2 dl - \rho g \iint_{S_f} \zeta(\vec{x} \times \nabla'\zeta) ds \\
&- \rho g \int_{WL} \zeta[(\vec{\xi} + \vec{\alpha} \times \vec{x}) \cdot \vec{n}'](\vec{x} \times \vec{k}) dl \\
&+ \frac{1}{2}\rho g \int_{WL} \frac{n_z}{\sqrt{1 - n_z^2}} (\vec{x} \times \vec{k})[\zeta - (\xi_3 + \alpha_1 y - \alpha_2 x)]^2 dl \\
&- \rho \iint_{S_c} [(\vec{x} \times \nabla\phi)\frac{\partial \phi}{\partial n} - \frac{1}{2}(\vec{x} \times \vec{n})(\nabla\phi \cdot \nabla\phi)] ds \\
&+ \rho \iint_{S_f} (\vec{x} \times \vec{k})(\zeta \frac{\partial \phi_t}{\partial z} + \frac{1}{2}\nabla\phi \cdot \nabla\phi) ds + \vec{M}_S^{(2)}
\end{aligned} \tag{15.61}
$$

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 hydrostatic components (15.56-15.57). The derivations of the equations (15.58-15.61) from (15.54-15.55) are given in Reference [28].

15.10 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

$$\phi = i\omega \sum \xi_j \phi_j, \tag{15.62}$$

and the first-order pressure at a fixed point on the tank surface is given by

$$P = -\rho g(z - Z_T + \xi_3 + \alpha_1 y - \alpha_2 x) - \rho \phi_t. \tag{15.63}$$

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

$$\mathbf{F} = \int\int_{S_T} P\mathbf{N} dS = \int\int_{S_T} P\left(\mathbf{n} + \alpha \times \mathbf{n}\right) dS, \tag{15.64}$$

$$\mathbf{M} = \int\int_{S_T} P\,(\mathbf{X} \times \mathbf{N})\,dS = \int\int_{S_T} P\,(\mathbf{x} + \xi + \alpha \times \mathbf{x}) \times (\mathbf{n} + \alpha \times \mathbf{n})\,dS, \tag{15.65}$$

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 (15.52-15.53) for the contributions from the hydrostatic pressure:

$$C_T(3,3) = \rho g \int\int_{S_T} n_3 dS,$$

$$C_T(3,4) = \rho g \int\int_{S_T} y n_3 dS,$$

$$C_T(3,5) = -\rho g \int\int_{S_T} x n_3 dS,$$

$$C_T(4,4) = \rho g \int\int_{S_T} y^2 n_3 dS - \rho g \forall_T z_c,$$

$$C_T(4,5) = -\rho g \int\int_{S_T} xy n_3 dS,$$

$$C_T(4,6) = \rho g \forall_T x_c,$$

$$C_T(5,5) = \rho g \int\int_{S_T} x^2 n_3 dS - \rho g \forall_T z_c,$$

$$C_T(5,6) = \rho g \forall_T y_c.$$

All other elements of the matrix C_T are equal to zero. Here

$$\forall_T = \text{VOLTANK}(1) = \int\int_{S_T} x n_1 dS,$$

$$\forall_T = \text{VOLTANK}(2) = \int\int_{S_T} y n_2 dS,$$

$$\forall_T = \text{VOLTANK}(3) = \int\int_{S_T} (z - Z_T) n_3 dS,$$

$$x_c = \frac{1}{2\forall_T} \int\int_{S_T} x^2 n_1 dS,$$

$$y_c = \frac{1}{2\forall_T} \int\int_{S_T} y^2 n_2 dS,$$

$$z_c = \frac{1}{2\forall_T} \int\int_{S_T} (z^2 - Z_T^2) n_3 dS.$$

In WAMIT the hydrostatic parameters of the hull and tanks are evaluated 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 3.1. 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):

$$\text{CTANK}(1) = \int\int_{S_T} n_3 dS,$$

$$\text{CTANK}(2) = -\int\int_{S_T} x n_3 dS,$$

$$\text{CTANK}(3) = \int\int_{S_T} y n_3 dS,$$

$$\text{CTANK}(4) = \int\int_{S_T} x^2 n_1 dS,$$

$$\text{CTANK}(5) = \int\int_{S_T} y^2 n_2 dS,$$

$$\text{CTANK}(6) = \int\int_{S_T} (z^2 - Z_T^2) n_3 dS,$$

$$\text{CTANK}(7) = \int\int_{S_T} x^2 n_3 dS,$$

$$\text{CTANK}(8) = -\int\int_{S_T} xy n_3 dS,$$

$$\text{CTANK}(9) = \int\int_{S_T} y^2 n_3 dS.$$

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.

When tanks are present, the header of the output file frc.out includes nondimensional values of the tank volumes and the contributions from the tanks to $C(3,3)$, $C(3,4)$ and $C(3,5)$.

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

$$A_{33} = \rho \int\int_{S_T} n_3\phi_3 dS = \rho\forall_T + C_T(3,3)/(gK). \tag{15.66}$$

$$A_{34} = \rho \int\int_{S_T} (yn_3 - zn_2)\phi_3 dS = \rho\forall_T y_c + C_T(3,4)/(gK), \tag{15.67}$$

$$A_{35} = \rho \int\int_{S_T} (zn_1 - xn_3)\phi_3 dS = -\rho \forall_T x_c + C_T(3,5)/(gK). \tag{15.68}$$

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.

15.11 BODIES WITH PRESSURE SURFACES

The boundary-value problem described in Section 15.1 is extended here to include problems where part or all of the body surface is one or more free surfaces, and a nonzero oscillatory pressure acts on these surfaces. On each of these surfaces the hydrostatic pressure is assumed constant, and non-negative. Thus the surface is in a horizontal plane at $z \leq 0$. The oscillatory component of the pressure is defined as the real part of $p_0(x,y)e^{i\omega t}$. The complete boundary surface $S_b = S_w + S_p$ includes the 'wetted' surface S_w and the 'pressure' surface S_p. To simplify the notation it is assumed here that NBODY=1 and there are no generalized modes. The Neumann boundary conditions (15.9) and (15.10) are applicable on S_w, and the boundary condition on S_p is

$$\varphi_z - K\varphi = -\left(\frac{i\omega}{\rho g}\right) p_0(x,y). \tag{15.69}$$

It is convenient to represent the pressure distribution by

$$p_0(x,y) = -\rho g \sum_{j=7}^{6+M_p} \xi_j n_j(x,y), \tag{15.70}$$

where, for $j > 6$, ξ_j is a normalized coefficient with the dimension of length and n_j is a nondimensional real function of position which represents the spatial dependence of the corresponding pressure mode. The number of modes M_p required to represent the pressure distribution will depend on the application. The total potential is given by (15.6), with the radiation potential extended in the form

$$\varphi_R = i\omega \sum_{j=1}^{6+M_p} \xi_j \varphi_j. \tag{15.71}$$

If the functions n_j are extended to apply on both S_b and S_p, with the definitions

$$n_j = 0 \quad \text{on} \quad S_w \quad (j > 6)$$

$$n_j = 0 \quad \text{on} \quad S_p \quad (j \leq 6)$$

the boundary conditions (15.9) and (15.69) can be combined in the form

$$\mathcal{L}\varphi_j = n_j \quad \text{on} S_w + S_p, \tag{15.72}$$

where

$$\begin{aligned} \mathcal{L} &= \partial/\partial n \qquad \text{on} \quad S_w, \\ &= \partial/\partial n - K \quad \text{on} \quad S_p. \end{aligned} \tag{15.73}$$

When necessary $\mathcal{L}_x$ and $\mathcal{L}_\xi$ will be used to indicate that the normal derivative is with respect to the corresponding coordinates.

The integral equation (15.11) for the velocity potential is replaced by

$$\begin{pmatrix} 2\pi \\ 4\pi \end{pmatrix} \varphi_j(\mathbf{x}) + \iint_{S_b} \varphi_j(\boldsymbol{\xi}) \mathcal{L}_\xi G(\boldsymbol{\xi}; \mathbf{x}) d\boldsymbol{\xi} = \iint_{S_b} n_j G(\boldsymbol{\xi}; \mathbf{x}) d\boldsymbol{\xi}. \tag{15.74}$$

The factor 2π is applied if the boundary surface is below $z = 0$, and 4π is applied if the boundary surface is on $z = 0$. In the latter case the integral over S_p on the left-hand-side of (15.74) vanishes, since the Green function satisfies the homogeneous free-surface condition. In the source formulation, where the potential is given by (15.24), the integral equation corresponding to (15.26) is

$$\begin{pmatrix} 2\pi \\ 4\pi \end{pmatrix} \sigma_j(\mathbf{x}) + \iint_{S_b} \sigma_j(\boldsymbol{\xi}) \mathcal{L}_x G(\boldsymbol{\xi}; \mathbf{x}) d\boldsymbol{\xi} = \iint_{S_b} n_j G(\boldsymbol{\xi}; \mathbf{x}) d\boldsymbol{\xi}. \tag{15.75}$$

In the diffraction problem the potential (15.8) satisfies the integral equation

$$\begin{pmatrix} 2\pi \\ 4\pi \end{pmatrix} \varphi_D(\mathbf{x}) + \iint_{S_b} \varphi_D(\boldsymbol{\xi}) \mathcal{L}_\xi G(\boldsymbol{\xi}; \mathbf{x}) d\boldsymbol{\xi} = 4\pi\varphi_0. \tag{15.76}$$

The definitions of the added-mass and damping coefficients in the first equation of Section 3.2 are unchanged, except for the extended range of the subscripts (i, j). In the definition of the Haskind exciting forces in Section 4.3 the normal derivative of the incident-wave potential is replaced by $\mathcal{L}\varphi_0$. The definition of the exciting forces from direct integration of pressure is given by the second equation in Section 3.3, without change.

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. J. N. Newman, "Wave Effects on Vessels with Internal Tanks". *20th International Workshop on Water Waves and Floating Bodies*, Spitsbergen, Norway, 2005.

28. C.-H. Lee, "Evaluation of quadratic forces using control surfaces". *2005 WAMIT Consortium Report*, www.wamit.com\report05.pdf, 2005.

29. J. V. Wehausen, "The motion of floating bodies". Annual Review of Fluid Mechanics, Vol. 3, pp 237-268 (1971).

30. F. T. Korsmeyer, H. B. Bingham and J. N. Newman, "A panel-method program for transient wave-body interactions", http://www.wamit.com/Publications/TiMITmanual.pdf

Appendix A

DESCRIPTION OF TEST RUNS

WAMIT V7.0 includes 37 standard test runs, including 12 low-order and 25 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 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). Test runs which are identical except for different input options are assigned the same number with a letter suffix. For example, 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 a rectangular barge is defined by the subroutine BARGE (IGDEF=-5), and in TEST16a the patches are defined by flat panels (IGDEF=0). Tests 17 and 17a-c illustrate alternative methods for analyzing a body with moonpools, as explained in Section A.17. Tests 01a, 09a, 13a and 22a are examples showing the use of trimmed waterlines (ITRIMWL=1).

tst	description	ILOWHI	other parameters
01	Circular cylinder	0	
01a	Circular cylinder	0	ITRIMWL=1
02	Circular cylinder	0	IRR=3
03	Circular cylinder	0	ISOR=1
04	Barge near wall	0	IWALLy0=1
05	Cylinder & spheroid	0	NBODY=2
05a	Cylinder & spheroid	0	NBODY=2, ISx=1
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	NPDIPOLE=(673 960)
09a	Spar with strakes	0	ITRIMWL=1
11	Circular cylinder	1	IGDEF=1
11a	Circular cylinder	1	IGDEF=-1
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
13a	Cylinder & spheroid	1	NBODY=2, ITRIMWL=1
14	ISSC TLP	1	IGDEF=-9
14a	ISSC TLP	1	NPER=101, IPERIN=2
15	Semi-sub	1	IGDEF=-10
16	Elastic barge	1	IGDEF=-5, NEWMDS=8
16a	Elastic barge	1	IGDEF=0, NEWMDS=8
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
17c	Cylinder & moonpool	1	IGDEF=-7, NMODESFSP=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, NPDIPOLE = 2 4 6
22	FPSO with 2 tanks	1	IGDEF=-21, NPTANK=(8-11) (12-15)
22a	FPSO with 2 tanks	1	ITRIMWL=1
23	Bank of wavemakers	1	IGDEF=0, ISOLVE=-1, NEWMDS=8
24	Motions of a hinged vessel	1	IGDEF=-32, NEWMDS=4
25	ACV with two pressure chambers	1	IGDEF=0, NMODESFSP=2

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 the perspective figures show the subdivisions into panels. For

the higher-order tests two perspective figures are included, to show the subdivisions into patches (upper or left) and into panels (lower or right).

All of the required input files for each test run, and the labeled output file (*.out) are included with the WAMIT software 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 extensions **.wam** and **.cfg**.

The additional configuration file `config.wam` is included with the test files, and is intended to supplement the separate .cfg file for each test. The standard version of this file is as shown below:

```
! generic configuration file:  config.wam
  RAMGBMAX=0.5
  NCPU=1
  USESRID_PATH=c:\wamitv7
```

The first line is a comment line which is ignored by the program. The parameters on the other lines are explained in Section 4.7.

Before running TEST*tst*, the user should copy the fnames files as follows:

```
copy test tst .wam fnames.wam
```

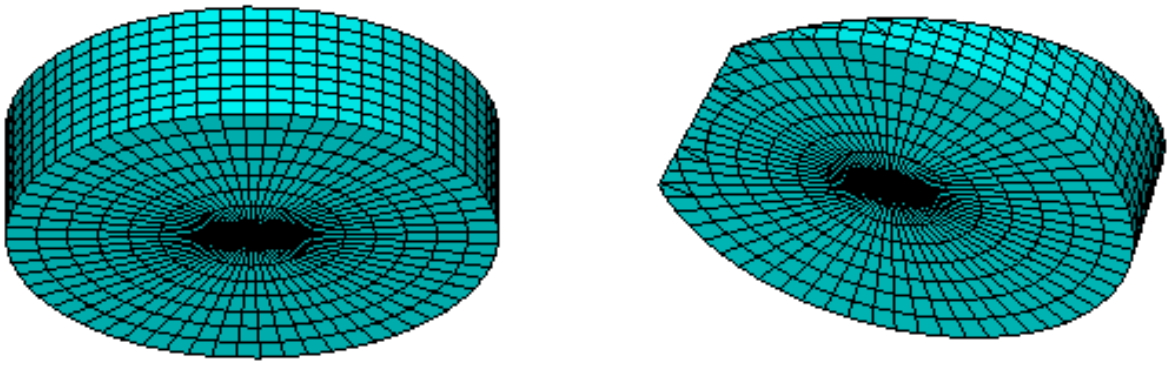
as explained in Chapter 2. Alternatively, the batch file `runtests.bat` can be used to run all tests in succession.

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.

In TEST01A, the option to trim the waterline is specified with the parameters ITRIMWL and XTRIM included in the TEST01A.CFG file. The other input files are unchanged, but the filenames TEST01A.POT and TEST01A.FRC are used so that the output files will be named accordingly. The cylinder is rotated 15 degrees about the $x-$axis and elevated 0.27m, as shown in the lower figure below. The vertical elevation is required in this case since the gdf file only extends up to the original waterplane, and trimming in roll or pitch about the center without vertical displacement would submerge half of the waterline with a gap above it.



```
 Input file:  fnames.wam
test01.pot
test01.frc
test01.cfg

 Input file:  test01.cfg
! TEST01.CFG -- cylinder R=1, T=0.5, ILOWHI=0, IRR=0
 ipltdat=5
 ISOR=1       (omit ISOR in POT file, include source formulation)
 ISOLVE=0     (use iterative solver)
 ISCATT=0     (solve for total diffraction potential, not scattering)
 ILOG=1       (omit ILOG in POT file, integrate log singularity)
 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)

 Input file:  test01.pot
TEST01.POT -- cylinder R=1, T=0.5, ILOWHI=0, IRR=0
 -1.       HBOT
   1   1                           IRAD,IDIFF
 3                               NPER (array PER follows)
 8.971402 2.006403 1.003033      PER
 1                               NBETA (array BETA follows)
 0.                              BETA
   1                             NBODY
 test01.gdf
 0. 0. 0. 0.                   HBOT, XBODY(1-4)
  1  1  1  1  1  1                 IMODE(1-6)

 First 10 lines of input file:  test01.gdf
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
   0.1250000      0.0000000E+00 -0.5000000
   0.1243981      1.2252143E-02 -0.5000000
```

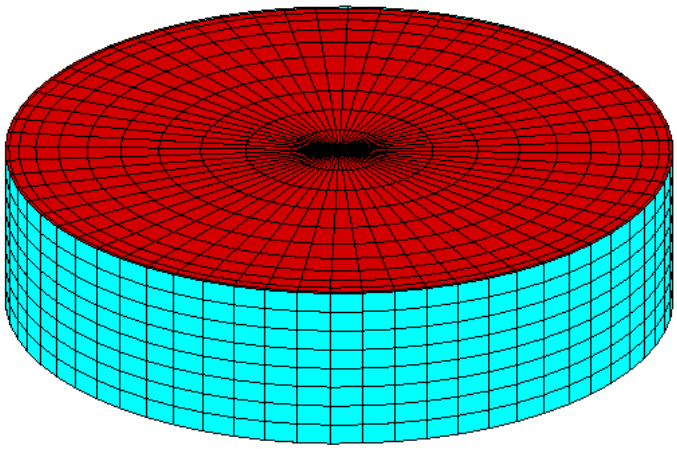
```
Input file:  test01.frc
TEST01.FRC  Circular cylinder, ILOWHI=0, IRR=0
  1   1   1   1   0   3   0   2   1
0.000000                                      VCG
1.000000      .0000000      .0000000
.0000000      1.000000      .0000000
.0000000      .0000000      1.000000        XPRDCT
0                                             NBETAH
2                                             NFIELD
1.5  0.  0.
1.5  0. -0.5                                  XFIELD
```

```
 Input file:  test01a.cfg
! TEST01A.CFG -- cylinder R=1, T=0.5, trimmed waterline
 ilowgdf=1
 ipltdat=5
 ISOR=1       (omit ISOR in POT file, include source formulation)
 ISOLVE=0     (use iterative solver)
 ISCATT=0     (solve for total diffraction potential, not scattering)
 ILOG=1       (omit ILOG in POT file, integrate log singularity)
 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)
 ITRIMWL=1
 XTRIM= 0.27 0.0 15.

 Input file:  test01a.pot
TEST01A.POT -- cylinder R=1, T=0.5, trimmed waterline
 -1.       HBOT
   1   1                          IRAD,IDIFF
 3                              NPER (array PER follows)
 8.971402 2.006403 1.003033     PER
 1                              NBETA (array BETA follows)
 0.                             BETA
   1                            NBODY
 test01.gdf
 0. 0. 0. 0.                  HBOT, XBODY(1-4)
  1  1  1  1  1  1                IMODE(1-6)

 Input file:  test01a.frc
 TEST01A.FRC  Circular cylinder, trimmed waterline
   1   1   1   1   0   3   0   2   0
 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                                XFIELD
```

A.2 IRREGULAR-FREQUENCY REMOVAL – TEST02

This test run illustrates the use of the irregular-frequency option described in Chapter 10. The geometry and most other inputs are the same as in TEST01. 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 direct solver (ISOLVE=1) is used since the iterative and block-iterative solvers do not converge reliably for the source formulation (ISOR=1).

The GDF input is the same as TEST01.GDF. The additional panels on the interior free surface, which are generated automatically by the program, are shown in red in the Figure below.



```
 Input file:  test02.cfg
!  TEST02.CFG -- Circular cylinder, ILOWHI=0, IRR=3, direct solver
 ilowgdf=1
 ipltdat=5
 IRR=3
 ISOR=1
 ISOLVE=1
 ILOG=1
 MONITR=0
 NUMHDR=1

 Input file:  test02.pot
 TEST02.POT -- Circular cylinder, ILOWHI=0, IRR=3
 -1.       HBOT
   1   1                          IRAD,IDIFF
 2
 1.182288    1.003025
 1
 0.0
   1                          NBODY
 test02.gdf
  0.0  0.0  0.0  0.0
  1  1  1  1  1  1

 First 10 lines of input file:  test02.gdf
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
   0.1250000      0.0000000E+00 -0.5000000
   0.1243981      1.2252143E-02 -0.5000000
```

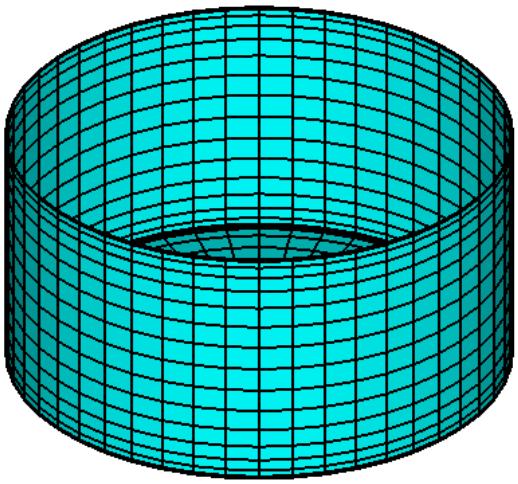
```
Input file:  test02.frc
TEST02.FRC  -- Circular cylinder, ILOWHI=0, IRR=3
  1   1   1   1   0   3   0   1   1
0.000000
1.000000      .0000000      .0000000
.0000000      1.000000      .0000000
.0000000      .0000000      1.000000
0
2
1.5 0.0  0.0
1.5 0.0 -0.5
```

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.



```
 Input file:  test03.cfg
! TEST03.CFG  Cylinder, R=T=1, ILOWHI=0, ISOR=1
 ilowgdf=1
 ipltdat=5
 ISOR=1
 ISOLVE=0
 ISCATT=0
 ILOG=0
 IRR=0
 MONITR=0
 NUMHDR=1

 Input file:  test03.pot
TEST03.POT  Cylinder, R=T=1, ILOWHI=0, ISOR=1
    7.14       HBOT
   1   1                          IRAD,IDIFF
    4
    2.837491 2.398118 2.006409 1.638226
    1
    0.0
   1                           NBODY
 test03.gdf
  0.0  0.0  -0.515 0.0
     1  1  1  1  1  1

 First 10 lines of input file:  test03.gdf
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
   0.1950903      0.0000000E+00 -0.4850000
   0.1934213      2.5464399E-02 -0.4850000
```

```
 Input file:  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
```

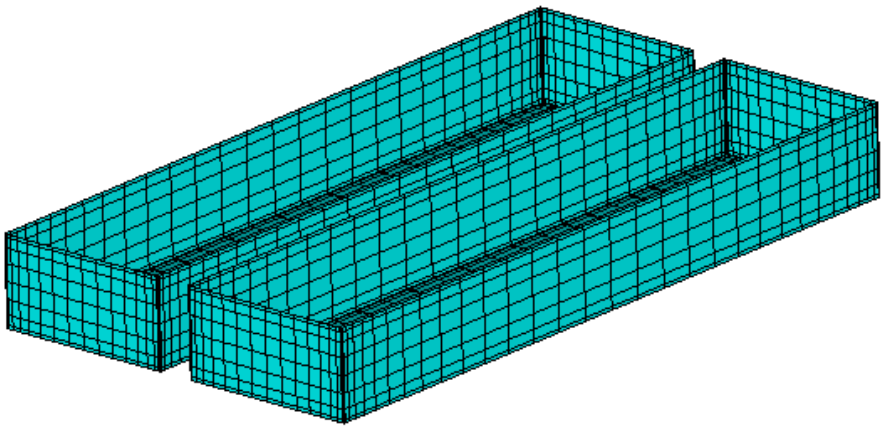
A.4 BODY NEAR A WALL – TEST04

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. The figure below shows the catamaran configuration or, equivalently, the original hull plus its image with respect to the wall.



```
 Input file:  test04.cfg
! TEST04.POT -- Barge near wall, ILOWHI=0
 ilowgdf=1
 ipltdat=5
 ISOR=1
 ISOLVE=0
 ISCATT=0
 ILOG=0
 IRR=0
 MONITR=0
 NUMHDR=1
 IWALLY0=1

 Input file:  test04.pot
TEST04.POT -- Barge near wall, ILOWHI=0
 -1.       HBOT
   0   0                          IRAD,IDIFF
 3
 6.  7.  8.
 1
 0.0
   1                           NBODY
 test04.gdf
 0. 12. 0. 0.
  1  0  1  0  1  0

 First 10 lines of input file:  test04.gdf
TEST04.GDF -- Barge near wall, ILOWHI=0
    40.00000       9.806650
     1     0       ISX, ISY
          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
    3.920686       10.00000      -1.464466
   0.0000000E+00   10.00000      -1.464466
```

```
 Input file:  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
```

A.5 MULTIPLE BODIES – TEST05

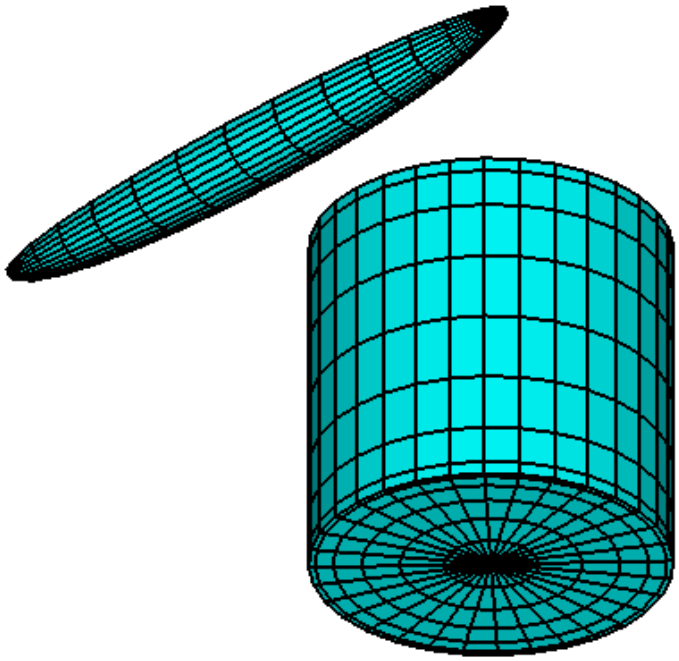
The NBODY option described in Chapter 7 is illustrated in this test run. Body one is a circular cylinder of radius 1 meter and draft 2 meters. Body 2 is a spheroid of length 4 meters and maximum radius 0.25 meters. The gap between these two bodies is set equal to the beam of the spheroid (0.5 meters) 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 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 11. 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.

In Test05a the relative orientations of the two bodies are the same, but they are positioned such that the 90 degree rotation of the spheroid is not required, and the plane X=0 is a plane of symmetry for both bodies, as explained in Section 8.5. This reduces the number of equations (NEQN) by a factor of one-half, and reduces the run time and storage requirements. The outputs from the two runs are essentially the same, except that the directions of the coordinates are changed with corresponding changes in the definitions of the force coefficients.



```
 Input file:  test05.cfg
! TEST05.CFG -- Cylinder + spheroid, NBODY=2, ILOWHI=0
 IPLTDAT=1
 ISOR=1
 ILOG=0
 IRR=0
 NUMHDR=1
 NOOUT=0 1 1 1 0 1 1 1 1
 IALTFRC = 3    !  Alternative Form 3 FRC
 IALTFRCN= 1 1

 Input file:  test05.pot
 TEST05.POT -- Cylinder + spheroid, ILOWHI=0
 -1.0       HBOT
   0   0                          IRAD,IDIFF
 2
 1.5 2.0
 1
 0.0
   2                          NBODY
 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

 First 10 lines of input file:  test05c.gdf
 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
   0.2588190      0.0000000E+00  -2.000000
   0.2538459      5.0493091E-02  -2.000000

 First 10 lines of input file:  test05s.gdf
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
    2.000000      0.0000000E+00 -0.0000000E+00
    1.961571      9.5150545E-03 -4.7835436E-02

 Input file:  test05.frc
 TEST05.FRC -- Cylinder + spheroid, ILOWHI=0
   1   1   1   1   0   3   1   1   1
  1.0
test05c.frc
test05s.frc
  0
  1
  0. 0. 0.
```

```
 Input file:  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

 Input file:  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

 First 10 lines of input file:  test05c.csf
cylinder R=1.2 T=2.2 -- low-order control surface
    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
  0.11769E+01  0.23411E+00  0.00000E+00
  0.11769E+01  0.23411E+00 -0.27500E+00

 Input file:  test05s.csf
ELLIPSOID CONTROL SURFACE defined by subroutine ELLIPSOID_CS
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)
```

```
 Input file:  test05a.pot
 TEST05a.POT -- Cylinder + spheroid, ILOWHI=0
 -1.0       HBOT
   0   0                          IRAD,IDIFF
 2          NPER
 1.5 2.0    PER
 1          NBETA
 90.0       Beta
   2                           NBODY
 test05c.gdf
   0.0 1.25 0.0 0.0
  1   1   1   1   1   1
 test05s.gdf
   0.0 -0.5 0.0 0.0
  1   1   1   1   1   1

 Input file:  test05a.frc
 TEST05a.FRC -- Cylinder + spheroid, ILOWHI=0
   1   1   1   1   0   3   1   1   1
  1.0
test05c.frc
test05s.frc
  0
  1
  0. 0. 0.
```

A.6 THE ISSC TENSION-LEG PLATFORM – TEST06

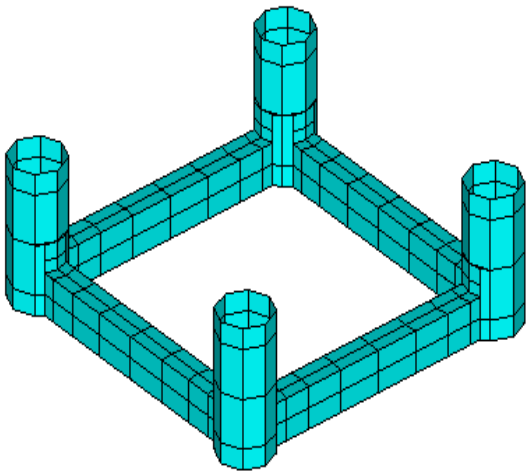
The added-mass, damping coefficients, exciting forces, motions and wave loads are evaluated for the ISSC Tension-Leg-Platform, in a finite water depth of 450 meters, for three wave periods and one wave heading. The TLP consists of four circular cylindrical columns and four rectangular pontoons as shown 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. There is a warning message for options 5-9, as explained in Section 10.1, since IDIFF=0.

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.



```
 Input file:  test06.cfg
! TEST06.CFG -- ISSC TLP, coarse discretization
 ipltdat=1
 ISOR=1
 ISOLVE=4
 ISCATT=1
 ILOG=0
 IRR=0
 MONITR=0
 NUMHDR=1
 IALTFRC=2

 Input file:  test06.pot
TEST06.POT -- ISSC TLP, coarse discretization
 450.       HBOT
   0   0                          IRAD,IDIFF
 3
 5. 10. 15.
 1
 0.
   1                          NBODY
 test06.gdf
 0. 0. 0. 0.
  1  0  1  0  1  0

 First 10 lines of input file:  test06.gdf
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
                49.09267       37.15733       -5.12567
                49.09267       37.15733      -17.50013
```

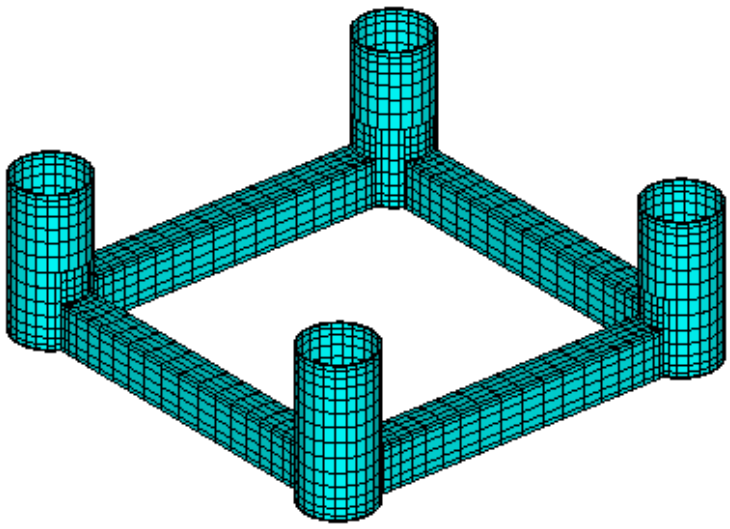
```
Input file:  test06.frc
TEST07.FRC  ISSC TLP -- ILOWHI=0, fine discretization -- IALTFRC=2
1  1  1 -2  0  0  0  1  1  IOPTN (IOPTN(4)<0 signifies fixed modes)
6                          NDFR
1 1 0 0 0 1                IMODE
1.                         RHO
0. 0. 3.0                  XCG
1                          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
```

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. As explained in Appendix A.6 a warning message is displayed for Options 8 and 9 since IDIFF=0.



```
 Input file:  test07.cfg
! TEST07.CFG  ISSC TLP -- ILOWHI=0, fine discretization
 ipltdat=1
 ISOR=1
 ISOLVE=4
 ISCATT=1
 ILOG=1
 IRR=0
 MONITR=0
 NUMHDR=1
 IALTFRC=2

 Input file:  test07.pot
 TEST07.POT  ISSC TLP -- ILOWHI=0, fine discretization
 450.       HBOT
   0   0                          IRAD,IDIFF
 3
 5. 10. 15.
 1
 0.
   1                          NBODY
 test07.gdf
 0. 0. 0. 0.
  1  0  1  0  1  0

 First 10 lines of input file:  test07.gdf
 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
     49.09267      37.15733      -0.33626
     49.09267      37.15733      -1.33212
```

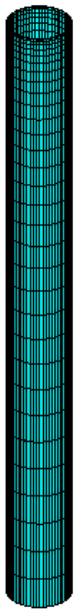
```
Input file:  test07.frc
TEST07.FRC  ISSC TLP -- ILOWHI=0, fine discretization -- IALTFRC=2
1  1  1 -2  0  0  0  1  1  IOPTN (IOPTN(4)<0 signifies fixed modes)
6                         NDFR
1 1 0 0 0 1               IMODE
1.                        RHO
0. 0. 3.0                 XCG
1                         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
```

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.



```
 Input file:  test08.cfg
! TEST08.CFG -- bending of vertical column with 4 generalized modes
 ilowgdf=1
 ipltdat=5
 NUMHDR=1
 NUMNAM=0
 ISOR=0
 IRR=0
 MONITR=0
 NEWMDS=4
 IALTFRC=2

 Input file:  test08.pot
TEST08.POT -- bending of vertical column at resonance, 200m depth
      200.0       HBOT
   0   0                          IRAD,IDIFF
           1
           6.5
           1
           0.0
   1                          NBODY
 test08.gdf
  0.0  0.0  0.0  0.0
        0   0   0   0   0   0

 First 10 lines of input file:  test08.gdf
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
    10.0000       0.000000       -190.186
    9.95185       0.980171       -190.186
```

```
 Input file:  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
```

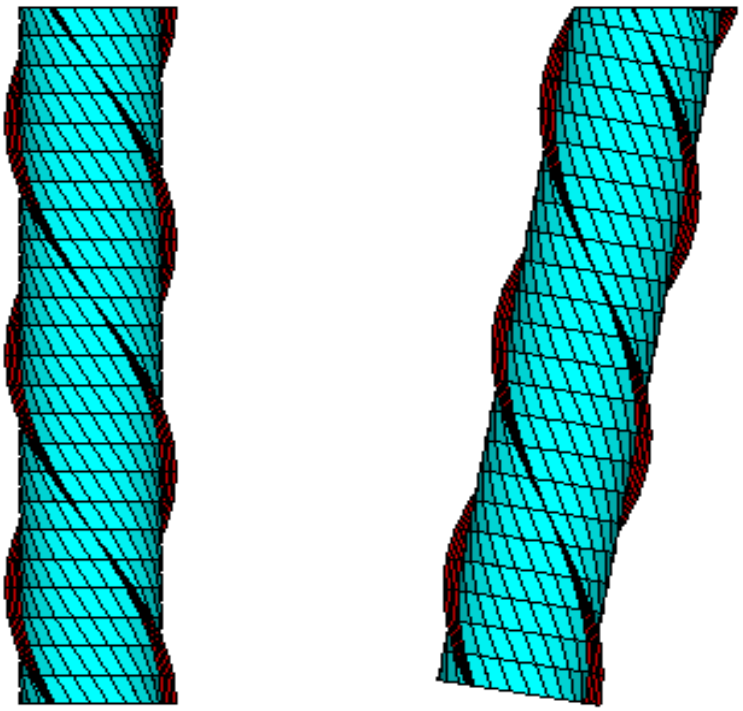
A.9 SPAR WITH THREE STRAKES – TEST09

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.

In TEST09A the trimmed waterline option is used, with the parameters ITRIMWL and XTRIM specified in the file TEST09A.CFG. These parameters specify a vertical trim of 10m and a pitch angle of 10 degrees. The same TEST09.GDF file is used for both tests. Only the filenames are changed in TEST09A.POT and TEST09A.FRC. Perspective views in the untrimmed and trimmed conditions are shown below.

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.

The figure below shows the submerged body surfaces for both TEST09 and TEST09a. The dipole panels are red.



```
 Input file:  test09.cfg
! TEST09.CFG (Spar with three strakes)
 ilowgdf=1
 ipltdat=5
 ISOR=0
 ISOLVE=0
 ISCATT=1
 ILOG=0
 IRR=0
 MONITR=0
 NUMHDR=1
 IPERIN=3
 IPEROUT=3
 NOOUT= 1 1 1 1 0 1 1 1 1
 NPDIPOLE = ( 673  960 )

 Input file:  test09.pot
TEST09.POT (Spar with three strakes)
 -1.       HBOT
   1   1                          IRAD,IDIFF
 3                               NPER (array PER follows)
 0.1 0.5 1.0
 2                               NBETA (array BETA follows)
 0.0 45.                         (end of file)
   1                           NBODY
 test09.gdf
 0. 0. 0. 0.                HBOT, XBODY(1-4)
  1  1  1  1  1  1               IMODE(1-6)

 First 10 lines of input file:  test09.gdf
 SPAR  R, D, W, T, NS, TWIST=   18.00  200.00    3.70  0.000000    3  1.000
   18.00000       9.806650
           0           0
     960
   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
   17.38667       4.658743      -8.333333
   15.58846       9.000000      -16.66667
```

```
 Input file:  test09.frc
TEST09.FRC (Spar with three strakes)
   1   1   1   1   1   3   0   2   0
 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)
```

```
 Input file:  test09a.cfg
! TEST09a.CFG -- Spar with trimmed waterline
 ISOLVE=0
 ISCATT=1
 ILOG=0
 IRR=0
 MONITR=0
 NUMHDR=1
 IPERIN=3
 IPEROUT=3
 NOOUT= 1 1 1 1 0 1 1 1 1
 ITRIMWL=1
 XTRIM=10. 10. 0.
 NPDIPOLE = ( 673  960 )

 Input file:  test09a.pot
TEST09A.POT -- spar - trimmed waterline
 -1.       HBOT
   1   1                          IRAD,IDIFF
 3                               NPER (array PER follows)
 0.1 0.5 1.0
 2                               NBETA (array BETA follows)
 0.0 45.                         (end of file)
   1                           NBODY
 test09.gdf
 0. 0. 0. 0.                HBOT, XBODY(1-4)
  1  1  1  1  1  1               IMODE(1-6)

 Input file:  test09a.frc
TEST09A.FRC (Spar with three strakes, trimmed waterline)
   1   1   1   1   1   3   0   2   0
 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)
```

A.11 CIRCULAR CYLINDER – TEST11

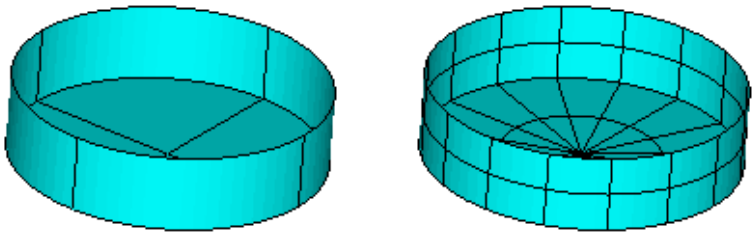
The same cylinder used for the low-order TEST01 is used here with the higher-order option (ILOWHI=1). Two alternatives are used for the geometry.

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.



```
 Input file:  test11.cfg
! TEST11.CFG  Cylinder R=1, T=0.5, igdef=1
 ipltdat=5
 ilowgdf=5
 ILOWHI=1
 IRR=0
 ISOLVE=2
 KSPLIN=3
 IQUADO=3
 IQUADI=4
 MONITR=0
 NOOUT= 1 1 1 1 0 1 1 1 1
 NUMHDR=1

 Input file:  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)

 First 10 lines of input file:  test11.gdf
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
  0.500000000000000        1.00000000000000        1.00000000000000
   1.00000000000000        1.00000000000000

 Input file:  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)
```

```
 Input file:  test11.frc
TEST11.FRC  Cylinder R=1, T=0.5, igdef=1
   1   1   1   1   3   3   0   2   2
 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)
```

```
 Input file:  test11a.cfg
! TEST11a.CFG  Cylinder R=1, T=0.5, igdef=-1
 ipltdat=5
 ILOWHI=1
 IRR=0
 ISOLVE=2
 KSPLIN=3
 IQUADO=3
 IQUADI=4
 MONITR=0
 NUMHDR=1
 NOOUT= 1 1 1 1 0 1 1 1 1

 Input file:  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
 1                            NBETA (array BETA follows)
 0.
 1                            NBODY
 test11a.gdf
 0. 0. 0. 0.                  XBODY
 1  1  1  1  1  1             IMODE(1-6)

 Input file:  test11a.gdf
TEST11a cylinder R=1 T=0.5 -- analytic geometry (npatch=2)
1. 9.80665  ULEN GRAV
1  1        ISX  ISY
2  -1       NPATCH  IGDEF
 2
1.0 0.5     RADIUS, DRAFT
0           UNIFORM MAPPING

 Input file:  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)
```

```
 Input file:  test11a.frc
TEST11a.FRC  Cylinder R=1, T=0.5, igdef=-1
   1   1   1   1   3   3   0   2   2
 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)
```

```
 Input file:  test11b.cfg
! TEST11b.CFG  Cylinder R=1, T=0.5, igdef=1
 ILOWHI=1
 IRR=0
 ISOLVE=2
 KSPLIN=3
 IQUADO=3
 IQUADI=4
 MONITR=0
 NUMHDR=1
 NOOUT= 1 1 1 1 0 1 1 1 1

 Input file:  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)

 Input file:  test11b.gdf
TEST11 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)

 Input file:  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)
```

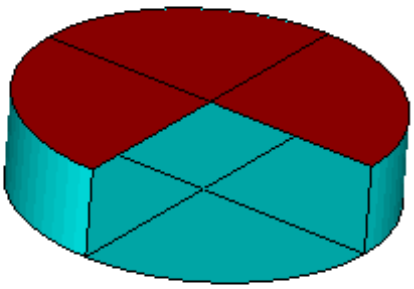
```
 Input file:  test11b.frc
TEST11B.FRC  Cylinder R=1, T=0.5, igdef=-1
   1   1   1   1   3   3   0   2   2
 0.000000                                  VCG
 1.000000      .0000000      .0000000
 .0000000      1.000000      .0000000
 .0000000      .0000000      1.000000      XPRDCT
 0                                         NBETAH
 2                                         NFIELD
1.5 0.  0.
1.5 0. -0.5                                (end of file)
```

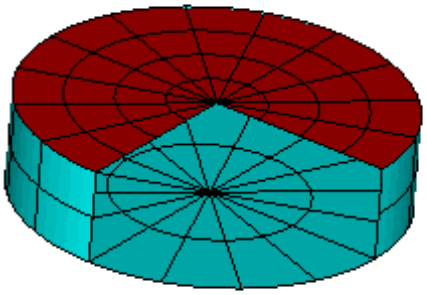
```
 Input file:  test11c.cfg
! TEST11c.CFG  Cylinder R=1, T=0.5, igdef=2
 ILOWHI=1
 IRR=0
 ISOLVE=2
 KSPLIN=3
 IQUADO=3
 IQUADI=4
 MONITR=0
 NUMHDR=1
 NOOUT= 1 1 1 1 0 1 1 1 1

 Input file:  test11c.pot
TEST11C.POT  Cylinder R=1, T=0.5, igdef=2
 -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)

 Input file:  test11c.gdf
TEST11 cylinder R=1 T=0.5 -- MultiSurf .ms2 input, nonuniform mapping
1. 9.80665  ULEN GRAV
1  1        ISX  ISY
0  2        NPATCH  IGDEF
3           NLINES
TEST11C.MS2
wetted_surfs
0  0  0     default settings: FAST, DivMult, outward normal

 Input file:  test11c.spl
TEST11c.spl - cylinder R=1 T=0.5 -- MultiSurf 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)
```

```
 Input file:  test11c.frc
TEST11C.FRC  Cylinder R=1, T=0.5, igdef=2
   1   1   1   1   3   3   0   2   2
 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)
```

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. In the figures below the patch and panels on the interior free surface are shaded red. One quadrant of the side and interior free surface are omitted to show the bottom surface.





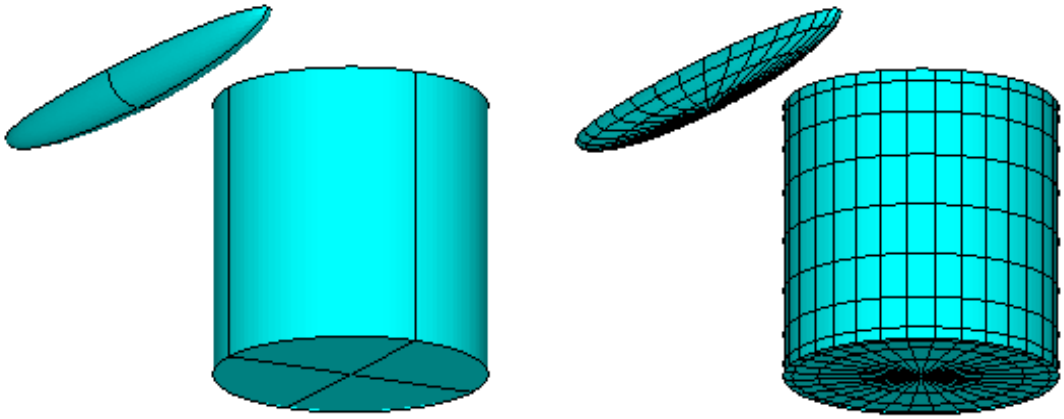
```
 Input file:  test12.cfg
! TEST12.CFG  Cylinder R=1, T=0.5, igdef=-1, npatch=3 (IRR=1)
 ILOWHI=1
 IRR=1
 ILOG=1
 ISOLVE=1
 KSPLIN=3
 IQUADO=3
 IQUADI=4
 MONITR=0
 NUMHDR=1
 NOOUT= 1 1 1 1 0 1 1 1 1

 Input file:  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)

 Input file:  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           UNIFORM MAPPING

 Input file:  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  4        NU NV (Parch 3, interior free surface)  RESET NV=2 FOR STANDARD
```

```
 Input file:  test12.frc
TEST12.FRC  Cylinder R=1, T=0.5, igdef=-1 (irr=1)
   1   1   1   1   3   3   0   2   2
 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)
```

A.13 MULTIPLE BODIES – TEST13

This test uses the same cylinder and spheroid as in the low-order TEST05. The geometry is defined by the input files TEST13C.GDF and TEST13S.GDF. TEST13C uses IGDEF=-1 as in TEST11a. TEST13S.GDF uses the ELLIPSOID subroutine (IGDEF=-4) with the semi-axes (2.0, 0.25, 0.25) specified. 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.)

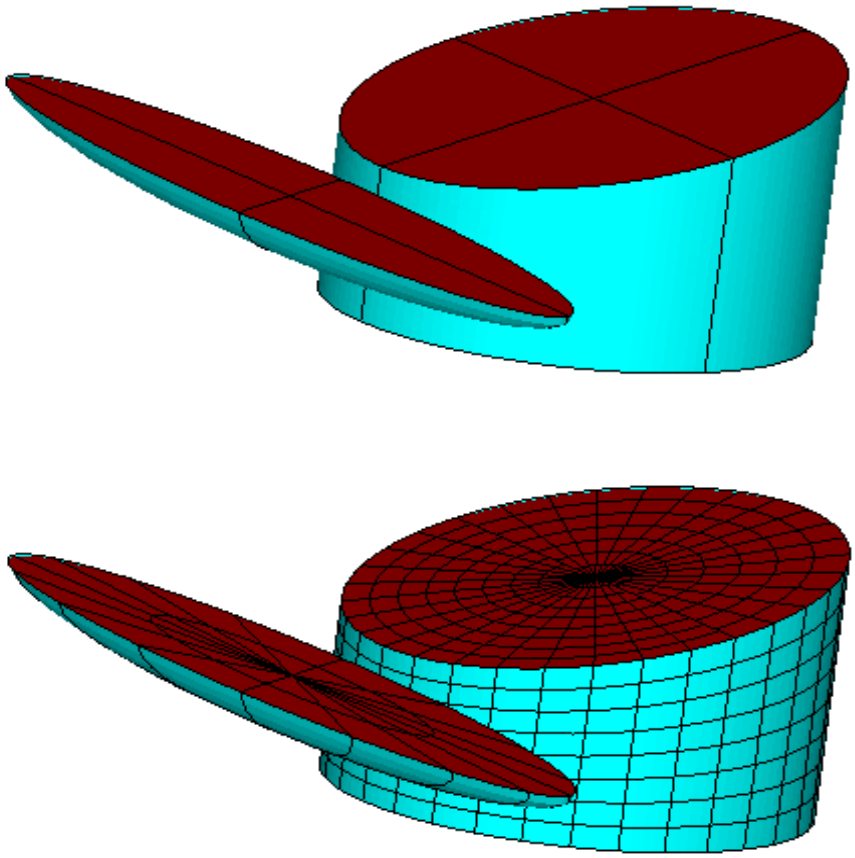
The option is used to evaluate the mean drift force and moment using a control surface, following the instructions in Chapter 11. The control surfaces surrounding the cylinder and spheroid are defined by the input files TEST13c.csf and TEST13s.csf. These control surfaces are generated by the subroutines 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.



```
 Input file:  test13.cfg
! TEST13.CFG -- Cylinder + spheroid, ILOWHI=1
 IPLTDAT=4
 ILOWHI=1
 IRR=0
 ISOLVE=2
 KSPLIN=3
 IQUADO=3
 IQUADI=4
 NUMHDR=1
 NOOUT=0 1 1 1 0 1 1 1 1
 IALTFRC = 3    !  Alternative Form 3 FRC
 IALTFRCN= 1 1

 Input file:  test13.pot
 TEST13.POT -- Cylinder + spheroid, ILOWHI=1
 -1.
 1           1               IRAD, IDIFF
 2                           NPER (array PER follows)
 1.5  2.0      PER
 1                           NBETA (array BETA follows)
 0.                          BETA
 2                           NBODY
 test13c.gdf
 1.25 0.0 0.0 0.0            XBODY
 1  1  1  1  1  1            IMODE(1-6)
 test13s.gdf
 -0.5 0.0 0.0 90.0           XBODY
 1  1  1  1  1  1            IMODE(1-6)

 Input file:  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
1           UNIFORM MAPPING

 Input file:  test13c.spl
TEST13C cylinder R=1 T=2 -- analytic geometry (npatch=2)
 8  8        NU NV (side)
 8  4        NU NV (bottom)
```

```
 Input file:  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

 Input file:  test13s.spl
TEST13S spheroid A=2 B=C=.25 -- analytic geometry (npatch=1)
 8  4        NU NV

 Input file:  test13.frc
 TEST13.FRC -- Cylinder + spheroid, ILOWHI=1
   1   1   1   1   1   3   1   1   1
  1.0
test05c.frc
test05s.frc
  0
  1
  0. 0. 0.

 Input file:  test13c.csf
test13c.csf   cylinder control surface
1   ILOWHICSF
1  1        ISX  ISY
3 -1001  1. NPATCH  ICDEF  PSZCSF
2
1.2 2.2   1.     RADIUS, DRAFT , Inner radius
0             uniform mapping


 Input file:  test13s.csf
ELLIPSOID CONTROL SURFACE defined by subroutine ELLIPSOID\_CS
1   ILOWHICSF
1  1        ISX  ISY
2 -1003  1.0  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)
```

TEST13A illustrates the use of trimmed waterlines, as specified by the last three lines of the file TEST13A.CFG. The cylinder is raised vertically by 1m, and rotated about the pitch axis by 15 degrees. The options IRR=3 and IRR=1 are used to remove irregular frequency effects, as described in Section 9.4. The interior free surface of the spheroid is defined by the GDF file and subroutine ELLIPSOID, with IRR=1 (See Sections 6.8 and 9.1). The interior free surface of the trimmed cylinder is defined automatically by the program, based on the trimmed waterline, with IRR=3 (See Section 9.4). (Since the cylinder is trimmed the waterplane is elliptical, and the IRR=1 extension of CIRCCYL in GEOMXACT is not valid.) Note that NPATCH=2 is assigned in test13as.gdf, to provide for the interior free surface, and test13as.spl includes the spline parameters NU,NV for this extra patch. Conversely for the cylinder the input files test13ac.gdf and test13ac.spl do not include the extra patch since this is added by the program using IRR=3.



```
 Input file:  test13a.cfg
! TEST13A.CFG -- Cylinder + spheroid with trim
 IPLTDAT=4
 ILOWHI=1
 IRR(1)=3
 IRR(2)=1
 ILOG=1
 ISOLVE=1
 KSPLIN=3
 IQUADO=3
 IQUADI=4
 NUMHDR=1
 NOOUT=0 1 1 1 0 1 1 1 1
 IALTFRC = 3    !  Alternative Form 3 FRC
 IALTFRCN= 1 1
 ITRIMWL=1    trim waterline
 XTRIM(1)=  1.0 15. 0.
 XTRIM(2)=  0.0  0. 0.

 Input file:  test13a.pot
 test13a.POT -- Trimmed Cylinder + spheroid, IRR=3
 -1.
 1           1               IRAD, IDIFF
 2                           NPER (array PER follows)
  1.00   2.00
 1                           NBETA (array BETA follows)
 0.
 2                            NBODY
 test13ac.gdf
 1.25 0.0 0.0 0.0             XBODY
 1  1  1  1  1  1             IMODE(1-6)
 test13as.gdf
 -0.5 0.0 0.0 90.0            XBODY
 1  1  1  1  1  1             IMODE(1-6)

 Input file:  test13ac.gdf
test13ac.gdf -- Cylinder, trimmed, no interior fs
1. 9.80665  ULEN GRAV
1  1        ISX  ISY
2  -1       NPATCH  IGDEF
2           NLINES
1.0 2.0     RADIUS, DRAFT
0           UNIFORM MAPPING
```

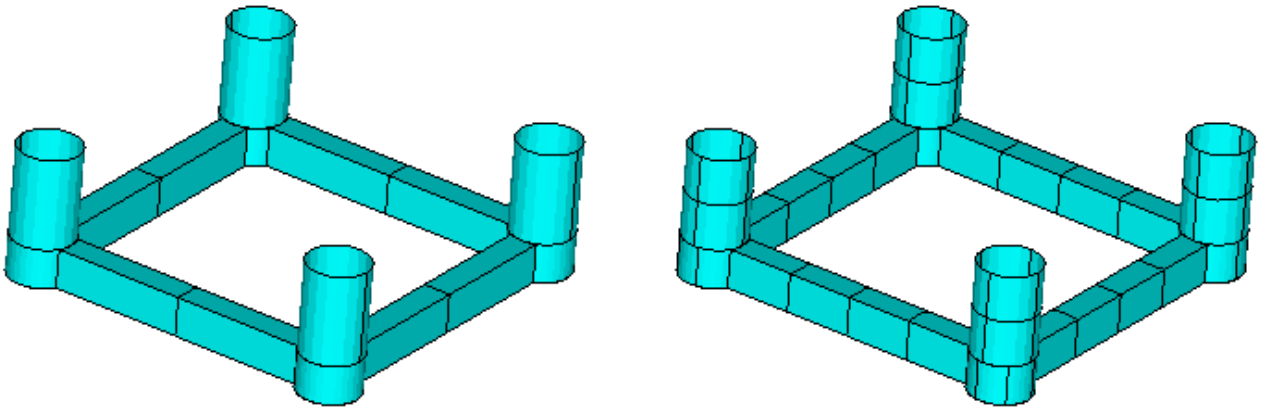
```
 Input file:  test13as.gdf
test13as.gdf -- untrimmed spheroid with interior fs IRR=1
1. 9.80665  ULEN GRAV
1  1        ISX  ISY
2  -4       NPATCH  IGDEF
1           NLINES
2.0 0.25 0.25    A, B, C

 Input file:  test13ac.spl
TEST13C cylinder R=1 T=2 -- analytic geometry (npatch=2)
 8  8        NU NV (side)
 8  4        NU NV (bottom)

 Input file:  test13as.spl
test13as.spl -- untrimmed spheroid with interior fs for IRR=3
 4  2     body patch   NU NV
 4  2     interior fs  NU NV

 Input file:  test13a.frc
 TEST13a.FRC -- Cylinder + spheroid, trimmed waterlines
   1   1   1   1   0   3   0   1   1
  1.0
test05c.frc
test05s.frc
  0
  1
  0. 1. 0.
```

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. As explained in Appendix A.6 a warning message is displayed for Options 8 and 9 since IDIFF=0.

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 99 uniformly spaced additional frequencies. IPERIN=2 is specified in the cfg file. In FRC, IOPTN(4) is set to output the RAOs in all six modes.

Since there are a large number of wave frequencies in TEST14a, it is a good example of the benefit of multiple processors (See Section 14.6). The runtimes in this case using NCPU=1,2,4,8 are almost inversely proportional to NCPU.



```
 Input file:  test14.cfg
! TEST14.CFG -- ISSC TLP (ILOWHI=1)
 ipltdat=5
 ILOWHI=1
 IRR=0
 ISOLVE=1
 IQUADI=4
 IQUADO=3
 KSPLIN=3
 NUMHDR=1
 IALTFRC=2

 Input file:  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)

 Input file:  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
```

```
 Input file:  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
```

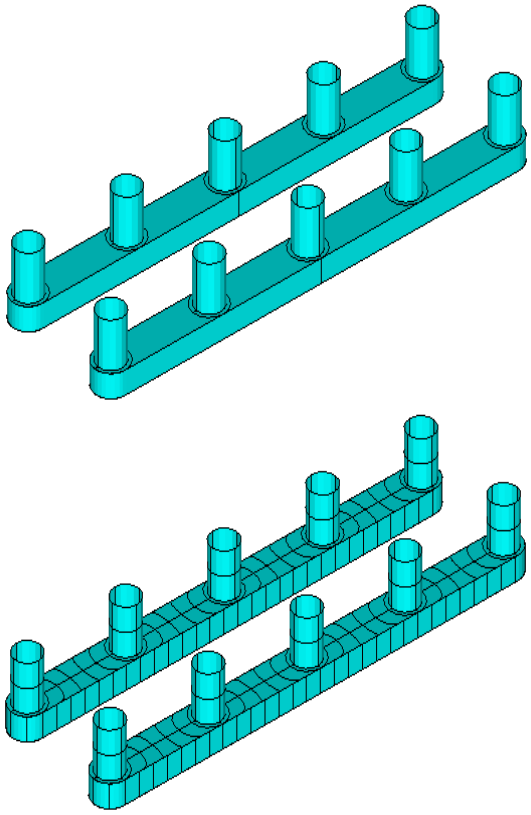
```
 Input file:  test14.frc
TEST14 -- ISSC TLP (ILOWHI=1, IALTFRC=2)
 1  1  1 -2  0  0  0  1  1  IOPTN (IOPTN(4)<0 signifies fixed modes)
 6                          NDFR
 1 1 0 0 0 1                IMODE
 1.                         RHO
 0. 0. 3.0                  XCG
 1                          IMASS
    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
```

```
 Input file:  test14a.cfg
 ILOWHI=1
 IRR=0
 ISOLVE=1
 IQUADI=4
 IQUADO=3
 KSPLIN=3
 NUMHDR=1
 IALTFRC=2
 IPERIN=2
 IPEROUT=2

 Input file:  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)

 Input file:  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
```

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.

Option 7 is used to evaluate the drift force and moment from the method described in Chapter 11. The control surface is defined by the program, using the automatic method described in Section 11.4, with the input parameters specified in the CSF file. The outer control surface is a rectangular box, and three inner partitions are defined to separate the columns, as discussed in Example 4 of Section 11.4.



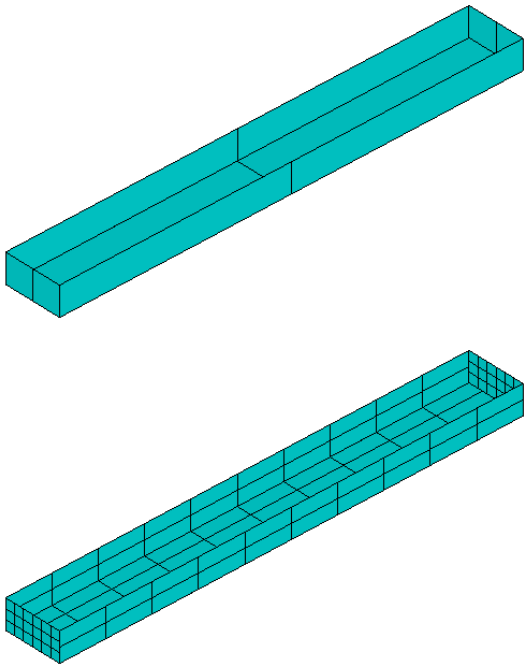
```
 Input file:  test15.cfg
! TEST15.CFG -- Semi-sub with five columns on each pontoon
 ipltdat=1
 ISOLVE=1
 NUMHDR=1
 KSPLIN=3
 IQUADI=4
 IQUADO=3
 ILOWHI=1

 Input file:  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)

 Input file:  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

 Input file:  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)

 Input file:  test15.frc
TEST15  Semi-sub with five columns on each pontoon
   1   1   1   1   0   0   1   1   1
       0.0000
       20.0      0.0        0.0
       0.        60.0        0.0
       0         0.         60.0
       0
       0

 Input file:  test15.csf
 test15.csf  semi sub, outer box 150*60*40
1   ILOWHICSF
1  1        ISX  ISY
0   0  10. NPATCH  ICDEF  PSZCSF   (1st two indicate this is automatic)
 0.0  40.0    RADIUS, DRAFT of outer box
  4   NPART
  3   nv0
  150.0   0.0
  150.0  60.0
    0.0  60.0   end of partition line 0 (outer boundary of control surface)
  3   nv1
   0.0  0.0
  30.0  0.0
  30.0  50.0
  4   nv2
  30.0  50.0
  30.0  0.0
  90.0  0.0
  90.0  50.0
  3   nv3
  90.0  50.0
  90.0  0.0
  150.0  0.0   end of partition line column 3
```

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.



```
 Input file:  test16.cfg
! TEST16.CFG elastic barge with 8 beam modes
 ipltdat=5
 ilowgdf=5
 ILOWHI=1
 IALTFRC=2
 ISOLVE=1
 IQUADI=5
 IQUADO=4
 MONITR=0
 NUMHDR=1
 IGENMDS=16
 NEWMDS=8

 Input file:  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)

 Input file:  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

 Input file:  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

 Input file:  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. 1.E+06 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. 1.E+06 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. 1.E+06 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. 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. 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
```

```
 Input file:  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)

 First 10 lines of input file:  test16a.gdf
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)
    40.0000        5.00000       -5.00000
   0.000000        5.00000       -5.00000

 Input file:  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
```

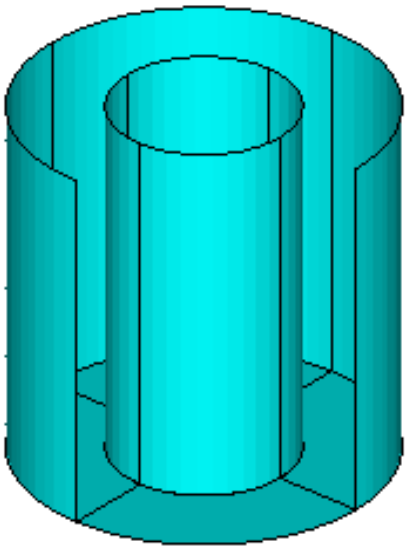
```
 Input file:  test16a.frc
TEST16a elastic barge with 8 beam modes (igdef=0)
 1  1  1  1  0  0  0  0  0
 1000.
 0. 0. 0.
 1
  4.00000E+06 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0.
  0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0.
  0. 0. 4.E+06 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0.
  0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0.
  0. 0. 0. 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
```

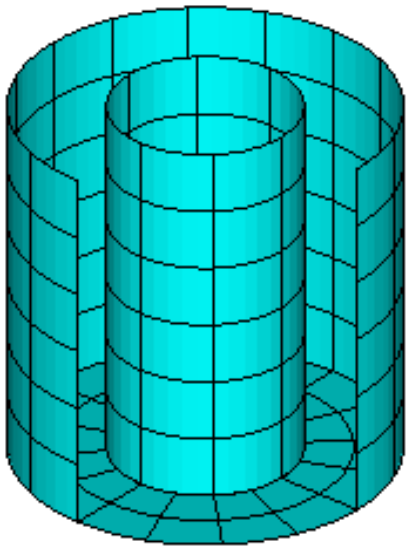
A.17 CYLINDER WITH MOONPOOL – TEST17

This test run illustrates three 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 Kd, the product of the wavenumber K and draft d 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 0.5m and the inner radius 0.25m. The geometry is represented analytically by the subroutine CYLMP (IGDEF=-7). In TEST17 three patches are used to represent the outer side r=RADIUS, the annular bottom z =-DRAFT, and the inner side r=RADMP. The free surface inside the moonpool is part of the physical free surface, and the appropriate free-surface boundary condition is satisfied by the Green function as described in Chapter 15. The side of one quadrant is omitted in the figures below, to show more clearly the bottom and the side of the moonpool.

To clarify the behavior near resonance, the wavenumber K is input in the POT file with the corresponding option IPERIN=3 specified in the TEST17.cfg file. 61 values of K are input in the range $0.1 \leq K \leq 1.5$ to describe the behavior near resonance. The computed hydrodynamic parameters include the force coefficients, RAO's, and the elevation of the free surface at the center of the moonpool.





When the structure is fixed in heave the resonant pumping mode in the moonpool occurs at $Kd = 0.85$. The outputs related to the vertical force component display singular features near this point. This includes large amplitudes of the heave damping and exciting force, and negative added mass. The heave RAO has two adjacent resonant peaks as shown in the Figure below, due to coupling between the heave mode and the moonpool pumping mode.

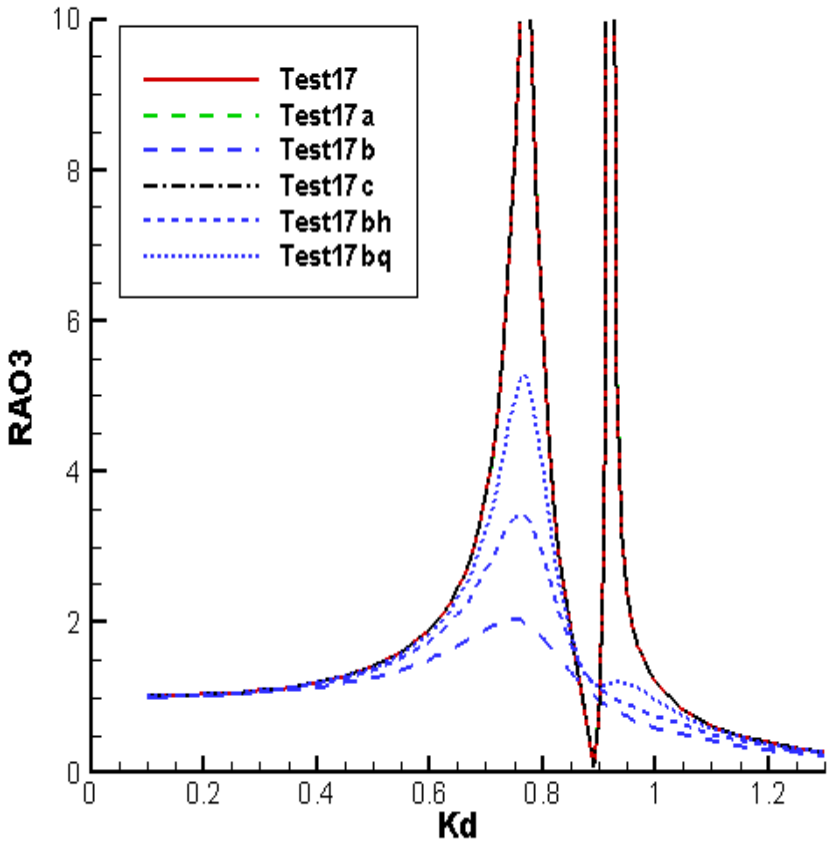


Figure A.1: Heave RAO for each of the test17 runs. The results for Test17, Test17a, and Test17c are practically identical. The results for Test17b (lid with damping) include three values of the external damping coefficients. In Test17b $b_{33} = 0.4$ and $b_{77} = 0.1$, as shown in the `test17b.frc` file below. In Test17bh these values are reduced by a factor of one-half and in Test17bq they are reduced by a factor of one-quarter.

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. Typical resonant amplitudes are likely to be in the range represented by the dashed lines in Figure A.1.

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 Test17c). The geomxact 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 $j = 7$. In Test17b a pitch rotation of the lid ($j = 8$) is also included to provide a more general deflection of the free surface. A more complete expansion can be introduced, but at the wavenumbers considered here and for head-sea incidence angle these two modes of motion are sufficient. 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, as shown in the Figure above, 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.

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 as shown in the Figure. 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.

In TEST17c the same geometry is used with the free-surface pressure option (FSP) described in Section 10.11. The FSP surface is the same as the lid described above, and the same gdf file is used as in TEST17a and 17b. In the CFG file the parameter IMODESFSP=1 is used to select the subroutine PRESSURE_FS in the NEWMODES.DLL file, and NMODESFSP=1. The pressure distribution on the FSP surface is constant. To correspond with the original Test17, the pressure represented by mode 7 is set equal to zero by assigning mode 7 to be fixed in the file `test17c.frc`.

```
 Input file:  test17.cfg
! TEST17.CFG file, cylinder with moonpool
 ipltdat=5
 ilowgdf=5
 ILOWHI=1
 IALTFRC=1
 ISOLVE=1
 PANEL_SIZE = 0.2       (use default .spl parameters)
 IPERIN=3               (input wavenumber)
 IPEROUT=3              (output wavenumber)
 NUMHDR=1

 Input file:  test17.pot
TEST17 cylinder with moonpool, NPATCH=3
 -1.
 0           0               IRAD, IDIFF
          61
    0.100000    0.150000    0.200000    0.250000    0.300000
    0.350000    0.400000    0.450000    0.500000    0.550000
    0.600000    0.610000    0.620000    0.630000    0.640000
    0.650000    0.660000    0.670000    0.680000    0.690000
    0.700000    0.710000    0.720000    0.730000    0.740000
    0.750000    0.760000    0.770000    0.780000    0.790000
    0.800000    0.810000    0.820000    0.830000    0.840000
    0.850000    0.860000    0.870000    0.880000    0.890000
    0.900000    0.910000    0.920000    0.930000    0.940000
    0.950000    0.960000    0.970000    0.980000    0.990000
    1.000000    1.050000    1.100000    1.150000    1.200000
    1.250000    1.300000    1.350000    1.400000    1.450000
    1.500000
 1                            NBETA (array BETA follows)
 180.
 1                           NBODY
test17.gdf
 0. 0. 0. 0.                 XBODY
 1  0  1  0  1  0            IMODE(1-6)
```

```
 Input file:  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

 Input file:  test17.frc
TEST17.FRC  Cylinder with moonpool
 1    1    1    1    0    1  0  0 0     IOPTN(1-9)
 0.000000                                   VCG
 0.500000      .0000000      .0000000
 .0000000      0.500000      .0000000
 .0000000      .0000000      0.500000       XPRDCT
 0                                          NBETAH
 1                                          NFIELD
0.0 0.0 0.0
```

```
 Input file:  test17a.cfg
! TEST17A.CFG file, cylinder with moonpool, free lid
 ipltdat=5
 ilowgdf=5
 ILOWHI=1
 IALTFRC=2
 ISOLVE=1
 PANEL_SIZE = 0.2       (use default .spl parameters)
 IPERIN=3               (input wavenumber)
 IPEROUT=3              (output wavenumber)
 ILOG=1
 NUMHDR=1
 IGENMDS=17
 NEWMDS = 2

 Input file:  test17a.pot
TEST17 cylinder with moonpool, NPATCH=3
 -1.
 0           0               IRAD, IDIFF
          61
    0.100000    0.150000    0.200000    0.250000    0.300000
    0.350000    0.400000    0.450000    0.500000    0.550000
    0.600000    0.610000    0.620000    0.630000    0.640000
    0.650000    0.660000    0.670000    0.680000    0.690000
    0.700000    0.710000    0.720000    0.730000    0.740000
    0.750000    0.760000    0.770000    0.780000    0.790000
    0.800000    0.810000    0.820000    0.830000    0.840000
    0.850000    0.860000    0.870000    0.880000    0.890000
    0.900000    0.910000    0.920000    0.930000    0.940000
    0.950000    0.960000    0.970000    0.980000    0.990000
    1.000000    1.050000    1.100000    1.150000    1.200000
    1.250000    1.300000    1.350000    1.400000    1.450000
    1.500000
 1                            NBETA (array BETA follows)
 180.
 1                           NBODY
test17a.gdf
 0. 0. 0. 0.                 XBODY
 1  0  1  0  1  0            IMODE(1-6)
```

```
 Input file:  test17a.gdf
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

 Input file:  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)
    0.589    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      0.147    0.0      0.0      0.0      0.0
    0.0      0.0      0.0      0.0      0.147    0.0      0.0      0.0
    0.0      0.0      0.0      0.0      0.0      0.147    0.0      0.0
    0.0      0.0      0.0      0.0      0.0      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
```

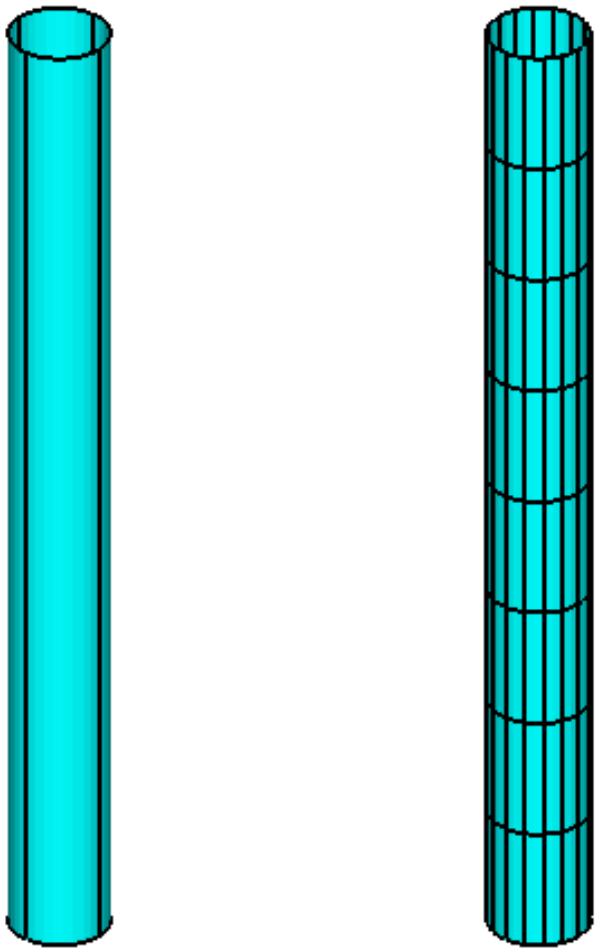
```
 Input file:  test17b.cfg
! TEST17B.CFG file, cylinder with moonpool, damped lid
 ILOWHI=1
 IPOTEN=0
 IALTFRC=2
 ISOLVE=1
 PANEL_SIZE = 0.2       (use default .spl parameters)
 IPERIN=3               (input wavenumber)
 IPEROUT=3              (output wavenumber)
 ILOG=1
 NUMHDR=1
 IGENMDS=17
 NEWMDS = 2

 Input file:  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)
    0.589    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      0.147    0.0      0.0      0.0      0.0
    0.0      0.0      0.0      0.0      0.147    0.0      0.0      0.0
    0.0      0.0      0.0      0.0      0.0      0.147    0.0      0.0
    0.0      0.0      0.0      0.0      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
```

```
 Input file:  test17c.cfg
 ILOWHI=1
 IALTFRC=1
 ISOLVE=1
 PANEL_SIZE = 0.2       (use default .spl parameters)
 IPERIN=3               (input wavenumber)
 IPEROUT=3              (output wavenumber)
 IRR=0
 ILOG=1
 NUMHDR=1
 IMODESFSP=1
 NMODESFSP=1
 NPFSP=(4 4)            (free surface pressure on patch 4)
 IMODESFSP=1            (use NEWMODES subroutine PRESSURE_FS)
 NMODESFSP=1            (1 pressure mode, same symmetry as heave)

 Input file:  test17c.pot
TEST17c  cylinder with free-surface pressure in moonpool
 -1.
 0       1               IRAD, IDIFF
          61
    0.100000    0.150000    0.200000    0.250000    0.300000
    0.350000    0.400000    0.450000    0.500000    0.550000
    0.600000    0.610000    0.620000    0.630000    0.640000
    0.650000    0.660000    0.670000    0.680000    0.690000
    0.700000    0.710000    0.720000    0.730000    0.740000
    0.750000    0.760000    0.770000    0.780000    0.790000
    0.800000    0.810000    0.820000    0.830000    0.840000
    0.850000    0.860000    0.870000    0.880000    0.890000
    0.900000    0.910000    0.920000    0.930000    0.940000
    0.950000    0.960000    0.970000    0.980000    0.990000
    1.000000    1.050000    1.100000    1.150000    1.200000
    1.250000    1.300000    1.350000    1.400000    1.450000
    1.500000
 1                            NBETA (array BETA follows)
 180.
 1                           NBODY
test17a.gdf
 0. 0. 0. 0.                 XBODY
 1  0  1  0  1  0            IMODE(1-6)
```

```
 Input file:  test17c.frc
TEST17c.FRC cylinder with surface pressure in moonpool, fixed mode 7
 1    1    1    -1    0    0  0  0 0     IOPTN(1-9)
 7
 1    1    1   1    1    1   0
 0.000000                                  VCG
 0.500000      .0000000      .0000000
 .0000000      0.500000      .0000000
 .0000000      .0000000      0.500000      XPRDCT
 0                                         NBETAH
 0                                         NFIELD
```

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.



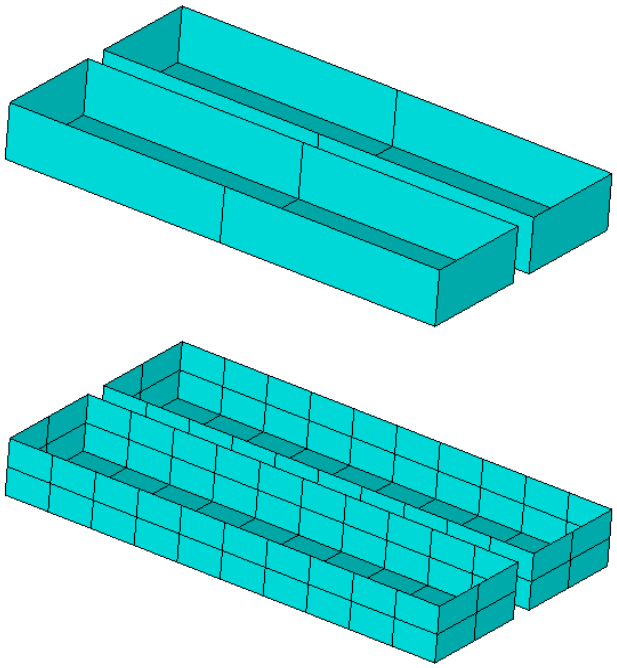
```
 Input file:  test18.cfg
! TEST18.CFG -- bending of vertical column with 4 generalized modes
 ipltdat=5
 ilowgdf=5
 ISOLVE=1
 NUMHDR=1
 NUMNAM=0
 NEWMDS=4
 ILOWHI=1
 IALTFRC=2
 KSPLIN=3
 IQUADO=3
 IQUADI=4
 IGENMDS=18

 Input file:  test18.pot
TEST18.POT -- bending of vertical column at resonance, 200m depth
      200.0       HBOT
   0   0                          IRAD,IDIFF
           1
           6.5
           1
           0.0
   1                          NBODY
 test18.gdf
  0.0  0.0  0.0  0.0
        0   0   0   0   0   0

 Input file:  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                   uniform mapping

 Input file:  test18.spl
TEST18.spl - bottom-mounted cylinder R=10 T=200 -- (npatch=1)
4  8        NU NV (Patch 1, side   u azimuthal v vertical)
```

```
 Input file:  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
```

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.



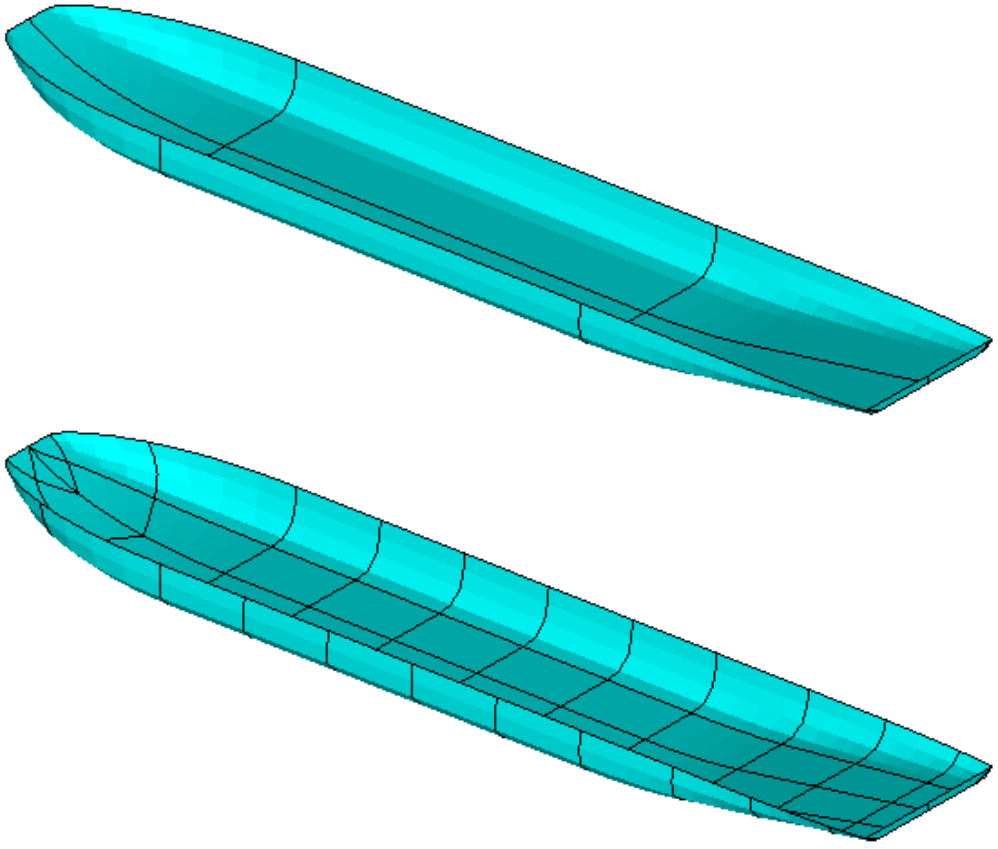
```
 Input file:  test19.cfg
! TEST19.CFG -- Catamaran barge
 ipltdat=5
 ILOWHI=1
 IALTFRC=1
 ISOLVE=1
 IQUADI=5
 IQUADO=4
 MONITR=0
 NUMHDR=1

 Input file:  test19.pot
TEST19.POT -- Catamaran barge, same geometry as TEST04
 -1.       HBOT
   0   0                          IRAD,IDIFF
 3
 6.  7.  8.
 1
 0.0
   1                           NBODY
 test19.gdf
 0. 0. 0. 0.
  1  0  1  0  1  0

 First 10 lines of input file:  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
```

```
 Input file:  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

 Input file:  test19.frc
TEST19.FRC -- Catamaran barge, ILOWHI=1 (same as TEST04.FRC)
 1    1    1    1    1    0    0    1    1
 3.0
 20.00000      0.000000      0.000000
 0.000000      5.000000      0.000000
 0.000000      0.000000      20.00000
 0
 0
```

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.

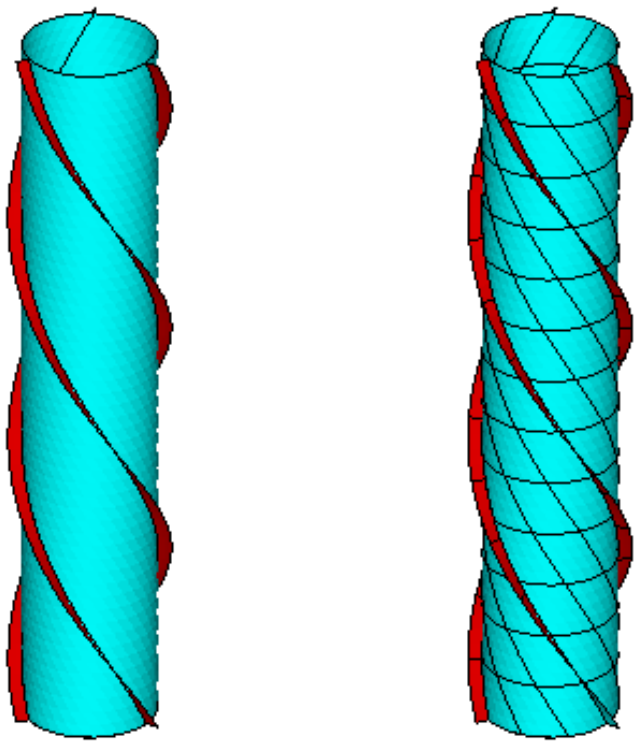


```
 Input file:  test20.cfg
! TEST20.CFG - MultiSurf barge
 ipltdat=4
 ILOWHI=1
 IALTFRC=1
 ISOLVE=1
 NUMHDR=1
 panel_size= 10.
 IPNLBPT=1

 Input file:  test20.pot
single barge based on MultiSurf model (igdef=2)
 -1.
  1     1                      IRAD, IDIFF
   3                           NPER (array PER follows)
 6. 9. 12.
 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)

 Input file:  test20.gdf
Test run for barge modelled with MultiSurf
 1.000000  9.80665  ULEN, GRAV
  0  1  ISX, ISY
  0   2  NPATCH, IGDEF
  3  NLINES
test20.ms2
wetted_surfs
0  0  0 FAST,DivMult, outward normals

 Input file:  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
```

```
First 10 lines of input file:  test20.bpi
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
  -40.8942     0.0000     2.1333
  -38.6878     0.0000     1.4815
```

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.



```
 Input file:  test21.cfg
! TEST21.CFG -- SPAR with three strakes
 ipltdat=4
 ilowgdf=4
 ILOWHI=1
 ISOLVE=1
 KSPLIN=3
 IQUADO=3
 IQUADI=4
 IPERIN=3               (input wavenumber)
 IPEROUT=3              (output wavenumber)
 MONITR=0
 NUMHDR=1
 NOOUT= 1 1 1 1 0 1 1 1 1
 PANEL_SIZE=18
 ILOG=1
 NPDIPOLE = 2 4 6

 Input file:  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)

 Input file:  test21.gdf
TEST21 SPAR2 with three strakes IGDEF=-12
18. 9.80665  ULEN GRAV
0  0        ISX  ISY
7  -12       NPATCH  IGDEF
5
18. 200.              RADIUS, DRAFT
3.7 0. 1. 3           WIDTH, THICKNESS, TWIST, NSTRAKE
0                     IRRFRQ
0  0.                 IMOONPOOL, RADIUSMP
0                     IMPGEN
```

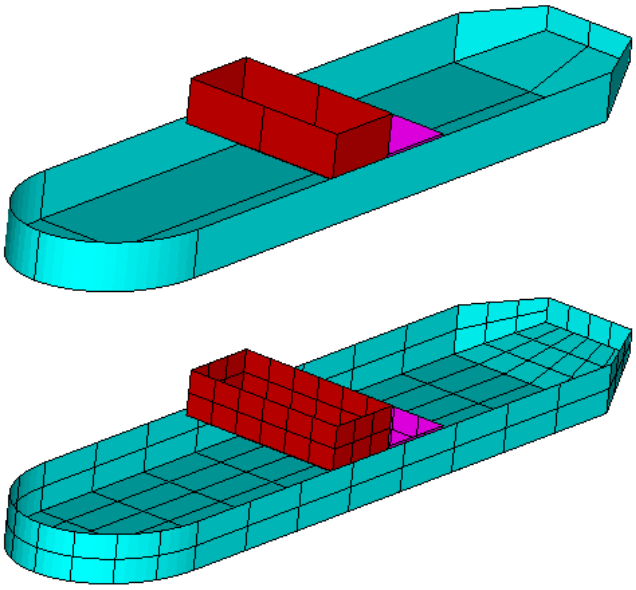
```
 Input file:  test21.frc
TEST21.FRC  SPAR with three strakes igdef=-12
   1   1   1   1   0   3   0   2   0
 0.000000                                    VCG
 100.000000      .0000000      .0000000
 .0000000      100.000000      .0000000
 .0000000      .0000000      10.000000        XPRDCT
 0                                           NBETAH
 2                                           NFIELD
23. 0.  0.
15. 15. -0.5                                 (end of file)
```

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.

Option 7 is used to evaluate the mean drift force and moment using a control surface. The control surface surrounding the FPSO is automatic, defined by the input file `test22.csf`. The parameter PSZCSF is negative, indicating that the subdivision of the control surface is determined by the parameters in the file TEST22.CSP. A second CSF file is contained in the file `test22.csf` for illustration, but it is not read by the program unless it is moved to the top of the file. In the second case the outer boundary is circular, and PSZCSF is positive, indicating that automatic subdivision of the control surface is performed as described in Section 11.4.

The corresponding output for the mean drift force and moment is contained in the file TEST13.7. Comparison of the results for the mean drift force in the sway direction from the files TEST22.9 (direct pressure integration) and TEST22.7 (control surface), with the far-field 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.

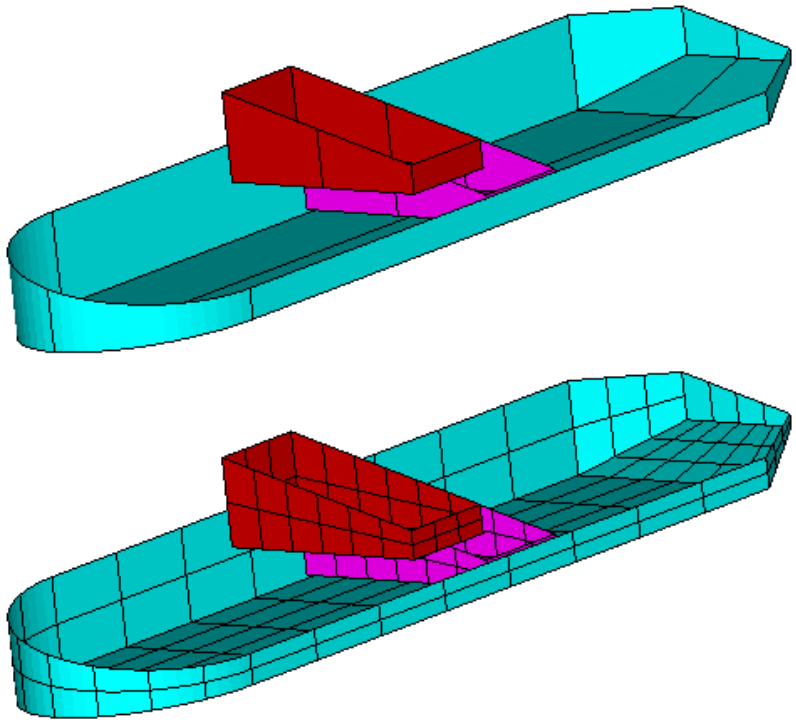


```
 Input file:  test22.cfg
! TEST22.CFG  fpso with 2 interior tanks
 ipltdat=4
 ILOWHI=1
 ILOG=1
 ISOLVE=1
 KSPLIN=3
 IQUADO=3
 IQUADI=4
 MONITR=0
 NUMHDR=1
 NOOUT= 1 1 1 1 0 1 1 1 1
 NPTANK=(8-11)  (12-15)
 RHOTANK= 1.0  1.0       (relative densities of tank fluids)
 ITANKFPT=1              (tank field points are in .frc file)

 Input file:  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)

 Input file:  test22.gdf
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        INONUMAP, NTANKS
   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

Input file:  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

 Input file:  test22.frc
TEST22.FRC fpso with 2 tanks, one field point on free surface in each tank
   1   1   1   1   0   3   1   1   1
 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

 Input file:  test22.csf
 test22.csf  FPSO, rectangular outer boundary
1   ILOWHICSF
0  1        ISX  ISY
0   0  -2. NPATCH  ICDEF  PSZCSF   (1st two indicate this is automatic)
  0.0  2.0   RADIUS, DRAFT of outer box (0.0 signifies outer bdry defined below
  1   NPART
  4   nv0
  12.0   0.0
  12.0   3.0
 -12.0   3.0
 -12.0   0.0

THE FOLLOWING IS AN ALTERNATIVE CSF FILE WHICH IS NOT READ BY THE PROGRAM
 UNLESS IT IS INTERCHANGED WITH THE FILE ABOVE.

 test22.csf  FPSO, circular outer boundary
1   ILOWHICSF
0  1        ISX  ISY
0   0  2. NPATCH  ICDEF  PSZCSF   (1st two indicate this is automatic)
  12.0  2.0    RADIUS, DRAFT of outer box
  0   NPART
```

In test22a the waterline is trimmed with a roll angle of 15 degrees. In order to preserve the same mean depth of the vessel, the draft is increased by 1m in the GDF file, and a vertical trim XTRIM(1)=1.0 is specified in the CFG file. In this manner one ensures that the entire submerged portion of the hull surface is correctly defined. Since the draft is increased in the GFD file it is necessary to lower the tank bottoms by the same amount, thus the lower edges of the tank patches in test22a.gdf are 1m lower than in test22.gdf.



```
 Input file:  test22a.cfg
! TEST22a.CFG  fpso with 2 interior tanks, trimmed waterline
 ipltdat=4
 ILOWHI=1
 ILOG=1
 ISOLVE=1
 KSPLIN=3
 IQUADO=3
 IQUADI=4
 MONITR=0
 NUMHDR=1
 NOOUT= 1 1 1 1 0 1 1 1 1
 NPTANK=(8-11)  (12-15)
 RHOTANK= 1.0  1.0       (relative densities of tank fluids)
 ITANKFPT=1              (tank field points are in .frc file)
 ztankFS= 1.0   0.0
 ITRIMWL=1
 XTRIM=1.0  0. 15.0

 Input file:  test22a.pot
TEST22a.POT  fpso with 2 interior tanks, trimmed waterline
 -1.0
   1         1               IRAD, IDIFF
     3
 2.0 2.5 3.0
 1                            NBETA (array BETA follows)
 90.
 1                            NBODY
 test22a.gdf
 0. 0. 0. 0.                  XBODY
 1  1  1  1  1  1             IMODE(1-6)

 Input file:  test22a.gdf
TEST22.GDF -- fpso with 2 tanks, trimmed waterline
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
 2.2  1.6   DRAFT, DTRANSOM
 0 2        INONUMAP, NTANKS
   2.000000      0.0000000E+00  1.0000000E+00
```

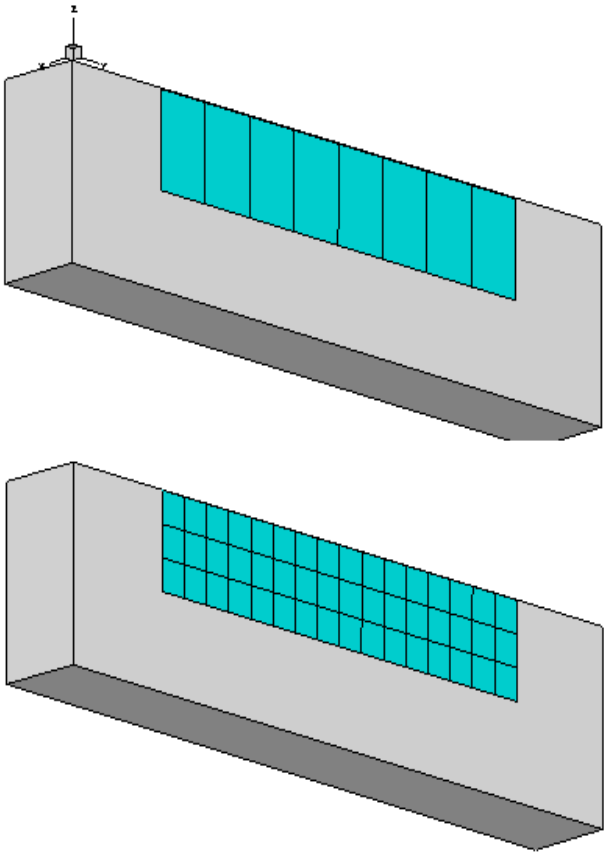
```
 2.000000       2.100000      1.0000000E+00
 2.000000       2.100000      -1.100000
 2.000000      0.0000000E+00  -1.100000
 2.000000       2.100000      1.0000000E+00
 0.000000E+00   2.100000      1.0000000E+00
 0.000000E+00   2.100000      -1.100000
 2.000000       2.100000      -1.100000
 0.000000E+00  0.0000000E+00  -1.100000
 2.000000      0.0000000E+00  -1.100000
 2.000000       2.100000      -1.100000
-0.000000E+00   2.100000      -1.100000
-0.000000      0.0000000E+00  -1.100000
-0.000000       2.100000      -1.100000
-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      -2.100000
 0.000000      0.0000000E+00  -2.100000
 0.000000       2.100000      0.0000000E+00
-2.000000E+00   2.100000      0.0000000E+00
-2.000000E+00   2.100000      -2.100000
 0.000000       2.100000      -2.100000
-2.000000E+00  0.0000000E+00  -2.100000
 0.000000      0.0000000E+00  -2.100000
 0.000000       2.100000      -2.100000
-2.000000E+00   2.100000      -2.100000
-2.000000      0.0000000E+00  -2.100000
-2.000000       2.100000      -2.100000
-2.000000       2.100000      0.0000000E+00
-2.000000      0.0000000E+00  0.0000000E+00

Input file:  test22a.spl
TEST22a.SPL  FPSO with two tanks, trimmed waterline
     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

 Input file:  test22a.frc
TEST22A.FRC fpso with 2 tanks, one field point on free surface in each tank
   1   1   1   1   0   3   0   1   1
 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
```

A.23 RADIATED WAVE FIELD FROM A BANK OF WAVEMAKERS – TEST23

Several variants are considered to illustrate the analysis of wavemakers in a wave tank.

In TEST23, following the procedure in Section 12.3, the option ISOLVE=-1 is used to compute the radiated waves from a bank of 'paddle' wavemakers. 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.



```
 Input file:  test23.cfg
! TEST23.CFG -- 8 wavemaker segments in wall x=0
 ILOWHI=1
 IALTFRC=2
 ISOLVE=-1              (skip POTEN solutions for wavemakers in walls)
 MONITR=0
 NUMHDR=1
 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)
 ipltdat=4
 NEWMDS = 8

 Input 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)

 First 10 lines of input file:  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
```

```
 Input file:  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.

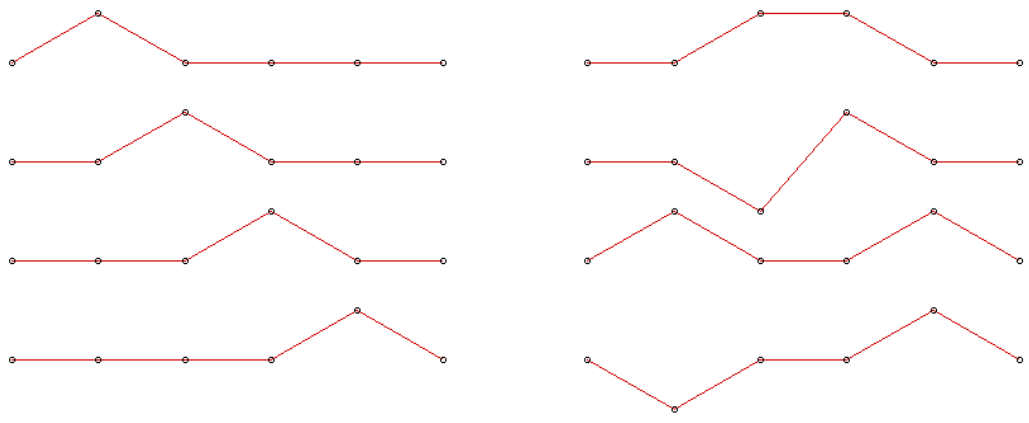
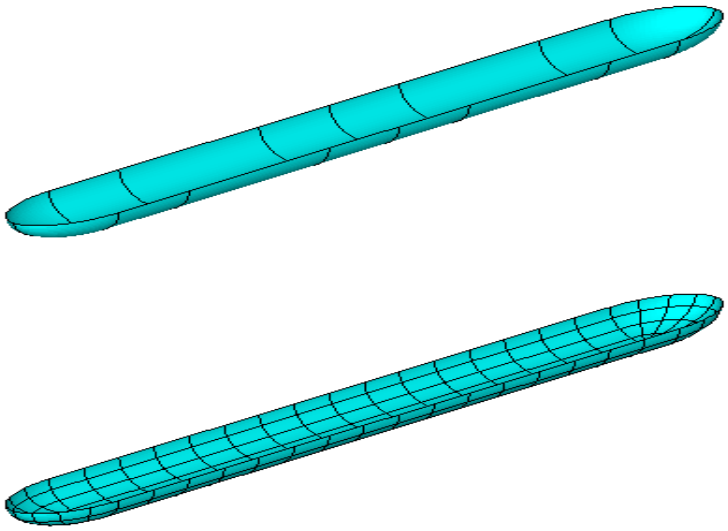


Figure A.2: Generalized modes used for the hinged barge with four hinges. The modes on the left are 'tent functions', suitable for use when ISX=0. When ISX=1 each mode must be either symmetric or antisymmetric, as shown in the right column. The latter modes are used for TEST24. The conventional rigid-body modes in heave and pitch represent the nonzero vertical motions at the two ends.

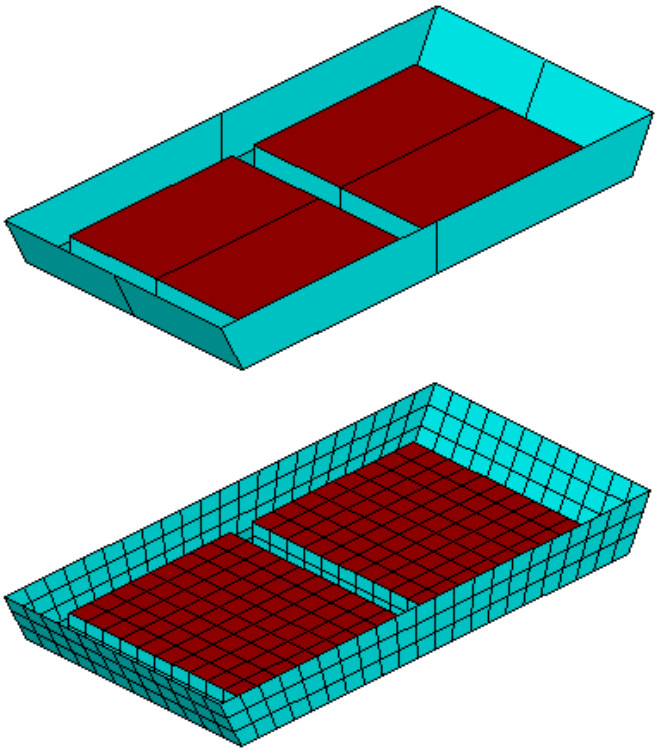


```
 Input file:  test24.cfg
! TEST24.CFG -- segmented vessel with 7 segments and 4 hinge modes
 ipltdat=5
 ILOWHI=1
 IALTFRC=2
 ISOLVE=1
 IQUADI=4
 IQUADO=3
 KSPLIN=3
 MONITR=0
 NUMHDR=1
 IGENMDS=22
 NEWMDS = 4

 Input file:  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)
```

```
 Input file:  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

 Input file:  test24.spl
TEST24 segmented vessel with 7 segments
 4 2      NU NV   mid cylinder
 4 4              next cylinder
 4 2              outer cylinder
 4 4              spheroidal end

 Input file:  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
```

A.25 AIR-CUSHION VESSEL WITH PRESSURE CHAMBERS – TEST25

The vessel consists of two air chambers surrounded by rectangular skirts. The length is 20m, beam 10m and draft 2m. The width of the skirts is 1m. The free surface in the chambers, shown by red shading in the figures below, is 1m below the exterior free surface. One quadrant of these surfaces is represented by flat quadrilateral patches in the GDF file, with two planes of symmetry. Two pressure modes are used to represent anti-symmetric and symmetric pressure distributions with constant pressure in each chamber, denoted by Modes 7 and 8 respectively. These pressure modes are defined by the NEWMODES subroutine PRESSURE_FS.



```
 Input file:  test25.cfg
! TEST25.CFG -- ACV (air-cushion vehicle) with 2 air chambers
 ipltdat=1
 ILOWHI=1
 IALTFRC=1
 ISOLVE=1
 PANEL_SIZE = 1.0       (use default .spl parameters)
 NUMHDR=1
 IMODESFSP=1            (use NEWMODES subroutine PRESSURE_FS)
 NMODESFSP=2            (2 pressure modes, sym & antisym about x=0)
 NPFSP=(9 9)            (free surface pressure on patch 9)

 Input file:  test25.pot
TEST25 ACV (air-cushion vehicle) with 2 air chambers
 -1.
 0       1               IRAD, IDIFF
 6                            NPER (array PER follows)
 5.0 6.0 7.0 8.0 9.0 10.0
 1                             NBETA (array BETA follows)
 180.
 1                            NBODY
test25.gdf
 0. 0. 0. 0.                  XBODY
 1  0  1  0  1  0             IMODE(1-6)

 First 10 lines of input file:  test25.gdf
TEST25 ACV (air-cushion vehicle) with 2 air chambers - igdef=0
     1.00        9.80665
           1           1
           9           0          NPATCH, IGDEF
     9.00        0.00       -2.00
     9.00        5.00       -2.00
    10.00        5.00        0.00
    10.00        0.00        0.00   (end)
     9.00        5.00       -2.00
     0.00        5.00       -2.00
```

```
 Input file:  test25.frc
TEST25.FRC  ACV (air-cushion vehicle) with 2 air chambers
 1    1    1    1    0    0  0  0 0     IOPTN(1-9)
 0.000000                                    VCG
 2.500000      .0000000      .0000000
 .0000000      5.000000      .0000000
 .0000000      .0000000      5.000000        XPRDCT
 0                                         NBETAH
 0                                         NFIELD
```

Appendix B

FILE CONVERSION USING THE UTILITY V6V7inp

The utility executable program `V6V7inp` is supplied with WAMIT Version 7, to facilitate the conversion of old input files from Version 6 to Version 7. These changes are summarized in Section 4.1.

This program is intended for use on Windows PC's. Reference will be made here to 'directories' (folders) and 'sub-directories' which are adjacent to the directories. The program is intended to process one or more sets of input files, identified by corresponding FNAMES.WAM files as described in Section 4.8. These are referred to here as the 'fnames' files, which should all have the extension '.wam' but may have different filenames corresponding to different sets of input files, as with the standard test runs described in Appendix A.

The program requires the same Intel/Microsoft DLL files as are required by WAMIT Version 7, listed in Section 2.1. These must be in the same directory as the program at runtime. The program can either be run locally from each directory where input files are processed, or alternatively (and more conveniently) in one directory of the PC which is identified in the system PATH.

Before executing the program two preparations must be made: (1) a sub-directory `V7inp` should be made, to save the new input files, and (2) the input file `directory_wam.txt` should be made with a listing of all fnames files to be processed. This can either be done manually by the user, or automatically using the batch file `V6V7.bat` which also runs the program. If the batch file is saved with `V6V7inp.exe` in a directory identified by the system PATH it can be executed from each directory containing old input files, simply by typing the command `V6V7`. This automatically performs the steps (1) and (2) above and then runs the program. Alternatively, these three steps can be performed manually by the user. (The simplest way to make the input file `directory_wam.txt` manually is to execute the DOS command `dir *.wam > directory_wam.txt` which produces a DOS directory list of all fnames files. The program ignores extraneous columns in this directory and only uses the character string containing '.wam' if this is found.)

When the program is run, a list of the fnames files is output to the monitor, with a list of modified input files appended. The same list is saved in the file `v6v7inp.log`. Input files which are modified are saved in the sub-directory `V7inp`. If the user wishes to update the input files for each run, simply execute the DOS command `copy v7inp\*.*` to overwrite the original files in the directory with the new files. Before overwriting with the new files it may be advisable to save the old input files, for example by copying them to a sub-directory `V6inp`.

Special situations which should be noted are as follows

- If the old configuration file uses the default name `config.wam` a new configuration file $fnamewam$`.cfg` is generated, where $fnamewam$ is the filename of the fnames file. In this case a new fnames file is generated, including the name of the new .cfg file.
- In cases where IALTPOT=1 is used for the old files and the .gdf file name is included in the .wam file, this filename is removed in the new .wam file.
- Configuration parameters which are no longer used are removed from the new cfg file.
- If the same input file is used with more than one fnames file in the directory, the new input file with the same name may not be correct except for the last run processed. The following example illustrates how this problem may occur: if two runs with fnames files RUNA.WAM and RUNB.WAM use the same POT file RUNAB.POT with IALTPOT=1 format in V6.4, and with different GDF files RUNA.GDF and RUNB.GDF assigned in RUNA.WAM and RUNB.WAM, if V6V7inp processes RUNA first and RUNB second there is only one new file RUNAB.POT with the gdf file specified as RUNB.GDF in this file.

The program `V6V7inp` is intended to work for all input files which were valid in Version 6, but it is impossible to test the program with all possible combinations of inputs. Users are advised to check the changes in the new input files, especially if error messages are generated in the WAMIT runs using these files, and to report problems by email to `info@wamit.com`.

Appendix C

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; revised Sept. 20, 2008; revised Oct. 20, 2011

Contents

Using the WAMIT-RGKernel Interface 1
Contents 1
1. Introduction 1
1.1 New features for WAMIT version 6.4 2
2. Supported features and options 3
3. Required files, versions and file locations 4
4. MultiSurf modeling considerations 4
4.1 Trimmed surfaces are excluded 4
4.2 Optional wetted-surface Entity List 5
4.3 Order of patches 6
4.4 Surface normal orientations 6
4.5 Base plane and waterline 7
4.6 Mirror symmetry 8
4.7 Rotational symmetry 8
4.8 “Fast” vs. “Accurate” evaluation 9
4.9 Divisions and Subdivisions 9
4.10 Irregular frequency removal 10
4.11 Coordinate singularities 10
4.12 Breaklines in surfaces 11
4.13 “Cosine spacing” 11
4.14 Parameters 12
4.15 Patch types and color coding 12
4.16 Internal tanks 13
4.17 Control surfaces for mean drift forces 13
5. GDF file format 14
6. Short GDF format for low-order analysis 16
7. Log file RGKLOG.TXT 16
8. Synopsis of operation 17
9. Error Conditions 18

1. Introduction

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

1.1 New features for WAMIT version 6.4

New features introduced to the RG2WAMIT interface for WAMIT version 6.4 include:

- Dipole patches (Section 4.15)
- Internal tanks (Section 4.16)
- Control surfaces for mean drift forces (Section 4.17)
- Transparent treatment of breaklines in MultiSurf surfaces (Section 4.12)
- Allowing NPATCH = 0 in the short GDF file (Section 5)
- Short GDF format for low-order analysis (Section 6)

1.2 New Features for WAMIT version 7.0

New features introduced to the RG2WAMIT interface for WAMIT version 7.0 include:

- Free-surface Pressure patches (Section 4.15)
- Internal tanks (Section 4.16)

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 entity 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 folder as WAMIT.EXE. Also, any model file named in an IGDEF=2 GDF must be located in the same folder as the GDF. We'll subsequently refer to this folder as the "working folder".

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 Trimmed surfaces are excluded

Trimmed surfaces (TrimSurf and TrimSurf2 entities) can only be handled for low-order analysis (ILOWHI = 0) 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. When ILOWHI = 1, RG2WAMIT checks for the presence

of Trimmed Surfaces in the wetted-surface Entity List, and returns an error when one is encountered.

SubSurfaces, on the other hand, are perfectly legal for RG2WAMIT, because they have (like all other RG surface types except Trimmed Surface) 4-sided topology and a full [0,1]x[0,1] parameter space. One or more SubSurfaces can always cover a complex-shaped region of a base surface, where a single Trimmed Surface might otherwise be used.

4.2 Optional wetted-surface Entity List

In order to be read and utilized through the WAMIT-RGKernel interface, an MS2 file ("model file") may include an Entity List naming the wetted surfaces. When this option is used, WAMIT needs to be given the name of this Entity List (in the GDF file, see below).

Note: In MultiSurf 5 and earlier, an Entity List was called an "ObjectList". 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 Entity List. 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 property), or located on a layer that is disabled in the MS2 file (unchecked in Settings / Layers dialog).

Entity List (ObjectList) is an entity type that has been supported since MultiSurf 4.0. It is just a list of other entities, of any type. It is a non-graphical entity; you may or may not be able to see the entities in the list -– that depends on whether each entity is made from a graphical entity type, whether it is visible, and whether it is on a turned-on layer. But, the Entity List itself is never drawn -– it is just a list.

In MultiSurf 6 and later, to create an Entity List, use Insert / Entity List. You can preselect the "parents", i.e. the entities to be in the list. You can add or subtract entities just as you would edit the multiple parents of any other entity. Select / By Name is often the best way to select an Entity List -– or 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 (different volumes

for the X, Y and Z directions). The interface does not currently produce a warning in these situations.

You can have two or more Entity Lists in your model. They (like all other entities) will be distinguished by their unique names. By changing the Entity List name specified in the GDF file, you can conveniently switch from one Entity List 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(3) 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 Entity List:

(1) If you use an explicit Entity List, the patch order will be the order in which the surfaces are listed in the Entity List.

(2) If you use the default identifier '*' for the Entity List, 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"; Select / All; and read the order in the Selection Set Manager.

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 *in WAMIT* 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 3 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 parent entities;

(2) the order in which the parents 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, you first need to have an option turned on: this is under Tools / Options, on the General tab, Auto Orientation needs to be "All selected objects". Then select all the wetted surfaces; for example, select the Entity List, then Select / Expand Entity Lists / 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 file 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 Entity List, or when * is used for the Entity List, 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 (in RGKLOG.TXT) 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" (Tools/ Settings/ Performance tab) 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 divisions multiplier of 1. 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” properties 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 is often 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 WAMIT's 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 Entity List of wetted surfaces, and in NPATCH. If you put them last in the Entity List, 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 direction. 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 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 (a "degree-1 breakline"), 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.

Before WAMIT version 6.4, we had to recommend in general that breaklines **not** be used on surfaces intended for WAMIT consumption. WAMIT was not be aware of the breakline and would have been modeling a smooth, continuous distribution of potential across the chine.

In the version of RG2WAMIT released with WAMIT version 6.4, breaklines are automatically taken into account. A MultiSurf surface with N u=constant degree-1 breaklines and M v=constant degree-1 breaklines is split along these breaklines into (N+1) x (M+1) "panes" and is presented to WAMIT as that many separate patches. This all takes place "under the hood", with little need for the user to be aware of it. (The splitting at breaklines will be visible in the patch counts seen in RGKLOG.TXT.)

(In the truncated cylinder example, modeling the side and bottom surfaces as two separate surfaces (rather than one surface with a breakline) would have the advantage of making it 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:
entity-name float-index value

For example, if Point '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.

4.15 Patch types and color coding

WAMIT recognizes several types of surface patches with different characteristics for analysis. In the RG2WAMIT interface these patch types are distinguished by color coding -- assigning specific colors to surfaces to designate different patch types, as follows:

(1) Color 11 (bright cyan) is reserved for dipole patches.
(2) Color 15 (bright white) is reserved for exterior free surface patches (used for FDF file, for the second-order option).
(3) Color 4 (red) is reserved for free-surface pressure patches.

Conventional patches (body patches) and interior free surface patches can be any other color besides 4, 11 and 15.

Surface colors are selected in the Properties Manager. The Color Property dialog has a control allowing colors to be specified by color number (0-255).

4.16 Internal tanks

Internal tanks with liquid contents and a free surface can be represented in a model for WAMIT analysis; see section 10.7 of the WAMIT User Manual.

In MultiSurf, the tanks are modeled with surfaces, and are communicated to WAMIT through a specially composed Entity List, called the "Tank list". The name of the tank list is included in the short-form GDF file as a second token on line 7, following the wetted surface entity list.

The tank list will have one entity for each internal tank; this must be either
(1) a surface entity (when a single surface is enough to define the tank), or
(2) an entity list of surfaces for the tank (in the usual case where multiple surfaces are required to define the tank).

It is possible to analyze the tanks alone, without the enclosing body. Just insert an empty Entity List (say its name is 'empty') in your model, and use it for the wetted-surface Entity List on line 7 of the GDF file:

```
empty  tank_list
```

Note that WAMIT requires the normals on tank surfaces to be opposite those for body surfaces, because the wetted side is the inside of the tank, as opposed to the outside of the body. Thus if your body surfaces have outward normals, the tank surface normals should be inward.

The density of the fluid in each tank is specified in the .CFG file (RHOTANK).

4.17 Control surfaces for mean drift forces

A new analysis option in WAMIT version 6.3 is evaluation of mean drift forces and moments by means of momentum flux through a control surface surrounding the body. The control surface is defined in a control surface file (.CSF extension), with two format options:
(ICDEF = 0) low-order, similar to a low-order GDF; or
(ICDEF = 1) higher-order, similar to a higher-order GDF.

The control surface can be modeled in MultiSurf. For ICDEF = 0, it can consist of any type of surfaces, including Trimmed Surface, triangle meshes or Composite Surfaces. For ICDEF = 1, it can consist of any type of surface *except* Trimmed Surfaces and Composite Surfaces.

The control surface can be part of the same MultiSurf model as the body surfaces, or it can be a completely separate model file.
CSF export is now on the MultiSurf menu, under File / Export3D / WAMIT.

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 surfaces in this body (not counting X and Y mirror symmetry images, but counting Z-axis rotational symmetry images)
(Starting with WAMIT v. 6.4, NPATCH can be specified as 0; see discussion below.)
IGDEF = 2, for using geometry from an .MS2 file

In an IGDEF=2 GDF file, there are always 4 more lines:

Line 5: NLINES = number of lines to follow this line (3 plus number of parameter lines)
Line 6: Model filename, including extension .MS2 (no embedded spaces)
Line 7: Name of Entity List 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.)
Optionally, line 7 can additionally have a second Entity List name, specifying internal tank surfaces.
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:
entity-name float-index value

entity-name is the name of an entity in the model that has one or more floating-point numbers in its properties. For example, a Point has 3 floats -- its X, Y and Z coordinates or offsets --; a Bead has just one, its t parameter or offset.

float-index is an integer from 1 to the number of floats the entity has, e.g. 1 to 3 for a Point; 1 to 1 for a Bead.

value is the floating-point value to substitute.

NPATCH = 0 option

Starting with WAMIT version 6.4, NPATCH can be specified as 0 in an IGDEF = 2 short-form GDF. This means "figure out the number of patches from the Entity List (or '*') found on Line 7". This option is highly recommended because it avoids the need to count surfaces, and the potential error of leaving surfaces out by specifying the wrong number for NPATCH.

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

where N is the number of rotational symmetry images.
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 (except line 7) 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. Line 7 is an exception, because presence of a tank-list on this line is optional; if there is a second token on line 7, Fortran will try to interpret is as a tank-list name.

Example GDF:

```
Tension-leg platform example
120. 32.2  ULEN, GRAV
1  1  ISX,ISY
0  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 specified as 0; the interface will count the surfaces in Entity List 'wetted_surfs' to determine NPATCH. 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 Entity List named ‘wetted_surfs’ which names the wetted surfaces. This example has no tank-list. 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. Short GDF format for low-order analysis

Beginning with WAMIT version 6.4, the "short" GDF format used for IGDEF = 2 higher-order analysis can also be used for low-order analysis (ILOWHI = 0). The format is almost exactly the same as detailed in the previous section for higher-order analysis. The only difference is that the first token on line 4 must be 0, in place of NPATCH (the number of patches). RG2WAMIT will calculate the number of panels from the model geometry (the sum of panel counts over the wetted-surface patches).

Advantages of this feature are:
(1) No long-form low-order GDF file needs to be created.
(2) Parameter lines can be used in the short GDF file to vary the geometry at WAMIT run time.
(3) Trimmed surfaces, triangle meshes and composite surfaces can be freely used along with other surface types for low-order analysis.

Note that a MultiSurf model that is suitable for higher-order analysis may not be 100 percent suitable for low-order analysis. The primary issue is divisions and subdivisions, which control the subdivision of surfaces into panels. A surface with very low divisions x subdivisions can be plenty accurate for higher-order analysis, but would result in very few panels. This aspect should be investigated, and divisions increased if necessary. CountPanels command will likely be a useful tool in such an investigation.

7. Log file RGKLOG.TXT

During operation of the WAMIT-RGKernel interface, a log file is opened and written to in the working folder (the same folder that contains the GDF file). 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
Entity List 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
```

8. 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 working folder. 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.

9. 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 (.MS2 extension) named in an IGDEF=2 GDF file was not found. Check to see that the file exists in the same directory as the GDF file. Check to see if the file name was spelled correctly in the GDF. Blanks (spaces) are not allowed in filenames for RG2WAMIT. (NOTE: Filenames are not case sensitive in Windows, but *are case sensitive* in Linux.)

4. The specified Entity List was not found in the model file.
An Entity List of wetted surfaces may be specified by name in the GDF file. Entity names in RG are case sensitive, and must use an underscore '_' in place of a space. Check to see that the Entity List name is correctly spelled and correctly capitalized in the GDF. Open the MS2 file in either MultiSurf or Notepad and see that the Entity List is there, and you have given its correct name in the GDF. You can find the Entity List in the Entities Manager under Entity Lists, or use Select/By Name.

5. Wetted surface Entity List 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 Entity List 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 may 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. **Wetted-surface Entity List has fewer than NPATCH surfaces**
   The GDF file specifies NPATCH, the number of surfaces to use in the solution for a given body; the Entity List 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 Entity List. If the number of surfaces is large, making counting difficult, select the Entity List in MultiSurf (often Select/ By Name is the easiest way), and the number of its parents will be indicated in the Properties Manager. Visually, you can select the Entity List and Select/ Expand Entity Listts/ First Generation; all surfaces in the list will then be highlighted.

9. **Wetted-surface Entity List contains a non-surface entity.**
   The wetted-surface Entity List must contain all the wetted patches and/or interior free-surface patches, and no other entities. (An Entity List, in general, can legally contain entities of any type.) Check the contents of the Entity List and see that they are correct in number and identity.

10. **Wetted-surface Entity List includes a Trimmed Surface.**
    Trimmed surfaces are not eligible for use in WAMIT models for higher-order analysis, because they do not provide a complete square parameter space for distribution of the unknown potential. You may be able to replace a Trimmed Surface with one or more SubSurfaces.

11. **Surface entity is in error.**

The RGKernel error code will be given. This error code is compatible with the MultiSurf error system, so it 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.

12. The specified tank list was not found in the model.

A tank list was specified by name on line 7 of the GDF file, but no entity with this name was found in the model. First, confirm that you are trying to perform analysis with internal tanks in this run; if not, line 7 should have only a single token (the wetted-surface Entity List name, or '*'). Check that the second token on line 7 is the correct name for the tank list. Entity names are case sensitive. Check that an Entity List of this name exists in the model.

13. The entity named as tank list is not an Entity List.

A tank list was specified by name on line 7 of the GDF file, but the entity with that name is not an Entity List. Check that the second token on line 7 is the correct name for the tank list. Entity names are case sensitive. Check that an Entity List of this name exists in the model.

14. The tank-list Entity List is in error.

A tank list was specified by name on line 7 of the GDF file, but the entity with that name is in error. An error code will be given; this can be looked up in the MultiSurf manual or help system. The most likely cause is that one of the parent lists or surfaces is in error; this will result in error 284 on the tank list. In the Entities Manager, select the tank list and expand the Parents tab to identify the error.

15. Tank list contents don't qualify for a tank list.

A tank list was specified by name on line 7 of the GDF file, but the Entity List with that name does not qualify as a tank list. Each element of a tank list must be either (1) a surface, or (2) an Entity List of surfaces.

16. The wetted-surface Entity List contains duplicate surfaces.

Duplicate surfaces in the wetted-surface Entities List will lead to singular or very ill-conditioned equations in the WAMIT solution. Select the Entity List and click on "Entities" in the left-hand column to review its contents. Click in the right hand column to edit its contents; remove any duplicates.

17. IBODY is out of range.

In a multi-body problem, each IBODY must be in the range 1 to NBODY. This error would probably indicate a bug in WAMIT's writing of RGKINIT.TXT.

18. Two or more bodies have the same IBODY.

In a multi-body problem, the IBODY's must be distinct. This error would probably indicate a bug in WAMIT's writing of RGKINIT.TXT.

19. A surface has a patch type that is invalid for analysis.

Patch types are controlled by color coding; for example, color 11 (bright cyan) is reserved to indicate a dipole patch. Color 15 (bright white) is reserved for exterior free surface patches; these are excluded from both low-order and higher-order analysis. Review the colors of your surfaces; don't use color 15.

20. Demo version, used with non-demo model file.

Wamit, Inc. distributes two different versions of rg2wamit.dll, both with the same filename. The demo version is restricted to only function with a few specific model files, and is freely distributed. The real version is only distributed to users who are licensed to use WAMIT and MultiSurf together. This error occurs when the demo version of rg2wamit.dll is used with a MultiSurf model file other than one of the authorized demo files.

Using WAMIT-RGKernel Interface1.doc