

## Chapter 3

# USING WAMIT

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

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

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

■ The optional 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.

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
<i>pot.pot</i>	POTEN	Potential Control File (Sections 3.1-2)
<i>gdf.gdf</i>	POTEN	Geometric Data File (Chapters 5,6)
<i>frc.frc</i>	FORCE	Force Control File (Sections 3.3-4)
<i>gdf.spl</i>	POTEN	Spline Control File (Section 6.11)
FNAMES.WAM	POTEN/FORCE	Filenames list (Section 3.8)
■ BREAK.WAM	POTEN	Optional file for runtime breakpoints (Section 3.11)
*.cfg	POTEN/FORCE	Configuration file (Section 3.7)
<i>gdf.ms2</i>	POTEN	MultiSurf geometry file (Section 6.7)
<i>gdf.csf</i>	FORCE	Control surface geometry file (Section 14.1)
<i>gdf.bpi</i>	FORCE	Specified points for body pressure (Section 4.11)

The first three input files are required in all cases. The others are required in some cases, or are optional in all cases, as explained below. Here *pot* is a user-specified filename assigned to the POT file. The same filename is assigned to the P2F file, which is output by POTEN and input to FORCE. Similarly, *frc* is a user-specified filename assigned to the FRC file and this filename is assigned to the principal output files from FORCE. *gdf* is the user-specified filename assigned to the GDF file. If the optional Spline Control File is used it must have the same filename.

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

Filename	Program	Description
<i>pot.p2f</i>	POTEN	P2F File (binary data for transfer to FORCE)
<i>errorp.log</i>	POTEN	Error Log File (Section 10.1)
<i>errorf.log</i>	FORCE	Error Log File (Section 10.1)
<i>wamitlog.txt</i>	POTEN/FORCE	Log file of inputs (Section 10.1)
<i>frc.out</i>	FORCE	Formatted output file (Chapter 4)
<i>frc.num</i>	FORCE	Numeric output files (Section 4.9)
<i>gdf_pan.dat</i>	POTEN	Panel data file (Section 10.6)
<i>gdf_pat.dat</i>	POTEN	Patch data file (Section 10.6)
<i>gdf.pnl</i>	FORCE	Panel data file (Section 4.9)
<i>gdf.hst</i>	FORCE	Hydrostatic data file (Section 8.4)
<i>rgklog.txt</i>	POTEN/FORCE	MultiSurf log file (Section 6.7)
<i>frc.fpt</i>	FORCE	List of field points (Section 4.9)
<i>gdf.idf</i>	POTEN	Interior free-surface panels (Section 9.3)
<i>gdf.bpo</i>	FORCE	Specified points for body pressure (Section 4.11)
<i>gdf_low.gdf</i>	POTEN	Low-order GDF file (Section 10.6)
<i>gdf_csf.dat</i>	FORCE	Control surface data file (Section 14.4)
<i>gdf_low.csf</i>	FORCE	Low-order control surface file (Section 14.4)

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

The primary generic data files are the two ‘control files’ input to POTEN and FORCE.

These are referred to as the Potential Control File (POT), with the extension '.pot', and the Force Control File (FRC), with the extension '.frc'.

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

Additional input files may be used to assist in using WAMIT, to select various options, and to optimize its use for specific applications. The optional file FNames.WAM is used to specify the names of the GDF, POT, and FRC input files to avoid interactive input of these names (see Section 3.6). The optional input file CONFIG.WAM may be used to configure WAMIT and to specify various options, as described in Section 3.7. (Starting in Version 6.2 this configuration file may be assigned a different filename by the user.) The optional input file BREAK.WAM may be used to set break points in the execution of POTEN, as described in Section 3.11. The optional Spline Control File (SPL), is used only in the higher-order method, as described in Section 6.11.

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

Two alternative formats for the POT and FRC files are described separately in Sections 3.1-2 and 3.3-4.

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

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

Several output files are created by WAMIT with assigned filenames. The output from

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

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

The filenames assigned to the various output files are intended to correspond logically with the pertinent inputs, and to simplify file maintenance. Thus the filename of the P2F file is derived in the program from the filename of the POT file, and similarly for the OUT output file from the FRC file. To make this convention more specific it is necessary to define the input filenames specified by the user. For the present discussion these are assumed to be, respectively *pot*.POT and *frc*.FRC where the italicized portion is user-specified. The resulting output filenames are then assigned as follows: *pot*.P2F is the output file from POTEN, input to FORCE. *frc*.OUT is the principal output file from FORCE. The various numeric output filenames are assigned the same name as the OUT file, but with extensions corresponding to each option number. (The alternative name OPTN may be specified by setting the optional switch NUMNAM=1 in the configuration file, as described in Section 3.7.) For Option 5, the panel data file is assigned the filename from the Geometric Data File, *gdf*.PNL, since this data is specific to the body identified in the GDF file. If NUMNAM=1, the generic name ‘gdf.pnl’ is assigned to the panel data file. For Options 6-7, the field point file is assigned the filename *frc*.FPT (or OPTN.FPT, if NUMNAM=1), to correspond to the force control file and other numeric output files. These conventions are illustrated in Figure 1.1.

### 3.1 THE POTENTIAL CONTROL FILE (Alternative Form 1)

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

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

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

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

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

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

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

**‘header’** denotes a one-line ASCII header dimensioned CHARACTER\*72. This line is available for the user to insert a brief description of the file.

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

ISOR= 0: Do not evaluate the source strength.

ISOR= 1: Evaluate the source strength.

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

**NPER** is the number of wave periods to be analyzed. **NPER** must be an integer. The following alternatives can be used starting in Version 6.2:

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

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

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

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

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

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

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

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

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

If  $N\text{PER} < 0$  a special convention is followed to assign a total of  $|N\text{PER}|$  wave periods, frequencies, or wavenumbers with uniform increments, starting with the value equal to  $\text{PER}(1)$  and using the increment equal to the value shown for  $\text{PER}(2)$ . In this case only two values  $\text{PER}(1), \text{PER}(2)$  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  $\text{IPERIO}$  values, irrespective of whether  $\text{PER}$  represents the wave period, frequency, or wavenumber. Special attention is required when zero-period or zero-frequency inputs are required, following the definitions

as specified above. For the example described in the preceding paragraph with  $IPERIO=2$ , the inputs  $NPER=-4$  and  $(PER = -1. 1.)$  will result in the sequence of wave frequencies equal to (zero, infinity, 1., 2.).

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

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

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

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

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

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

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

and

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

## 3.2 THE POTENTIAL CONTROL FILE (Alternative Form 2)

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

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

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

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

Definitions of the other data above are as follows:

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

**GDF(K)** is the name of the Geometric Data File of the *K*th 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 *k*th body relative to the **global coordinate system**, input in the units of the length ULEN. The incident-wave

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

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

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

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

### 3.3 THE FORCE CONTROL FILE (Alternative form 1)

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

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

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

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

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

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

‘**header**’ denotes a one-line ASCII header dimensioned CHARACTER\*72. This line is available for the user to insert a brief description of the file.

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

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

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

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

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

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

#### **IOPTN(4)**

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

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

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

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

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

#### **IOPTN(5)**

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

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

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

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

#### **IOPTN(6)**

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

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

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

### **IOPTN(7)**

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

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

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

### **IOPTN(8)**

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

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

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

### **IOPTN(9)**

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

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

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

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

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

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

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

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

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

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

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

- 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, as explained in Section 3.1. 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 10.1, 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  $3 \times 3$  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  $m_{ij} = m \times \text{XPRDCT}(i - 3, j - 3) \times |\text{XPRDCT}(i - 3, j - 3)|$ . 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 (see Section 4.3). This is a useful feature since a relatively small number of wave headings for the diffraction problem may be specified in the Potential Control File and the time required to solve many diffraction problems in POTEN greatly exceeds the time required to evaluate the Haskind exciting forces in FORCE. Since the number of Haskind wave headings will affect the subsequent READ statements for data in the Force Control File, it is important to ensure that this number corresponds with the prescribed integer NBETAH. In particular, if NBETAH= 0 no values of BETAH should be included and NFIELD should appear on the next line of the Force Control File. If NBETAH> 0 is specified, the settings of the IOPTN switches are automatically set equal to 0 for options 3,5,6,7,8.

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

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

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

### 3.4 THE FORCE CONTROL FILE (Alternative form 2)

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

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

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

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

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

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

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

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

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

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

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

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

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

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

```

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

```

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

```

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

```

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

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

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

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

I=1: matrix follows on subsequent lines

I=2: file name follows on next line

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

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

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

### 3.5 DEFINITION OF FIXED OR FREE MODES

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

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

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

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

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

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

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

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

### 3.6 FILENAMES LIST ‘FNAMES.WAM’

An optional input file may be used to specify the filenames of the primary input files CFG, POT, FRC, and GDF. Use of this optional file is recommended, particularly to facilitate batch processing. The optional file must be named FNAMES.WAM. (The name ‘fnames.wam’ must be used for this file if the system is case sensitive. Source-code users can redefine this name if it is desired to do so.) The optional file is simply a list of the other input file names, including their respective extensions ‘.cfg’, ‘.pot’, ‘.frc’, and ‘.gdf’. If this file does not exist, or if it is incomplete, the user is prompted to supply the missing filenames interactively. In Version 6 the number and order of these file names is arbitrary, but they must include the specified extensions. If the alternative (IALTPOT=2) format is used, as explained in Section 3.2, the GDF filename(s) are specified in the POT file and it is not necessary to specify any GDF filename in the FNAMES.WAM file. (In this case a GDF filename that exists in the FNAMES.WAM file is ignored.) Starting in V6.2 the filename of the configuration file may be input in the FNAMES.WAM file, with the extension ‘.cfg’, as explained below.

### 3.7 CONFIGURATION FILE ‘CONFIG.WAM’ OR ‘\*.CFG’

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

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

Parameter	Alternate File	Default value
IALTFRC		1
IALTFRCN		1,1,1,...
IALTPOT		1
■ IBODYW		0
ICTRSURF		0
IDIAG†	POT	
IFIELD_ARRAYS		0
IFORCE		1
■ IGENMDS(n)		0
ILOWGDF		0
ILOWHI		0
ILOG†	POT	
INUMOPT5		0
INUMOPT6		0
INUMOPT7		0
IPERIO		1
IPLTDAT		0
IPNLBPT		0
IPOTEN		1
IQUAD†	POT	0
IQUADI†	SPL	4
IQUADO†	SPL	3
■ IRREADRAO		0
■ IRR(n)†	POT	
ISCATT		0
ISOLVE		0
ISOR†	POT	
ITANKFPT		0
■ ITRIMWL		0
■ IWALLX0		0
■ IWALLY0		0

	KSPLIN	SPL	3
	MAXITT		35
	MAXMIT		8
	MAXSCR		2
	MODLST		0
	MONTR		0
	NEWMDS(n)†	POT <sub>2</sub>	0
	NOOUT		1
■	NPDIPOLE(n)	GDF	
	NPTANK(n)		
	NUMHDR		0
	NUMNAM		0
	PANEL_SIZE		-1.0
	RHOTANK		
	SCRATCH_PATH		current directory
■	TOLGAPWL		1.E-3
	USERID_PATH		current directory
	XBODY(n)†	POT or GGDF	
■	XTRIM(n)		
■	ZTANKFS		

When IALTPOT=1, parameters marked † must be input once and only once, either in the configuration file, or in the other files specified following the symbol †. When IALTPOT=2 the input of these parameters is optional in the CFG file. All other parameters in the list above are optional, with the specified default value assigned when the parameter is not included explicitly.

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

- IGENMDS(n), IRR(n), NEWMDS(n), NPDIPOLE(n), NPTANK(n) XBODY(n) and XTRIM(n) denote the elements of the corresponding arrays for body number n. When NBODY=1 these can be written more simply as IGENMDS, IRR, NEWMDS, NPDIPOLE, NPTANK, XBODY and XTRIM. If NBODY>1 the number of each body should be indicated in parenthesis, as illustrated in the example in Section 7.6. If IGENMDS≠0 the nonzero values must be input for each body in the configuration file. If IALTPOT= 1 and NEWMDS>0 the nonzero values of NEWMDS must be input for each body in the configuration file. If IALTPOT= 2 the parameters NEWMDS and XBODY must be input for each body in the POT file as shown in Section 3.2.

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

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

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

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

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

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

The default value is IALTFRC= 1.

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

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

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

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

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

The default value is IALTPOT= 1.

- **IBODYW** is an integer specifying the body number of the first body which represents a wavemaker in a wall (see Section 10.10).

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

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

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

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

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

The default value is ICTRSURF= 0.

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

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

IFIELD\_ARRAYS= 1: Additional field point data is assigned in the conventional FRC

files, as explained in Section 3.10.

The default value is `IFIELD_ARRAYS= 0`.

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

`IFORCE= 0`: Do not execute FORCE.

`IFORCE= 1`: Do execute FORCE.

The default value is `IFORCE= 1`.

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

`IGENMDS= 0`: use a separate 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≠ 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`.

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

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

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

The default value is `ILOWGDF= 0`.

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

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

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

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

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

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

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

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

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

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

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

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

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

IPERIO= 1: Input periods in seconds for PER

IPERIO= 2: Input radian frequencies for PER

IPERIO= 3: Input infinite-depth wavenumbers for PER

IPERIO= 4: Input finite-depth wavenumbers for PER

The default value is IPERIO= 1.

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

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

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

The default value is IPLTDAT= 0.

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

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

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

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

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

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

IPOTEN= 0: Do not execute POTEN.

IPOTEN= 1: Do execute POTEN.

The default value is IPOTEN= 1.

**IREADRAO** is an integer parameter specifying if the RAO's are to be input from an external file, as explained in Section 3.12.

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 without using RAO's computed by WAMIT for Options 5-9

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

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

ISCATT= 0: Solve the diffraction problem

ISCATT= 1: Solve the scattering problem

The default value is ISCATT= 0.

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

ISOLVE= 0: Use the iterative solver

ISOLVE= 1: Use the direct solver

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

The default value is ISOLVE= 0.

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

The block iterative solver is based on the combination of the local LU decomposition for each diagonal block and the algorithm of the iterative solver. This option may be used in cases where the the iterative solver does not converge sufficiently fast, but where the

available RAM does not permit assigning  $\text{MAXSCR}=\text{NEQN}$ , or where  $\text{NEQN}$  is so large that the direct solver is too time-consuming (the cpu required for the LU decomposition is proportional to  $\text{NEQN}^3$ ). In general, the number of iterations is reduced as the block size increases.

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

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

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

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

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

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

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

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

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

The default value is  $\text{ITANKFPT}=0$ .

- **ITRIMWL** is the integer parameter used to specify the trimmed-waterline option (See Section 10.8). When  $\text{ITRIMWL}=1$  the array  $\text{XTRIM}$  must be included for each body.

The default value is  $\text{ITRIMWL}=0$ .

- **IWALLX0** is the integer parameter used to specify that the plane  $X=0$  is a reflecting wall (See Section \*\*.\*)). When  $\text{IWALLX0}=1$  the fluid is bounded by a reflecting wall at  $X=0$ .

The default value is  $\text{IWALLX0}=0$ .

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

The default value is  $IWALLY0 = 0$ .

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

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

**MAXMIT** is the maximum number of iterations in the adaptive integration used to evaluate the momentum integral for the drift force and moment (Option 8). (The maximum number of integration ordinates in the azimuthal direction is equal to  $2*MAXMIT$ .) The default value  $MAXMIT=8$  is assigned unless a different value is input in the **CFG** file; this value is recommended for general use. A warning message is issued if the integration does not converge to 4 significant digits within the specified number of iterations. See Section 10.1 for further information regarding the interpretation and control of this warning message.

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

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

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

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

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

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

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

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

The default value is  $MONITR=0$ .

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

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

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

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

NOOUT= 1 1 1 1 0 1 1 1 1

This option can be useful to avoid very long OUT files, since the data for option 5 is generally much more extensive than for the other options. The default value NOOUT(I)= 1 for  $I = 1, \dots, 9$  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. In WAMIT Versions prior to V6.4 this data must be input in the GDF file, as explained in Sections 5.4 and 6.10. In Version 6.4 the option is introduced to input the dipole data instead in the CFG file, with somewhat more flexibility. If dipole elements are used, this data must be input for each body where dipole elements are used, either in the GDF or CFG file, not in both files. (If NBODY>1 it is permissible to mix these conventions, using the GDF option for some bodies and CFG for other bodies.) If the CFG option is used, there are no restrictions on the number of lines of input but each line must begin with ‘NPDIPOLE =’. If the CFG option is used in the low-order method it is not required for the dipole panels to be at the end of the panel array for each body, as in the case where the number of dipole panels are specified in the GDF file. If the CFG option is used, lines which begin with ‘NPDIPOLE =’ 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 correspond to dipoles:

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.

Since the CFG option is more general, and does not require an explicit input of the total number of dipole patches, it is recommended to use this option although GDF files with dipole data used in previous versions of WAMIT may still be used in Version 6.5. The option to define dipole panels or patches in the GDF file may be discontinued in future versions of WAMIT.

**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` – this requirement is new in Version 6.4.** More than one line can be used for multiple tanks, and/or multiple tanks can be defined on the same line. If `NBODY`>1, the body number(s) for each body containing tanks must be appended to the parameter name. Only integer data 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 10.7.

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

NUMHDR= 0: No headers are included.

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

The default value is NUMHDR= 0.

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

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

NUMNAM= 1: Numeric filenames are assigned as ‘optn’.

The default value is NUMNAM= 0.

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

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

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

**RHOTANK** is a real array used to specify the density of fluid in internal tanks. The density specified is relative to the density  $\rho$  of the fluid in the external domain outside the bodies, as defined in Chapter 4. The data in the array `RHOTANK` must be input in the same order as the data in the array `NPTANK`. Multiple lines of this parameter may be used, with an arbitrary number of data on each line, but each line must begin with ‘RHOTANK=’. The total number of tanks `NTANKS` is derived from the inputs `NPTANK` in the `POTEN` run. If fewer than `NTANKS` values of `RHOTANK` are specified, the remainder of the array is assigned the last non-negative value. Thus if the density is the same for all tanks, only the first value is required. Zero may be assigned, but negative values of the density must not be assigned. `RHOTANK` is only used in the `FORCE` run,

and may be changed if separate FORCE runs are made using the same POTEN outputs. Further details and examples are given in Section 10.7.

- **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 (Section 9.4) and automatic representation of control surfaces (Section 14.4). The default value is  $10^{-3}$ .
- **XTRIM(n)** is the real array used to specify the trimmed-waterline option for body number n. When NBODY=1 XTRIM may be used instead of XTRIM(n). XTRIM includes three real numbers, specifying the vertical displacement, pitch, and roll of the body in the trimmed condition, as explained in Section 10.8.
- **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, as in Versions 6.2 and 6.3. ZTANKFS is only used when the waterline trimming parameter ITRIMWL=1. Further information is given in Sections 10.7 and 10.8.

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

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

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

The syntax of the configuration file is illustrated in the following example. To specify each of the desired inputs, the corresponding parameter is displayed, followed by an '=' sign, followed by the value of the parameter. These lines may be in any order. Lines which do not contain an '=' sign are ignored. Comments may be inserted following the

value of a parameter on the same line, separated by at least one blank space. (Since a blank space is used to designate a comment, the names of directories or folders in SCRATCH\_PATH and USER\_PATH cannot include blank spaces.) All elements of the arrays IALTFRCN(NBODY), NOOUT(9) and XBODY(4) should be displayed in order on one line. All other parameters must be input singly on separate lines.

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

- IALTFRC=1 (use FRC format in Section 3.1)
- IALTPOT=1 (use POT format in Section 3.3)
- IBODYW =0 (No wavemakers are present in walls)
- ICTRSURF=1 (use Alternative 1 evaluation of control surface drift forces)
- IDIAG=0 (omit IDIAG from POT file)
- IFIELD\_ARRAYS=1 (arrays of field point data is assigned, as in Section 3.10)
- IFORCE=1 (run the FORCE subprogram)
- IGENMDS=1 (run DEFINE subroutine in .dll file for generalized modes)
- ILOG=0 (omit ILOG from POT file)
- ILOWGDF=0 (do not output low-order GDF from higher-order geometry)
- ILOWHI=0 (use low-order method)
- INUMOPT5=1 (output the separate components of the body pressure/velocity)
- INUMOPT6=1 (output the separate components of the field pressure)
- INUMOPT7=1 (output the separate components of the field velocity)
- IPERIO=1 (POT file contains wave periods in seconds)
- IPLTDAT=0 (do not output data for plotting geometry)
- IPNLBPT=1 (evaluate body pressure at points specified in body coordinates)
- IPOTEN=1 (run the POTEN subprogram)
- IQUAD=0 (omit IQUAD from POT file)
- IQUADI=4 (inner Gauss quadrature order in higher-order method)
- IQUADO=3 (outer Gauss quadrature order in higher-order method)
- IREADRAO=2 (input modified RAO's from external file)
- IRR=0 (omit IRR from POT file)
- ISCATT=1 (Solve for scattered potential)
- ISOLVE=2 (Use two blocks in iterative solver)
- ISOR = 1 (solve for source strength – omit ISOR from POT file)
- ITANKFPT=1 (input field points in internal tanks with special format)
- 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)
- MAXSCR = 1024 (store  $\leq 1024 \times 1024$  LHS matrix elements in RAM)
- MODLST =1 (Outputs in same order as left-hand-sides)
- MONITR=1 (display all FORCE output to monitor)

NEWMDS=0 (No generalized modes)  
 NOOUT=1 1 1 1 0 1 1 1 1 (output all but body pressure/velocity data)  
 ■ NPDIPOLE= 2 4 6 (dipole panels/patches)  
 ■ NPTANK= (8-11) (12-15) (range of panels/patches on two internal tanks)  
 NUMHDR=1 (write headers to numeric output files)  
 NUMNAM=1 (Numeric filenames are assigned as 'OPTN')  
 PANEL\_SIZE = 0.1 (automatic subdivision of patches in higher-order method)  
 RHOTANK = 0.6 1.0 (relative fluid densities in the two tanks)  
 SCRATCH\_PATH=D:\TEMP  
 ■ TOLGAPWL= 0.01  
 USERID\_PATH=C:\WAMIT  
 XBODY = 0.0 0.0 0.0 0.0 (omit XBODY from POT file)  
 ■ XTRIM = 1. 2. 0. (Trim the body by 1 vertically and 2 degrees pitch angle)  
 ■ ZTANKFS = 1. 2. (Elevations of tank free surfaces above the plane Z= 0.)

## 3.8 FILE NAMES

The names of the input data files should not exceed **20 characters** in length including extensions and should have **one period** separating the filename and extension. Filenames must be continuous, with no 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 CYL.FRC is retained in Session 2 as described in Section 2.2, instead of a modified name such as CYL2.FRC, the user will be prompted at the start of the FORCE subprogram with the choice of either overwriting the old file CYL.OUT or specifying another name for the new OUT file. If the default setting NUMNAM=0 is used, the same safeguard will apply to the numeric output files, minimizing the possibility that these are lost during a subsequent run. Otherwise, if NUMNAM=1, the OPTN output files are assigned the same generic names for all runs, and old OPTN files are overwritten without warning when a new run is made; this option avoids the proliferation of old output files, but requires the user to rename or otherwise preserve the contents of OPTN files which are to be saved.

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

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

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

Starting in V6.2 a separate utility program F2T.EXE is provided. 'INPUTS.F2T' is reserved as a generic input filename to this program.

### 3.9 FILE FORMAT

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

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

### 3.10 UNIFORM ARRAYS OF FIELD POINTS

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

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

The additional data is defined as follows:

**NFIELD\_ARRAYS** is the number of separate arrays. **NFIELD\_ARRAYS** must be an

integer, greater than or equal to zero.

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

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

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

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

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

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

Test23 illustrates the use of this option.

### ■ 3.11 USING THE OPTIONAL FILE 'BREAK.WAM'

The principal computational time required in most 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 output file.) This display during the run permits the user to monitor the progress, and estimate the total time to complete the run.

Starting in V6.4 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.

## ■ 3.12 ASSIGNING RAO'S IN AN EXTERNAL FILE

In some special circumstances it may be appropriate to modify the Response Amplitude Operators (RAOs), to account for physical effects that are not included in the WAMIT computations. Examples include mooring loads, forces on risers, wind loads, thrusters, and empirical corrections based on model tests. Often in these circumstances it is appropriate to use the hydrodynamic force coefficients evaluated by WAMIT in Options 1-3 (added-mass, damping, and exciting force coefficients), as well as hydrostatic forces evaluated by WAMIT.

Starting in V6.4 the option exists 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 3.7. If IREADRAO=0 (default), the RAO's computed by WAMIT in Option 4 are used to evaluate the other hydrodynamic parameters (Options 5-9), as in previous versions of WAMIT.

In a typical application where this option is used, 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 skipped. (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 other software. 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 conform to the contents of the WAMIT output file *frc.4*.

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 4.9 for OPTN.4. (Only the real and imaginary components are used as inputs, so that 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 decimals. 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 3.5, 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 should be noted that it is not necessary to assign *IREADRAO*=1 in place of the default *IREADRAO*=0, but this avoids unnecessary computations for Options 5-9 in situations where the RAO's will be modified by the user.

The use of this option is illustrated in Test15a, as described in Appendix A.15.