FS2000

FS-Wave Hydrodynamic Loading

Advanced Structural Analysis
for Windows
(c) A.E.S. Ltd 1988,2022

Email:support@aes-uk.com

Table of Contents

1.0 Introduction

FSWave is a pre-processor for the generation of hydrodynamic loading on framed structures created in FS2000. The program can be used to generate a static 'snapshot' of loading due to wave action, current action. Gravity and buoyancy can also be included. It can also interface with the dynamic solution modules of FS2000 to enable full dynamic analysis to be undertaken.

The main capabilities of the program are:

- Wave motions may be evaluated by Airy(Linear), Stokes 5th Order or Stream Function Wave Theories.
- Current velocity can be defined as a function of depth and specified in a different direction to the wave direction.
- Load coefficients may be specified by individual definition, global definition, definition as a function of Reynolds Number, or definition as function of depth.
- Drag and inertia coefficients may be defined relative to "Y" & "Z" directions of the element local co-ordinate system.
- Marine growth (coatings) thickness may be individually specified for each element or defined as a function of depth.
- Contents density may be specified by individual definition for each element or defined globally. A contents limit in terms of depth can also be specified i.e. depth above which the element will be assumed empty.
- Loading on non-circular hollow sections and solid sections is possible.
- Loading due to non-structural items can be applied as attached nodal areas or volumes.

The program will read the default model data from the wave and other related data is then defined using the various input forms. There a few requirements when creating model for wave load generation. These are addressed in Section 4.

The Program is started from the Waveloader command in TASK:Load Generator menu. It can also be run in **Batch** mode.

-o-

2.0 Basic Program Operation - Overview

When started the FSWave program reads the model data files of the active FS2000 model. Default properties are assigned to various coefficients and parameters of FSWave.

Wave, current and other related data is defined using the various input forms. Data associated with the wave load evaluation is saved in two data types;

Hydrodynamic Model Data Files Data associated with the physical model

Environmental Load Data Files Data associated with the environment.

The program can be activated to evaluate the loading at any specified wave phase angle or alternatively the wave may be stepped through the structure. The total shear and vertical forces are printed for each evaluated phase interval. The centroid of force is also evaluated for single phase evaluation.

Loads Cases can be created during evaluation if the Create Load case option is activated.

-o-

2.1 Hydrodynamic Model Data

The ***Hydrodynamic Model Data*** File (file extension ".WAV") is a definition file that contains data relating to the hydrodynamic properties of the model i.e. drag coefficients, coating profiles etc. If a file exists it will be loaded automatically. This file will only exist if saved during earlier sessions in the FSWave with the same model.

The data file is an interpreted file that uses the command line instructions given in Section 8. When the file is save within FSWave extra description lines are added to make the file more readable.

It is always recommended when saving to use file extension **.WAV** for this file.

If additional Hydrodynamic Model Data File are required e.g. using different profiles on the same model, then the file extension .**WAV*** should be used e.g WAV2. This will ensure that all definition data is archived.

-o-

2.2 Environmental Load Data

Data relating to the environment is saved in the ***Environmental Load Data*** Files. Generally there will be one for each load case being generated.

The data file is an interpreted file that uses the command line instructions given in Section 8. When the file is save within FSWave extra description lines are added to make the file more readable.

These files can be saved using any legal filename and extension. However, to ensure that the file is archived with the model, the name must be the model name and the first character of the extension must be a @. The file extension may be used to identify direction i.e. **".@NW"** or **".@SW"**. Always include a file extension when saving.

It is recommended that before Wave Load Cases are created both the Hydrodynamic Model Data and the Environmental Load Data are saved.

-o-

3.0 Background Theory

3.1 Wave Theory

The user is given the option of evaluating wave profiles and particle motions by :

- Linear (Airy) wave theory.
- Stokes 5th Order wave theory.
- Steam Function (Dean) wave theory
- Solitary Wave theory

Stokes 5 th

The Stokes 5 th (Skjeldreia & Hendickson version) theory has been modified so that numerical difficulties in evaluating hyperbolic functions (high magnitudes) for deep water waves are avoided. This modification uses an effective depth of 2 times the wavelength for the evaluation of wave motions when the depth of water exceeds 2 times the wavelength.

Stream Function

Order - The order of the Stream function theory can be varied from 3 to 30. The more shallow the water (Breaking Limit) the higher the order. Wave Applicability charts are usually used to select the appropriate order.

Damping - Low damping (1.0) will increase convergence for near linear waves. High damping (0.1) will provide a more stable solution for more non-linear waves. The default value of 0.3 should be suitable for virtually all waves.

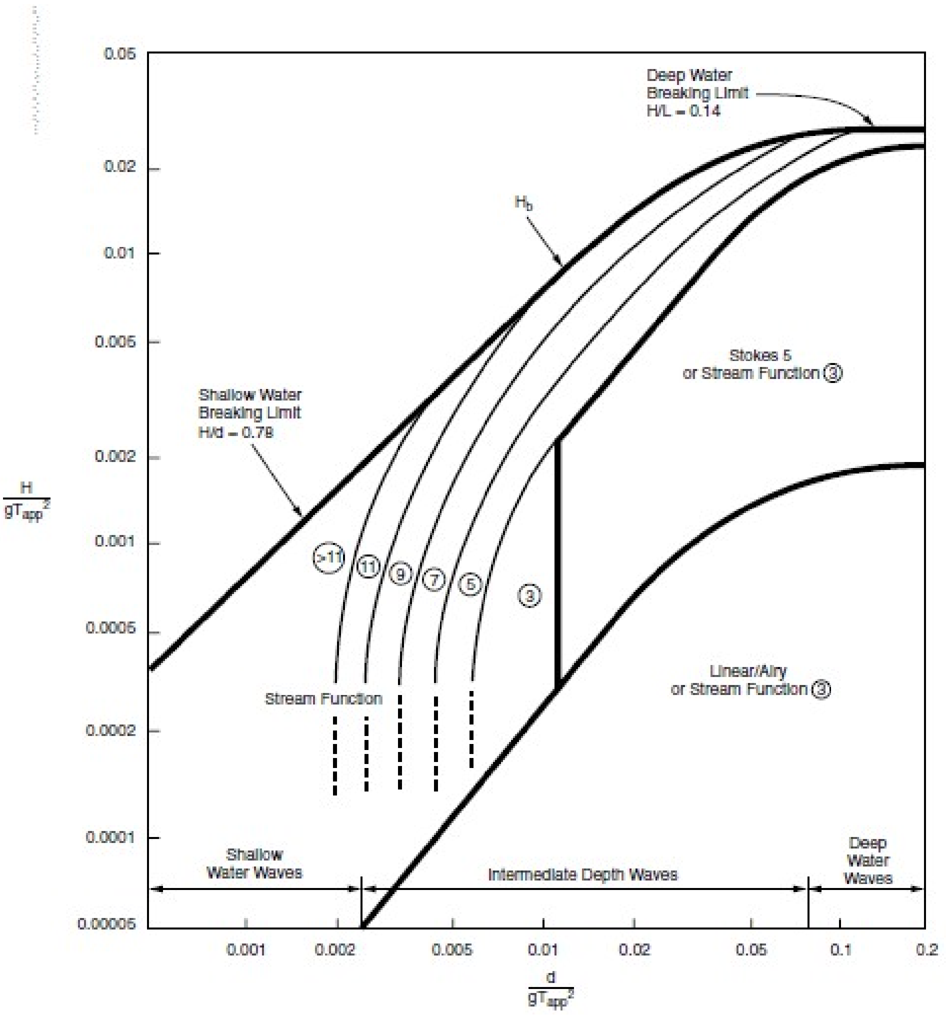
NPts - This defines the number points in wave cycle at which the Bernoulli constant is evaluated. Permissible range is 42 to 122. Value outside of this range will be assigned the value 62. Linear interpolation is used to evaluate wave properties between wave cycle points.

Solitary Wave Theory

The wave profile and celerity are based on Boussinesq and the particle motions are based on McCowan. A solitary wave being non-periodic does not have a specific wave length i.e its infinite. However there is a requirement to employ effective wave length so that the spatial effect of a passing wave is suitably captured and loading can be obtained at any point as the wave passes. This is achieved in the program by defining a suitable wave period which is then used to obtain a wavelength based on Linear wave theory. This period should be sufficiently high that the profile of the wave captured i.e between points where the surface elevation above SWL approach zero. this can be easily verified by clicking the **View Prof Elev** button in the Environment input form.

Wave Applicability

The wave theory applicability chart shown below is from API RP2A (Barltrop & Adams MTD 1991)


0.05
0.02
0.01
0.005
0.002
0.001
0.0005
0.0002
0.0001
0.00005
H / gT_app^2
Deep Water
Breaking Limit
H/L = 0.14
H_b
Shallow Water
Breaking Limit
H/d = 0.78
Stokes 5
or Stream Function ③
>11
11
9
7
5
3
Stream Function
Linear/Airy
or Stream Function ③
Shallow
Water Waves
Intermediate Depth Waves
Deep
Water
Waves
0.001
0.002
0.005
0.01
0.02
0.05
0.1
0.2
d / gT_app^2

-o-

3.2 Wave and Current Interaction

Both the wave particle velocities and current velocities are resolved into the directions of the global co-ordinate system. The respective velocity vector components are added prior to being used for load evaluation. Three wave/current interaction methods are available.

- *Stretched Profile* - The defined profile is stretched to the crest of the wave. This is the recommended (default) method
- *Stretched + Mass Continuity.* As the above but with the velocity is adjusted to maintain mass continuity.
- *Cutt-Off + Uniform Addition* - Profile is cut off at still water level and maintained constant at that value to the top of the wave crest.

-o-

3.3 Element Hydrodynamic Loading

The evaluation of hydrodynamic wave and current loading is based on the use of Morrison's Equation. This implies that the characteristic dimensions of the structural elements should not greater than 0.2 x the wavelength of the wave.

Transverse Loading (Local Y and Z directions)

The application of the Morrison's Equation to inclined elements is that most consistent with Morrison's Equation (derived only for vertical cylinders) i.e. the resultant velocity and accelerations are resolved into components tangential and normal to the element axis and only the normal components are used for evaluation in Morrison's Equation. This form of Morrison's equation is shown below (only y direction shown).

Fy = CDy.RHO.Dz.vn.uy/2 + (1+Cm'y).RHO.V.duy/dt

For non-tubular (note that if WINT=1 then non-tubular will be evaluated the same as tubular - this is default on all new models)

Fy = CDy.RHO.Dz.U.uy/2 + (1+C'my).RHO.V.duy/dt

Where

CD = Drag coefficient for y comp flow normal to the member

Cm'y = Added Mass coefficient for y comp flow normal to the member

Dz = Projected width in local z direction

V = Displaced volume of the member

RHO = Fluid mass density

vn = Normal component of fluid particle velocity $= (uy^2 + uz^2)^{1/2}$

uy = Fluid particle velocity in the local y direction

du/dt = Fluid particle acceleration in the local y direction

U = Resultant velocity component$= (vn^2 + vt^2)^{1/2}$

Note that V=Vr. Cm' = 1 therefore inertia Coefficient Cm = [1+Cm'] = 2

NOTE: The program uses the added mass coefficient rather than the inertia coefficient for inertia force evaluation. Be aware of the difference between inertia coefficient Cm and added mass coefficient Cm'. This is done because the program has been configured to accommodate dynamic solutions and non-tubular sections and it is more convenient to use the added mass coefficient for evaluation of inertia force.

Note that for tubular sections V=Vr. Cm' = 1 therefore the often quoted Inertia Coefficient Cm = [1+Cm'] = 2

For non tubulars the inertia force uses the following program defines parameters

Inertia Force (I.F) = Cm.a.V = (V + Cm'.Vr).a

Where

a = Fluid particle acceleration in the local y direction

V = Volume of liquid displaced by member (section volume+internal volume+coating - defined by area)

Vr = Reference volume to which the added mass coefficient is related

$= \pi$.Dz**2 / 4

Cm' = Added mass coefficient

Axial Loading (Local x direction) Loading

Loading in the axial direction is based on the following

Fx = CDx.RHO.Lp.vt..vt/2 + RHO.V.du/dt

Where

CDx = Drag coefficient for x comp flow (axial to the member)

Lp = Wetted perimeter (Coating perimeter for non-circular sections)

V = Displaced volume of the member

RHO = Fluid mass density

Vt = Axial(x) component of fluid particle velocity

V = Displaced volume of the member

RHO= Fluid mass density

du/dt = Fluid particle acceleration in the local x direction

Lift Loading (Global y direction) Loading

If a lift coefficient is defined on tubular elements the lift loading in the positive y global direction is based on the following.

$Fx = CL.RHO.D.Vnxz^2/2$

Where

CL = Lift coefficient

D = Effective Diameter

V = Displaced volume of the member

RHO= Fluid mass density

Vnxz = Comp. of fluid velocity parallel to the x-z plane and normal to element

RHO = Fluid mass density

It is only evaluated for elements within 1.5D of the seabed and vertical angles of less than 60 degrees to the seabed.

-o-

3.4 Nodal Hydrodynamic Loading

Loading due to non-structural attachments can be accounted for by the use of nodal Areas and Volumes. Area and Volumes and their associated local direction coefficients can be assigned to specific nodes in the model. The directional properties are defined by reference to the orientation of an existing element or by reference to a local coordinate system.

The evaluation of these types of load are based on the following

Drag Force = CDn.RHO.A.vn.vn/2

where

CDn	= Drag coefficient for comp flow in the direction of the local axis
A	= Area
RHO	= Fluid mass density
vn	= Component of fluid particle velocity in the local direction

Inertia Force = (1+C'm).RHO.V.du/dt

where

C'm	= Added mass coefficient for comp flow in the direction of the local axis
V	= Reference Volume
RHO	= Fluid mass density
du/dt	= Fluid particle acceleration in the local direction

-o-

3.5 Linearisation of Element Loads

The hydrodynamic loading is not linear along the element, to linearise the loading for analysis each element is divided into segments for load evaluation. The loading is then approximated by a series of linear trapezoidal loads distribution across each segment. This linearisation process is always conservative i.e. loads will always be grater that the exact distribution.

The default number of divisions is 3 but this can be increased to up 10 for long elements or cases with large load variation. The segments are generally equal except in the case of surface penetrating elements in which case a division boundary is positioned at the interface.

The evaluation of nodal deflections and nodal forces is always EXACT relative to the linearised trapezoidal loading applied.

The evaluation of mid span loading (moment plots etc) has an additional degree of linerisation in that the mid span load is assumed to be linear between the end points. In cases where the element length is long relative to the variation in load e.g. small waves on a long element or elements passing through the splash zone the level of accuracy will decrease. In such cases the element should be sub-divided to include a node at the point of interest. The plotting of the moment distribution along an element will show the level of accuracy (use at least 10 plot points). If an abrupt step appears it represents the degree of error. If this is too great and is an area of interest then additional nodes should be introduced.

On large models such as a jacket structure one mid span node in the elements above and blow the splash zone is likely to provide sufficient accuracy for overall shear loading. On smaller features such as a detailed riser model more mid-span nodes would be required.

For small models there is no reason to economic with mid-span nodes. On large models they can be inserted to asses their effect in areas of interest.

-o-

4.0 Creating Structural Models for Wave Analysis

Restrictions and/or cautionary measures when creating structural models in FS2000 for wave loading are;

- The seabed must be in the X-Z plane with the Y direction positive in the direction from the seabed towards the surface. This is shown in the Figures.
- When saving wave load cases ensure that different load case numbers to existing load cases are used otherwise they will be overwritten. It is good practice to keep wave load case numbers high e.g. 50 upwards.
- FSWave creates secondary binary load files which contain trapezoidal patch loads for each element and a conventional load case definition file (.L"m"). The .L"m" file only contains the load case parameters from FSWave it does not contain the actual loads. If a model is changed or saved in the Model Definition TASK all wave load cases must be regenerated since the secondary load files are purged (deleted).
- If the Purge Results option is not active when the model is saved, the existing load cases do not require to be regenerated. This is not a recommended practice
- If the model is recovered from archive, the wave load cases require to be regenerated
- If the number of elements in the model is changed the wave load cases will require to be re-generated. A warning will be given if this happens.
- The loading on tapered elements is based on the mean width.

Long Elements

A long element is an element whose loading distribution is highly non-linear i.e. an element passing through the splash zone.

Forces and stresses at nodes are always evaluated exactly corresponding to the trapezoidal loads applied to the element.

Forces and stresses at points along the element are based on the fixed end moments corresponding to the load intensity at the element ends. If load distribution along a long element is of prime importance then additional nodes should be introduced to ensure that this approximation is within the desired accuracy.

The plotting of the moment distribution along an element will show the level of accuracy (use at least 10 plot points). If an abrupt step appears it represents the degree of error. If this is too great introduce additional nodes.

UNITS

The units used in FS-Wave are dependent on whether the model was created using **SI-Units** or **USA-Units** . The unit table shows the unit that must be used and the default values.

Dynamic Time History Analysis

See section 6.2.6 in the main Help file for more information on time history wave loading.

-o-

4.1 Unit

The following table shows the units that must be used in the FS-Wave and the default values.

FS2000 Model Definition	**SI-Units**	**US-Units**
Depth & Seabed Datum	m	ft
Wave Height	m	ft
Current velocity	m/s	ft/s
Profiles	m	ft
Densities	1027 kg/m3	64.2 Lbs/ft3
Dynamic viscosity	1.64E-3 Ns/m2	1.1E-3 Lbs/s ft
Gravity	9.81 m/s2	32.2 ft/s2
Element Dia and Perimeter	m	ins
Element areas	m2	ins2
Nodal areas	m2	ft2
Nodal volumes	m3	ft3
Marine Growth Thickness	m	ins
Marine Growth Density	1300 kg/m3	81 Lbs/ft3

-o-

5.0 Loading Properties

5.1 Buoyancy/Gravitational Loading

Positive loading due to the buoyancy of submerged elements will be included. The displaced volume is based on the following.

Circular Tubes	Volume based on OD + coating thickness
Non-Circular Tubes	Property code area + User defined internal area + coating thickness
Other sections	Property code area + User defined internal area + coating thickness

If a lining or coating thickness is defined in the extended geometric property codes of FS2000 then an effective OD and ID will be used for buoyancy evaluation, which will include the additional thickness.

To include the effect of a coating thickness a coating perimeter must be specified for non-circular sections.

The only gravitational loads accounted for are those due to content densities and the weight of coatings.

The effects of buoyancy and gravitational loading can be excluded.

More detailed descriptions are given in Section 5.2

-o-

5.2 Element Coefficients & Data

By Geometric Property Code Reference

By default all element coefficients are defined in terms of their geometric property code. The may also be define on an individual element basis.

The default coefficients to be assigned by default can be re-defined by the user. These defaults are be applied to all codes.

The use of these parameters is descibed in Section 3.3.

The parameters can be defined using the interactive form described in Section 6.4 or by command line definition described in Section 8.0.

The individual properties assigned to the property codes can be modified. Properties that are modified can be identified in the View List by the * character. Modified and non-modified properties are written to different sections of the WAV file. Non-modified properties are written to the WAV file solely to record the values being used. They are never interpreted.

The property code parameters used are:

Cd-y	Drag coefficient. in the local element y-direction.
Cd-z	Drag coefficient. in the local element z-direction.
Cm'-y	Added mass coefficient. in the local element y-direction.
Cm'-z	Added mass coefficient. in the local element z-direction.
Dy	Characteristic dimension in the local y-direction (for local z flow).
Dz	Characteristic dimension in the local z-direction (for local y flow).
Cont Dens	Contents density (takes precedence over global definition)
Cl	Pipe Lift coefficient.

For **pipe elements** the O.D. will be assigned to Dy & Dz by default and Cd & Cm1 values will be assigned 0.7 and 1.0 respectively.

For **non- pipe sections** then default values for Cd and Cm' are 2 and 1.51 respectively. These defaults can be re-defined by the user. The Dy and Dz will be assigned to 2Ly(1) and 2Lz(1). This equates to the depth and width of the section for symmetrical library sections. The reference volume use for the is describe in Section 3.3.

The other properties defined by geometric property code are

Internal Area

Coating Perimeter

The Internal Area is used to evaluate the gravitational load of any defined contents. Therefore for hollow non-circular sections ensure that the internal area is defined.

The **Coating Perimeter** is used to evaluate in water gravitational weight of marine growth on non-circular sections. It is also used for the displaced volume evaluation and the evaluation of axial drag in non-circular sections.

Box sections are identified if property GType parameter is a B (GTAB1 command). In such cases the Coating Perimeter is evaluated using (Dy + Dz)*2 and the Internal Area using (Dy -t*2)*(Cz-t*2) where t = ave t = CSA/Perimeter.

For **non-circular sections** the displaced volume is evaluated as the sum of the property CSA + **Internal Area** + **Coating Perimeter*CoatingThickness**. ***Therefore for hollow non-circular sections ensure that the internal area is defined.***

If the **Contents Density** is entered as a negative value then the element will be free flooding and the contents elevation will follow the surface elevation and and no external pressure will be applied.

IMPORTANT COMMENT

When property code referenced coefficients are modified and are saved, these values are fixed. They can only be changed by subsequent re-definition and re-save. Any future modifications to the property codes in the FS2000 model will not be change the coefficients already saved. If Property Codes data is changed in FS2000 such that the coefficients in the Hydrodynamic Model Data are invalid that the values will require to be modified. The Upgrade form Prop Codes button will do this all codes within the model.

By Element Reference

Coefficients may be defined by element reference. Element reference takes precedence over property code definition. The following properties can be defined by element reference;

Contents Density

Cd-y Drag coeff. in the local element y-dir.

Cd-z Drag coeff. in the local element z-dir.

Cd-x Drag coeff. in the local element x-dir (axial).

Cm1-y Added mass coeff. in the local element y-dir.

Cm1-z Added mass coeff. in the local element z-dir.

If the **Contents Density** is specified as a -ve value the contents elevation will follow the surface elevation i.e. be free flooding. Empty above STW for a DyNoFlex solution.

Cd by Depth Profile Definition

If the Cd-y property code coefficient is a negative value the property code drag coefficients will be disabled and the coefficients will assigned using the active profile method i.e. Cd depth profile or Reynolds Number correlation.

Nodal Coefficients

Nodal Areas are used to attract drag loading and Nodal Volumes are used to attract inertia and buoyancy loading. Three coefficients that correspond to the principle directions x, y and z may be defined for each type. Orientation of the coefficients is defined by equating them to one of the following.

- Global axis
- Reference to an existing element
- Reference to a local co-ordinate system (not applicable to a DyNoFlex solution)

-o-

6.0 Operation

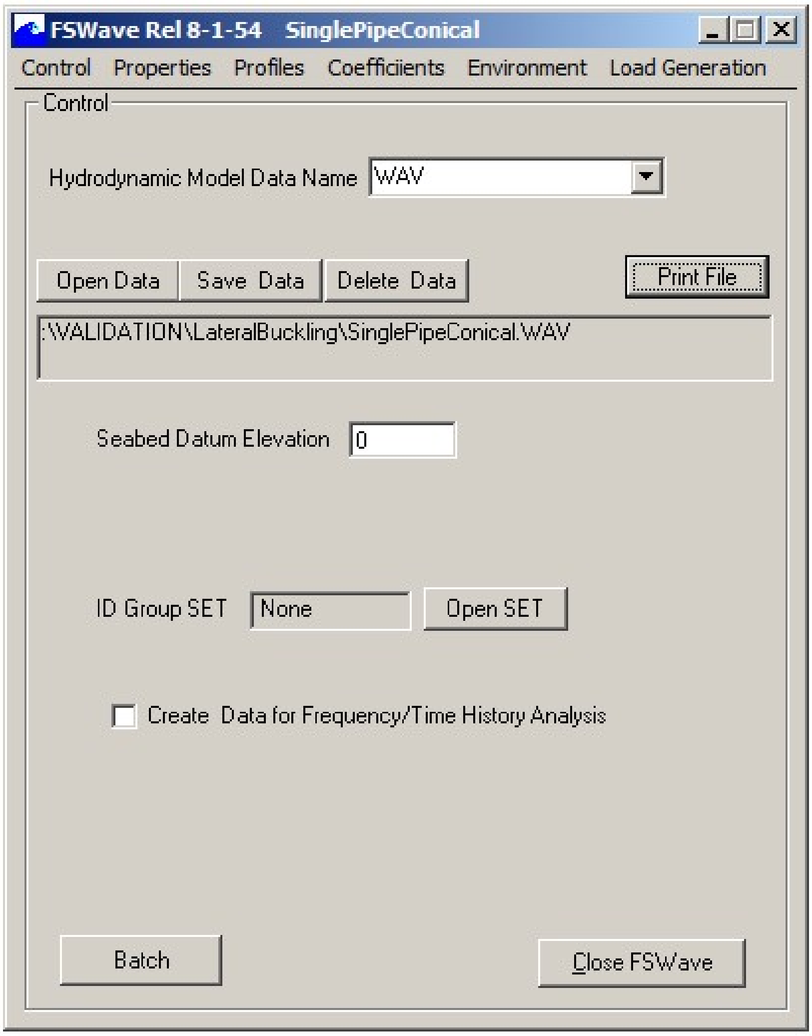
The progam would normally be run in interactive mode when data is being added.

When running in interactive mode a data entry from will appear once the program is loaded. The top menu bar used to select the any of the following sub-forms for data entry and program operation.

Control	Definition Data File management
Environment	Definition of wave and current loading
Properties	Basic property definition
Coefficients	Definition of load coefficients
Profiles	Defines load related depth profiles
Evaluation	Initiates and Controls load evaluation and load case creation

-o-

6.1 Operation - Control



The unit table in Section 4 shows the units that must be used.

Hydrodynamic Model Data

On entry the default Hydrodynamic Model Data File (WAV) will be loaded, if one exists. The text box indicates if one is loaded. I none are loaded None will displayed.

The **Open Data** button is used to open the user defined Hydrodynamic Model data file whose name is entered in Name box. Existing data can be selected from the drop down list.

The **Save Data** is used to save the Hydrodynamic Model Data currently in memory.

When saving the Hydrodynamic Model Data File it is always essential to start the name with **WAV** (WAV files are identified by the file extension .WAV).

If additional Hydrodynamic Model Data File are required e.g. using different profiles on the same model, then **WAV** must always form the first 3 characters of the name of the additional data e.g WAV_MaxMGrowth. This will ensure that all definition data is archived.

The **Seabed Datum Elevation** is used to define the position of the seabed relative to the model co-ordinate system origin using the model co-ordinate system. See Figures. If load cases are created it is important that the Hydrodynamic Model Data is saved so that a current record of the data used is recorded and is available for subsequent repeat analysis. If a water depth is defined in the Environmental form a graphical plot of the model orientation can be displayed using the View Prof Elev button.

The **ID Group SET** is used to load a Group Set that can be used to identify element for property assignment. Group identification can be used to select elements for data assignment.

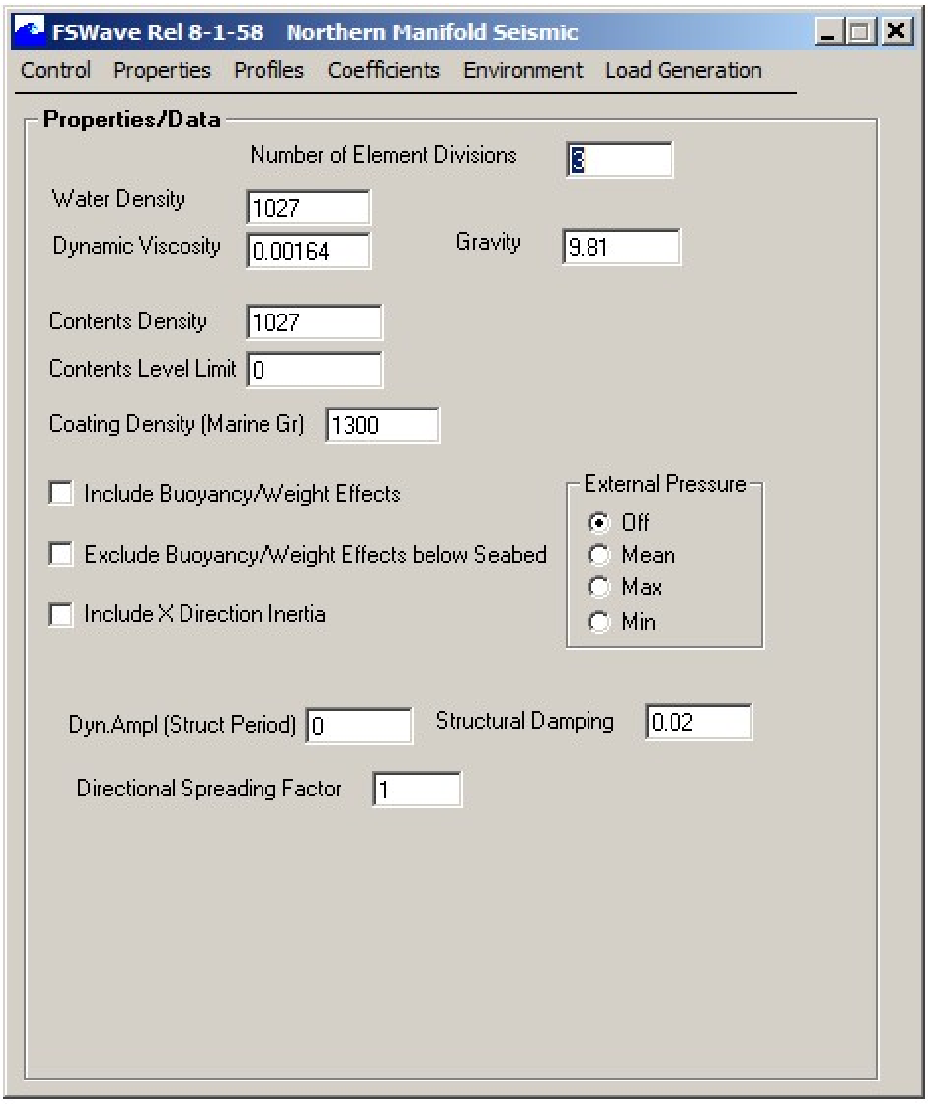
The **Batch** button is used to append the batch command **WAVELOAD WAVx** to the active BRM batch run file where WAVx is the current data file. This line is usually the first line in a batch file.

The **Create Data for Fequency/Time History Analysis** option is used to generate hydrodynamic and mass related data for time history analysis using DyNoFlex. It is also used for the Eigen frequency solution (additional mass). Note that if the additional mass is not required for an Eigen frequency solution, the <model>.WMass file should be deleted. If not active, the time history data will be deleted when the Data is is saved. The data is also deleted when the model is re-saved. See section 6.2.6 in the main Help file for more information on time history wave loading. A summary of the mass in the WMass file is included at the bottom of the WAV file.

Note that if the **Create Data for Fequency/Time History Analysis** option is active, the batch command line will be **WAVELOAD DYNWAVx.** The prefix **DYN** identifies the requirement to generate the additional data.

-o-

6.3 Basic Properties/Data



Most of the above entities are self explanatory apart from the following, which are described in more detail.

The unit table in Section 4 shows the units that must be used.

Number of Element Divisions This option allows the user to define the number of element divisions for force evaluation. The program divides the element into a series of trapezoidal loads for load application. The larger the number of divisions the greater the accuracy but the longer the computational time. The maximum number is 10. Use higher values when interest is in the splash zone.

Contents Density This is used to specify a global contents density for all hollow elements. This can be overwritten by individual contents definition. Note that this contents density is independent to the contents density in the model geometric code which is used in standard gravitational load cases.

Contents Level The contents level limit can be used to define an upper limit above which the hollow section will be assumed to have zero density. Elevation below this will be assigned the Contents Density. This limit is ignored in a DyNoFlex solution.

The elevation should be defined in terms of its position relative to SWL i.e. -10 would be 10 below SWL. The default value of 0 implies that all elements below SWL are full. If the value 1E10 is entered the contents elevation will follow the wave surface elevation for all elements ie they will be free flooding.

Individual element definition takes precedence. See 6.5 for surface elevation definition of individual elements.

Gravity is used to evaluate wave properties and buoyancy effects. Do not adjust to suit dead load requirements. Only change if units change.

Include Buoyancy/Weight Effects This is used to switch on/off the effects of buoyancy, contents weight and marine growth. Only hydrodynamic loads will be applied if this is not checked. This activation can also be applied on a wave basis in the Environment tab. .

Exclude Buoyancy/Weight Effects below seabed This is used to switch on/off the effects of buoyancy and contents weight for elements located below the seabed.

Include X Direction Inertia This is used to switch on/off the effects of axial inertia in elements due to the displaced volume.

External Pressure - The effects of external pressure can be included. This is applied as a constant pressure ever each element and the value used dependents upon the selected option (DyNoFlex always uses the mean). Pressure is only applied to empty pipe elements.

Dynamic Amplification Dynamic amplification factors based on the period of the wave and the period of the structure will be used if a non zero value is entered for the structure period. The DAF is based on the following which is the response for a viscous damped SDOF system.

DAF = 1 / [((1 - f2))2 + 4(SD.f)2]-.5

where **f** is the structure to wave frequency ratio

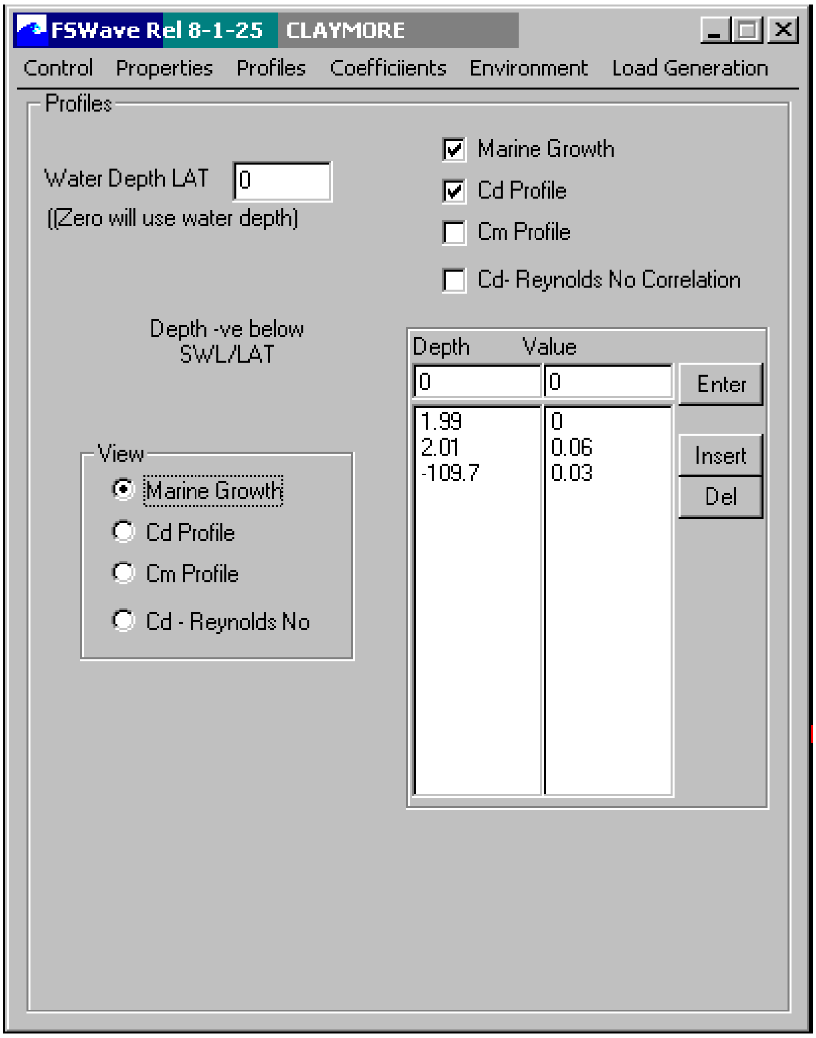
SD is the defined **Structural Damping** factor

The **Directional Factor Spreading** is used to factor the wave and current particle motions (velocities+current and accelerations). The environmental load dependent Wave Spreading and Current Blockage factors in the Environment Tab are more likely to be used for this purpose.

-o-

6.3 Profile Definition

The Profiles Tab form is used to define Growth-Depth profiles, Cd-Depth profiles and Reynolds No-Cd profiles.



The unit table in Section 4 shows the units that must be used.

The **View** option is used to select the profile to be displayed in the profile list.

The **Check** boxes at the bottom of the screen are used to select the profiles to be used for load evaluation. Only one of the CD profiles can be active at one time.

Profile data is entered using the input boxes to the left of the **Enter** button. Use the **Enter** button to add the displayed data to the profile list. The list may be edited as required using the **Insert, Del** and **Enter** buttons.

The **LAT Depth** box is used to define the depth of water at LAT. LAT is the reference elevation for the growth and Cd profiles. If a zero value is used the water depth will be used with the reference elevation being STW (Still water level).

Profile Convention

The **Depth** is entered as negative quantity when defining values below SWL . The uppermost profile point must be at the top of the list.

Up to 10 points may be used to define profiles. At least two data point must be entered.

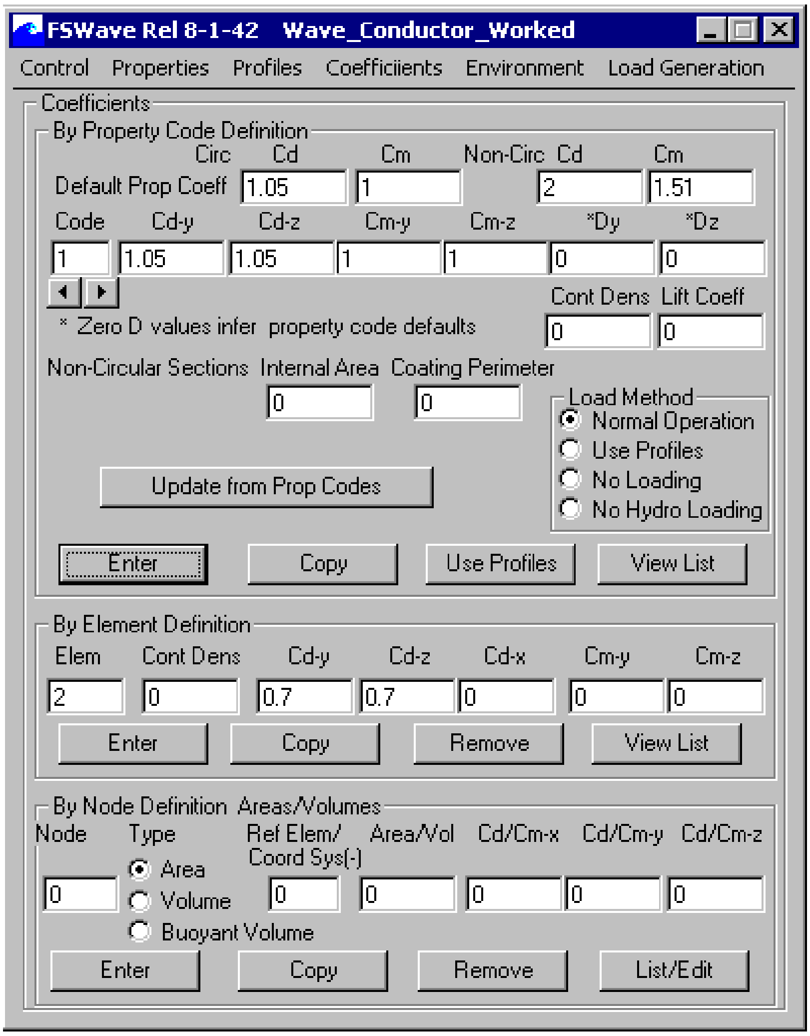
Values at elevations above or below the defined profile will take on the values of the adjacent end profile points. i.e. constant to seabed and constant to surface elevation. Hence ensure that the uppermost profile point is at the top i.e. towards the surface.

-o-

6.5 Coefficient Definition

The Coefficients input form is divided into two basic sections.

- The **By Property Code Definition** section is used to enter or edit Hydrodynamic Coefficients referenced by geometric property code.
- The **By Element Definition** is used to enter or edit Hydrodynamic Coefficients referenced by element label. Data defined by element definition always takes precedence.
- The **By Node Definition** is used attach area or volume to which hydrodynamic coefficients can be assignes to nodes.



The unit table in Section 4 shows the units that must be used.

See Section 5.1 and Section 5.2 for more details on these properties.

By Property Code Definition

IMPORTANT COMMENT

When specific property code referenced coefficients are modified and are saved, these values are fixed. They can only be changed by subsequent re-definition and re-save. Any future modifications to that specific property code in the FS2000 model will not be change the coefficients and D's already saved. If the Property Code data is changed in FS2000 such that the coefficients in the Hydrodynamic Mode Data are invalid that the values will require to be modified. The Upgrade form Prop Codes button will do this for all codes within the model. Note that this only applies to individual property codes that have been modified.

Property codes which have been modified are indicted by a * in the **View List.**

If **Dy** or **Dz** is specified as zero (the default) then these will always be assigned the default values from model property code. The **View List** will display the values used.

The **Cont Dens** is used to define contents density. Note that this contents density is independent to the contents density in the model geometric code which is used standard gravitational load cases. If the contents density is entered as a negative value then the element will be free flooding and the contents elevation will follow the surface elevation in surface piercing elements and and no external pressure will be applied.

The **Default Prop Coefficients** are the default values that will be applied to non-modified circular and non-circular sections. Note that elemnt axial inertia effect is a global setting - see Section 6.2.

When the **Code** number is entered the data may be edited. See Section 5.1 and Section 5.2 for more details on these properties.

The **Load Method** options are used to define the method by which the loading is evaluated for elements of a given property code.

Normal Operation - Used the defined coefficients

Use Profiles - The coefficients for that property code will be de-activated and the coefficients will be evaluated by the active profile method i.e. CD depth profile or Reynolds No correlation. Not applicable to DyNoFlex solutions.

No Loading - completely ignored by FS-Wave generated loading i.e. only applicable to DyNoFlex loadings. This option takes precedence over any element definition.

No Hydro Loading - Elements with this property code will shielded from hydrodynamic loading. Only buoyancy and contents weight effects will be applied. Marine growth will be ignored. This option takes precedence over any element definition.

No Buoyancy - Buoyancy and all weight affects will be ignored. This includes the effects of contents and marine growth. This option takes precedence over any element definition.

The **Enter** button is used to enter the data shown in the boxes.

The **Update from Prop Codes** button is used to initialise the whole list based on the current property codes and the current defaults. This would be used in cases where the model property codes have been changed for codes that are currently saved in the definition list. **Note**: Individual entries can be changed by editing the WAV file i.e. delete the appropriate PCOF line and then re-start FSWave.

The **Copy** button is used to copy the displayed entries to other property codes. When selected the user is required to input a code number range.

The **Use Profiles** button can also be used to select the profile coefficient method using a code number range.

The **View List** button is used to display the list of all the Code data. The list has the following button controls.

Edit This button enables selected entries to be edited

Update This button is used to update the list following data changes

By Element Definition

Zero entries will be ignored. .

The **Enter** button is used the enter the displayed data shown in the boxes.

The **Copy** button is used to copy the displayed entries to other elements. When selected the user is required to input either an element label range or an identification element group.

The **Remove** button is used to remove all entries on selected elements (by range or group)

The **View List** button is used to display the list of all the element data. The list has the following button controls.

Edit This button enables selected entries to be edited

Update This button is used to update the list following data changes

By Nodal Area/Volumes

The Nodal Areas/Volumes frame is used to define areas or volumes to be attached to nodes. An area and a volume can be attached to the same node.

The **Node** box is used to identify the recipient node. A double mouse click will enter the last node queried in FS2000.

The **Type** options are used to identify the type attachment. Areas attract drag loading and volumes attract inertia loading.

The **Ref Elem/Coord Sys(-)** is used to define the orientation of the attachment by equating the directions of the local drag and inertia coefficients to this entry.

- -1 Aligns to the global axis
- >0 Aligns to the orientation of the specified element (by element label)
- <0 Aligns to the orientation of local coordinate system (e.g. -6 is CSys 6)

The **Area/Volume** box is used to define the magnitude of the attachment type.

The **Cd/Cm-x**, **Cd/Cm-y** and **Cd/Cm-z** boxes are used to define the local coefficients appropriate to the attachment type.

The **Enter** button is used to assign the current data setting to the node.

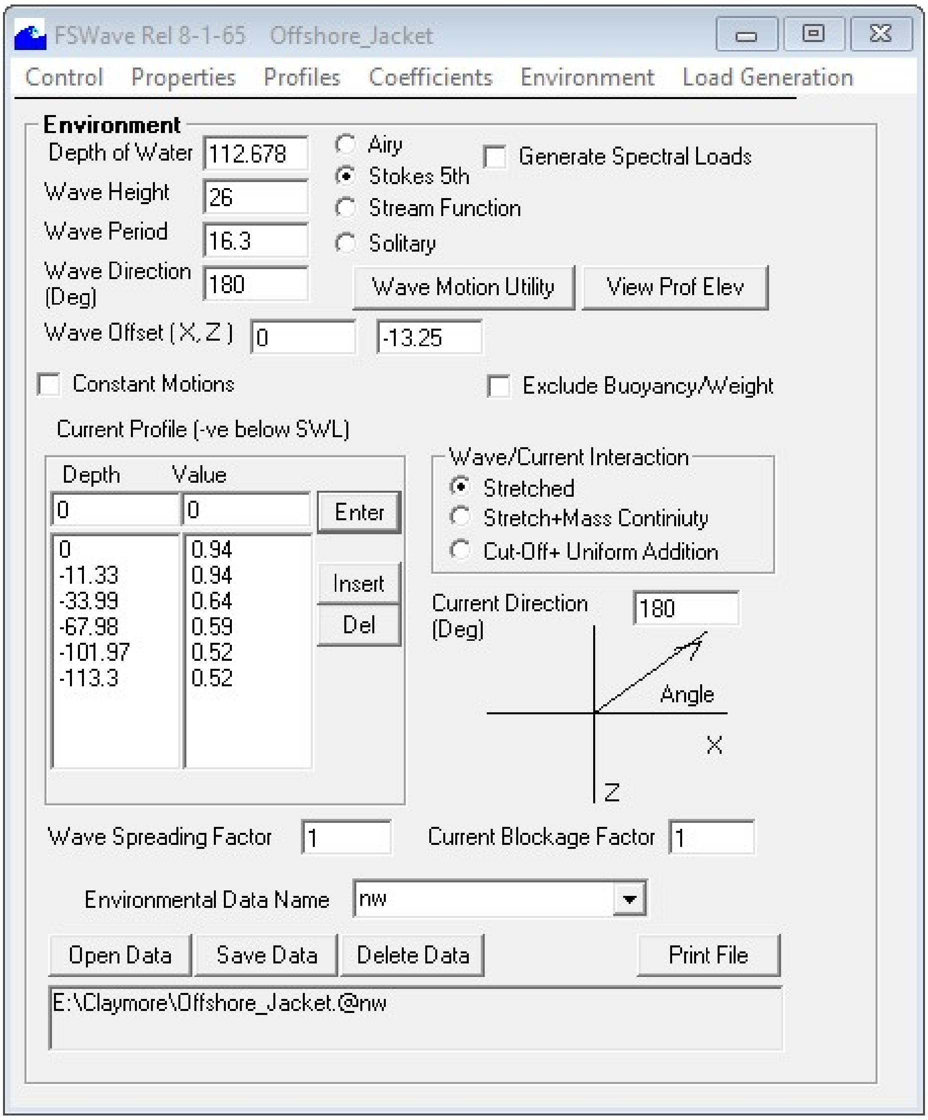
The **Copy** button is used to assign the current setting to a range of nodes that can be identified by label or group attribute.

The **Remove** button is used to delete the node attachment to a range of nodes that can be identified by label or group attribute.

The **List/Edit** button is used to show a list of all the nodal attachments. The list has Edit and Delete options

-o-

6.5 Environmental Data Input



The unit table in Section 4 shows the units that must be used.

Depth of Water This is used to define the depth of water. Water depth is always a positive quantity.

Wave Height, Wave Period and **Wave Direction** are used to define the wave parameters. Wave direction follows the convention of that shown in the figure on the form (45 deg shown)

Wave Theory. The wave motions may be based on Airy (Linear), Stokes 5 th, Stream Function Wave Theory or Solitary (see Section 3.1).

The **Wave Motion Utility** can be used to display then wave motions for the selected wave - ***very useful***.

Wave Offsets (Xoff,Yoff) are used to define the spatial position of the Start Phase relative to the "X-Z" origin of the structure. See Figures

Constant Motions If this option is active all wave motions will be made constant with respect to the spatial x and z model co-ordinates. The motions will be evaluated at a used defined wave phase (s).

Current Profile data is entered using the input boxes to the left of the **Enter** button. **Note the convention used**. Use the **Enter** button to add the displayed data to the profile list. The list may be edited as required using the **Insert, Del** and **Enter** buttons.

Exclude Buoyancy/Weight This is used to switch off the effects of buoyancy and contents weight. Only hydrodynamic loads will be applied if this is not checked. This option is not applicable to time history solutions where both type of loading are always applied unless the Include **Buoyancy/Weight Effects** in the Properties tab is not active.

Generate Spectral Loads This is used for spectral fatigue analysis but can be used for any dynamic response analysis. If active, a file containing dynamic loading commands (harmonic drag and inertia) for the modal response module will be created and called <modelname>.WSP'CaseNo'.

The **Wave Spreading Factor** is use to factor the wave particle velocities and wave particle accelerations.

The **Current Blockage Factor** is used to factor the current velocities.

Current Profile Convention

The direction convention for **Current Direction** is the same as that for wave direction.

The **Depth** is entered as negative quantity when defining values below SWL . The uppermost profile point must be at the top of the list and the list must be ordered by depth.

Up to 10 points may be used to define profiles. At least two data point must always be entered.

Values at elevations above or below the defined profile will take on the values of the adjacent end profile points. i.e. constant to seabed and constant to surface elevation. Hence ensure that the uppermost profile point is at the top i.e. towards the surface. The profile shown below would imply 1 for surface down to -10 and 0.5 from -20 to the seabed.

-10, 1

-20,0.5

Wave/Current Interaction Options - Three interaction methods are available.

Stretched Profile - The defined profile is stretched to the crest of the wave. This is the recommended (default) method

Stretched + Mass Continuity. As the above but with the velocity is adjusted to maintain mass continuity.

Cutt-Off + Uniform Addition - Profile is cut off at still water level and maintained constant at that value to the top of the wave crest.

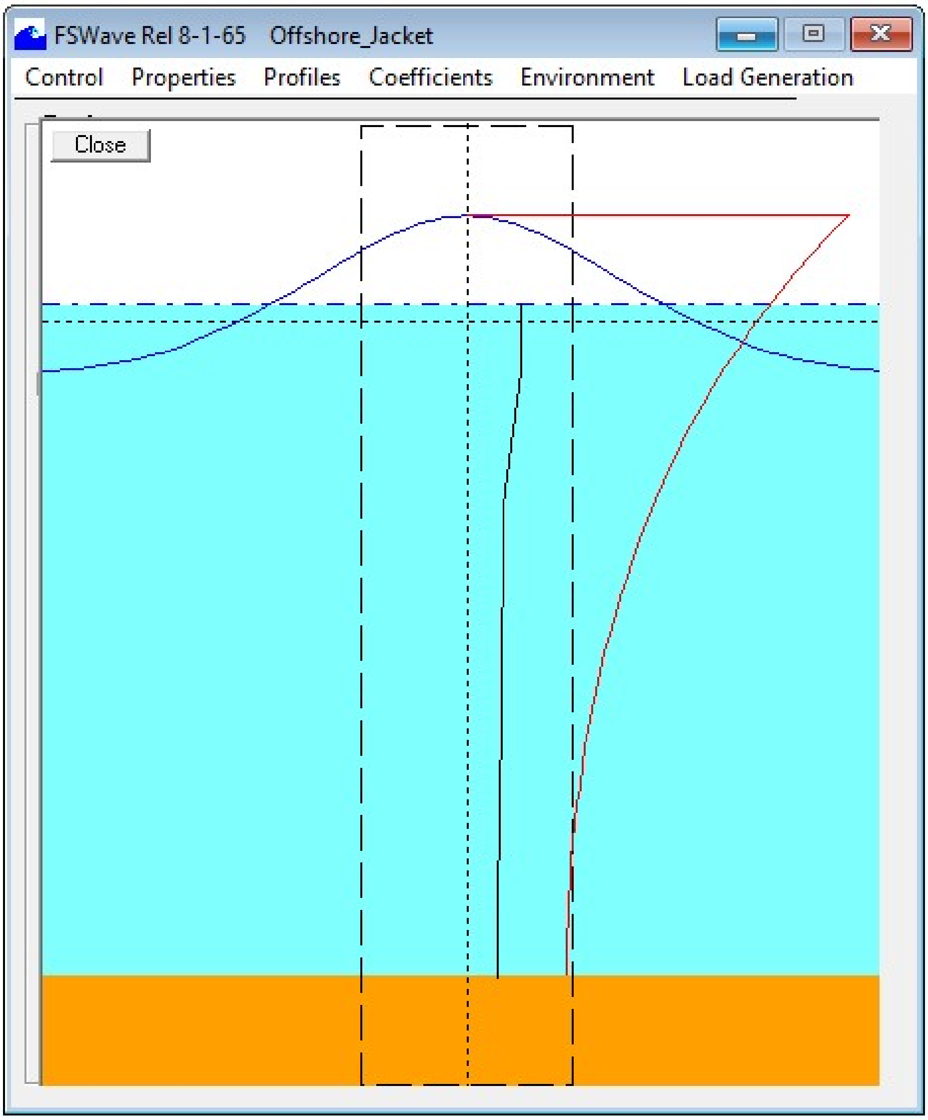
Environmental Data

The lower **Open Data** button is used to open the existing Environmental Load Data file whose name is entered in Name box. Existing data can be selected from the drop down list. The lower **Save Data** is used to save environmental data currently in memory.

Environmental data is the data shown when the Environmental Tab is selected all other data is Hydrodynamic Model related data.

To ensure that the Environmental Load Data File is archived with the model, the name must be the model name and the first character of the extension must be a @.. When entering a data name it is not necessary to include the @, it will be added by the program.

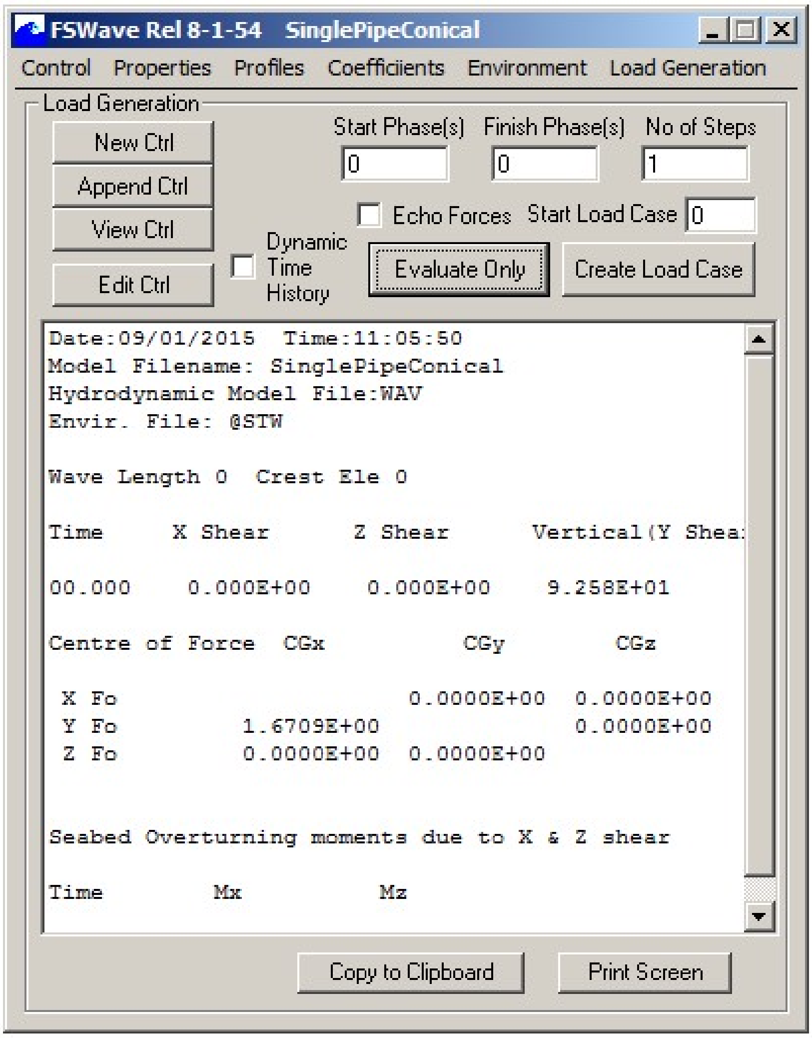
The **View Prof Elev** button enables the orientation to be displayed. The display show the elevations of the the model origin, sea bed, still water and water surface profile. The horizontal wave velocity profile and current profile are also shown and indicates that they are suitably defined. The dashed line indicates the outline of the the structure (only the vertical distances are to scale).


FSWave Rel 8-1-65 Offshore_Jacket
Control
Properties
Profiles
Coefficients
Environment
Load Generation
Close

-o-

6.6 Load Evaluation

This Tab form is used to initiate load evaluation.



Wave Step Parameters

Start Phase, Finish Phase and No of Steps are used to define the wave phase angle in terms of a time period (secs). This enables a wave is to be stepped through the structure.

Start Phase (s) is the initial phase in seconds of the wave, **Finish Phase** (f) the final phase and **No of Steps** (No) is the number of phase intervals. Load evaluation will be evaluated No+1 points.

If s = 0, f = 0 and no = 1 then the wave forces will be evaluated with the wave crest at the position defined by (Xoff, Zoff).

To evaluate forces at a given phase i.e. no stepping, always define s = f the required phase angle.

If s = 0, f = 9.375 and no = 15 then the wave forces will be evaluated at phase increments of 0.625 secs. For a 10s period wave phase increments of 22.5 deg.

Start Load Case This box is used to define the Load Case number. If more than one Load Case is to be created (e.g. a stepped wave) then the load cases will be sequential from the start case.

The **Evaluate Only** button is used to evaluate the shear loads on the structure without creating a load case. The shear loads will be displayed in output list.

If the **Echo Forces** is active, a file (**<ModelName>.WLD'Load Case No')** containing the element local distributed loads will be created. The loads are in the ED command line format.

The **Create Load Cases** is used to generate a load case using the **Start Load Case** as the load case number.

The following buttons are used for batch operation. For a more detailed description of batch control files see Command Line Operation.

The **Batch** button is used to append the FSWave command line to the **<ModelName>.BRM** batch file.

The **New Ctrl** button is used to create a new waveload batch control file **<ModelName>.BCW** using the current settings.

The **Append Ctrl** button s used to append the current settings to the existing waveload batch control file **<ModelName>.BCW .**

The **View Ctrl** button is used to view the existing waveload batch control file **<ModelName>.BCW.**

The Dynamic Time History option is used to create the **<ModelName>.UWData'Load Case No'** file. This file is used by when undertaking a DyNoFlex time history solution. This file is described in Section 6.2.6.6 of the main FS2000 Help file.

-o-

7.0 Command Line (Batch) Operation

The following describes the command line syntax that may be used to activate FSWave.

To use the command line activation it will be necessary for a ***WAVFILE*** to exist and an additional ***Batch Control File*** (Wave Control Files) called "modelname".BC **WAV** which is used to control the specific load case generation, to also exist. **WAV** is the default WAV file extension name. More that one file may be used e.g. WAV2.

The ***WAVFILE*** and any associated ***Batch Control File*** must be linked by name e.g. **WAVE2** matches BC **WAV2** and both must exist.

The command line instruction to run FSWave is

WAVELOAD *WAVFILE*

WAVFILE is used to define the Hydrodynamic Model Data File to be loaded. Only the model file extension should be used e,g. **WAV2**. NOTE that the extension name must start with the 3 characters **WAV**

If MODEL is used for ***WAVFILE*** then the <modelname>.WAV data will be loaded (legacy command).

Note that the Batch button in the FSWave form uses the current WAV file extension name when the WAVELOAD command is entered in the batch file.

The **WAVELOAD** command will start the FSWave, load the **Hydrodynamic Model Data File** and then look for the **Batch Control File** which is described below.

Creating Data for Fequency/Time History Analysis

The batch command line requires **DYN** to be prefixed to the ***WAVFILE*** *name* e.g. **WAVELOAD DYNWAV.** When this line is processed the **WAV** file will be loaded and the additional data for Fequency/Time History Analysis will be created.

Wave Control File (Batch Control File)

A control file <modelname>.BC**WAV** must exist in the same directory as the model. This text file controls the operation of FSWave when running in batch mode. The format for this file is given below.

NU number of wave file to be read
ENVNAME environmental file name e.g. WEXP.@NW better to use MODEL.@NW
S,F,NO wave step
START wave load case start number

The latter 3 entries require to entered NU times.

The **ENVNAME** can be any file specification and need not be related to the model. If FILENAME is entered as MODEL, then the default model name will be assumed (this makes the command portable i.e. model independent - which is highly desirable when copying models i.e.Save-as).

A typical <ModelName>.BCWAV is below. (MODEL is used to infer the <modelname>)

```
2
MODEL.@NW
0,8,1
10
MODEL.@SW
0,6.3,1
12
```

The **New Ctrl** and **Append Ctrl** buttons in the Load Evaluation Tab form are used to transfer the current settings to the control file.

-o-

8.0 Data File Command Formats

This section describes the command line instructions used by FS-Wave for its definition files.

Note that earlier versions of FS2000 used a fixed data file structure. These can still be read by FS2000 and will be converted to the latter format if re-saved.

The defaults for all command arguments is zero.

The Hydrodynamic Model Data File

STDRD, *Time*

Defines the structural period - used in dynamic amplification

NDIV, *Number*

Defines the number of element divisions for force evaluation. Range is 2 to 10

ZDAT, *Coordinate*

Coordinate of seabed relative to model origin

LATD, LAT Depth

Water depth at LAT. LAT is the reference elevation for the marine growth and Cd profiles.

BUOY, *Switch*

Buoyancy and contents weight effects 0 - Off 1 - On

XIN, *Switch*

X direction inertia for elements. 0 - Off 1 - On

GRAV, *Value*

Gravitational acceleration - always 9.81 unless units are changed from S.I

WPROP, *Density, Viscosity*

Seawater properties

WCONT, *Contents Density, Elevation Limit*

Properties of global contents

WINT, *Value*

Velocity components used in drag load evaluation (see Section 3.3) 0 - U.uy 1- vn.uy

CDEN, *Marine Growth Density*

Density (in-air) of marine growth

DEFC, *Cd Circ, Cm Circ, Cd Non-Circ, Cm Non-Circ*

Default coefficients applied to property code table

PCOF, *Code, Cd-y, Cd-z, Cm-y, Cm-z, Dy, Dz*

Coefficients assigned to specific geometric property codes

If Dy=0 or Dz=0 then model property code values will always be used.

PCLM, *Code, Method*

Load evaluation method - By property code

Method 1 - Use Profiles 2 - No Loading 3 - No Hydrodynamic loads

ECOF, *Elem, Contents Density, Cd-y, Cd-z, Cm-y, Cm-z*

Coefficients assigned to specific elements

NCI, *Code, Internal Area, Coating Area*

Additional properties for non-circular sections assigned to geometric property table

NA, *Node, Type, RefElem/CordSys, Area/Volume, Cdm-x, Cdm-y, Cdm-z*

Definition of nodal areas and volumes.

Type 0 - Area 1 - Volume 2 - Buoyant Volume

CDPR, *Depth, Cd*

Definition of a Cd profile point. Data points must be in order,

MGPR, *Depth, Growth Thickness*

Definition of a marine growth profile point. Data points must be in order,

REPR, *Re, Cd*

Definition of Cd as a function Reynolds Number. Data points must be in order.

SPREAD, *Factor*

This is used to factor the wave velocities, wave accelerations and current velocities.

Environmental Load Data Files

WDEPTH, *Water depth*

Water depth

WTH, *Wave theory number, Order, Damp, NPoints*

1 - Airy(Linear) 2 - Stokes 5th 3 - Stream Function

Order, Damp & NPoints are Stream Function parameters.

WAVE, *Height, Period, Direction*

Wave definition

WPOS, *X-Off, Y-Off*

Position of wave crest at zero phase angle.

CUINT, *Interaction Method*

Stretched profile etc

CUDIR, *Current direction*

Current direction

CU, *Depth, Value*

Defined current as a function of depth. Data points must be in order

SPECTL, *Value*

This is used for spectral fatigue analysis.

If this is set to 1, a file containing dynamic loading commands (harmonic drag and inertia) for the modal responses module will be created. <modelname>.WSP'CaseNo'

WASF, *Factor*

This is used to factor the wave velocities and wave accelerations.

WUBF, *Factor*

This is used to factor the current velocities.

-o-

9.0 Archiving Wave Related Data

The archive facility of FS2000 will automatically archive the following definition files which are the default naming conventions when using the program..

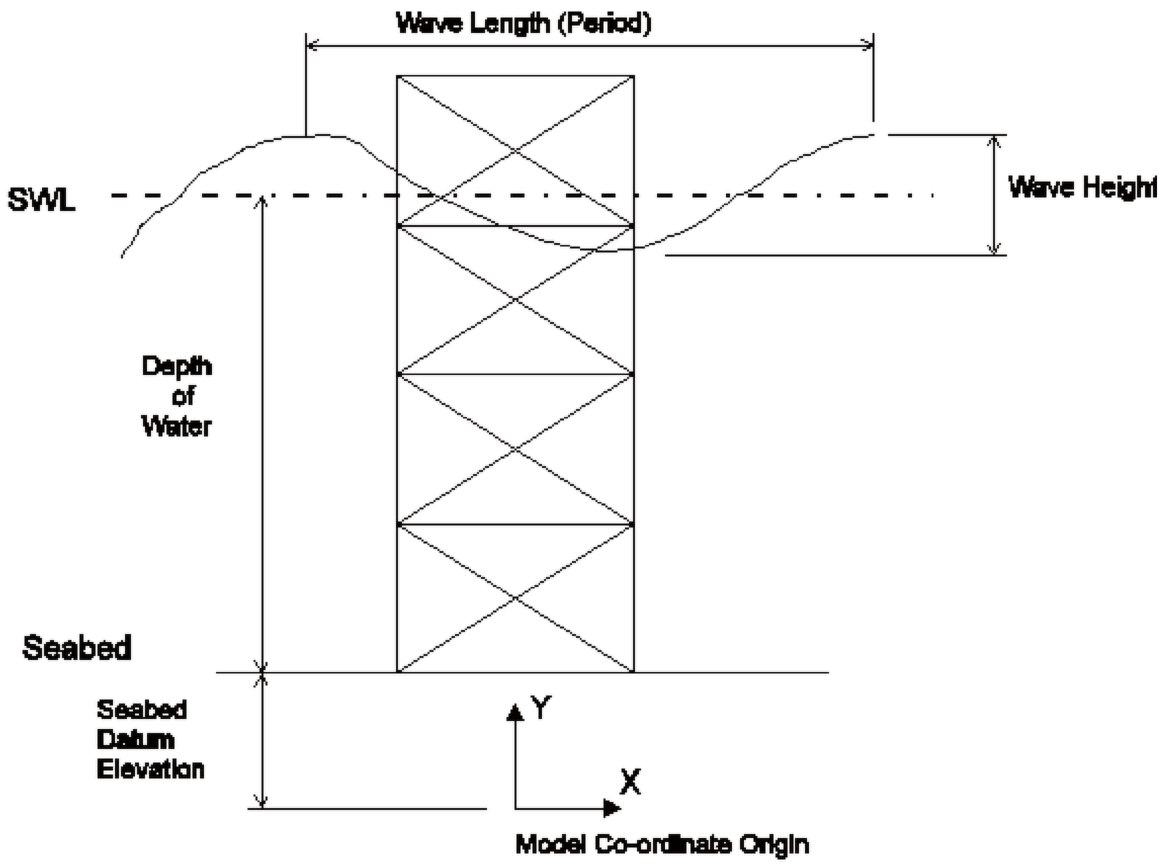
Hydrodynamic Mode Data files i.e. the **.WAV** files with the same file name as the model.

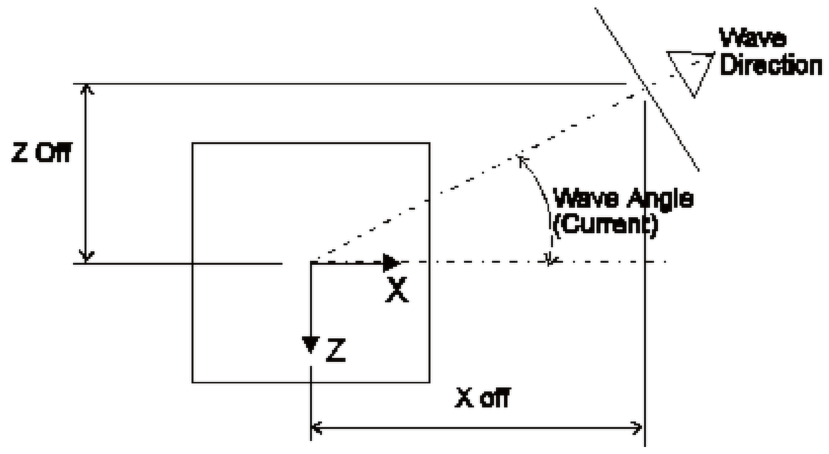
Environmental Load Data files with the same file name as the model and the first character of the extension is a **'@'** character.

If additional Hydrodynamic Mode Data files are required to be used use **<ModelName>.WAV*** since this type of file will also be archived.

-o-

10.0 Figures





-o-