HYBER
User Manual

Version 2.2

HYBER is developed by Norsk Hydro, Sintef and Ceetron. The development is paid by the Norwegian Deepwater Programme, "The Riser and Mooring Project", (1998-2003).

The 3D graphics functionality of HYBER is based on GLview API by Ceetron. The Qwt library is used for plot functionality. Qwt is released under the GNU Lesser General Public License (http://www.gnu.org/copyleft/lesser.html).

This product uses some of the PuTTY executables, both directly and in modified versions, to provide SSH, SFTP, Secure Copy, and Telnet functionality. We are obligated to include the license text for PuTTY:

This document is valid for HYBER version 2.2.

Contact information:

Kjell Herfjord, (Kjell.Herfjord@hydro.com)

Tore Holmås, (Tore@Holmas.com)

Trondheim, Norway, March 2006

Table of Contents

1 Introduction

This document is the user manual for HYBER. Thus the document specifies the graphical user interface and the functionality of HYBER. First there is a chapter called “Getting Started”, which describes how to use HYBER step by step. This is useful if you are new to HYBER since it helps you get up and running quickly. The major part of this user manual is a walkthrough of the elements in the graphical user interface in an ordered manner, and the functionality is described along the way. In addition there is a chapter that describes the file formats used by HYBER.

2 Getting Started

Introduction

After a normal installation, all necessary information is stored on the computer, (typically in file folder: C:/Program Files/hyber)

The computer tool HYBER contains following modules:

- Model generation/analysis set up tool, ("pre-processor")
- Simulator (the "engine")
- Result inspection tool, ("post-processor")

In addition, documentation (this manual), and some examples are available, and could be accessed from HYBER.

HYBER is activated from the start menu or by double clicking on the icon, (if a short cut is created).

Except for the simulation module "the engine", which also could be run on a remote LINUX server, the tool is running on standard PCs.

Simple Collision Check

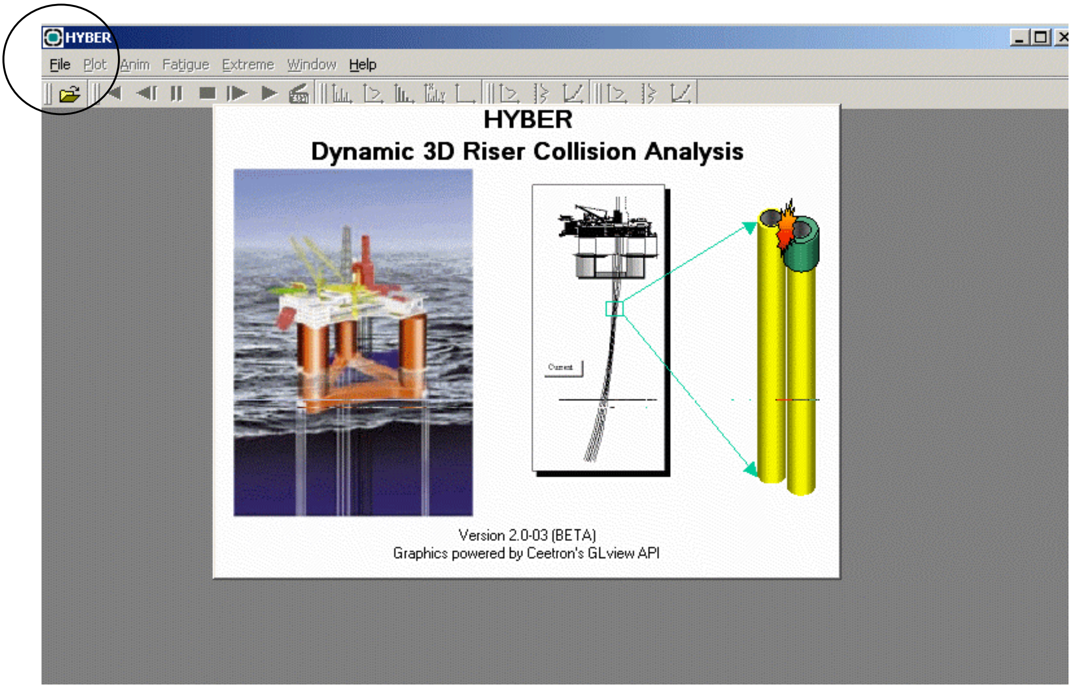
A step by step description on how to use HYBER will be given below. The case is a simple riser collision check, and the question to be answered is:

Will the two actual risers collide for the actual current condition?

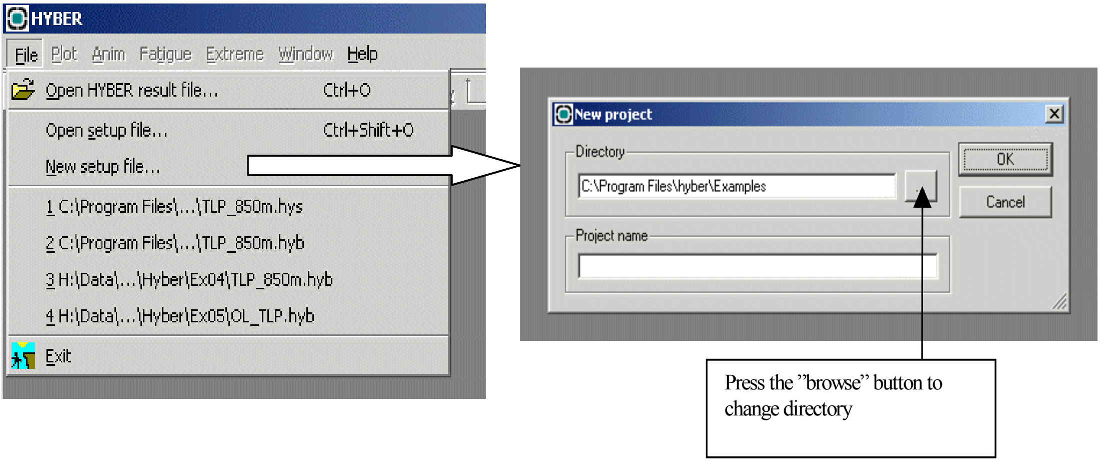
<u>Data:</u>

- Water depth : 300 m
- Spacing: top: 3m, bottom: 4m
- Cross Section: Outer: Do=0.25m, T=14mm, Inner: d=0.18, t=10mm
- Internal Fluid: Annulus: 1100, inner tubing 600 kg/m3
- TopTensionFactor: 1.5
- Spring Factor: 0.4
- Current: Surface: 1.5 m/s, from z=-100, gradually reduced to 0.5m/s
- No floater motion

HYBER opens as shown in the figure below. Select File and "New setup file..." when starting from scratch, (having no set up files to import and edit).



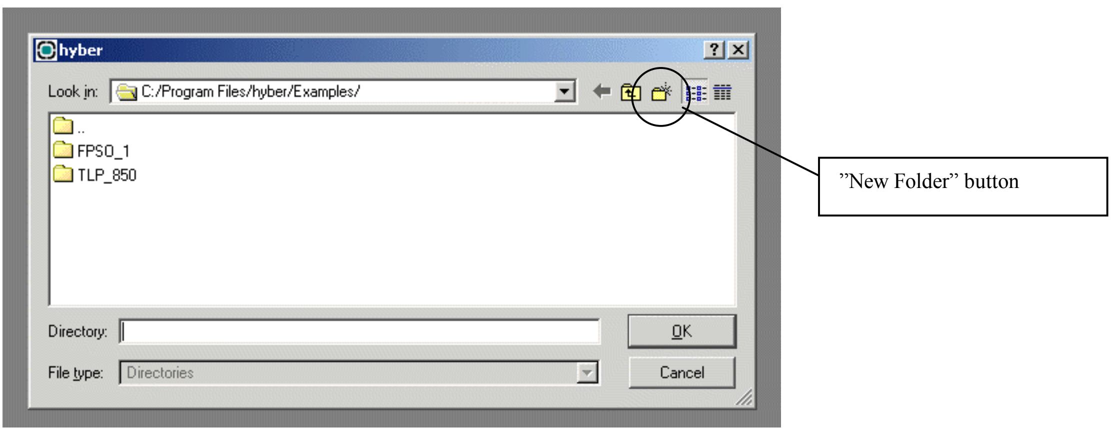
Hyber opening scene.



File dialog, selecting new set up file.

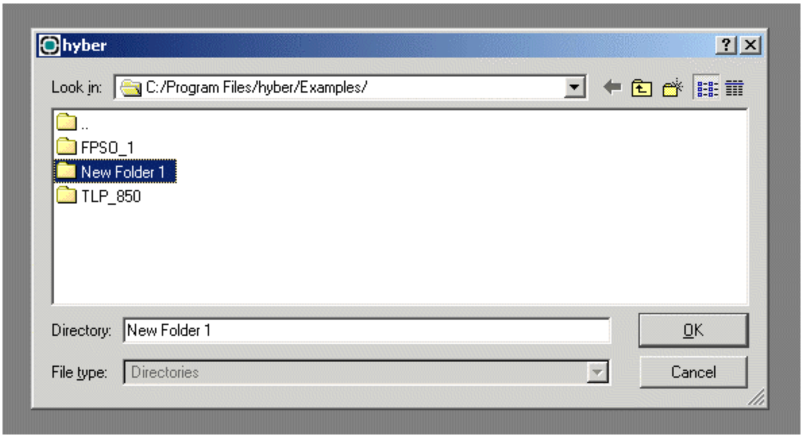
By default, HYBER will start in the “examples” file folder, and in the present example, the data files produced should be collected in a separate folder, which, in this case, does not exist in advance.

Pressing the “browse” button, gives a new dialog box , see the figure below.

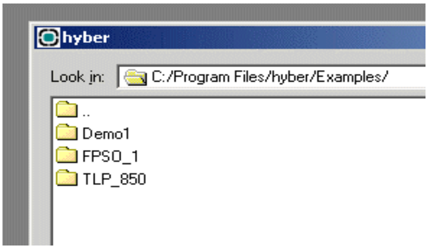


File dialog. "Browsing" for a folder to store the example files.

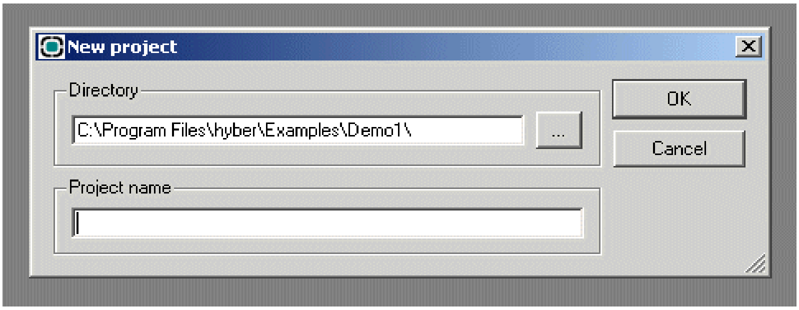
The folders in current directory are shown, and by pressing the “New folder” button, the field “New Folder” appears. Type in the actual folder name, (in this example, “Demo1”), and the new folder is created and available, (click in the white field). Double-click on this new folder, (Demo1), and define a project ***name*** to be used in this new folder, see the following figures.



File dialog. Typing the folder name.

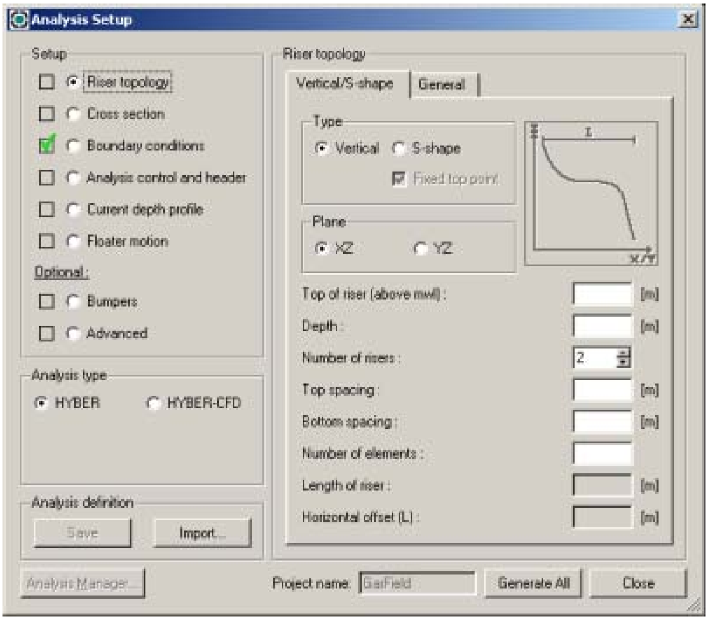


File dialog. New folder “Demo1” is created.

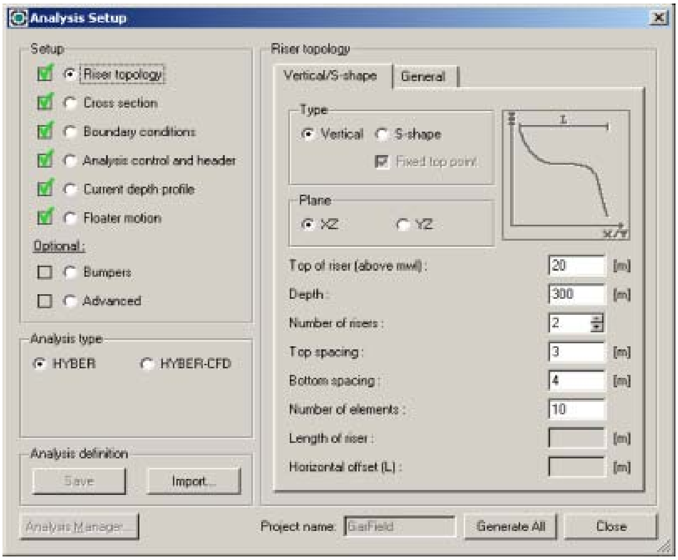


File dialog. Define project name to be used in the new folder.

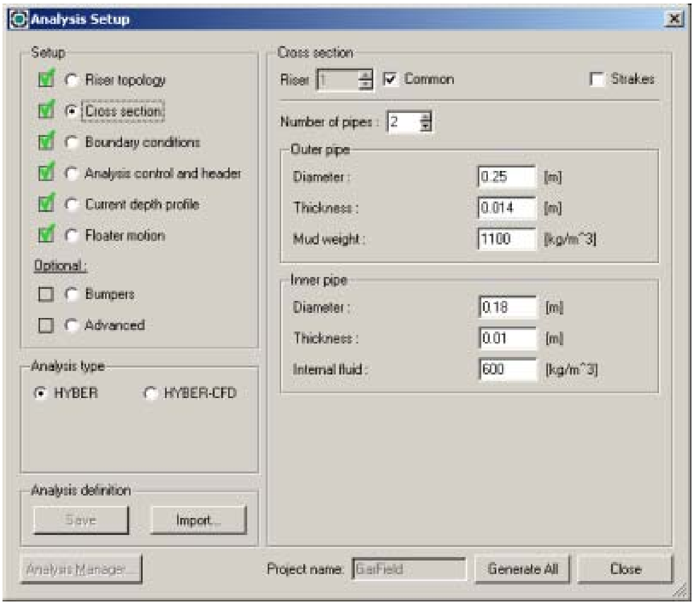
The project name "GarField" is defined as a project-name, and this name will prefix all files produced by HYBER. Press "OK", and the empty "Analysis Setup" sheet appears. In our case, no set up data exist in advance, (if old sheets were available, the "Import" function could have been used), and all data has to be typed in.



Empty Analysis Setup sheet.



Riser topology sheet completed.

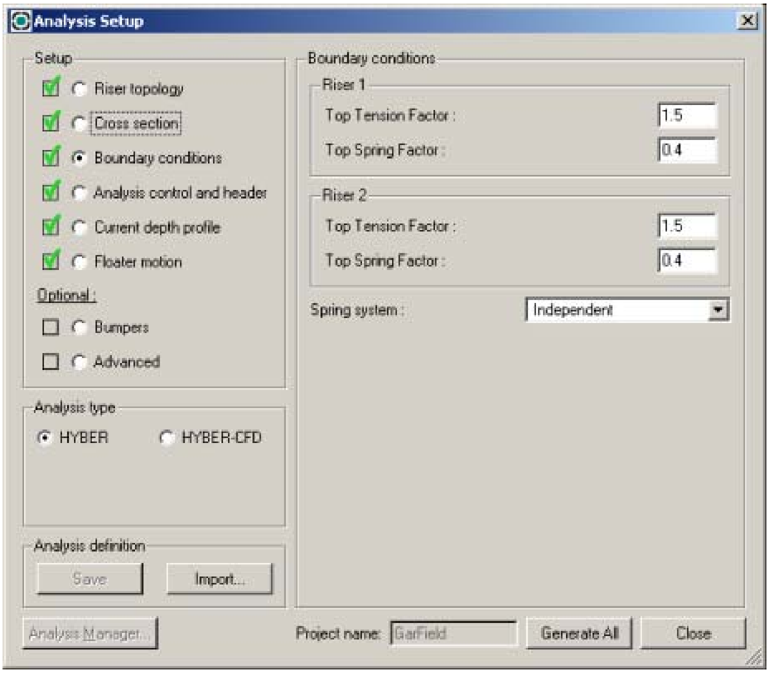


Cross section sheet completed.

Type in actual topology data. In the case, the top point of the riser is located 20m above the water surface. When the sheet is complete, a green tag appears for the particular sheet.7

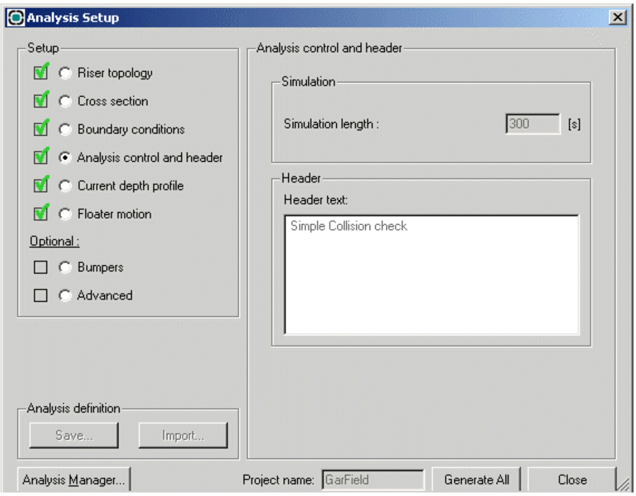
The cross sectional data for the pipe with inner tubing is specified in the "Cross Section" sheet.

The boundary conditions at the riser top in terms of top tension, (the Top Tension Factor), and the top spring stiffness, **K** = (Top Tension) x (top spring factor).



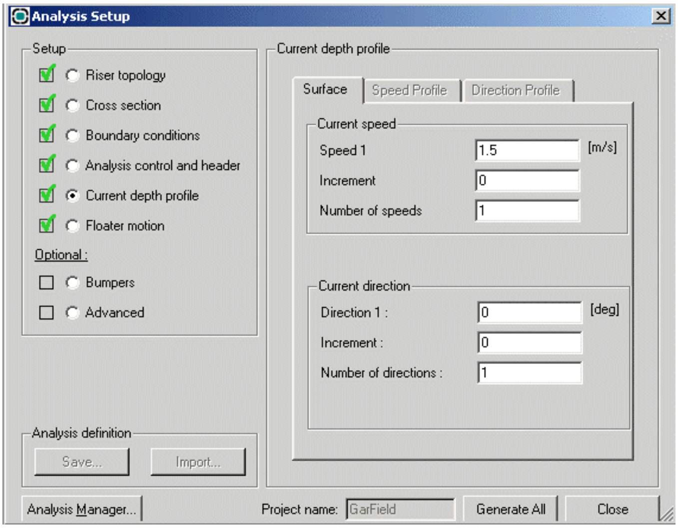
Boundary condition sheet completed.

The total simulation length is specified in the next sheet, and the heading text, (which will appear on plots etc), has to be typed into the Header field.



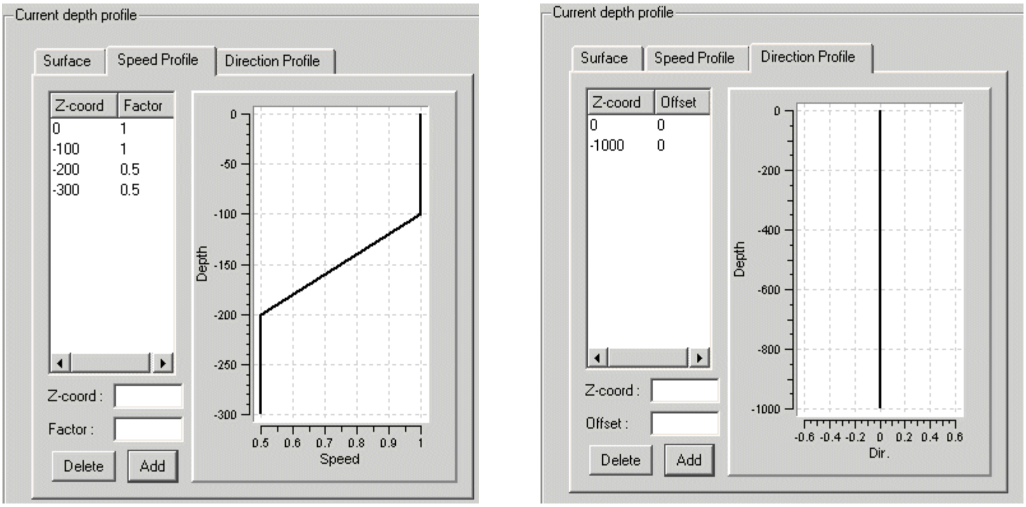
Analysis control sheet completed.

The current surface speed and direction is specified in the "Current depth profile" sheet (see the following figure). A series of speeds and directions could be specified in once, but in the present example, only one speed and direction is defined. The depth variation of the speed, as well as depth variation of the current direction, have to be specified.



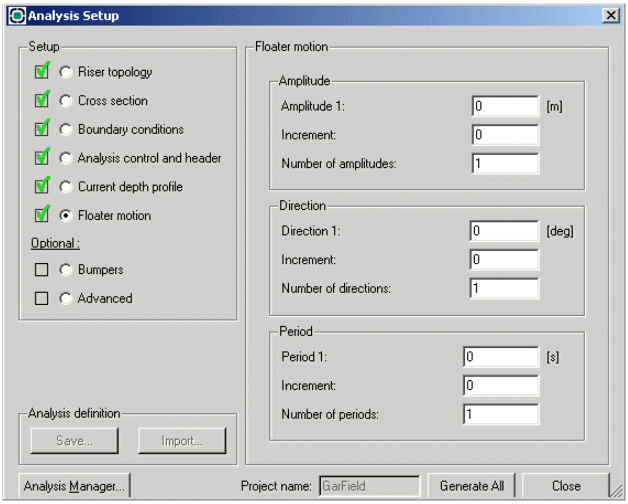
Current Depth Profile sheet completed.

NOTE that the actual current speed is the *product* of the surface speed and the value of the depth profile for the various depths. Similar for the current direction, where the direction of the current at a given depth is the surface direction *plus* the direction offset at the actual depth. In the present example, same current direction is used for all depths, (offset = 0).



Selected speed profile and direction offset profile.

There is no floater motion, and Amplitude1, Direction1 and Period1 are all set to 0.

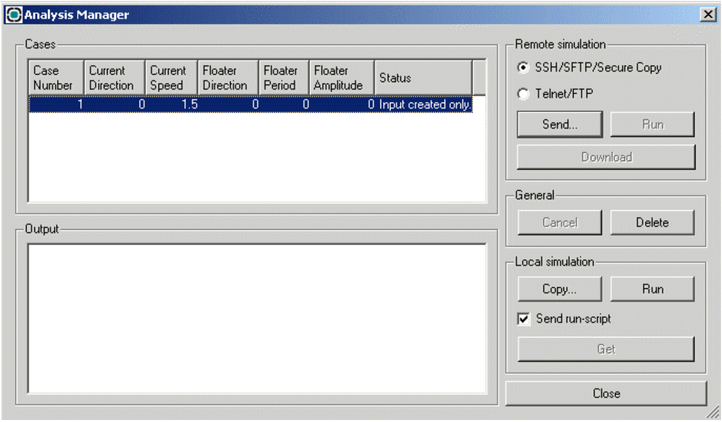


Floater motion sheet completed.

All input sheets are completed, and press "Save" to save the sheet parameters to a separate file, (which will be named GarField.hys automatically). This ".hys" file, (HYBER set-up file), could be "imported" and modified in connection with a later project. (It is also possible to save an incomplete work sheet set-up, for later completion.)

When all required sheets are defined, the "Generate All" button becomes active, and pressing this button will produce all necessary input to the simulation "engine" collected in one file per case. Since only one case is specified, only one file will be created, (GarField_001.fem).

Press the "Analysis Manager" button and the "Analysis Manager" dialog box appears.

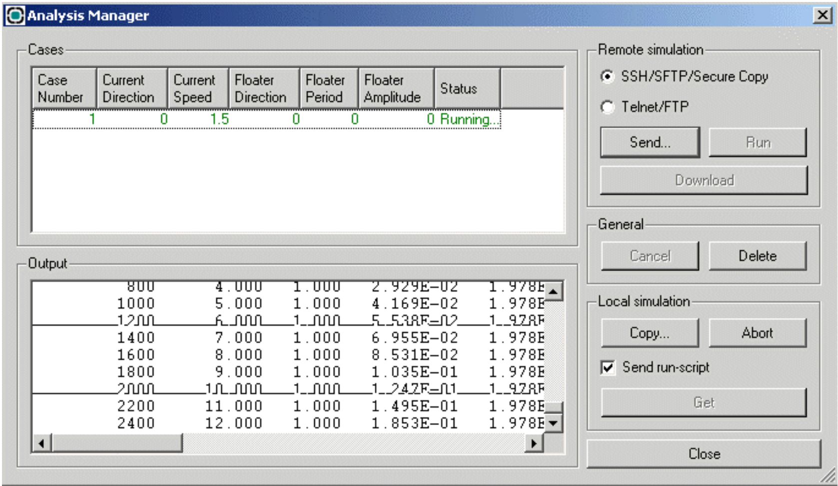


Analysis Manager dialog box.

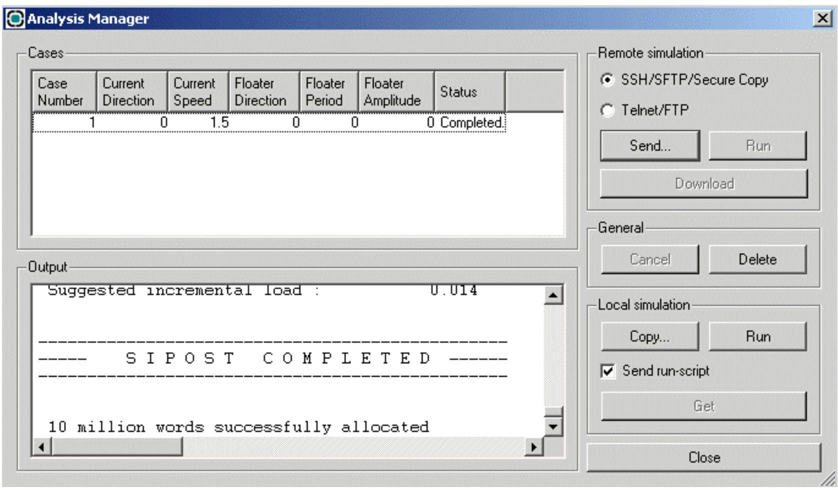
Select actual case(s), and how to further process the data. Two main alternatives exist:

- Sending data to a remote LINUX server for "batch" processing
- Running the simulation on the local PC.

Since the actual example is small, the "Local" alternative is selected, and after the "Run" button is pressed, some simulation key information is shown in the "output" field, see the following figure. When the simulation is completed, the Status changes from "running" to "completed", see the two following figures.



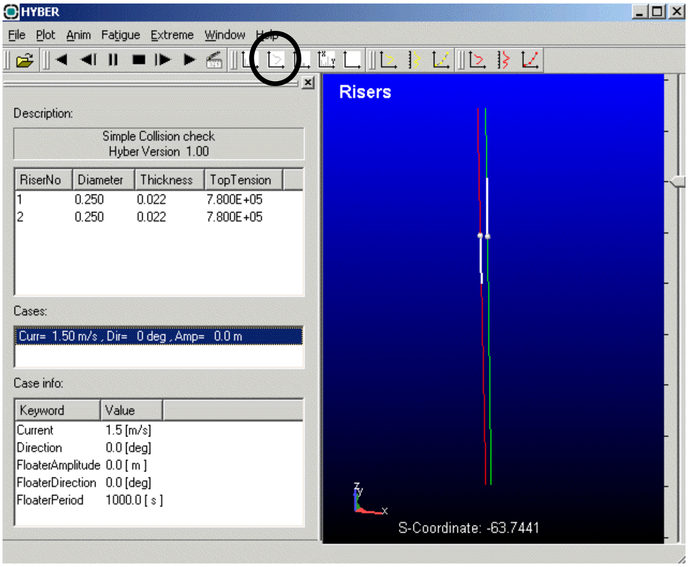
Running the simulation locally.



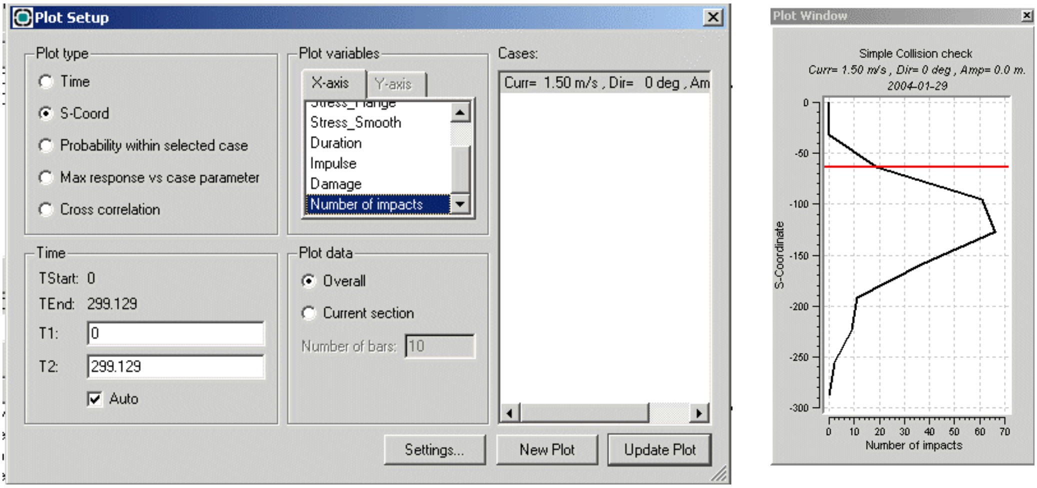
Simulation completed.

Now, the results are ready for inspection, and by selecting the File/Open HYBER result file from the HYBER menu, and selecting the file "GarField.hyb" the results from the simulation case(s) become available.

The question to be answered in this example was "collision or not". Selecting the "s-coordinate plot", (indicated button on the task bar), gives the "Plot Setup". Selection of "number of impacts" gives an overview of the number of hits along the riser length.



HYBER results. Selecting "S-coordinate plot".



Plot setup and selected "Number of Impacts".

As seen from the plot:

<u>"With the actual configuration, the risers are likely to clash"</u>

3 Graphical User Interface

Introduction

This chapter describes the graphical user interface and the functionality of HYBER. It is a walkthrough of the contents of the graphical user interface, which includes the main window with the docking windows it may contain, the menus in the menu bar, the toolbar, and a number of dialogs that are available through menu items.

The Main Window and its Contents

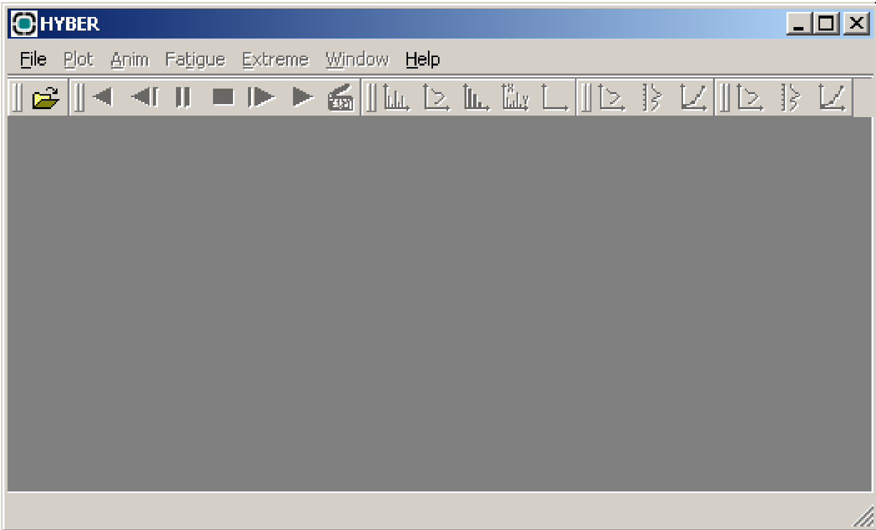
This section describes the contents of the main window.

Main Window

Initially, when no project is opened, the main window is empty, showing only a gray background along with a menu bar and a toolbar. As long as no project is opened, the only available actions are to open a project file, to view the "about box", or to exit the application.

When a project file is opened, project information will appear in a project window that is docked inside the main window. The geometry of the risers will be presented in a risers view that fills the area of the main window that is not covered by docking windows.

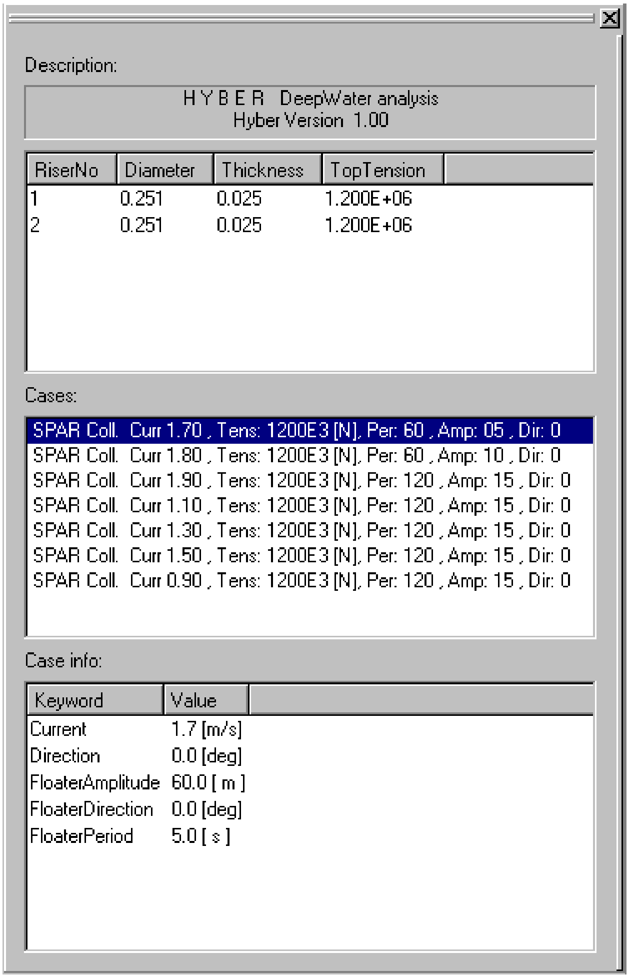
The main window looks as illustrated by the figure below.



Project Window

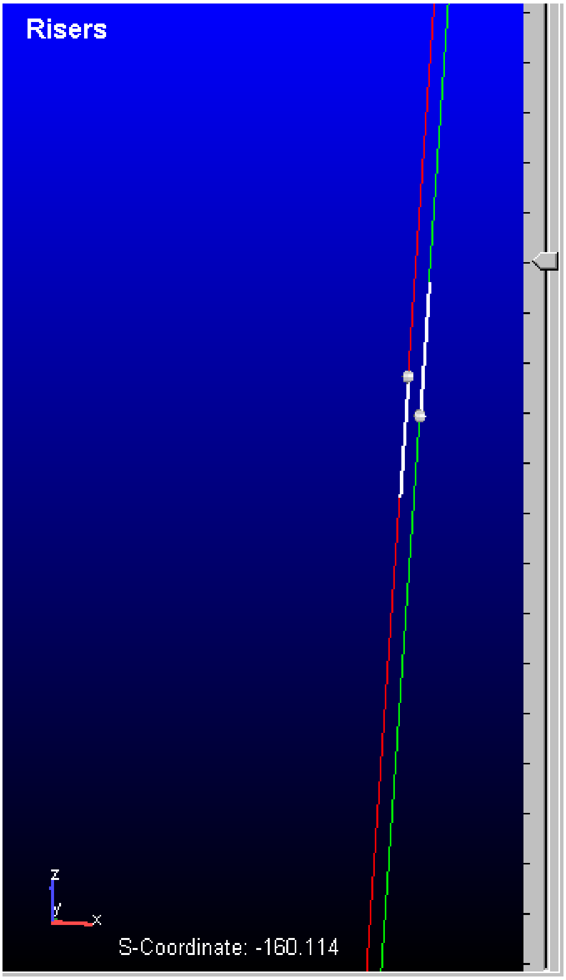
When a HYBER Project File is opened, a docking window will appear inside the main window. This is the project window, and it contains project description, parameters for the risers, and case information. Case descriptions are inserted in a list box. Selection of a case in this list box results in presentation of characteristics of a given case, i.e. data about current and floater.

The following figure shows an example of how the project window may look.



Risers View

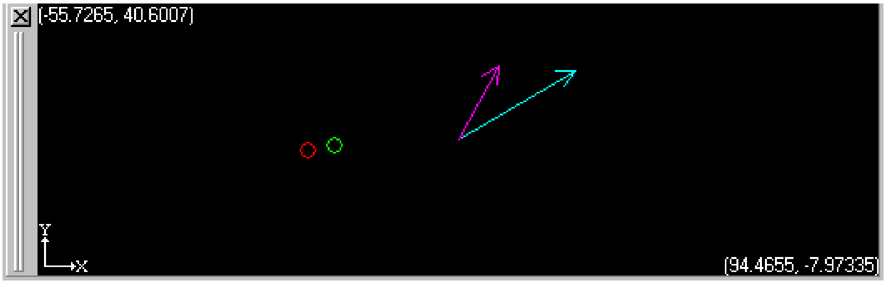
The risers view looks as illustrated by the figure below.



The risers view is a 3D graphics window that presents the geometry of the risers. A slider control is placed on the right side of the window. This slider control selects a section of the risers, and the current section is indicated in the 3D graphics window. To be more specific, the slider control selects an element in the red riser, and the first node in the selected element. The node and the element in the green riser which are closest to the selected node in the red riser will also be selected. The S-coordinate of the current section is written in the bottom of the view. An axis cross is located in the bottom left corner, to indicate the orientation of the view. Navigation inside the view is done with the mouse. Holding down the left mouse button and moving the mouse gives panning, the right button gives rotations, while the middle button (or left and right button simultaneously) gives zooming. When an animation is played, the risers will move according to the displacements, and the frame number and the time number are shown instead of the S-coordinate.

Animation View

When an animation is started, the animation view appears inside a docking window, and it looks as illustrated by the example in the figure below.



The animation consists of two circles that illustrate the sections. They have the same colors as the risers in the riser view, of course, and they move according to the displacement history. The animation window's rectangle is based on the bounding rectangle of all of the movements of the sections. To avoid wrong scaling in either the vertical or the horizontal direction, the direction that has the largest maximum displacements controls the scaling. This means that the area covered by the other direction changes according to the size of the “master direction”. As can be seen in the figure, the dimensions of the rectangle are shown by coordinates in the upper left and bottom right corners. An axis cross is located in the bottom left corner to indicate the axis that is X and the one that is Y. If specified, the current direction is shown as a light blue arrow, while the floater direction is shown as a purple arrow.

A context menu from which the animation view may be customized will appear if the right mouse button is clicked.

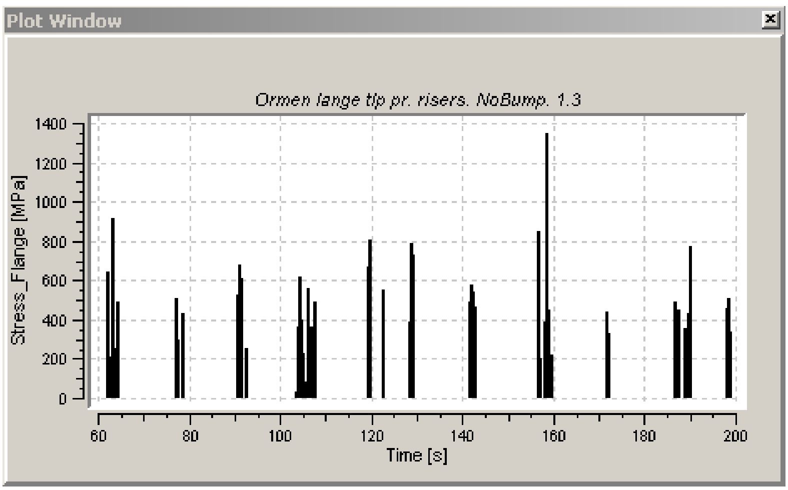
By default, the real movements (absolute displacements) are shown. If desired, the relative displacements can be shown instead. A checkable item in the context menu turns on/off relative displacements mode. In this mode, the red circle is kept still, while the green circle moves according to the relative displacements.

The diameters of the circles are based on the real diameters of the risers by default. However, the diameters may be so small in relation to the area in which the sections move that they almost become invisible. A pop-up menu inside the context menu called “Riser diameters” can be used to select a specific diameter (in pixels), or to use real diameters.

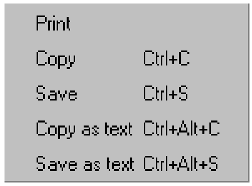
The visibilities of the current direction, the floater direction, the axis cross, and the dimensions of the rectangle, can be turned on or off through checkable menu items in the context menu.

Plot Window

When a plot is generated, either by plotting contact results or by computing fatigue or extreme estimates (described later in this manual), the plot will appear inside a docking window in the main window. A plot window may look as illustrated by the figure below.



The plot window has a context menu (whichs pops up when the right mouse button is clicked) that makes it possible to export the plot itself or the plot data. This menu looks as shown in the figure below.



“Print” prints the plot. A standard print dialog appears. “Copy” copies the plot as a bitmap to the clipboard. This is equivalent to pressing Ctrl+C when the plot window is active. “Save” brings up a file dialog where the plot may be saved to file. The plot may be saved as “.bmp”. “.png”, “.xbm”, “.xpm”, or “.pnm”. Pressing Ctrl+S is equivalent to selecting this menu item. The plot data may also be saved as text to a file or copied as text to the clipboard. This text contists of the plot title and the axis titles, followed by (x, y) data organized in two columns where the columns are separated by tabulators and the rows are separated by line breaks. These menu items ar called “Save as text” and “Copy as text”, with the keyboard shortcuts “Ctrl+Alt+S” and “Ctrl+Alt+C”, respectively.

The File Menu

This section describes the items in the file menu.

Open HYBER Result file...

Selection of this menu item opens a file dialog where HYBER Project Files (*.hyb) may be opened. When a project file is opened, the directory where this file is located will be used as current working directory. It is assumed that HYBER Case Files (*.hc1), VTF files (*.vtf), HYBER Damage Curve Files (*.hcu), and HYBER Damage Distribution Files (*.hdd) related to the project are located in the same directory, i.e. the current working directory.

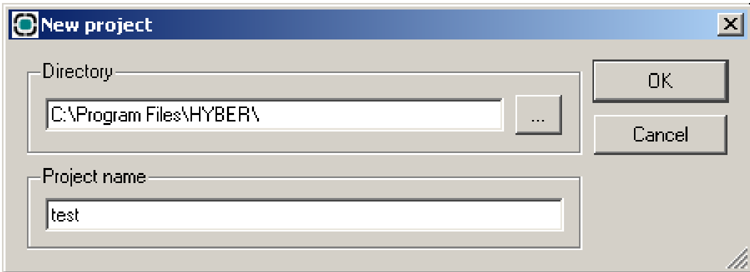
If the project file is opened successfully, HYBER searches for HYBER Case Files (*.hc1). All case files found in the working directory are read, and case information, along with project information, is presented in the project window as described earlier in this chapter. The first case is selected by default, and information for this case is shown.

Open setup file...

Selection of this menu item opens a file dialog where HYBER Setup Files (*.hys) may be opened. When a project file is opened, the Analysis Setup Dialog is opened and its controls are filled with information from the setup file. See section “The Analysis Setup Dialog” for more information.

New setup file...

Selection of this menu item opens a dialog where a directory and a project name may be specified. The figure below shows what this dialog looks like.



This dialog is used to specify a new HYBER Setup File (*.hys). When a directory and a project name have been specified, a file named “<project name>.hys” is created in the given directory. Note that you may browse for a directory by clicking the browse button ([...]), and the directory browser also lets you create a new directory. Once the setup file has been created, an empty Analysis Setup Dialog is opened and you may start specifying parameters for a new analysis. See section “The Analysis Setup Dialog” for more information.

Recent Files

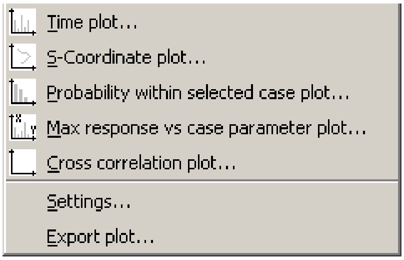
The 4 most recently used project files or setup files are available through menu items below the “Open...” menu item. Selection of one of those is equivalent to selecting the corresponding file in a file open dialog. The list of recent files is empty until the first time a file is opened.

Exit

Exits the application.

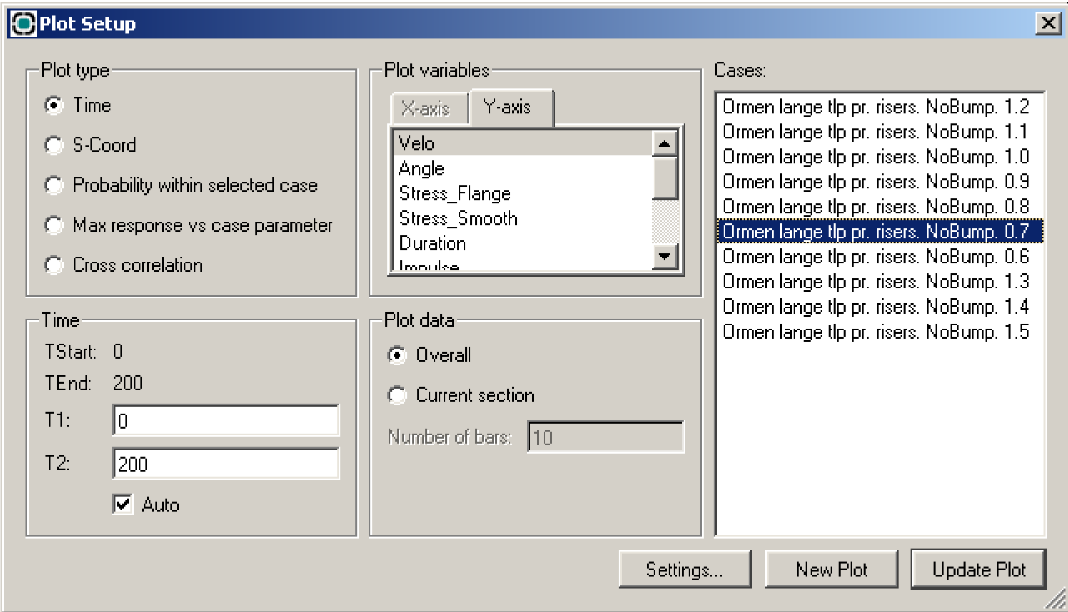
The Plot Menu

This section describes the items in the plot menu.



Time plot...

This menu item opens the plot setup dialog, which looks as illustrated by the following figure.



All of the available plot variables for the selected case(s) (selected in the list to the right) are listed in the "X-axis" and/or the "Y-axis" tab(s).. There are 5 radio buttons for selection of plot type. This is the place in which to select whether to plot over time, to plot over S-coordinates, to create a probability distribution, to create a max response vs case parameter plots, or to create a cross correlation plot.

In the lower left corner right the start/end times (T1 and T2, respectively) for plots may be set. This enables selection of a specific time interval for which to get data for plotting. The minimum value of T1 is "TStart", and the maximum value of T2 is "TEnd". These values are default. T1 cannot be larger than T2.

Time is selected by default. If "Time" is selected, plotting of values will be done over time. To be more specific, values for the selected variable are plotted at points in time where there have been hits.

If "S-Coord" is selected, plotting of values is over the entire length of the risers. The maximum value will be plotted if there have been several hits at the same S-coordinate. The variable "Number of impacts" is always present for S-coordinate plotting, and is generated automatically. This is simply a count of the number of impacts at specific elements, represented by S-coordinates.

If "Probability within selected case" is selected, the "Number of bars:" edit control is activated. This type of plot is a histogram where there is a number of bars as specified, where the height of each bar is the probable relative number of hits resulting in values within the interval covered by the width of the bar.

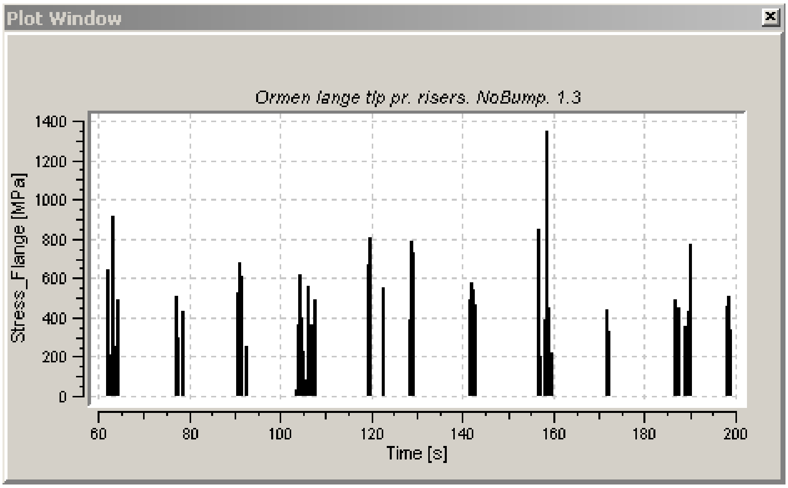
If “Max response vs case parameter” is selected, both the X-axis tab and the Y-axis tab are activated. The case parameter to plot along the X-axis may be selected in the X-axis tab and the variable to plot along the Y-axis may be selected in the Y-axis tab. The maxium value of the Y-axis variable is plotted for each case parameter value. One or more cases may be selected in the list of cases.

If “Cross correlation” is selected, both the X-axis tab and the Y-axis tab are activated. This is simply a general cross correlation plot where any X-axis variable may be plotted against any Y-axis variable. One or more cases may be selected in the list of cases.

Inside the "Plot data" frame in the dialog there are two radio buttons for choice of an "overall" plot or a plot for the current section only. If "Overall" is selected, all values are plotted, no matter where the corresponding collisions have occurred along the risers. If "Current section" is selected, only values resulting from hits for the current section are plotted. Values resulting from all other hits are counted as 0.

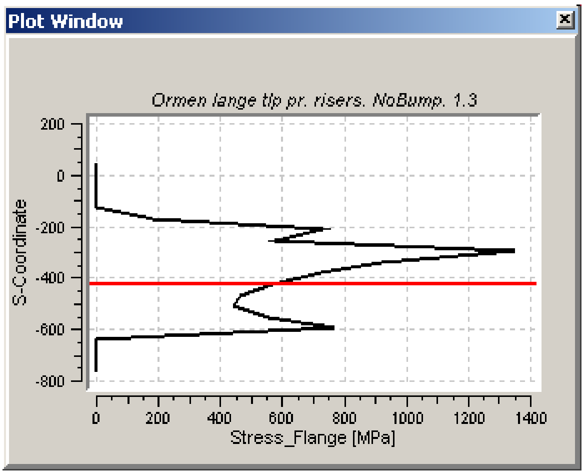
Pressing "Generate" generates a plot. The plot is placed in a docking window that appears inside the main window. The plot window may be undocked so that it floats outside the main window.

The following figure shows how time plots may look.



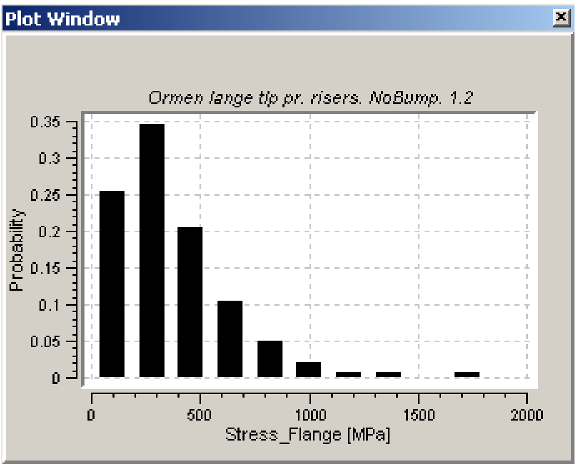
Time plots are presented as "spikes" with time as the X-axis variable and the selected variable as the Y-axis variable. Variable names are written along each of the axes, including units if specified.

S-coordinate plots may look as illustrated by the example in the figure below.



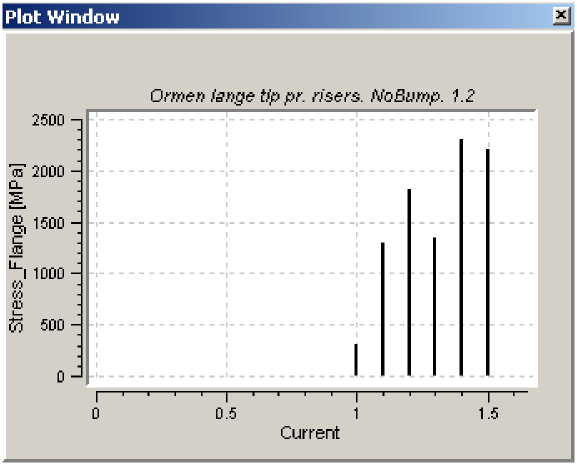
S-coordinate plots are presented as a curve consisting of connected line pieces along S-coordinates. S-coordinate is the Y-axis variable and the selected variable is the X-axis variable. S-coordinates where there have been no hits get the value 0. A red marker line is shown at the S-coordinate that corresponds to the currently selected section in the risers view. If a point on the curve is selected, the red line will be drawn through this point, and the new selection will also be selected in the risers view.

The following figure shows an example of how a probability within selected case plot may look.



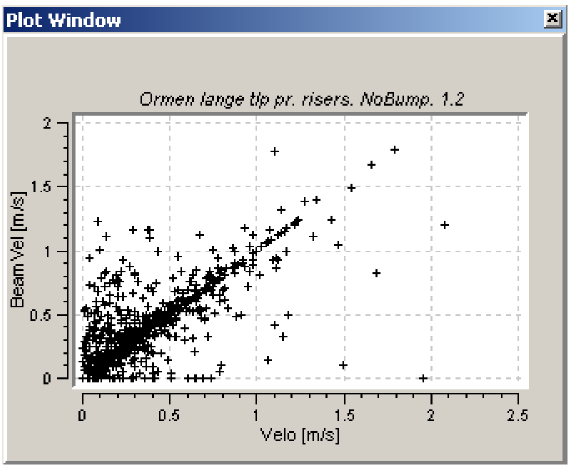
Max response vs case parameter plots are presented as "spikes" with the case parameter as the X-axis variable and the selected variable as the Y-axis variable. The maxium value of the Y-axis variable is plotted for each case parameter value.

Max response vs case parameter plots may look as illustrated by the figure below.



Cross correlation plots are presented as crosses and are simply general cross correlation plots where any X-axis variable may be plotted against any Y-axis variable.

The following figure shows an example of a cross correlation plot.



S-Coordinate plot...

This menu item opens the same dialog as when “Time plot...” is selected. The only difference is that the “S-Coord” radio button is selected by default. See the description of “Time plot...” for more information.

Probability within selected case plot...

This menu item opens the same dialog as when “Time plot...” is selected. The only difference is that the “Probability within selected case plot” radio button is selected by default. See the description of “Time plot...” for more information.

Max response vs case parameter plot...

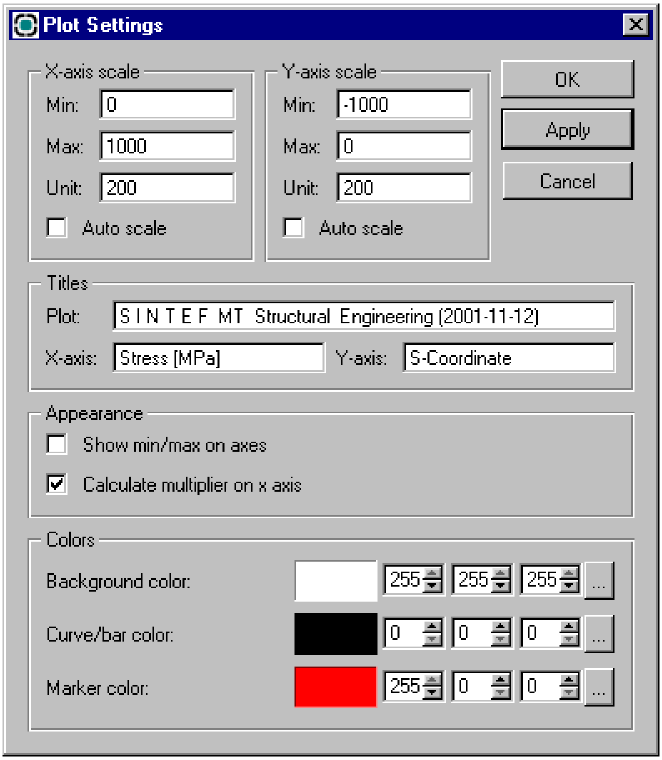
This menu item opens the same dialog as when “Time plot...” is selected. The only difference is that the “Max response vs case parameter plot” radio button is selected by default. See the description of “Time plot...” for more information.

Cross correlation plot...

This menu item opens the same dialog as when “Time plot...” is selected. The only difference is that the “Cross correlation” radio button is selected by default. See the description of “Time plot...” for more information.

Settings...

The plot settings dialog makes it possible to change the appearance of the plot. Default values are given initially. The dialog looks as illustrated by the figure below.



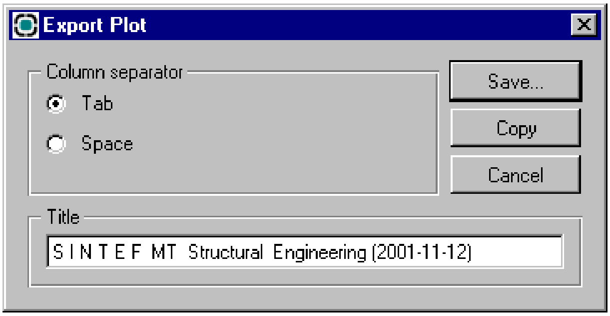
The minimum values, the maximum values, and the units of the axes and the units may be set manually, or this may be done automatically, which is the default setting. The titles of the plot and the axes may also be set manually even though they are created automatically by default.

The maximum and minimum values found in the plot data may be written in parentheses behind the axis titles. This option is turned off by default. By default, HYBER attempts to write nice numbers along the X-axis, and a multiplier is shown in parentheses behind the X-axis title. This option may be turned off.

In the bottom of the dialog there are controls for changing the background color, the curve/spike/bar color, and the marker color.

Export plot...

This menu item opens a dialog where the data in the current plot may be exported. This dialog is shown in the figure below.



The data is written as floating point numbers in two columns. In other words, each (x, y) pair is written on an individual line. By default, the column separator is tabulator, but space may be selected instead. The plot title, which is written in the first line, before the data, may also be changed. Clicking "Save..." brings up a file dialog where a name of the file to write to may be selected. If "Copy" is selected, the export data is put on the clipboard, so that it may be pasted in a different application.

The Animation Menu

This section describes the items in the animation menu, which looks as shown in the following figure.

Note that all “play” and “step” actions will result in reading data for animation setup if an animation hasn’t been set up for the selected case already. This may take a few seconds, depending on the number of time steps and the granularity of the riser geometries. The risers view will show the 3D animation of the risers, while the 2D animation view shows the displacements of the selected sections (in the risers view) for the selected case. When an animation has been set up for the selected case, all displacements for the risers are kept in memory and setting up an animation for a different section is therefore fast.

Play backward

Starts playing the animation backward.

Step backward

Goes back to the previous frame.

Pause

Pauses the animation.

Stop

Stops the animation.

Step forward

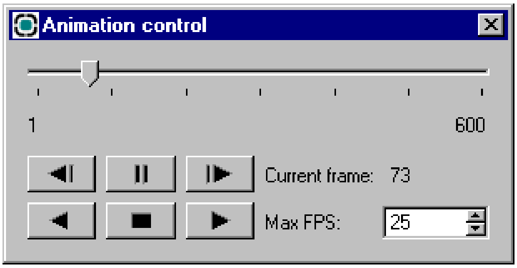
Advances the animation to the next frame.

Play

Plays the animation (forward).

Animation control...

This menu item brings up the animation control dialog, which looks as shown in the figure below.



Note that this dialog contains buttons that are equal to the animation control icons in the animation menu. The functionality of these buttons are exactly the same as the corresponding menu items. These buttons are also found in the animation toolbar.

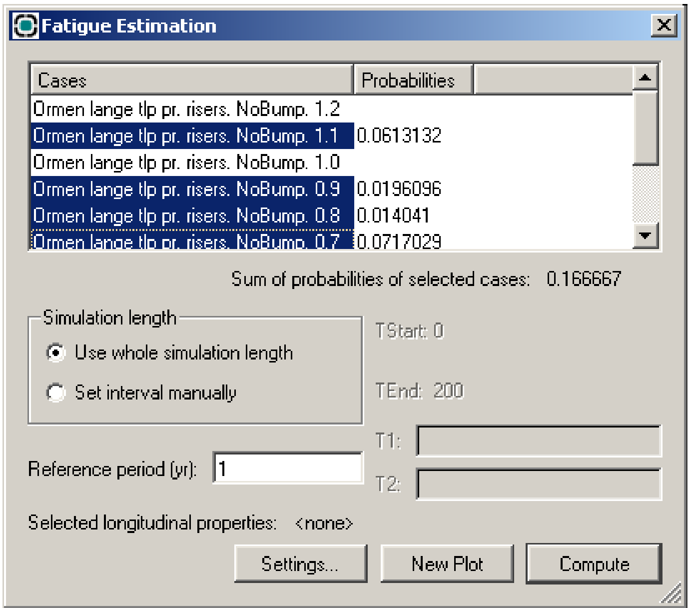
The slider control makes it possible to select a specific animation frame. The current frame number is shown in this dialog, and this number and the position of the slider are updated continuously during the animation. Maximum number of frames per second can be set manually. The default value is 25. Note that the system may be too slow to be able to play the animation at the specified maximum speed.

The Fatigue Menu

This section describes the items in the fatigue menu.

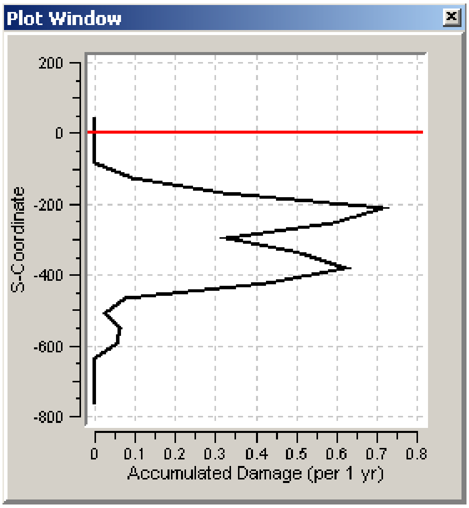
Calculate...

This menu item opens a dialog where fatigue estimations can be done. The following figure shows the appearance of this dialog.



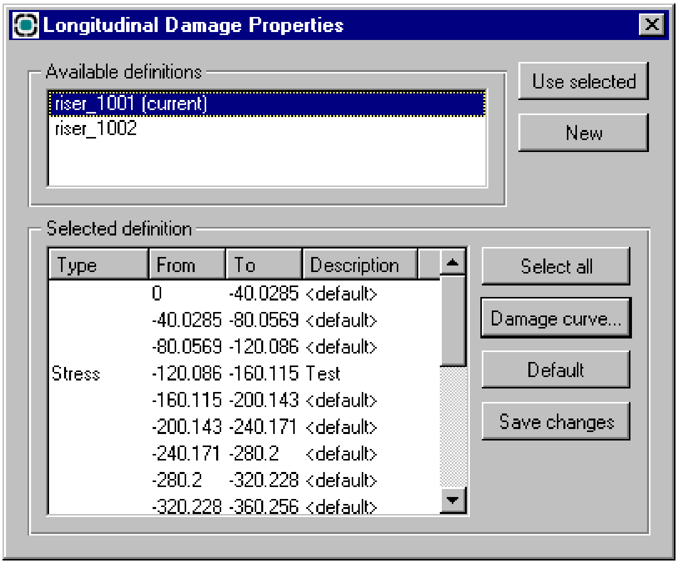
There is a list containing all cases. Multiple cases may be selected. The probabilities of the selected cases are shown in a column, and the sum of the probabilities is shown below the list of cases. When the selection has changed, the "TStart" and "TEnd" values will change so that they cover the time intervals for all of the selected cases. By default, the entire simulation lengths of all selected cases are used when calculating fatigue. If "Set interval manually" is selected, T1 and T2 may be changed within TStart and TEnd. This will limit the time intervals for the selected cases. The simulation length for each case is based on the part of its time interval that is covered by the range from T1 to T2. T1 can't be larger than T2. In addition, there is an edit control where the reference period may be specified. The default value is 1 year. The name of the selected longitudinal properties is shown, or “<none>” is written instead if default properties are used.

When the "Compute" button is pressed, the fatigue estimate will be calculated, and the plot will look as illustrated by the figure below. The curve shows the estimated fatigue at S-coordinates along the entire length of the riser. The figure below shows an example of a fatigue estimate.



Longitudinal properties...

This menu item brings up the longitudinal properties dialog, which looks as illustrated by the figure below.



All HYBER Damage Distribution Files (.hdd) found in the working directory are listed in the "Available definitions" list. By default, HYBER will use the damage distribution file with the same name as the project file, except for the extension, which is ".hdd" instead of ".hyb". If there is no such file, no damage distribution definition will be used initially. This is equivalent to specifying "Default" for all S-coordinates. To use a different damage distribution file, select a filename in the list and click "Use selected". The definition that is currently used is indicated by the text "(current)" behind the name of the definition.

If the "New" button is clicked, a new damage distribution file will be created. A name must be specified to use for the definition.

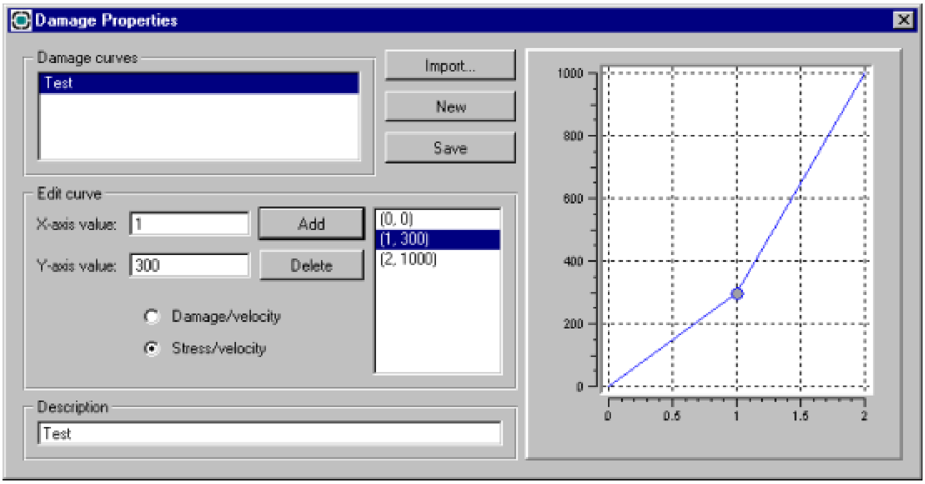
All editing of the damage distribution is done inside the "Selected definition" frame. The damage distribution is presented as a table with 4 columns. The first column contains the type. If a stress/velocity curve is associated with an S-coordinate interval the type is "Stress", and if it is a damage/velocity curve, the type is "Damage". The next 2 columns are from-to values of the S-coordinates along the first riser. There is one row per element (beam) in the riser, and the contents of the list are sorted on decreasing S-coordinate value. The fourth column contains the name/description of the corresponding stress/velocity or damage/velocity curve, or "<default>" if no curve is associated with this S-coordinate interval.

To change the damage properties for S-coordinate ranges, one or multiple rows in the table must be selected. Then "Damage curve..." must be clicked to select a specific damage curve, or "Default" may be clicked to use default damage instead of a damage curve. The "Select all" button selects all rows in the list.

The "Save changes" button saves the definition to the selected filename.

Property definitions...

This menu item opens the properties dialog, which is shown the figure below.

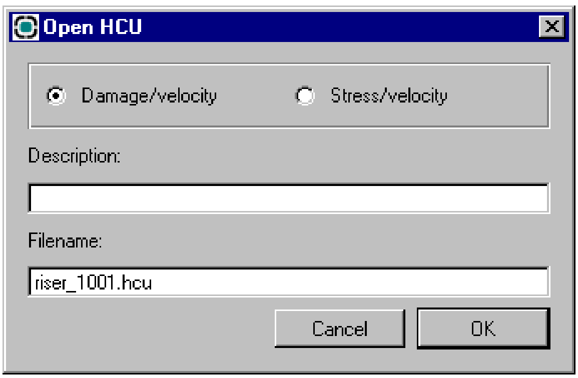


All damage curves found in HYBER Curve Files (.hcu) in the working directory will be listed in the list in the upper left corner. This list shows the description of each curve. When a curve is selected, it will be read from file and the contents of it will be presented in the dialog.

The curve may be edited by using the controls inside the "Edit curve" frame. Points on the damage curve are inserted by specifying an X-axis value and a corresponding Y-axis value, then clicking "Add". The points are shown in the list to the right of the "Add" button, and they are sorted on increasing X-axis value (velocity). An existing point can be deleted by selecting it in the list box and clicking the "Delete" button. There are two radio buttons that indicate whether it is a Damage/Velocity curve or a Stress/Velocity curve. A textual description of the curve must be given in the edit control inside the "Description" frame.

The damage curve is drawn in the right side of the dialog, and the curve is updated every time an edit operation is done. The mouse pointer may be moved to a point on the curve, which makes it possible to select the nearest point when the left mouse button is clicked. This point will be indicated in the curve and the corresponding item in the list will be selected.

Clicking the "Save" button will save the current damage curve definition. Clicking the "New" button will create a new curve. Before editing the curve, one must specify whether it is a "Damage/velocity" curve or a "Stress/velocity" curve, and a description must be given. A filename based on the name of the project file is suggested, but a different filename may be specified. This is done in a dialog that looks as illustrated by the figure below.



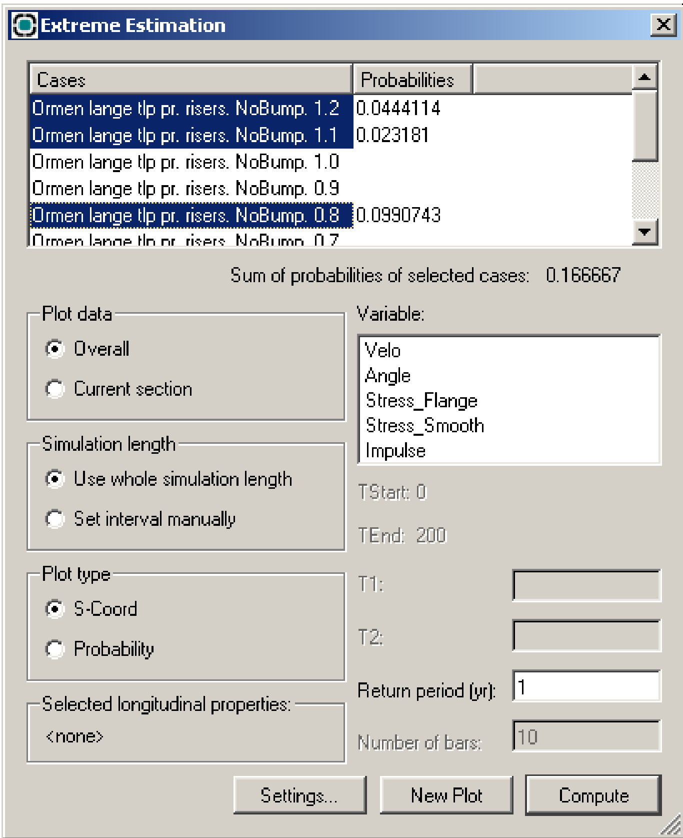
Clicking "Import" will import data from ASCII files that contain floating point number data organized in two columns, where the first column contains the X-axis values and the second the Y-values.

The Extreme Menu

This section describes the items in the extreme menu.

Calculate...

This menu item opens a dialog where extreme estimations can be done. The dialog looks as illustrated by the figure below.



There is a list containing all cases. Multiple cases may be selected. The probabilities of the selected cases are shown in a column, and the sum of the probabilities is shown below the list of cases.

When the selection has changed, the "TStart" and "TEnd" values will change so that they cover the time intervals for all of the selected cases. By default, the entire simulation lengths of all selected cases are used when calculating the extreme estimate. If "Set interval manually" is selected, T1 and T2 may be changed within TStart and TEnd. This will limit the time intervals for the selected cases. The simulation length for each case is based on the part of its time interval that is covered by the range from T1 to T2. T1 can't be larger than T2.

The "Variable:" list contains all variables that are specified as variables that we may calculate extreme values for. The list contains the extreme variables that are common for all selected cases. Extreme values are calculated for the variable that is selected when the "Compute" button is pressed.

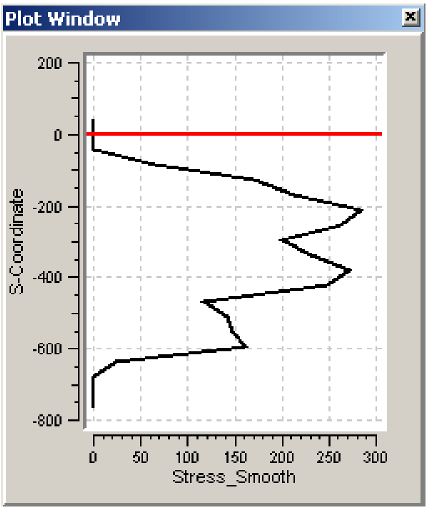
One may choose to calculate for the currently selected section, or for all sections (the entire riser).

In addition, there is an edit control where the return period may be specified. The default value is 1 year. Number of bars for long-term histograms is specified in the "Number of bars" edit control.

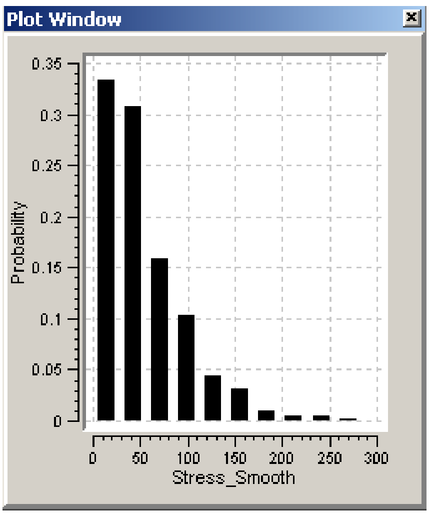
The name of the selected longitudinal properties is shown, or “<none>” is written instead if default properties are used.

When the "Compute" button is pressed the calculations will be performed, and the results of the calculations will be shown in a plot. If "S-Coord" is selected, extreme values are calculated with the given return period for all sections along the riser and those are plotted along S-coordinates. If "Probability" was selected, a long-term histogram is calculated. The plot will show a histogram that contains the specified number of bars. Each bar's height is determined by the relative probable number of hits within the interval of the selected variable covered by the bar.

The figure below shows an example of an extreme estimate.

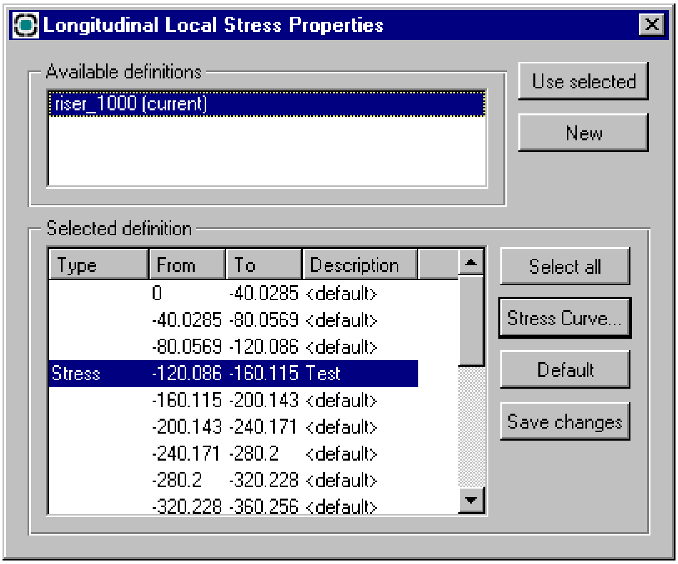


Long-term histograms may look as illustrated by the following figure.



Longitudinal properties...

This menu item opens the longitudinal local stress properties dialog, which is shown in the figure below.



All HYBER Damage Distribution Files (.hdd) that are found in the working directory and that contain references to stress/velocity curves or default stress are listed in the "Available definitions" list. By default, HYBER will use the file with the same name as the project file (except for the extension, which is ".hdd" instead of ".hyb"), unless it contains damage/velocity curves. If there is no such file, no definition will be used initially. This is equivalent to specifying "Default" for all S-coordinates. To use a different damage distribution file, select a filename in the list and click "Use selected". The definition that is currently used is indicated by the text "(current)" behind the name of the definition.

If the "New" button is clicked, a new stress properties file will be created. A name must be specified for the new longitudinal local stress properties.

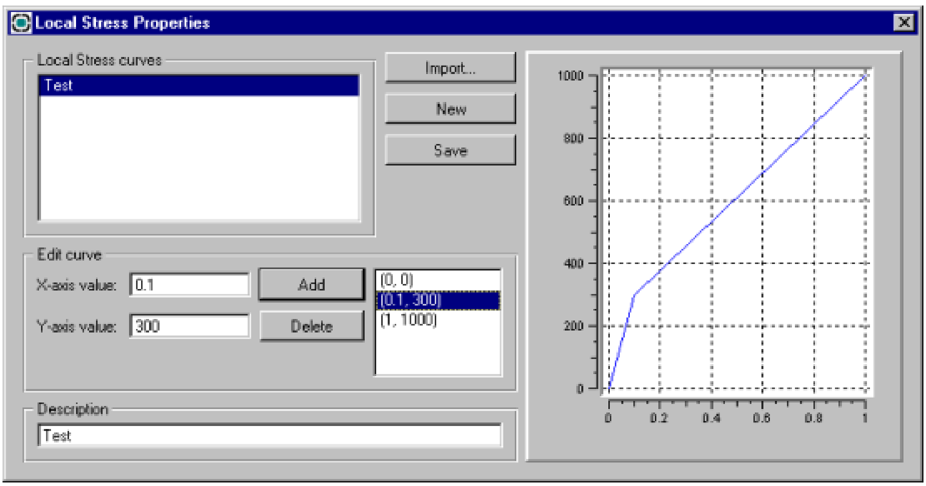
All editing of the definition is done inside the "Selected definition" frame. The definition is presented as a table with 4 columns. The first column contains the type. If a stress/velocity curve is associated with an S-coordinate interval the type is "Stress", and if it is a damage/velocity curve, the type is "Damage". The next 2 columns are from-to values of the S-coordinates along riser number one. There is one row per element (beam) in the riser, and the contents of the list are sorted on decreasing S-coordinate value. The fourth column contains the name/description of the corresponding stress/velocity or damage/velocity curve, or "<default>" if no curve is associated with this S-coordinate interval.

To change the definition for S-coordinate ranges, one or multiple rows must be selected in the table. Then "Stress Curve..." must be clicked to select a specific stress/velocity curve, or "Default" may be clicked to use default stress instead of a stress/velocity curve. The "Select all" button selects all rows in the list.

The "Save changes" button saves the definition in the selected stress properties file.

Property definitions...

This menu item opens the local stress properties dialog. This dialog is very similar to the damage properties dialog found in the "Fatigue" menu (read the description for it before reading further), except that only stress/velocity curves may be generated, and only stress/velocity curves are shown in the list of curves that have been defined. The local stress properties dialog looks as illustrated by the following figure.



The Window Menu

This section describes the items in the window menu.

Project

Toggles display of the project window.

Plot

Toggles display of the plot window.

Animation

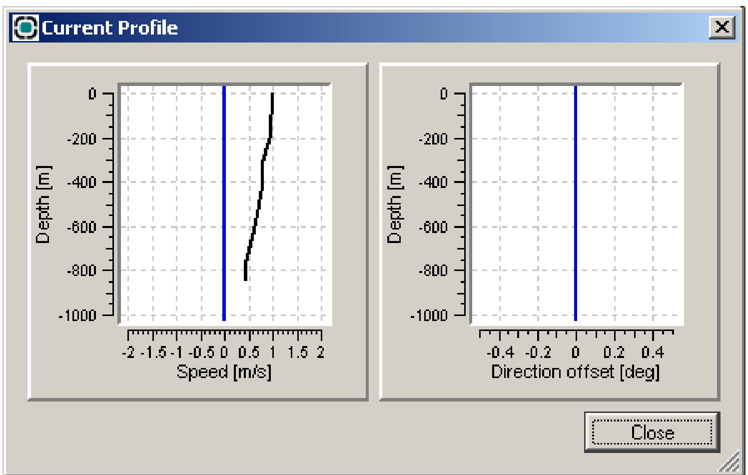
Toggles display of the animation view.

Plot

Toggles display of the plot window.

Current profile

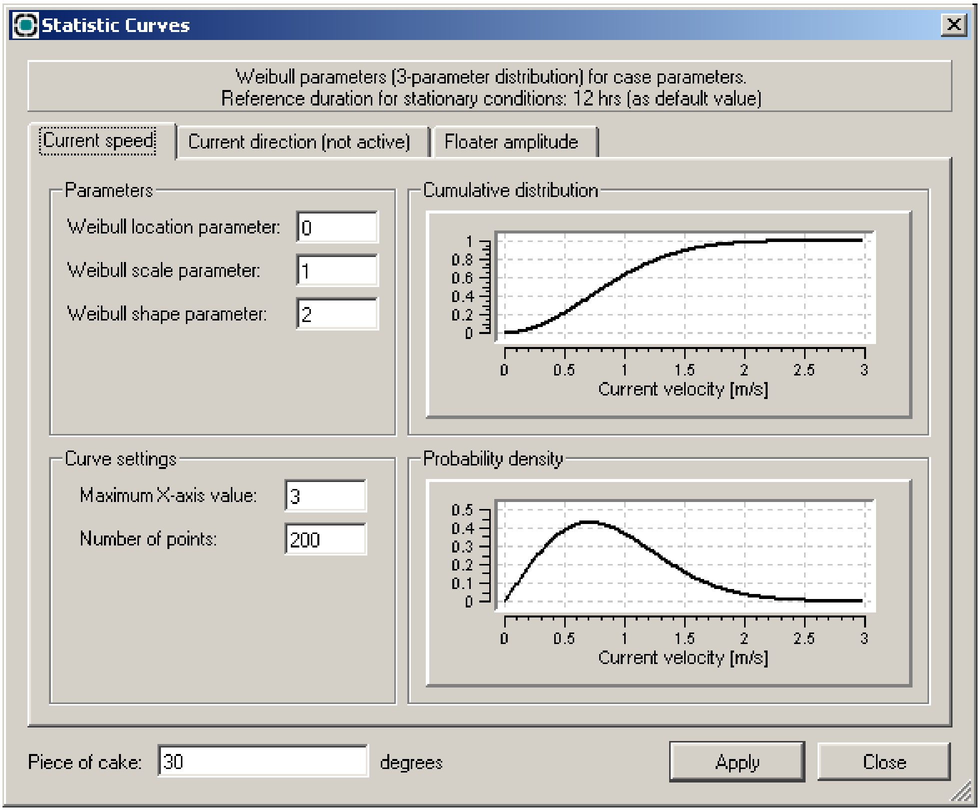
Toggles display of the current profile window. The current profile window shows the current profile for the currently selected case, and it may look as shown in the figure below.



If no current profile is given for the current case, the speed and offset profiles will be blank.

Statistic curves

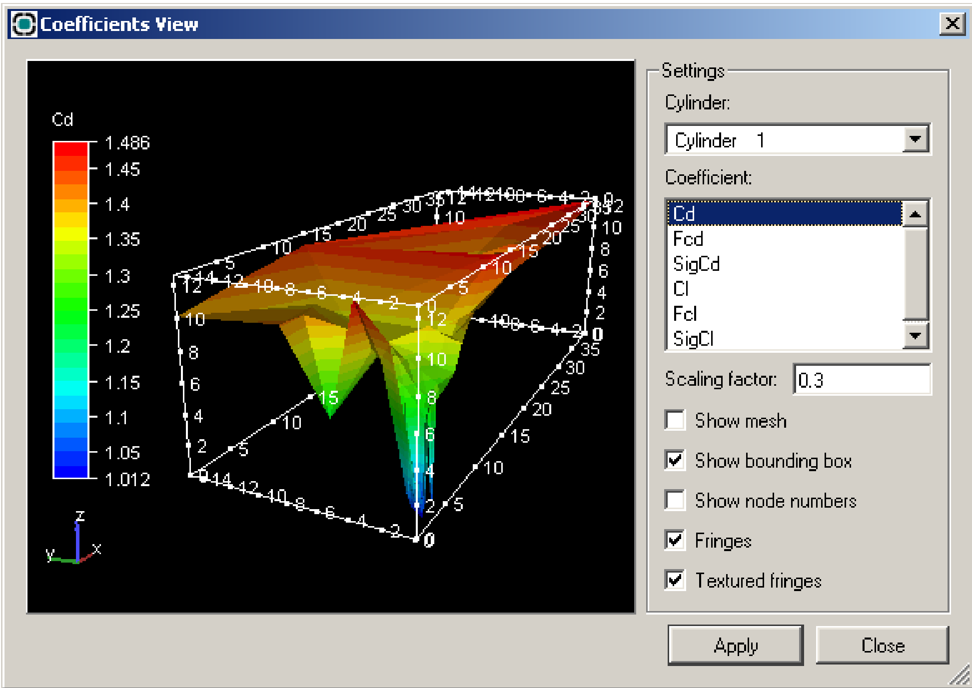
Toggles display of the statistic curves dialog. The figure below shows how this dialog may look.



This dialog specifies the cumulative distributions and the probability densities for case parameters. Those are used to compute the probabilities of the individual cases to be used when computing fatigue and extreme estimates. The shape of the curves are specified through the Weibull location, scale, and shape parameters. The maximum X-axis value and the number of points may be specified. These have influence on the presentation of the curves in this dialog only - the Weibull parameters are the only values that have effect on the case probabilities. "Piece of cake" may be set here. This parameter gives the circular sector of possible angles for hits, in degrees. 30 degrees means +/- 15 degrees, for example.

Coefficients view

Toggles display of the coefficients view, which may look as illustrated by the figure below:



The coefficients view shows the variations of a given coefficient for a given cylinder as a surface in the XY-plane, where the Z-values of the points in the surface are based on the coefficient value at each point. The surface is colored based on the coefficient values as well.

To the right of the view there are controls for selection of cylinder and coefficient.

By default, the height range of the surface is scaled so that it equals to 0.3 times the extent of the bounding rectangle of the points in the XY-plane, but this factor may be changed.

Display of the bounding box of the surface, the mesh, and the node numbers may be toggled. Coloring of the surface based on coefficient values may be turned on and off, and one may choose between textured fringes and fringes (colors interpolated within triangles).

The “Apply” button updates the view according to the settings. The coefficients view may be navigated just like the risers view, which is described earlier in this manual.

The Help Menu

This section describes the items in the help menu.

User manual...

Displays the user manual (this document).

Install on remote...

Opens a dialog for installation of HYBER Engine on a remote computer. See section called “Installing HYBER Engine on a Remote Computer“.

About

Displays the about box.

Toolbars

A number of toolbars are located in the top of the main window, right below the menu bar. These are described in the following sections.

The File Toolbar

This toolbar contains only one button. Clicking this button is equivalent to selecting "Open..." in the file menu.

The Plot Toolbar

This toolbar contains 5 buttons. They are equivalent to the items "Time plot...", "S-Coordinate plot...", "Probability within selected case plot...", "Max response vs case parameter plot...", and "Cross correlation plot..." in the plot menu.

The Fatigue Toolbar

This toolbar contains 3 buttons. They are equvalent to the items "Calculate...", "Longitudinal properties...", and "Property definitions..." in the fatigue menu.

The Extreme Toolbar

This toolbar contains 3 buttons. They are equvalent to the items "Calculate...", "Longitudinal properties...", and "Property definitions..." in the extreme menu.

The Animation Toolbar

This toolbar contains 7 buttons. They are equivalent to the items "Play backward", "Step backward", "Pause", "Stop", "Step forward", "Play", and "Animation control" in the animation menu.

The Analysis Setup Dialog

The Analysis Setup Dialog appears when a HYBER Setup File (.hys) is opened or created from the File menu. The purpose of this dialog is to generate input to HYBER Engine. This section describes the functionality of the dialog through an example.

Example: TLP, 850m water depth

In the following example, two vertical risers connected to a TLP should be simulated with the following key data.

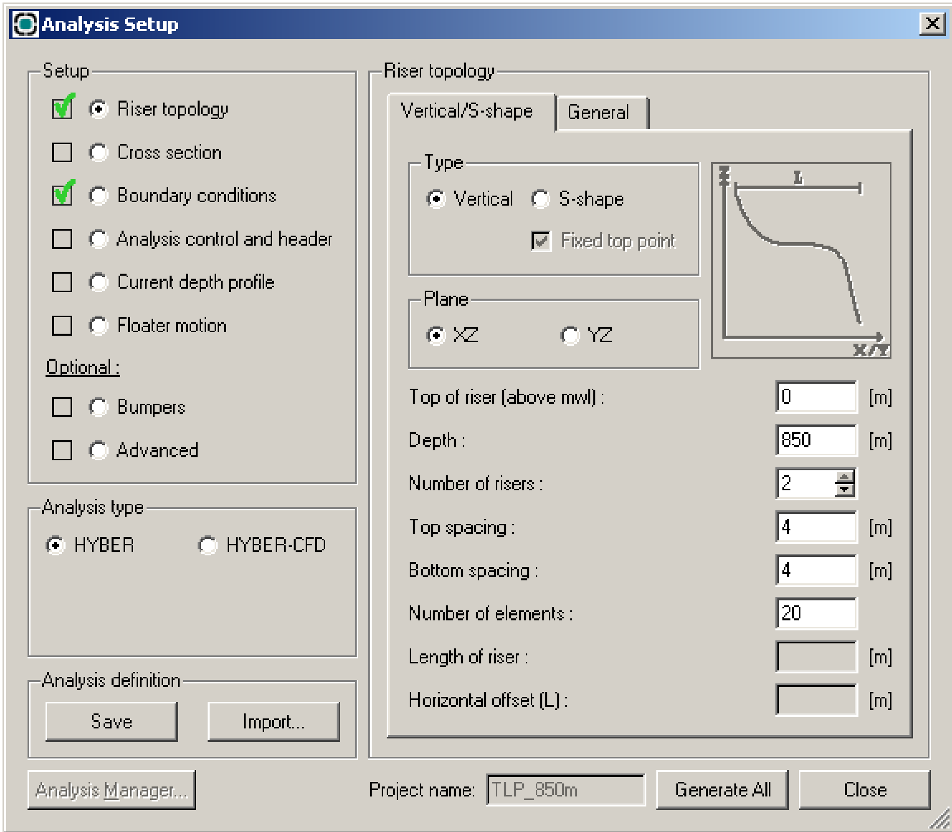
Depth	850m (distance from bottom to top end of riser)
Spacing	4m (same at bottom and on top)
Outer Pipe	D = 0.340 m, T = 0.0155m
Inner Pipe	D = 0.178 m, T = 0.0081m
Fluid density	Between pipes : 1100kg/m3. Inside tube: 200kg/m3
Top tension	TTF = 1.4
Top spring	k = 0.4 x Top Tension.
Simulation length	600 sec
Current speeds	0.8 to 1.3m/s with increment 0.1 m/s (5 cases)
Current dir	0 deg to 10 deg with increment 5 deg (3 cases)
Floater motion	Amp = 4m, Period = 142.2s, Direction 0 deg. (1 case)
Depth profile	Current speed varies vs depth. Current dir. is constant vs depth.

When you want to generate analysis input, the following options exist:

- Start from scratch. Fill in the different data types. (File/New setup file...)
- Load an existing setup file, and modify (File/Open setup file...)
- Load an existing setup file and use directly (File/Open setup file...)

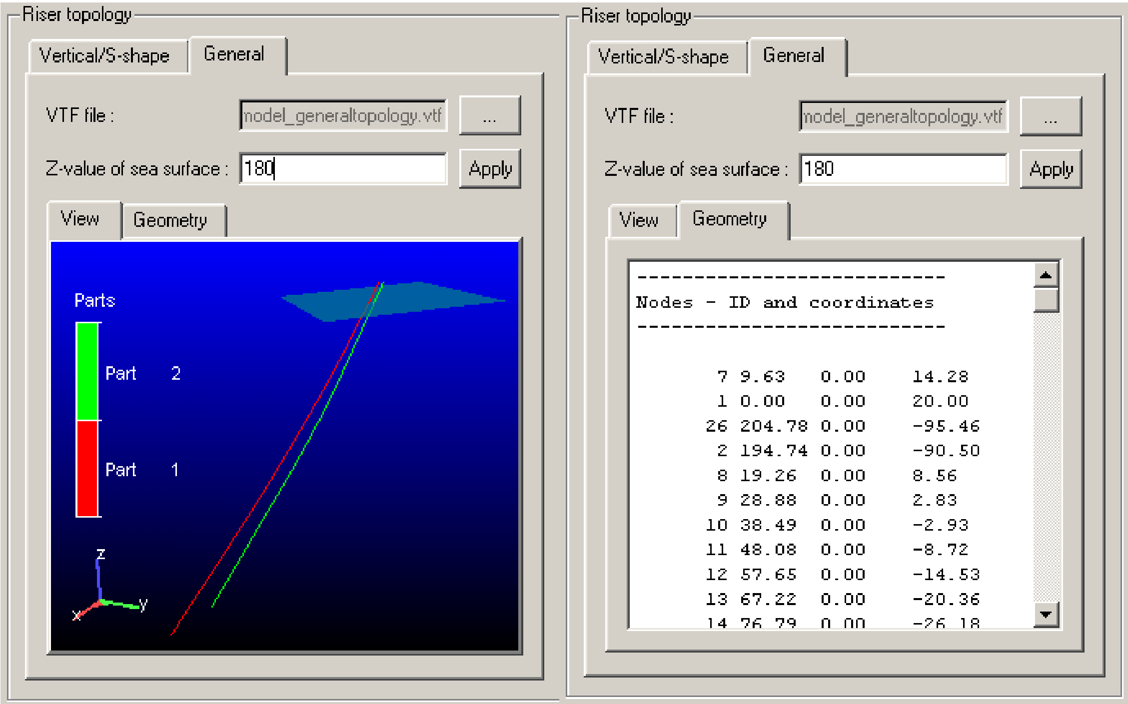
If you start from scratch (File/New setup file...), the Analysis Setup Dialog appears. Sheets are selected by clicking the radio buttons on the left side. As the different sheets are filled in, a green tag is inserted. By default, the boundary conditions are filled in (default values exist). As soon as any sheet is modified, the "Save..." button, (see the figure below), becomes active, indicating that the sheets are not saved on file.

To use parameters from an existing setup file, click the “Import...” button and select the file.



In the figure above the riser topology is defined. The riser topology type in this case is “vertical”, which means that the top and bottom depth values must be defined, along with top and bottom spacing, the number of elements per riser and the plane in which the risers are located. If the number of risers is 1, top and bottom spacing are irrelevant, of course. For the “S-shape” type of riser topology the riser length and the horizontal offset must be defined. If “Fixed top point” is selected then “Top of riser” becomes irrelevant.

The following figure shows how general topology is defined. Note that the figure has nothing to do with the example described in this section, since the example has a “vertical” toplogy. The riser geometry is read from a VTF file and is shown in the “View” tab window if the VTF file is read successfully. The VTF file must meet the following requirements: There must be at least one part consisting of 2-node beams only. The first 1 or 2 parts of the model in the VTF file consisting of 2-node beams only will be interpreted as risers. However, all parts are shown in the view.



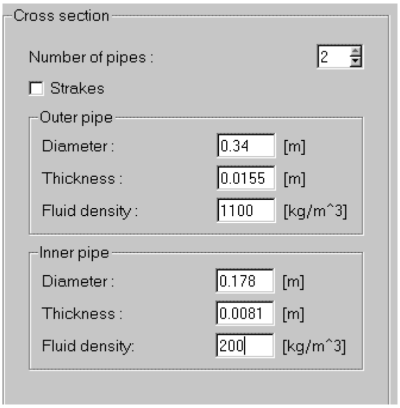
If the VTF file meets the requirements, the nodes and elements are listed in the “Geometry” tab window (shown on the right side of the figure above). However, the geometry of the VTF file is always shown in the “View” tab window as long as it is a valid VTF file.

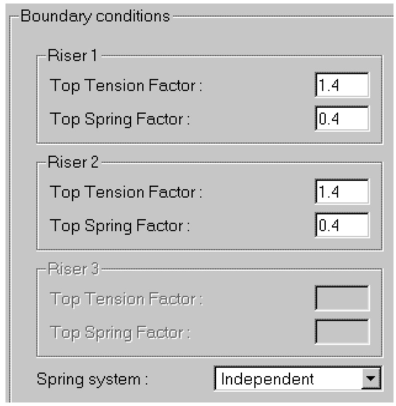
The Z-value of the sea surface may be adjusted. This value is indicated by a transparent rectangular surface in the plane defined by the current Z-value. Node coordinates read from the VTF file will be adjusted according to this value when saving the analysis setup file.

An example of a VTF file is given in the File Formats appendix, in a section called “VTF Files”. Note that a VTF file to be used for general topology doesn’t have to contain displacements, just the geometry of one or two risers.

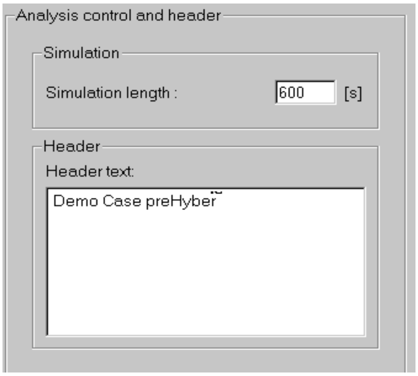
The next figure shows dialogs for definition of riser cross section and boundary conditions. The riser consists of two pipes (number of pipes is selected to 2). The fluid density for the Outer Pipe is used for the annulus between the outer pipe and the tubing. Based on the data, an equivalent cross section is computed where the outer diameter and total cross section area are kept. The material density is artificially scaled in order to get correct dry mass per length unit of the pipe. (Added mass is accounted for separately in HYBER Engine).

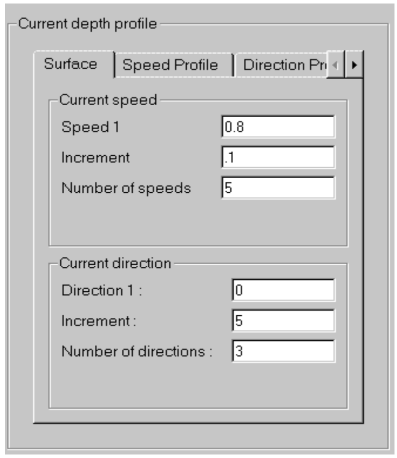
The top tension is specified through the top tension factor (Tension= TTF x Submerged weight). Flexibility of the top riser support is specified relative to the top tension, (k=Tension x TSF).





The following figure describes the simulation and current sheets. Simulation length is specified in seconds. Heading, which will appear on the different results are typed in, (max 3 lines of text).

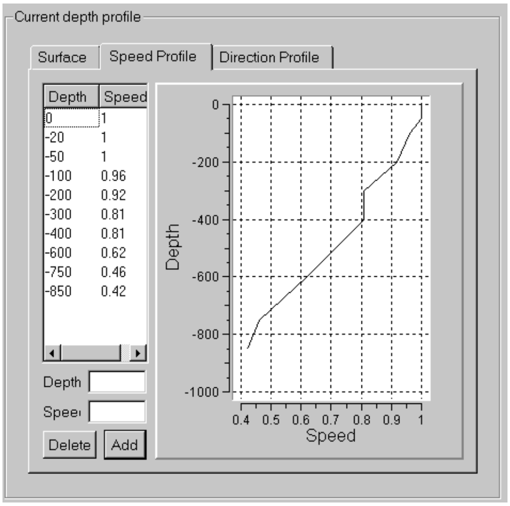


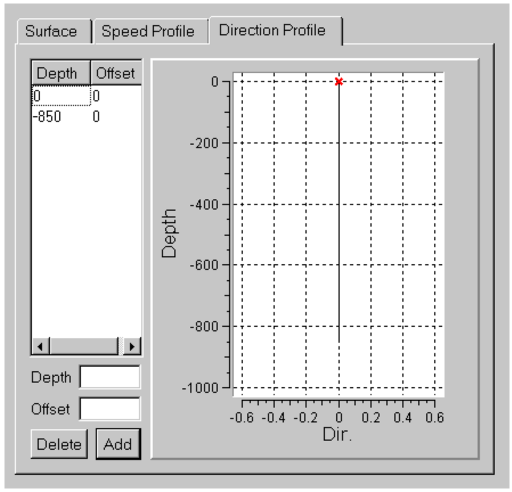


The current is specified as follows:

- Surface speed and direction
- Depth profile for speed (product of surface speed and curve value gives the actual speed).
- Depth profile for direction offset, (relative to the surface direction).

In the previous figure, 5 different surface speeds and 3 different surface directions are specified, (which gives totally 15 different analysis cases in HYBER Engine).





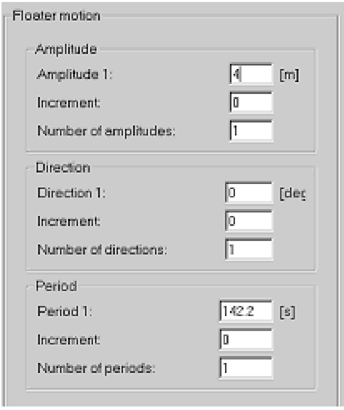
The depth variation of the current speed and direction is specified in separate sheets (see the previous figure). The current speed used at a given depth is the product of the surface speed and the curve value at the actual depth. The "Speed Profile" is then a scaling curve, which typically is kept constant for varying surface speed.

The ocean current direction is seldom (never) constant over the entire water depth. Significant changes in direction may occur for the different water levels. The direction offset from the surface current direction is specified as a depth profile. The depth curve is specified in degrees, and both positive and negative direction offset could be specified. The current direction in at a given depth is then the sum of the surface direction and the curve value at the actual depth.

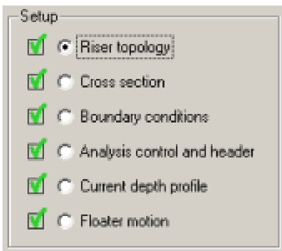
Direction = 0 deg is along global X-axis.

The motion of the top end of the risers could be prescribed using a harmonic function (sine), and the amplitude and period are specified. In addition, the direction of the floater motion could be specified.

In the actual demo example shown in the following figure, only one amplitude motion is specified. As for the current, it is possible to generate a series of combinations of the 3 parameters.



When all data sheets are properly defined, all fields have a green tag (as shown in the following figure). It is recommended to save the setup using the "Save" button at this stage.

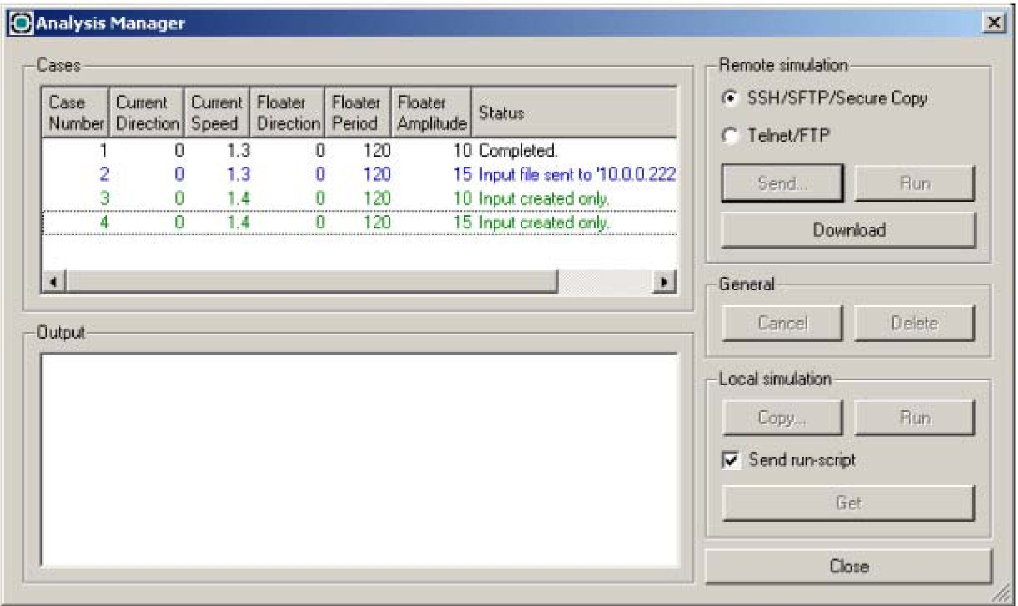


Create all input to HYBER Engine by pressing the “Generate All” button. In this demo example, 5 x 3 individual cases are generated. Totally 15 HYBER Engine input files are generated automatically ready for execution. The case number is a part of the file name, and is used to keep the individual files apart.

To get an overview of the cases that have been generated and/or to run them, click the “Analysis Manager...” button. This opens the Analysis Manager Dialog, which is described in the next section.

The Analysis Manager Dialog

The Analysis Manager Dialog is available through a push button in the Analysis Setup Dialog (see previous section). The following figure shows what it looks like:

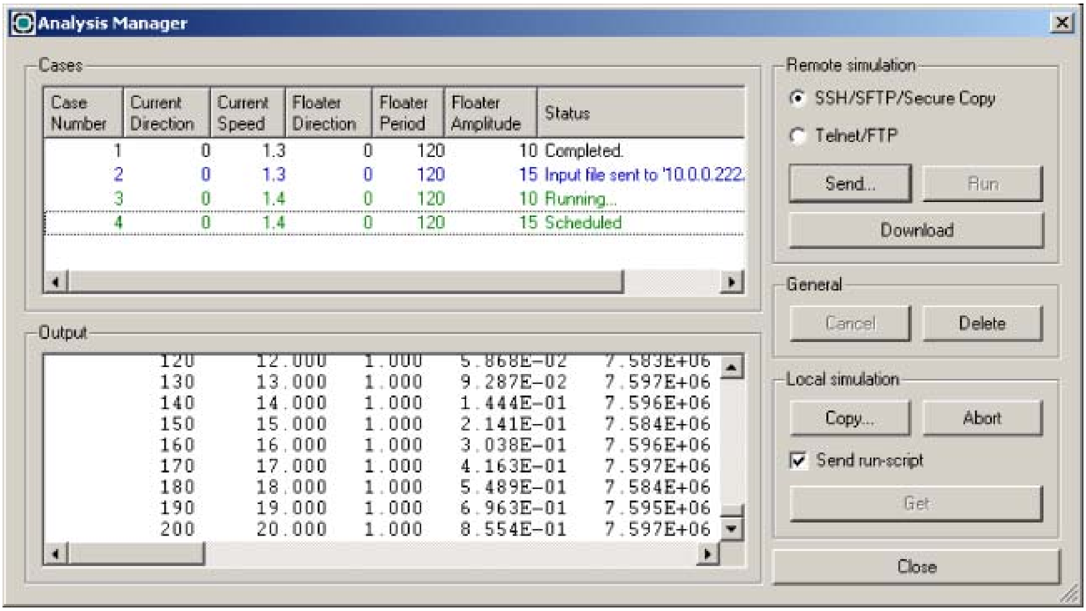


It gives an overview of all cases, with case numbers, parameters and status texts.Only parameters that may differ between cases in a project are shown here. The buttons on the right side of the dialog are actions which are enabled or disabled depending on the currently selected cases. Multiple cases may be selected by pointing and clicking while pressing the CTRL or SHIFT key. There are two general buttons: “Cancel” and “Delete”. “Delete” deletes the selected cases. “Cancel” is described in the following two subsections.

This dialog lets you run the analyses on the local computer, or you may run them on a remote computer. Note that Windows is not very well suited for running CPU-intensitive programs like HYBER Engine in the background - you may experience that the system “hangs” when using other programs simultaneously. It might be a good idea to have an Intel-based Linux computer in the network to do this job (see section “Installing HYBER Engine on a Remote Computer”). Another alternative is to use a Windows-based PC which is dedicated for this job and has access to the project directory (where the HYBER Setup File (.hys) is located). Then your local Windows-based PC may be used to do post-processing only, while one or more other computers do the hard work.

Simulations on the local computer

The buttons inside the “Local simulation” frame lets you run analyses on the local computer. This is very simple: just select one or more cases that haven’t been run yet, and click the “Run” button. See the figure below.



The figure shows an example where case numer 3 and 4 have been selected and the “Run” button for local simulations has been clicked. The status text for case 3 says “Running...”, which means that this case is currently being run. The status for case 4 says “Scheduled”, which means that it is scheduled and will be run when case 3 is finished. Output from the current run is shown in the “Output” frame.

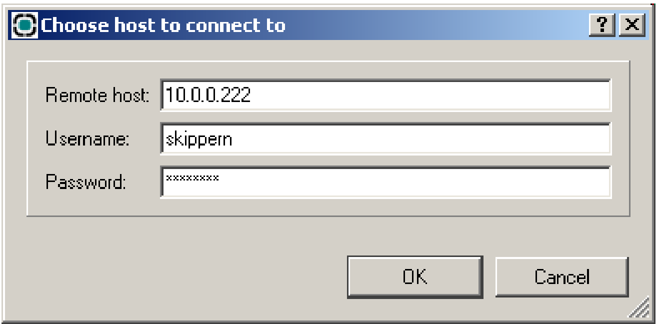
The “Run” button turns to “Abort” while running analyses on the local computer. Note that the run may be aborted at any time by clicking “Abort”. This will abort the current run and all scheduled runs. The status text of cases that have been run is “Completed.”

The “Local simulation” also has a “Copy...” button and a “Get” button. They may be used if you want to copy analysis input files to a directory that may be reached on the local network area. A run-script may be included if desired. This lets you run analyses manually on a different computer having access to the directory to which the input files have been copied. The “Get” button lets you copy the output files back to the local computer. “Cancel” removes the information about where the selected cases have been copied and you may do whatever you want with them again.

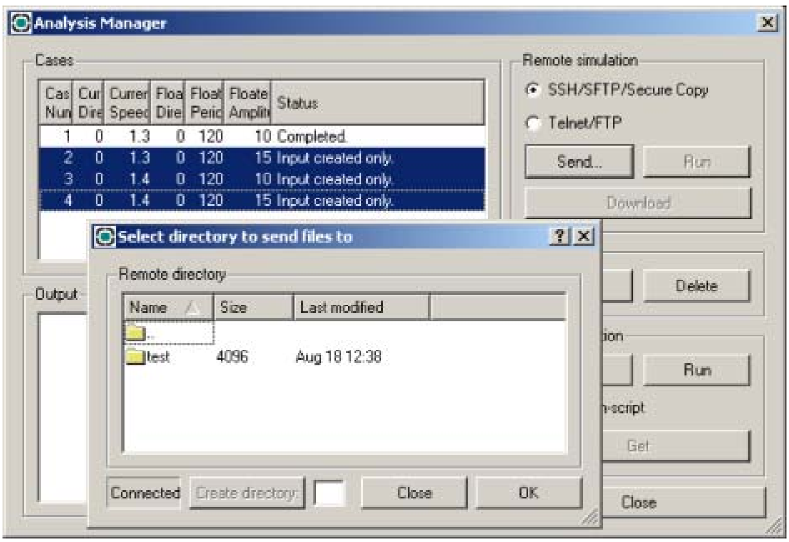
Simulations on a remote computer

The buttons inside the “Remote simulation” frame lets you run analyses on a remote computer. There are two different modes for communication with remote computers: One uses SSH, Secure FTP and Secure Copy, the other uses Telnet and FTP. The first one is default.

Cases that haven’t been run yet (the status text says “Input created only.”) may be selected and sent to a remote computer by clicking the “Send...” button. In order to log on to a remote computer you need to specify host name or IP address, user name, and password:



If the login process succeeds, a dialog appears. This dialog lets you go to a directory in which to put the input files, and you may create a new directory if desired. See figure below.



When you are in the target directory, just click OK and the input files are uploaded to the target directory on the remote computer. A progress dialog is visible during the upload, showing the number of files to upload and which file is currently being uploaded.

The status texts of cases that have been uploaded say “Input file sent to” along with the host name or IP address and the target directory of the remote computer.

If you select one or more cases that have been sent to a remote computer, the “Run” button for remote simulations is enabled. This button starts analyses on remote computers.

If you click the “Download” button, the program will log on to the remote computer(s) and check if any output files are available. Output files for the uploaded cases are downloaded if they exist, and a progress dialog gives information about the download process. The status text for cases whose output files have been downloaded is “Completed”.

Note that if cases have been uploaded by mistake you may click the “Cancel” button to remove the information about where the selected cases have been uploaded and you may do whatever you want with them again.

Installing HYBER Engine on a Remote Computer

You may easily install HYBER Engine automatically on an remote Intel-based computer running Linux through the graphical user interface of HYBER. Just select “Install on remote...” from the Help menu. This gives you a dialog that lets you start the installation by clicking the “Install...” button.

First you have to specify the name or IP address of the remote computer, along with a user name and a password. If the login process is successful, the installation process starts. The “Messages” window shows what’s going on and a message box appears and tells you when the installation is completed. After this you may run simulations on the remote computer. Note that the installation is done locally (in the home directory) for the given user.

The installation dialog looks as shown in the following figure.

File Formats

This chapter describes the file formats used by HYBER.

External Data Communication

This section describes the external file formats used by HYBER.

HYBER Project Files

HYBER is able to read HYBER Project Files (.hyb). HYBER Project Files are ASCII files. HYBER recognizes a number of keywords and reads the information connected to them. A keyword is placed at the start of a line. The table below describes the keywords that are recognized.

Keyword	**Description**
Identification	Project description/name. Can have multiple such lines.
PlotText	The project information that should appear in plots. Text string with max. 40 characters.
Version	HYBER version information. Text string.
RiserLabels	Labels for riser information. Each label consists of one word of text.
Riser	Riser information. One word of text per riser label.
StartNode	Integer value that gives the ID of the start node (where the S-Coordinate is 0).

HYBER Case Files

HYBER is able to read HYBER Case Files (.hc1). There is one case file per case, and the case files must be named "<project file name>_<case number>.hc1", where <project file name> is the name of the HYBER Project File, without the filename extension ".hyb" and <case number> is the number of the case, starting at "001" for the first case, and then "002", "003", and so on. As indicated, the case number has 3 characters.

HYBER Case Files are ASCII files. HYBER recognizes a number of keywords and reads the information connected to them. A keyword is placed at the start of a line. The table below describes the keywords that are recognized.

Keyword	**Description**
Case	Case description/name. Text string.
PlotInfo	Additional plot info to be given below the title in plots. Text string.
BeginCaseInfo	All lines of text following this keyword down to "EndCaseInfo" are case information.
EndCaseInfo	All lines of text before this keyword up to "BeginCaseInfo" are case information
NumberOfHit	Integer that gives the number of collisions for which we have parameters/results.
NColumns	Integer that gives the number of parameters/results (labels) in the file.
Labels	Names of the labels. One word per parameter/result. There are NColumns labels.
ExtremeLabels	Names of the labels for which extreme values may be calculated.
Units	Units of the labels. One word per label. Special value "(none)" means no unit.
BeginCurrent	Specifies the start of a current profile definition block. See specification of current profile definition block below.
EndCurrent	Specifies the end of a current profile definition block. See specification of current profile definition block below.
BeginCoefficients	Specifies the start of a coefficients block. See specification of coefficients block below.
EndCoefficients	Specifies the end of a coefficients block. See specification of coefficients block below.

Some labels are required. These are listed in the table below.

Keyword	Description
Time	Time when a collision occurred. Floating point number.
ElementID	The ID of the element for which the collision occurred. Integer value.
Damage	The damage as a result of the collision. Floating point number.

It is assumed that line with the “Units” keyword is followed by lines of text where we have NColumns parameter/result values for each line. There should be NumberOfHit such lines. Blank lines and lines starting with "#" are ignored.

Current profile block

A current profile block starts with the keyword “BeginCurrent” and ends with the keyword “EndCurrent”.

The following keywords are recognized inside a current profile block:

Keyword	Description
ProfileSpeed	Specifies profile speed. Syntax: <num points> <Z1> <speed 1> <Z2> <speed 2> ... <ZN> <speed N>
ProfileDirection	Specifies profile direction. Syntax: <num points> <Z1> <dir 1> <Z2> <dir 2> ... <ZN> <dir N>

Coefficients block

A coefficients block starts with the keyword “BeginCoefficients” and ends with the keyword “EndCoefficients”. The first non-empty line after the “BeginCoefficients” keyword not starting with a coefficients block keyword is used as the name of the cylinder for which coefficients are given.

The following keywords are recognized inside a coefficients block:

Keyword	Description
nColumns	Specifies the number of columns (number of coefficients). Syntax: nColumns <number>
nLines	Specifies the number of lines (number of coefficient points). Syntax: nLines <number>
Labels	Names of the coefficients. One word per coefficient. There are nColumns names.
Units	Units of the coefficients. One word per coefficient. Special value “(none)” means no unit.

It is assumed that line with the “Units” keyword is followed by lines of text where we have an X-value and a Y-value, followed by nColumns coefficient values for each line. There should be nLines such lines. Blank lines and lines starting with "#" are ignored.

VTF Files

HYBER is able to read ASCII VTF files containing the geometry of the risers. When a HYBER Project File is opened, there should be at least one case file (.hc1) in the same directory. Each case file should have one VTF file connected to it. Such a VTF file has the same name as the corresponding case file, except for the extension, which is ".vtf". HYBER reads the geometry of the risers from the VTF file related to the first case file found in the working directory.

The risers must be written as series of beam elements. There must be two risers. Each riser must be defined as a separate part, by writing one element block for each riser. The elements (and the nodes) of the first riser on the VTF file will be used when calculating S-coordinates. The elements of the risers must be listed in a sorted manner within the element block. To be more specific, the least deep element is listed first, then the second least deep element is listed, and so it goes up to the deepest element, which is listed at the end of the element block. The node connectivities of the elements (beams) have the same requirement: the first node is the least deep node, and the second node is the deepest. Note that each element is defined by a line of text where we have the element ID, and then the connectivity (the IDs of the nodes). See example below.

```
*VTF-1.00
NODES 1
%WITH_ID
      1 .00000E+00 .32000E+02 -.85000E+03
      2 .00000E+00 .23529E+02 -.62500E+03
      3 .00000E+00 .75295E+01 -.20000E+03
      4 .00000E+00 .00000E+00 .10000E+03
      5 .10000E+02 .32000E+02 -.85000E+03
      6 .84375E+01 .24000E+02 -.63750E+03
      7 .53125E+01 .80000E+01 -.21250E+03
      8 .37500E+01 .00000E+00 .10000E+03
*ELEMENTS 1
%NAME "Riser 1"
%NODES #1
%WITH_ID
%BEAMS
      1 1 2
      2 2 3
      3 3 4
*ELEMENTS 2
%NAME "Riser 2"
%NODES #1
%WITH_ID
%BEAMS
      4 5 6
      5 6 7
      6 7 8
*GLVIEWGEOMETRY 1
%NAME "Risers"
%ELEMENTS
1, 2
```

Note that this example only illustrates how to define the geometry of the risers. VTF files connected to case files should have nodal displacements for all of the timesteps in addition to the geometry. This information must be present in order to be able to set up animations.

FEM Files

HYBER generates FEM Files (.fem). These files are input files for HYBER Engine. The format is similar to the input format of USFOS. Detailed information about this format can be found in USFOS User Manual.

Import of Data for Damage Curves

HYBER is able to import ASCII files with floating point numbers given in columns, where columns are separated by white space and rows are separated by newline. Numbers in the first column are interpreted as velocities, while numbers in the second column are either damage or stress. The user must specify the type of data in the second column when importing such files. HYBER exports the imported data to a HYBER Curve File (.hcu).

Internal Data Communication

This section describes the file formats used internally by HYBER.

HYBER Damage Curve Files

Filename extension: .hcu

Type: ASCII

Format:

Line 1: Damage curve name/description

Line 2: Type (either "Damage" or "Stress")

Lines 3-N: <floating point number> <floating point number>

HYBER Damage Distribution Files

Filename extension: .hdd

Type: ASCII

Format:

Lines 1-N: <Element ID> <HYBER Damage Curve File (.hcu)>

Note: Only elements for which a user-defined damage curve is specified, are listed. Elements that are not listed have default damage, which means that damage results from the HYBER Case File (.hc1) should be used.

HYBER Setup Files

HYBER is able to read HYBER Setup Files (.hys). HYBER Setup Files are ASCII files. HYBER recognizes a number of block types and block-specific keywords with parameters within each block. The structure of a HYBER Setup File is as follows:

```
<block 1>
<block 2>
[...]
<block N>
```

The structure of each block type is as follows, with a few exceptions, documented for each block type:

```
<block begin keyword>
<block-specific keyword 1> <parameters>
<block-specific keyword 2> <parameters>
[...]
<block-specific keyword N> <parameters>
<block end keyword>
```

<u>Block descriptions:</u>

Topology block

A topology block starts with the keyword “BeginTopology” and ends with the keyword “EndTopology”.

The following keywords are recognized inside this block type:

Keyword	Parameters
Configuration	“Vertical”, “S-shape <“Disp” | “Force”> Control”, or “General”
TopOfRiser	Floating point number.
Depth	Floating point number.
NumberOfRisers	Integer.
Spacing	Floating point numbers. Syntax: Spacing <bottom> <top>

Keyword	Parameters
NumberOfElements	Integer.
LengthOfRiser	Floating point number.
HorizontalOffset	Floating point number.
Plane	“XZ” or “YZ”

If “Configuration” is “General”, the topology block defines a general topology. A general topology is defined in terms of nodes and elements. The following keywords/parameters are found in such a topology block:

Keyword	Parameters
RiserNodes	Integer. Tells the number of “Node” keywords that will follow.
Node	<Node ID> <X-coordinate> <Y-coordinate> <Z-coordinate>
RiserElements	<Riser number> <Number of elements in this riser>
Element	<Element ID> <Node ID 1> <Node ID 2>. Defines an element in the riser specified by the number in the previous “RiserElements” keyword. Node IDs refer to nodes specified by “Node” keywords.

Cross section block

A cross section block starts with the keyword “BeginCrossSection” and ends with the keyword “EndCrossSection”.

The following keywords are recognized inside this block type:

Keyword	Parameters
NumberOfPipes	Integer.
D_OuterPipe	Floating point number.

Keyword	Parameters
T_OuterPipe	Floating point number.
Fluid_OuterPipe	Floating point number.
D_InnerPipe	Floating point number.
T_InnerPipe	Floating point number.
Fluid_InnerPipe	Floating point number.
Strakes	String. “On” or “Off”

Boundary block

A boundary block starts with the keyword “BeginBoundary” and ends with the keyword “EndBoundary”.

The following keywords are recognized inside this block type:

Keyword	Parameters
SpringSystem	Floating point number.

In addition, this block type contains sub-blocks. A sub-block starts with “Riser <N>”, where N is the riser number. The two next lines contain keywords with parameters for the sub-block:

Keyword	Parameters
TopTensionFactor	Floating point number.
TopSpringFactor	Floating point number.

Heading block

A heading block starts with the keyword “BeginHeading” and ends with the keyword “End-Heading”.

Heading blocks may contain up to 3 lines of text.

Analysis control block

An analysis control block starts with the keyword “BeginAnalysisControl” and ends with the keyword “EndAnalysisControl”.

The following keywords are recognized inside this block type:

Keyword	Parameters
AnalysisModule	Text string. “Hyber”.
SimLength	Floating point number.

Current block

A current block starts with the keyword “BeginCurrent” and ends with the keyword “End-Current”.

The following keywords are recognized inside this block type:

Keyword	Parameters
CurrDirection	One integer and two floating point numbers.
CurrSpeed	One integer and two floating point numbers.
ProfileSpeed	<number of points> <x1 y1> <x1 y2> [...] <xN yN>
ProfileDirection	<number of points> <x1 y1> <x1 y2> [...] <xN yN>

Floater motion block

A floater motion block starts with the keyword “BeginFloaterMotion” and ends with the keyword “EndFloaterMotion”.

The following keywords are recognized inside this block type:

Keyword	Parameters
Amplitude	One integer and one floating point number.
Direction	One integer and one floating point number.
Period	One integer and one floating point number.

HYBER Log Files

HYBER creates a HYBER Log File (.hyl) when cases are generated from the Analysis Set-up Dialog. This file contains names of the parameters that are variable for the cases within a project, and the parameter values for each case.

The format of HYBER Log Files is as follows:

```
BeginParameterNames
<parameter 1>
<parameter 2>
[...]
<parameter N>
EndParameterNames
<case number 1> <parameter 1> <parameter 2> [...] <parameter N>
<case number 2> <parameter 1> <parameter 2> [...] <parameter N>
[...]
<case number N> <parameter 1> <parameter 2> [...] <parameter N>
```

HYBER Remote Files

HYBER creates a HYBER Remote File (.hyr) when analysis input files for cases are copied to a directory on the local network area or uploaded to a remote computer from the Analysis Manager Dialog. This file keeps information about where each input file has been copied or uploaded so that analysis output files may be copied back or downloaded when they have been created. There is one block type for case input files that have been copied to a directory on the local network area and one for case input files that have been uploaded to a remote computer. There may be multiple blocks of each type.

The format of the local network area block is as follows:

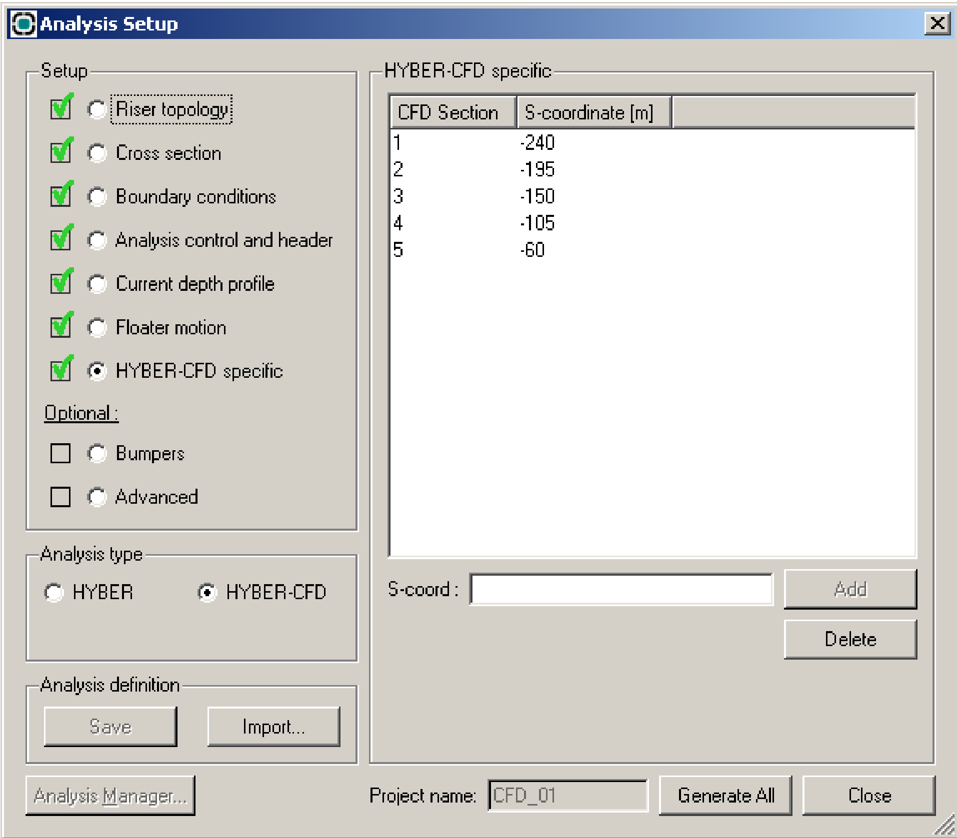
```
BEGINLOCALNETCASES
Directory <directory name>
Case <case number 1>
Case <case number 2>
[...]
Case <case number N>
ENDLOCALNETCASES
```

The format of the remote computer block is as follows:

```
BEGINREMOTEHOSTCASES
Host      <host name or IP address>
Directory <directory name>
User <user name>
Case <case number 1>
Case <case number 2>
[...]
Case <case number N>
ENDREMOTEHOSTCASES
```

HYBER-CFD

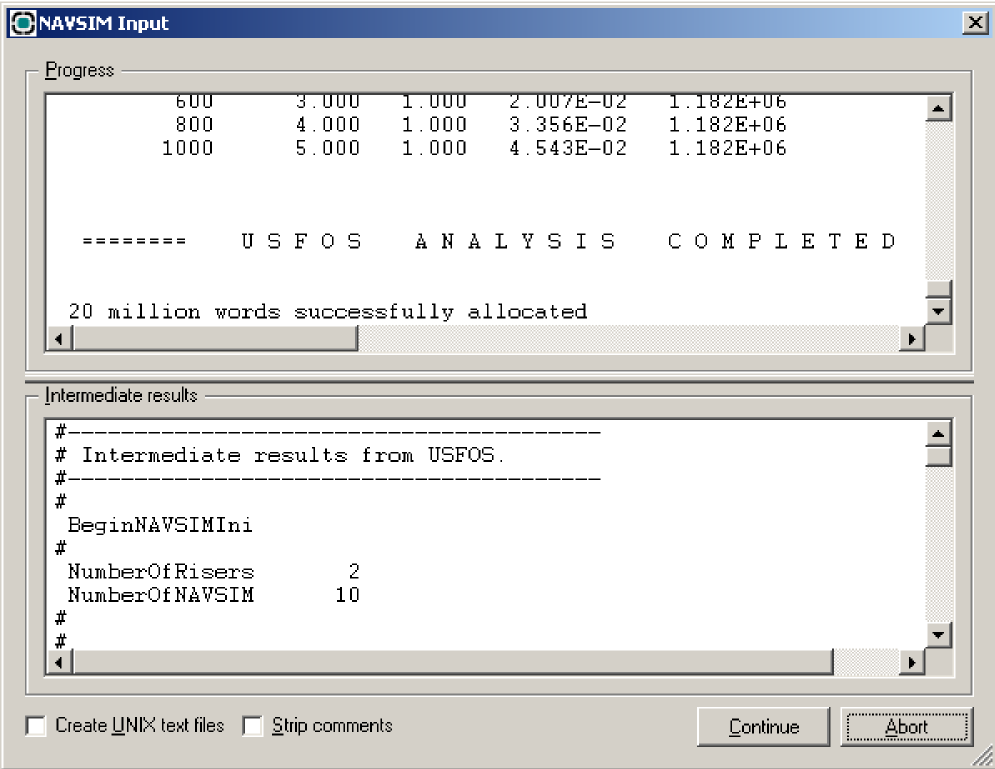
This appendix descripes the analysis setup type “HYBER-CFD”. This is used internally by HYDRO and is therefore described in an appendix and is irrelevant for others.



The HYBER-CFD analysis type is selected by clicking the “HYBER-CFD” radio buttton. An additional setup sheet called “HYBER-CFD specific” is available for this analysis type, as shown in the figure above. It gives the opportunity to add/delete CFD sections and each CFD section is defined by a number and an S-coordinate. CFD sections are sorted on S-coordinates so you don’t have to worry about the numbering of the sections.

Note that the Analysis Manager is unavailable for this analysis type, since analyses of this type can't be started from within this application. The input files are just prepared for analyses to be handled outside HYBER.

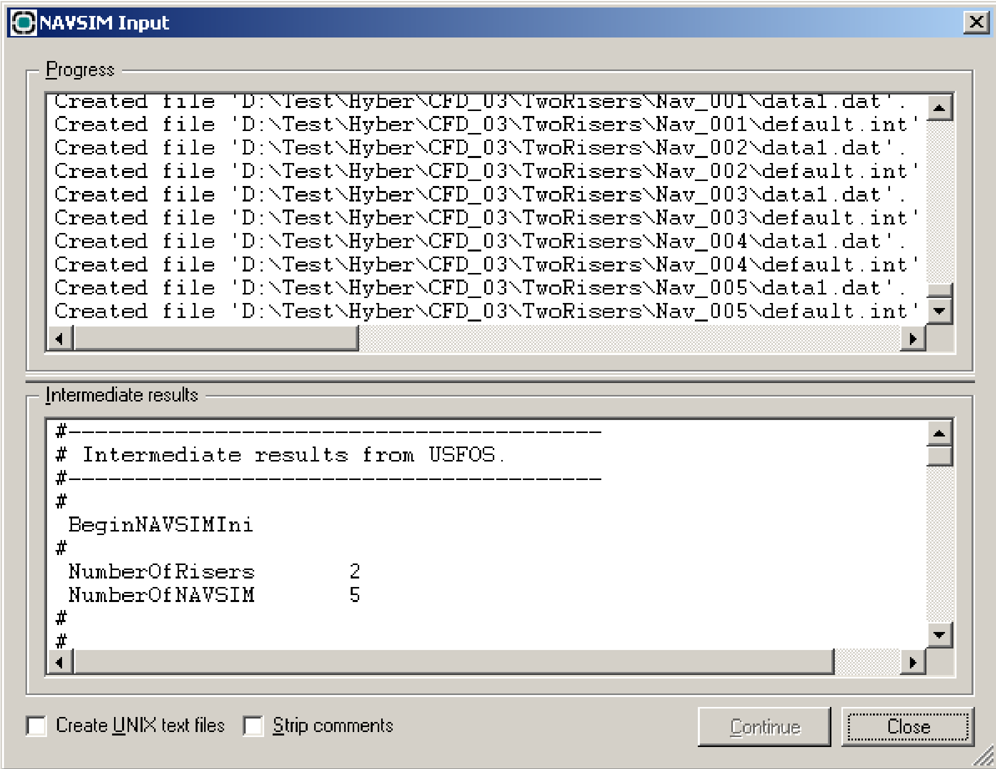
When "Generate All" is clicked, input files are created. The following figure shows the dialog that appears when "Generate All" is clicked.i



USFOS will start generating intermediate results and a message box tells when they are ready. The progress is shown in the upper part of the dialog. The intermediate results can be inspected in the lower part of the dialog. If the results are OK, just click "Continue" to generate all input files. If the results aren't OK, just click "Abort" to go back to the analysis setup dialog where parameters can be modified. Then intermediate results can be generated based on the new parameters and the new intermediate results can be accepted or rejected.

To generate input files, a directory called “Templates” must exist in the directory where the setup file is located, and a file called “Navsim_template.dat” and one called “Griddler_template.dat” must exist in the “Templates” directory. The files “data1.dat” and “default.int” will be generated for each CFD section, based on the template files “Templates/Navsim_template.dat” and “Templates/Griddler_template.dat”, respectively. The following tags are replaced by values in the input files if they are found in the template files:

Tag	**Description**
{Velo}	(sqrt(UX*UX + UY*UY))
{NCyl}	Number of risers
{Dt}	Time increment
{NStep}	The number of steps
{CylCoord}	Coordinates of the section. One line of {X Y Z} per riser.
{Navsim}	Project name



When all input files have been generated, the dialog may look like shown in the figure above. The progress window tells which files have been created. There is one directory for each CFD section and each directory contains a file for the section, based on the template file. A directory called “USFOS” is also created. This directory contains the intermediate results and the final simulation input file.