

A01 Catenary and wave systems

Introduction

In these examples, four types of riser system are shown: a simple catenary, a lazy wave, a steep wave and a pliant wave.

On opening each simulation file, the default workspace will present wire frame and shaded graphics views of the system.

The examples also show:

- Variation of drag coefficient with Reynolds number.
- Bend stiffener modelling.
- Modelling of buoyancy modules.
- Extending the simulation.
- Modelling tethers and their clamps.

All the examples in this set have regular waves applied. These are repeated waves with the same height and period. The Dean Stream non-linear wave theory has been applied because it is accurate over a wide range of water depths.

The build-up (stage 0), where wave heights are ramped up from zero to the required value, is given a duration of one wave cycle. Less than this and the rapid application of the wave could generate unwanted transient loads.

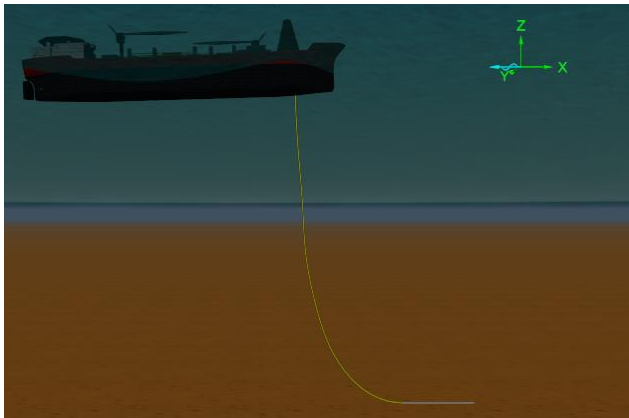
The main analysis (stage 1) is given a duration of five wave cycles. This is sufficient for most systems of this type to achieve a settled response.

Catenary riser

The example has a single riser in a simple catenary extending from a vessel to the seabed.

When you open the simulation file you will see two side elevation views of the catenary; one in wire frame and one in shaded graphics format. Beside them are two graphs. The upper graph is of top connection *end force* against *end force Ez-angle*. The lower is a range graph of normal drag coefficient along the catenary length.

Building the model



Open the *model browser* via the *model* menu (or use the shortcut *F6*) then double click on *Catenary Hose* to open its data form.

The *end connections* table shows that end A is connected to the vessel (*FPSO*) and end B is *anchored*.

The *anchored* option gives access to an additional data item called *height above seabed* which is important to get the correct seabed interaction at that connection.

Seabed contact takes account of the line contact diameter. This diameter is found on the *contact* page of the *line type* data form (note that a line contact diameter of “~” means that the line type *outer diameter* is used as the *contact diameter*). If the connection z coordinate was set at zero (centreline directly on the seabed) then the line bottom edge would be placed one radius beneath the seabed.

If you set *height above seabed* to zero then OrcaFlex offsets the end connection z coordinate by the line contact radius so the bottom edge of the line sits flush on the seabed.

The end orientations have also been set in this model. These specify the directions of the line end fittings. If the riser is aligned with that direction at the end fitting then no moment is produced. It is therefore called the *no-moment direction*. Further details about this can be found on the [Modelling, data and results | Lines | Line data | End connections](#) page of the OrcaFlex help and the tutorial video [Setting line end orientations](#), available on the [Orcina website](#).

You can view the local axes, including the line end orientations, by selecting [Axes and labels | Local axes](#) from the *View* menu (or use the shortcut *Ctrl+Y*). The key point is that OrcaFlex adopts an ‘end A to end B’ convention. This means that the end direction (i.e. the local z-axis direction on which the no-moment direction is based) points into the line at end A and out of the line at end B.

In this example, the riser end connections are pinned (zero connection stiffness). This means the connection is free to rotate and no end moments will be produced, so changing the end orientations will not change the behaviour of the system.

In spite of this, it is good practice always to set the line end orientations. The line model is then set up appropriately in case you change to non-zero connection stiffness at a later stage. Also bend stiffener design loads need *end force* and *end force Ez-angle*, its direction relative to that no-moment direction.

Therefore, to obtain the correct design loads, the no-moment direction needs to be set. A more detailed explanation can be found in the [Modelling, data and results | Lines | Line results | End loads](#) page of the OrcaFlex help.

Establishing the appropriate level of line segmentation is a balancing act. You want sufficient refinement that the model can capture the response accurately but not so much that simulation run times are slow with no significant improvement in results. It is best to consider refined segmentation only where it is needed. Where the detail is required will depend on what you are modelling.

In this example the area of interest is the touchdown point i.e. where curvature is expected to be high. On the [Catenary Hose](#) line data form, look at the segmentation for the line on the [structure](#) page.

The riser is made up of three sections with the same properties but different segmentation. There is detailed segmentation in the middle section, the touchdown region, where it has a [target segment length](#) of 0.5m. There is coarser segmentation either side where the riser remains fairly straight.

From the [model browser](#), open the [Generic drag](#) dataset from the [Variable data](#) → [Drag coefficient](#) group. This contains a table relating [Reynolds number](#) to [drag coefficient](#). Note, the data used here is intended for illustrative purposes only. Select the [profile](#) button to see the data plotted on a graph.

Returning to the [model browser](#), double-click on the line type named [10' flexible variable Cd](#) to open the [line type](#) data form. Go to the [drag & lift](#) page – here the [Generic drag](#) dataset is selected for the normal drag coefficient. Note that the drag coefficient in the y direction is set to '~' which means 'use the same value as the x direction'. In other words, the drag is isotropic. At each time step OrcaFlex will calculate the Reynolds Number (Re) at each line node and apply the appropriate drag coefficient (Cd) value.

The [Reynolds number calculation](#) data item on the [sea](#) page of the [environment](#) data form is used to determine how OrcaFlex interprets the data specifying Cd vs Re. This data item should be set to match the conventions used by your Cd vs Re data source. For more information, see the [Modelling, data and results | Environment | Sea data](#) page of the OrcaFlex help.

Results

The [edit replay parameters](#) option, accessible from the [replay](#) menu, will open the replay settings. Choose [latest wave](#) to see the final cycle of the run. You will see the line nodes turn white as they contact the seabed.

Look at the range graph of the line [x drag coefficient](#) for the [latest wave](#). This shows how the normal drag coefficient varies along the line length. It gives the maximum, mean and minimum values experienced through the final wave cycle. The graph shows a variation between 0.4 and 1.2. There is significant variation along the whole length. Results are also available for [drag force](#) and its components.

The graph of the riser's [end force](#) against [end Ez-angle](#) at the FPSO (end A) shows the bend stiffener design loads. The moving cross shows where on the curve the animation has reached. In this case it is a closed loop, indicating that the system has settled into a steady response.

The simulation duration is five wave cycles. You can extend the run without having to start at the beginning again. Select the [extend dynamic simulation](#) option from the [calculation](#) menu and enter how much more simulation time is required. The model state will change from [completed](#) to [paused](#) and

you can set it running for the additional time by choosing *run dynamic simulation* from the *calculation* menu, or by using the keyboard shortcut *F10*.

The graphs discussed above were all set up in advance using the workspace facility. To create a range graph of effective tension, choose *select results* from the *results* menu (or press *F5*). This takes you to the OrcaFlex results form. In the *result type* area at the left of the form are the types of plots available. Greyed out options are those that are not valid for the type of analysis carried out in this example.

Make sure *range graph* is selected. This result is only valid for lines and turbine blades so the only option in the *object* area is *Catenary Hose*. OrcaFlex also needs to know what time period of the run you want results for. In this example, select *latest wave* in the *period* area.

Finally you need to select the variable you want plotted. For lines there are many results so they have been sorted into groups. By selecting the group in the *show* list, you can hide and show different variables. Select the *forces* group, then choose *effective tension* from the *variable* area.

Press the *show* button to produce the plot. The range graph will now be at the top left. It shows the expected tension distribution for this simple catenary.

Lazy wave riser

A lazy wave configuration is similar to catenary but this option has mid-water support provided by distributed buoyancy modules. 'Lazy' means that the riser centreline is near parallel with the seabed on contact while 'wave' describes the line shape as a result of the buoyancy modules.

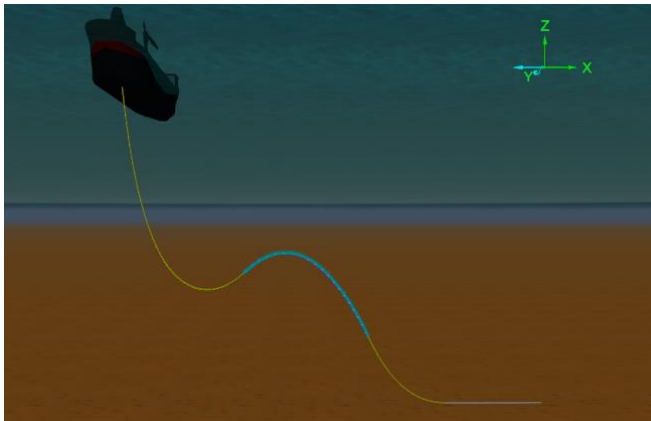
This example has two risers positioned exactly on top of each other to allow comparison of two different modelling methods. These are:

- **10" Lazy Wave Distributed:** Buoyancy is applied using a line section that has the equivalent smeared properties of the line and buoyancy modules combined. This approach is more efficient and convenient for the design stage and load analysis. It allows easy variation in buoyancy application and does not need as much refinement of line segmentation.
- **10" Lazy Wave Discrete:** Buoyancy is applied as discrete modules attached to the line. This approach is more useful in the final stages of an analysis when the as-built module details are known. It allows a check that module size and pitch (the spacing between consecutive buoyancy modules) do not lead to excessive riser sag between the modules; often called the 'curtain-railing' effect.

A wire frame plan view and a shaded side elevation view of the model will be presented when you open the simulation file. You will also see range graphs of curvature for both options.

The plan view shows that this is a cross-weather case. Make sure that the [Environment axes](#) option is selected from the [Axes and labels | Local axes](#) part of the [View](#) menu. The weather heading is shown in the top right-hand corner of the model views. Here, the current is in the riser plane (straight blue arrow) while the waves are normal to it (wavy blue arrow). The vessel has also been rotated such that it points into the waves i.e. to represent a weathervaning system.

Building the model



Both risers have the same end connection settings and the same number of sections. As with the simple catenary model, refined segmentation has been applied where needed.

For this example, it is the sagbend, hogbend and touchdown regions that require refined segmentation. The suspended length below the hang-off, and the length lying on the seabed, do not experience significant curvature and so allow coarser segmentation.

We will first consider the riser with smeared properties: **10" Lazy Wave Distributed**. Open the line data form for this line. You can do this by double clicking the line name on the [model browser](#). Now look at the [structure](#) page on the form.

The riser is made up of five sections for the suspended length, sagbend, hogbend, touchdown and length on the seabed. Segmentation differs between the sections as mentioned above, however the

hog region (section 3) also has a different line type. It uses *10"+Floats* instead of *10" flexible*. This *+Floats* line type contains the smeared properties of the riser and floats.

To see the details of these properties, select the *line types* button from the bottom left-hand corner of the form. It will take you to the *line type* data form. You can also access it directly via the *model browser*.

At the top of the form, set the *view mode* to *all* so you can compare the properties of the two line types. If you move through the pages on the form you will see that *10"+Floats* has increased outer diameter, mass, hydrodynamic coefficients and contact diameter. The drag and stress diameters are set to match the values of *10" flexible*. There is a variable drag coefficient called *10"+Floats drag* which is based on the *Generic Drag* specified for *10" flexible* and adjusted to account for the extra normal drag from the buoyancy modules.

There is no need to calculate any of these changes, we can simply run the OrcaFlex *line type wizard* to create them. The *line type wizard* is only fully accessible when the program is in the *reset* state. So, exit the form and select *reset* from the *calculation* menu (or press *F12*). Now open the *line type* data form again.

If you are going to use the line type wizard, then you first need to select the line type you want to modify. So select the *10"+Floats* line type and then select the *wizard* button from the top right-hand corner of the form.

As we have already set up an example, the wizard has remembered what we have done and will be showing you the *special category* option *line with floats*. Select the *next* button at the bottom right to move to the page where you input data.

This facility allows you to easily experiment with module properties and pitch (spacing) to get the configuration you wish. You can find more information on how the properties are calculated in [Modelling, data and results | Lines | Line with floats](#) in the OrcaFlex help.

Note the hydrodynamic properties for the buoyancy module in this example are for illustration only and do not reflect actual values.

Now we will consider the riser with discrete modules attached: *10" Lazy Wave Discrete*. Open its line data form via the *model browser*. If we look at the *structure* page the only difference is that section 3 (the hogbend region) now has *10" flexible* line type and its segmentation is more refined (*target segment length* = 0.625m).

Next, move to the *attachments* page. There are 20 attachments from an arc length of 81.25m to 128.75m. The spacing is 2.5m which is the same *pitch* considered for the distributed method. The finer segmentation of 0.625m is to make sure there is a node at each location for each module to be attached to. Smaller segments also mean that four segments exist between each module, to help assess the riser sag in between modules.

Each attachment is called *Module* and represents an individual buoyancy module. To see the properties, select *attachment types* from the bottom of the data form. This will take you to the *attachment types* data form. On the *clump types* page, set the *view mode* to *individual* to see all the module data at once.

The properties are not quite the same as those used by the line type wizard:

- Line type wizard: total drag = drag of modules + drag of exposed riser length
- Line and attachments: total drag = drag of modules + drag of total riser length

Therefore, the attachments should only contain the additional properties the modules would add to the bare riser; otherwise you would be applying loads twice.

In addition, notice the *align with* option is set to *line axes*. The module z-axis will align with the line node z-axis and move with it. This is appropriate for a module that wraps around a line. The alternative (*global axes*) is more appropriate for clumps attached by a swivel or tether i.e. those which do not rotate with the line.

The detailed profiling within the module is not included in this example. You could add a curved plate shape to each module to model this refinement. If so then the clumps will need to be replaced by *6D buoy* objects. This is a far more complex model which should only be built if detailed analysis of certain effects is essential e.g. "curtain-railing".

Results

If you have reset the simulation then you must reload the simulation file in order to access results.

Look at the animation through the latest wave. The blue line section represents the distributed buoyancy, the pink clumps represent the individual module attachments. The two lines are moving in unison, even though the plan view shows waves being produced at the hangoff and moving down the line due to vessel surge motions.

Look at the *curvature* range graphs for both lines through the latest wave. The lower plot is the distributed method and the upper plot is the discrete method. Note that these curvature plots show the magnitude of the curvature vector and so are rectified, i.e. always positive. Therefore the curve for the trough region has been reflected to become a peak.

There are sudden changes in curvature at about 80m and 130m: these are points of inflection where the buoyed length begins and ends. The other sudden drop is around 165m, the touchdown point. The variation of curvature in the discrete buoyed region shows the "curtain-railing" effect. Each little peak is where a buoyancy module is located.

There is also a slight curvature increase near the seabed termination. This is because the anchored end (end B) is being held exactly on the seabed. In fact the riser would sink into the seabed slightly and the node in front of end B is able to do so. Hence the slight increase of curvature. This slight rise is not significant to the system response.

The two range graph plots show there is little difference in extreme curvature between the two modelling methods.

Steep wave riser

A steep wave configuration is similar to the lazy wave but it has a near vertical connection at the seabed. 'Steep' means that the riser centreline is near vertical at the lowest end while 'wave' describes the line shape as a result of the buoyancy modules.

When you open the file you will see four views of the system. The riser extends from the side of the vessel down to a pipeline end manifold (PLEM). The plan view shows weather is near head seas for the vessel but normal to the riser plane – a transverse case. This is typical for West African systems.

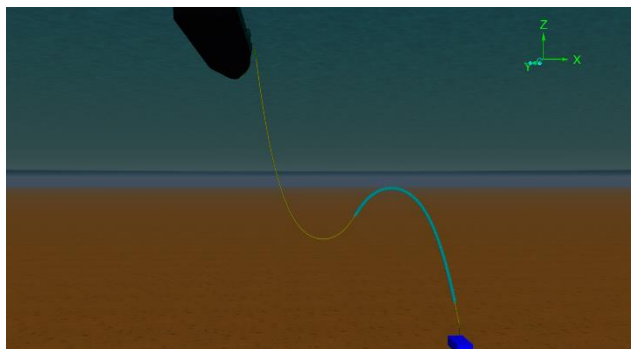
The two detailed shaded views at top and bottom right show bend stiffeners (BSRs) located at the hang-off and PLEM connections respectively. There are two methods available for modelling BSRs in OrcaFlex; both are demonstrated here for comparison.

Building the model

Two duplicate risers are included in this model, one for each style of BSR. When you first open up the model, one of them is hidden. Make sure the *model browser* is in *view by groups* mode; this is set by right-mouse clicking in the *model browser* and checking that the *view by groups* option is ticked.

There are two groups shown on the bottom of the *model browser*: *Separate Line BSRs* and *Attachment BSRs*. Select the *Separate Line BSRs* group and press *Ctrl + H* to show the objects in this group. The two systems are modelled directly on top of each other, so very little changes in the model views except for red coloured BSRs now being visible on top of the green coloured ones in the two right-hand views.

Press *Ctrl + H* again to hide this group while we first consider the *Attachment BSRs* group in detail.



In each group, the riser is built in a similar manner as for a lazy wave (discussed previously), however the 'steep' configuration means that the lower end is vertical rather than horizontal when it reaches the seabed.

The buoyancy has been applied by distributed properties but it could also be modelled as discrete modules if required; see the lazy wave example for details.

The PLEM is represented by two shapes set as *drawing* types. They have no contact stiffness so will not interact with the riser; they are present for visualisation only.

Go to the *10" Steep Wave* line data form. The *z* coordinate for end B says 5m but the *height above seabed* says 4.822m. This is because OrcaFlex assumes lines have hemispherical ends and so for clearance purposes the height above seabed still takes account of line contact radius. However in this example the *z* coordinate is what we are interested in because it defines the pivot point for the line.

Both line ends in this example have built-in connections (infinite connection stiffness). Local bend protection is provided by bend stiffeners (BSRs), included in the model.

Modelling bend stiffeners as attachments

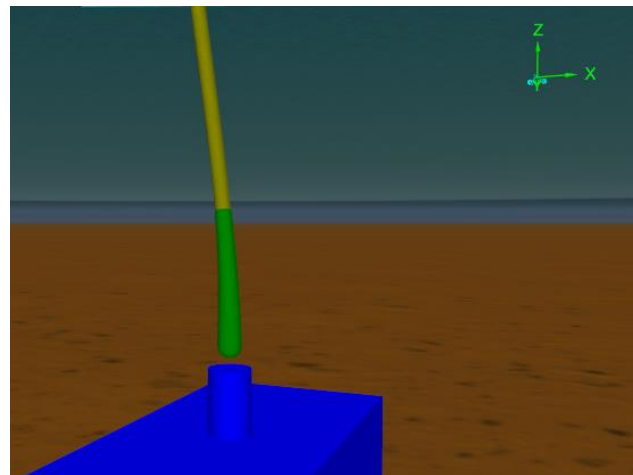
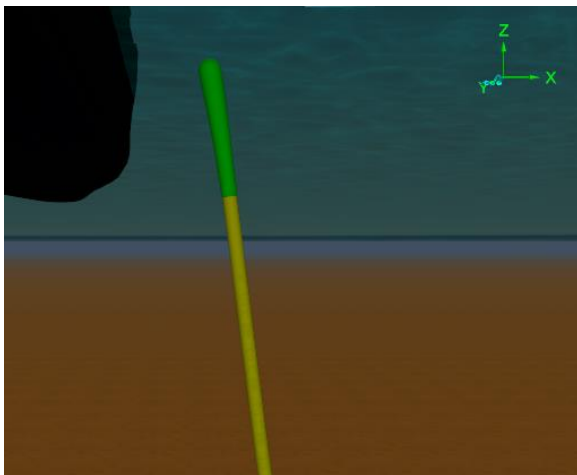
The simplest way to model a BSR is as an attachment to a line. This method allows you to model the polymeric section of the BSR, but not the rigid steel mount section. So, if fitting loads are required they must be obtained through a separate calculation outside of OrcaFlex. Go to the [attachments](#) page of the *10" Steep Wave* line data form. Two BSRs are attached, one at end A (hangoff) and one at end B (PLEM). Each has been given a unique name (the *name* column of the table) so they are easily identified when extracting results.

Now look at the [structure](#) page. The riser is made up of six sections which represent different regions of the riser: BSR, suspended, sag, hog, suspended, BSR. Select the [profile graph](#) button at the bottom of the form. This will show you how the diameters of the riser vary. You can see the red line showing the riser. The buoyed section is very clear with its much larger OD. The blue lines at right and left are the BSRs. You can see how they taper and wrap around the riser.

It is very important to remember when creating a BSR in this way that the refinement of the BSR depends on the segmentation of the section it wraps around. If the section has 0.5m segmentation then that is also the segmentation of your BSR, even if you input BSR profile data at 0.1m intervals. Always check this profile plot to check if the resulting profile is appropriate. It is also a good way to check that your BSRs are the right way round, i.e. with the root at the end of the line.

The BSRs can be clearly seen in the shaded views. Views can be switched between wire frame and shaded by selecting [Shaded graphics mode](#) from the [View](#) menu, or by using the keyboard shortcut, [Ctrl+G](#).

The views below show the hang-off and the PLEM BSRs (both in green). You can see how the bend stiffener tapers as it extends away from the line end.



To see the attachment properties, select the [attachment types](#) button at the bottom of the line data form and go to the [Stiffener types](#) page. Each attachment has properties determined by a line type and a length of 3m, as determined by the BSR profile – see below.

To see the physical properties of the BSRs, close the attachments form and select the [line types](#) button instead. Take a look at the line properties for the lines [Topside BSR Properties](#) and [Seabed BSR Properties](#).

On the [Category](#) page, they have been given the [homogenous pipe](#) category, with typical BSR polymer material properties, so OrcaFlex can calculate the appropriate mass, stiffness, etc.

As BSRs typically have an outer diameter that changes with arc length, it is important to be able to model this geometry. Looking at the [Geometry, mass & expansion](#) page you will see that the outer diameter is a variable data item rather than a constant value. To see the details right click on the form and select [edit variable data](#) from the menu. You will now be in the [variable data](#) form, which can also be opened directly from the [model browser](#).

One of the [data source type](#) options, listed on the left-hand side of the form, is [line type diameter](#). This contains two variable data sources, one for each BSR. They have identical profiles except [Seabed BSR OD](#) is the reverse of [Topside BSR OD](#). This is because Topside BSR has its root closest to end A while the Seabed BSR has its tip closest to end A. The [reverse](#) button at the bottom of the page allows OrcaFlex to invert the table automatically for you.

Select [profile](#) at the bottom of the table to take a look at the distribution. Note also that each OD table gives values up to an arc length of 3m. This maximum arc length is what determines the attachment length so you do need to input the data for the whole polymeric BSR length.

Back on the [structure](#) page of the line data form, note that the first and last sections of line have fine segmentation. These sections are 5m long, the first 3m of which are protected by the BSRs. The fine segmentation has been deliberately continued for 2m beyond the tip of the BSRs. It is good practice to check the detailed curvature at the point where the riser exits the BSR, because over bending can occur just beyond the BSR tip if it is loaded beyond its design capability.

More details on BSRs, and bend limiter modelling can be found in the [Modelling, data and results | Lines | Modelling bend restrictors](#) section of the OrcaFlex help.

Modelling bend stiffeners as separate lines

If you require more detail in your bend stiffener model, for example if you need to include the steel root section, or model the 'rattle space' or friction effects between the BSR and the protected line, then you need to model the BSR as a separate line rather than as an attachment.

In this method, we utilize the [line contact](#) model to define the relationships between the stiffeners and the protected line. This method allows the BSR to be modelled as separate sections to represent both the steel mount and the profiled cone section.

In the [model browser](#), hide the [Attachment BSRs](#) group and show the [Separate Line BSRs](#) group. The BSRs are now shown in red, with a short grey section (the steel root) at the end where they connect to the vessel or the PLEM. The two BSR lines are called [TopEnd BSR](#) and [Seabed End BSR](#).

Open the data form for the line [TopEnd BSR](#), via the [model browser](#). If we look at the [structure](#) page, there are two sections in the line: one of line type [Steel Root](#) and one of type [BSR Cone](#). These line types have been defined on the [line type](#) data form, and in fact the [BSR Cone](#) line is identical to the [Topside BSR Properties](#) line used in the attachments method, with the exception that it has been made a different colour, for clarity.

Now look at the line data form for either BSR line. On the [structure](#) page, note that the BSR Cone section of line has been given relatively short segments, 0.25m in length. This allows the changing properties of the BSR to be modelled accurately, as it reduces in diameter down its length. Both BSRs

have end A built-in to their connection points (they have 'Infinite' end connection stiffness) and end B is *free*. The *end orientation* of end A has been set appropriately, as discussed in the catenary riser example. The BSR lines also *include torsion*, as indicated by the radio button at the top of the line data form. Torsion is required by the *line contact* model, which is discussed below. The end positions of both BSRs take into account the 0.5m steel root section.

Look at the *contents* page and note that the BSR lines have *free flooding* contents.

Close the data form and reset the model (press *F12*). The two right-hand views of the BSRs now show that the BSRs and riser lines are separate lines and are hanging independently. Run statics (press *F9*) and you'll see the riser quickly gets pulled inside the BSRs. This happens because a *line contact* relationship exists between them.

Double click on the *line contact data* item, located towards the top of the *model browser*. A relationship is defined on the *relationships* page for each of the BSRs; these specify that the *10" Steep Wave1* line is positioned *inside* each of the BSRs. The *containment enabled* box is ticked for both, indicating that the inner line (riser) is shielded by the outer line (BSR). *Stiffness* and *friction coefficient* values are also assigned here. The *line contact* model is described in more detail on the [Modelling, data and results | Lines | Line contact | Modelling](#) page of the OrcaFlex help, and in the example [D04 J-tube pull-in](#).

Note that the option exists on the *line contact data* form to have each line contact relationship enabled (i.e. the *enabled* box is ticked) or disabled (box not ticked). This is useful when problem solving, for example when trying to solve statics convergence problems. Sometimes it can help to see how the model solves without a relationship active, in order to help determine what is causing the convergence issues.

Results

Re-open the simulation file and load *A01 Steep wave riser attachment BSR.wrk* from the *Workspace* menu. This will present the curvature and bend moment range graphs for the attachment type BSR at the hang-off, and the portion of riser that is inside it. The right-hand graphs are for the riser, the left-hand graphs for the BSR. The curvatures for the two lines are very similar (top two plots), as you'd expect, however the bend moments for the two lines are very different (bottom two plots), showing that the BSR experiences significantly more of the load than the riser does, as it is designed to do.

Open the *A01 Steep wave riser line BSR.wrk* workspace to see the same results for the separate line type BSR. This time the curvature plots are quite different. The steel root of the BSR is clearly visible, with zero curvature (remember the steel root is not modelled with the attachment method). Also, the riser is free to move in a different manner to the BSR as there is a small annular clearance between the two. The separate line model allows the two lines to move independently, limited only by the contact between the outer diameter of one and the bore of the other, whereas in the attachment model the BSR and riser act as one, with no relative movement between them.

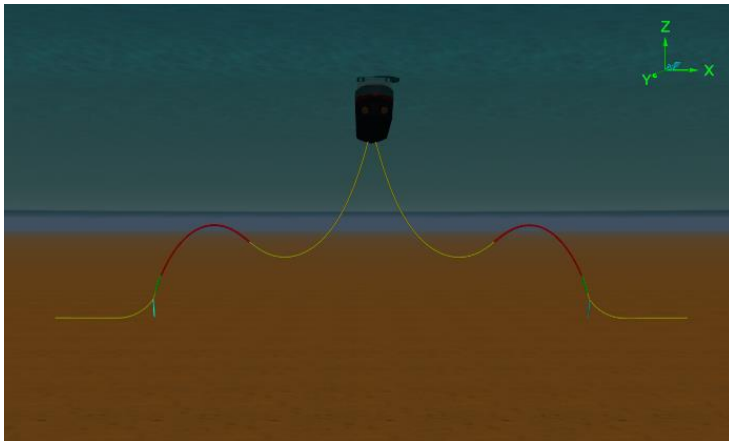
Pliant wave riser

A pliant wave configuration is a lazy wave with the addition of a tether restraining the touchdown point. A clamp acts as the interface between the tether and the riser. This example shows two methods of modelling the tether arrangement:

- Simple Tether (Port side): The riser is a continuous line. A link restrains the riser and a clump attachment gives the clamp properties. This is sufficient for most cases. The link component has no mass or hydrodynamic properties but if the link is small compared to the riser length this is not expected to make significant changes to the response. If more detail is required, then the tether could be modelled using a line instead.
- Detailed Tether (Starboard side): The riser is split into two lines connected together by a 6D (6 degrees of freedom) buoy that represents the clamp. This allows you to determine bend protection requirements either side of the clamp and look at more complex tethering arrangements. In this example we have modelled the tether using a line, to give more detail in this region.

When you open the file, you will see a series of views of the system. To the left are a shaded side elevation and a wire frame plan view of the two risers. To the right are wire frame elevation views of the tether attachment methods. The upper view shows the simple method, the lower shows the detailed method.

Building the model



Make sure the *model browser* is in *view by groups* mode.

As with the preceding examples, this is not a coupled analysis. Risers in shallow water have negligible effect on the motions of an FPSO so a fully coupled analysis is not required.

Instead the vessel motions are applied through displacement RAOs and moorings need only be included if interference is of interest.

The plan view in the OrcaFlex model shows the vessel offset out of the riser plane. This represents the mean and 2nd order offsets due to the environmental loads, which are applied as the vessel initial position.

The 1st order motions are then applied in the dynamic simulation using vessel displacement RAOs. These oscillations are applied about the initial position.

The risers extend from the vessel to the seabed in a wave formation and have an additional tether restraint near the touchdown. The risers extend to Port and Starboard of the vessel, and so will experience the same loading in the head sea conditions applied here. This allows for easy comparison.

The riser hog bend has distributed buoyancy properties in this example but discrete modules could be used if preferred. See the lazy wave example for details of this.

Open the *10" Pliant Simple* line data form via the *model browser*. On the *structure* page there are two smeared buoyancy sections, *10"P+SmallBuoys* which is shown in red in the view and *10"P+BigBuoys* which is shown in green.

It is common practice to vary the size and pitch of modules along the riser length. Smaller modules closer to the hang-off can reduce the risk of compression ahead of the modules i.e. where the increased drag and inertia of the buoyed length means it cannot fall through the water as rapidly as the bare length. Larger modules closer to the seabed termination or tether (as in this example) increase the mean tension and reduce the risk of compression at the termination or the tether going slack.

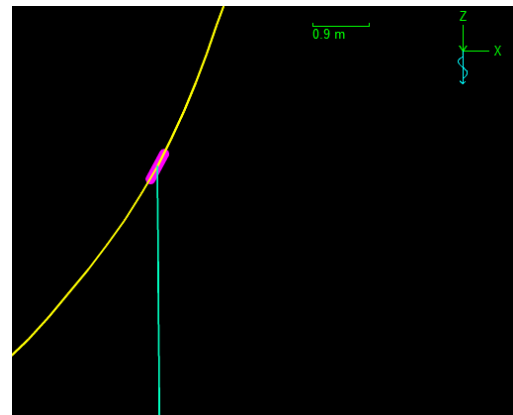
Go to the *line type* data form using the button at the bottom left of the line data form. Select either of the buoyed line types and select the *wizard* button at the top right of the form.

You will not see the full *line type wizard* settings unless you reset the simulation but the summary form you will see is sufficient to show what the buoyancy module settings were. The two line types have the same pitch but the small modules have diameter and length of 1.2m and the big modules have diameter and length of 1.45m.

Close the wizard and look at the *drawing* page of the *line type* data form. Each line type has a different colour and pen thickness so they can be identified easily in the view.

The components of the simple model are contained in the *Simple Tether* group in the *model browser*. This set extends to Port. The attachment is shown in the figure to the right.

The riser is a continuous line of 232.5m length and the clamp is a clump attached at a node 176m along it. See the *attachments* page of the *10" Pliant Simple* line data form to see this location. Section 4 of the riser stops at this arc length to ensure there is a node at that precise location.



Open the *attachment types* data form via the *attachment types* button at the bottom left of the line data form to see the clamp settings. The physical properties are for those of the steelwork alone. As for the buoyancy modules discussed in the lazy wave example, the clump attachment should only contain loads that are additional to the riser's loads.

Hydrodynamic properties for *Clamp Type* have not been set in this example because they rarely have a significant effect on the response of the system.

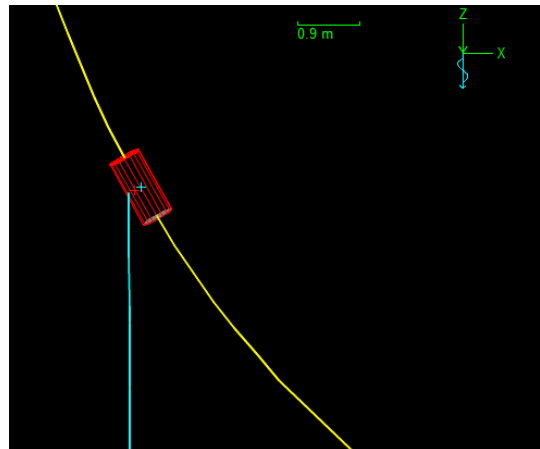
The tether is modelled as a 9.9m long link and attached at the same node as the clamp. Select *Tether Pliant Simple* from the *model browser* to open the link data form. The link type is *Tether* so it will apply tension when stretched but will go slack rather than enter compression.

This arrangement applies clamp and tether loads at a single point i.e. the node they are attached to. Curvatures local to this location will have lower accuracy.

If you are interested in the distribution of the local load and identification of any bend protection requirements, then a more detailed model might be required. An example of this is in the [Detailed Tether](#) group. The clamp arrangement is shown in the figure to the right.

Here, the riser is split into two separate lines, above and below the clamp. They are connected via a 6D buoy. The end connections are built-in so that moments are transferred from one riser length, to the other, via the buoy.

This buoy represents the clamp and so has the properties of the clamp, the riser within it and also any riser contents.



As the clamp is 1m long, each riser length has been shortened by 0.5m so the total length is still 232.5m. Hydrodynamic properties assigned to the buoy exclude any contribution from the clamp itself but include contributions from the riser to allow comparison with the simpler clamp method above.

You could model the clamp as a line type, and therefore model the entire length as a single line, but the addition of a 6D buoy allows convenient extraction of bend protection loads either side.

In the detailed model, the tether is now a line ([Tether Pliant Detailed](#)). Its connection is offset from the centreline of the riser by 0.2m so its length has been shortened by the same amount. The tether properties were also produced by the line type wizard but this time the *special category* was *rope/wire*.

Look at the [limits](#) page of the [line type](#) data form. You will notice that *compression is limited* has been ticked for the [Tether](#) line type. This means that the line will go slack rather than enter compression, just like the link. The tether axial stiffness also matches the stiffness of the link used in the simple version.

The detailed version therefore allows assessment of the resulting loads on the clamp and the effectiveness of any intended bend protection. Bend stiffeners can be modelled using either of the methods shown in the steep riser example.

Results

The views compare the two modelling methods. The detailed views to the right are of the simple method (top) and the detailed method (below). The resulting configurations are very similar.

An efficient model of a system will contain details critical to the system response but simplify or exclude details that have negligible effect. In this example, the simple clamp had no hydrodynamic properties while the detailed clamp included them. The difference in results is negligible, so clamp hydrodynamics could be excluded.

Now load [A01 Pliant wave riser curvature.wrk](#) from the workspace menu. It will present four curvature range plots. On the left are the results for the length above the clamp. The upper plot is for the simple configuration, the lower plot for the detailed configuration.

To make the axes match, double-click on the upper plot to access the *graph properties form*. Set the Y axis maximum value to 0.12 on the *Axes* page then select OK. This shows there is slightly more curvature in the detailed model at the clamp, but other differences are small.

The right hand plots are for the length below the clamp. There is a slight difference in curvature in the touchdown region (10m to 30m on the detailed plot) but again this is small.