A05 Steel catenary riser systems

Introduction

This example contains three examples of steel catenary risers (SCRs). These are:

- Catenary with spar.
- Catenary with semisub.
- Lazy wave with FPSO.

The example also illustrates:

- User specified units.
- Homogeneous pipe properties.
- Adding coatings and linings to lines.
- Modelling strakes.
- Modelling tapered stress joints (TSJs).
- Modelling flex joints.
- API RP 2RD stress code check.
- Vessel drawing techniques.

Catenary with spar

Open model *A05 Catenary with Spar.sim*. A steel riser descends from a spar platform to the seabed in a simple catenary configuration transitioning to a flowline after 7800ft. The riser is hung off the spar at the soft tank with a flexjoint, and the upper 4000ft of riser is straked to suppress vortex induced vibration (VIV). Note VIV analysis is not considered in this example.

When you open the simulation file, you will see a wire frame elevation view of the system and three results plots.

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.
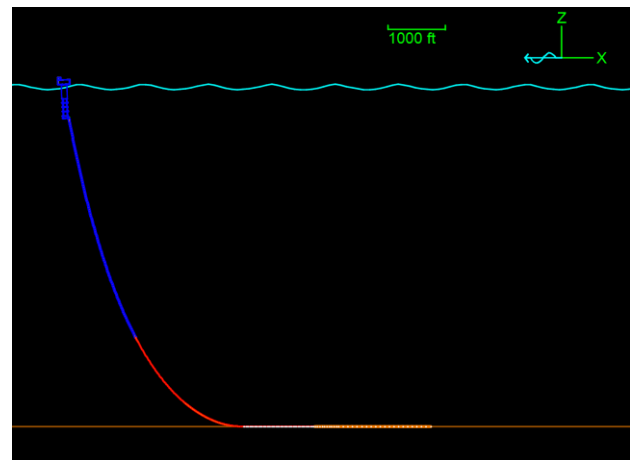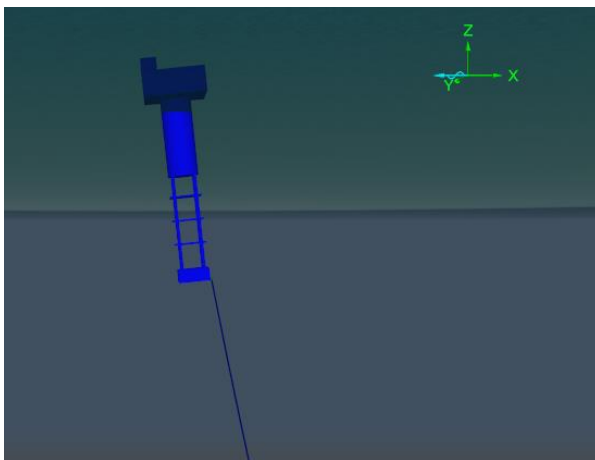
Building the model

Details on constructing a simple catenary configuration are contained in the *Catenary riser* example of the [A01 Catenary and wave systems](#) set. This discussion will only consider differences from that example.

The example has been set up with user defined units. Open the *general* data form via the *model browser*. OrcaFlex allows you to use SI, US or user defined (customised) units. If the units are changed, then OrcaFlex converts all the data in the model into the new units. This is useful if project data is supplied in a range of units. Data can be input in the units supplied, then the model can be converted back to the preferred unit system. More information can be found on the [Modelling, data and results | General data](#) page of the OrcaFlex help.

The *model browser* shows a *Spar Group* and a *Riser*. The *Spar Group* contains a vessel object called *Spar* and a number of shapes. The shapes are all *drawing* type, which means they are for visualisation only, generating the spar appearance seen in the wire frame and shaded views. The *Spar* vessel object is hidden in all the views. *Hide* and *show* options are accessed by right clicking on an object in the *model browser* and selecting from the dropdown menu (or by using the keyboard shortcut *Ctrl + H*).

Now double click on *Spar* in the *model browser* to access the *vessel* data form. The vessel has been offset on the same heading as the waves and current. There is also a trim of 6º with the weather.

The offset and trim represent the mean and 2nd order contributions from environmental effects not explicitly calculated in the analysis e.g. the wind, wave drift and current.

The *calculation* page shows that static analysis and primary motion are turned off. The vessel 1st order dynamic motions are included through displacement RAOs specified by the superimposed motion. These 1st order dynamic motions are applied relative to the spar offset position.

Select *vessel types* at the bottom left of the form and the *vessel type* data form will open. The *displacement RAOs* page shows data has been supplied for wave incident angles of 0º, 45º and 90º only.

Now look at the *conventions* page. At the bottom of the page symmetry has been specified as being in the xz and yz planes. These planes are both vertical because the OrcaFlex vessel z-axis points upwards. OrcaFlex will determine the RAOs for the required heading based on these rules of symmetry.

Now open *Riser* from the *model browser*. The riser line data form shows the top connection (end A) attached to the spar and the bottom connection (end B) on the seabed.
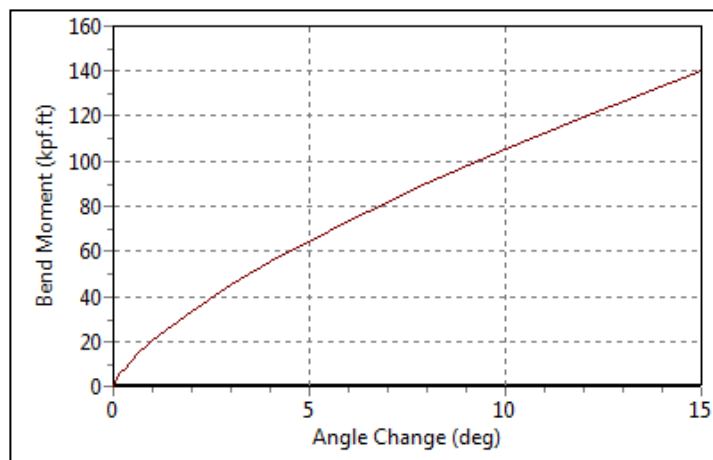
This example has a flex joint at end A, which is modelled in two parts.

The *connection stiffness* table states that end A has variable bend stiffness *Flexjoint Stiffness*. Right click anywhere on the form and select *edit variable data* from the dropdown menu.

The *profile* button at the bottom right of the *bending connection stiffness* data source type (see the *Structure* folder) shows the specified nonlinear relationship between angle change and bend moment. End A has a no-moment direction 8º from vertically down and pointing towards end B (azimuth 0º and declination 172º). Angle changes are calculated relative to this no-moment direction.

It is important to input the correct no-moment direction and bend connection stiffness relationship because they will affect the stresses generated near the top of the riser.

Close the *variable* data form to return to the *line* data form. The remaining properties of the flex joint, both physical and hydrodynamic, are represented by a line type – named *Flexjoint* – which is applied to the top 9ft of the riser. Note that this is one method of creating a flex joint, which is appropriate when the joint coincides with a line end. In OrcaFlex you can also create mid-line flex joints by using the flex joint *attachment type*: see the help file section Modelling, data and results | Lines | Attachments | Flex joints for further details.

Move to the *contents* page of the *line* data form. Contents density and internal pressure have been specified. Both must be set correctly to obtain the correct wall tension along the riser length.

Move back to the *structure* page. The rest of the line is made of *Straked*, *Riser Pipe* and *Flowline* line types. There are several sections to each line type. These allow the segmentation to be optimised. More detail is included in regions where significant curvature is expected or where a change in line properties takes place. Less detail is needed in the regions where the line has less curvature and is not subject to significant variation in loading during dynamics.

Select *line types* from the bottom left-hand corner of the *line* data form. Set the *line type* data form *view mode* to *all* then select the *category* page. Note that all four line types have been set as *homogeneous pipe*. This means OrcaFlex will determine the overall properties based on the dimensions and material properties we supply.

The construction of a *homogeneous pipe* is a load-bearing pipe with non-load-bearing coatings and linings added. Further details about this can be found on the Modelling, data and results | Lines | Homogeneous pipe page of the OrcaFlex help.

The *geometry, mass & expansion* page shows the outer and inner diameter of the load bearing pipe and its material density. The *structure* page is where the material Young's Modulus is specified so pipe stiffness can be calculated.

The *coatings and linings* page shows the additional layers applied. These will change the outer and inner diameters and increase the mass accordingly. The *Flexjoint* line type has no coating or linings, while the *Riser Pipe* and *Flowline* have about 0.25" of abrasion protection. However, the *Straked* type has two coatings, the abrasion protection and the strake. Therefore, it has a variable coating thickness called *Coating and Strake* applied.

Right click anywhere on the form to access the *variable* data form. The *coatings or linings* data source type shows the properties of these layers. The layers are listed in order of application, so the first layer is the abrasion protection and the second is the strake. The strake thickness has been set to give the correct displacement. The actual diameter of the structure would be greater than this.

Close the *variable* data form and move to the *drag & lift* page of the *line type* data form. This shows a high normal drag coefficient for the strake (= 1.6), as would be expected. The drag coefficients use the diameter of the coated pipe as the reference; this diameter is shown in the *drag / lift diameter* columns. A value of '~' indicates that no coatings have been applied and the bare steel diameter, specified on the *geometry, mass & expansion* page will be used. You may need to adjust your drag coefficient if its source uses a different reference diameter. For further details, see the Modelling, data and results | Lines | Line types | Drag & lift data page of the OrcaFlex help.

Now select the *code checks* button at the bottom of the data form. Various code checks have been built into OrcaFlex; in this case we are using *API RP 2RD*, so click on this tab to display the details. In this case the yield stress (SMYS) has been set for steel and a design case factor (Cf) of 1.2 applied (appropriate for extreme conditions).

Return to the *model browser* and open the *environment* data form. The data on the *waves* page shows that this example is a regular wave analysis. The default Dean Stream theory has been selected because this is robust for a wide range of water depths. The *current* page shows the current direction is collinear with waves in this example i.e. they have the same direction. Note that OrcaFlex applies wave and current *towards* a given direction.

On the *seabed* page the normal and shear stiffness have been increased from default to represent a stiffer seabed. Seabed stiffness has a significant effect on SCR fatigue so these settings must be set carefully.

In this example, a linear seabed has been applied. OrcaFlex has the option for a more complex nonlinear seabed, but it is recommended this only be used on a case-by-case basis. This seabed model requires a significant increase in computational effort. More details on the seabed options can be found on the [Theory | Environment theory | Seabed theory](#) page of the OrcaFlex help.

Now move to the *general* data form via the *model browser*. The *analysis* page shows a build-up period (stage 0) of 1x wave period, then a run for 5x wave periods (stage 1). This is usually sufficient to obtain a regular wave settled response for most systems.

The *dynamics* page shows that a fixed step implicit time integration scheme has been applied. This is the default setting and will provide fast, good quality results for typical SCR systems.

Results

Three results plots are shown in OrcaFlex. The right-hand plots are instantaneous range graphs of utilisation along the riser length. Utilisation is a result from the API RP 2RD code check and a value greater than 1.0 indicates the limit has been exceeded. The lower plot shows the variation along the full riser length from the hang-off, at 0ft, down to the seabed termination at 9810ft. The upper plot shows the response for the first 50ft of riser.

The lower left-hand plot is of utilisation against *end force Ez-angle* at the flexjoint. The *end force Ez-angle* is the rotational deflection from the no-moment direction. As would be expected an increased angle gives an increased utilisation. However, the nonlinearity of the flexjoint stiffness means at greater angles the increase in utilisation is less severe.

Stress results are also calculated, and plots can be produced from the results form (*results* dropdown menu).

Run the replay via the *replay* dropdown menu. The instantaneous range graphs are synchronised with the view to the left and show how the utilisation distribution changes with time. Note the step change in utilisation at the transition of flex joint to riser and riser to flowline.

Variation with time is most significant at the hang-off and touchdown regions. This is typical for an SCR configuration.

The plot of utilisation against *Ez-angle* is for the latest wave. A moving cross hair will appear on the plot when the replay reaches that latest wave. This cross denotes where on the plot the animation has reached.

Catenary with semisub

Two steel risers descend from a semisub pontoon in a simple catenary configuration, transitioning to a flowline after 9210ft. The risers are hung off the outer edge of the forward pontoon. One SCR has a flexjoint connection at the hang-off; the other has a tapered stress joint (TSJ). Otherwise, the risers are identical.

When you open the simulation file (*A05 Catenary with semisub.sim*) you will see two wire frame views of the system. One is a side elevation of the configuration and the other is a detailed view of the semisub and connections. There are also two results plots on the right hand side; these are discussed later.
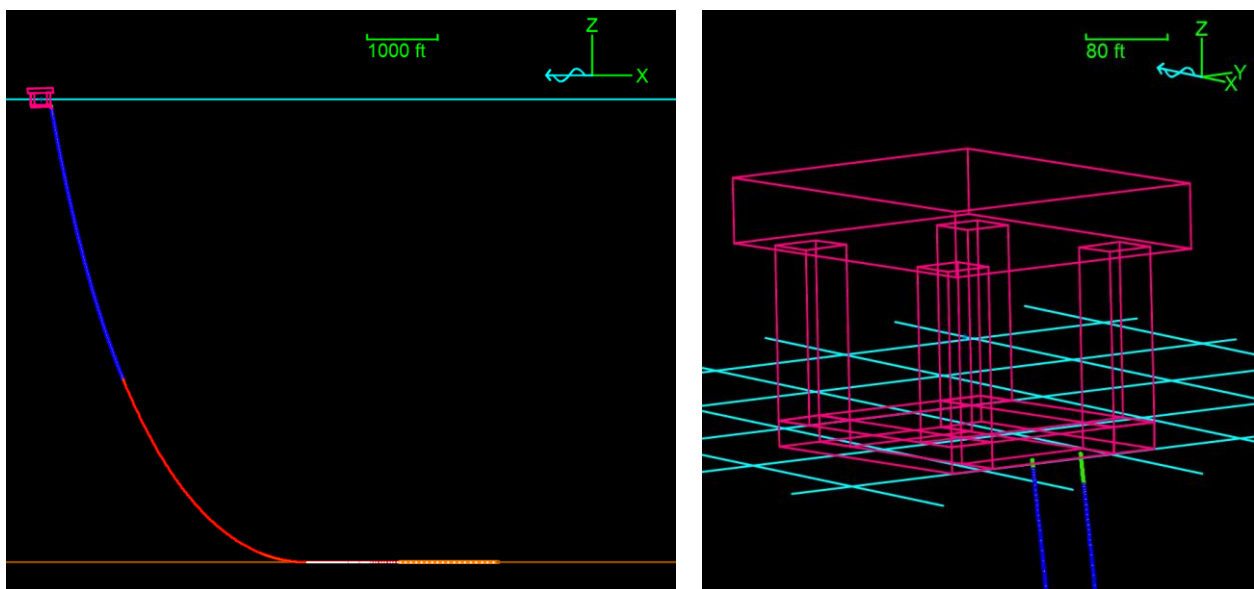
Building the model

The SCR configuration is similar to the one attached to the spar in the earlier example. In this section, only the differences will be discussed.

Both risers hang off the semisub with an angle of 7° from vertical. With the semisub offset on a far heading, by 10% of water depth, the riser deflection from vertical will increase. This results in increased curvature at the hang off, so local bend protection is used to prevent excessive stress.

The *Riser with Flexjoint* has local stress relief provided by a nonlinear end connection stiffness. This method is discussed in more detail in the spar example earlier.

The *Riser with TSJ* provides local reinforcement to the riser wall with a tapered stress joint. This is a line section with a tapered outer diameter. At the hang-off, the root of the TSJ has a larger outer diameter and it then decreases linearly to the tip where the outer diameter matches the rest of the riser. The section therefore needs to be assigned a line type with a variable outer diameter.

Open the *line type* data form via the *model browser*. Put *view mode* on *all* so you can compare the properties of line type *TSJ* with the others and move to the *geometry, mass & expansion* page. Instead of a constant outer diameter, *TSJ* has *TSJ Profile*.

Right click on the form and select *edit variable data* to move to the variable data form. The *line type diameter* data source type can be found in the *structure* folder. A table shows the TSJ outer diameter. The diameter order is defined following the usual line conventions (end A to end B). So, in this example the taper is root to tip. If you wanted to reverse this (because the TSJ was at end B for instance) then the *reverse* button at the bottom of the table will do this for you.

Close the variable data form and move to the *structure* page. Note that the bending, axial and torsional stiffness for the TSJ are not specified because they will vary along any line sections the line type is assigned to. OrcaFlex will calculate these properties accordingly.

Click on the *code checks* button. On the *API RP 2RD* page the design case factor, Cf, has been set at 1.0 for operational conditions and the corrosion thickness and SMYS data have been assigned for all lines.

Open *Riser with TSJ* from the *model browser*. On the *structure* page the first section has the line type TSJ. The section length is greyed out because this is specified on the *line type diameter* page of the variable data form.
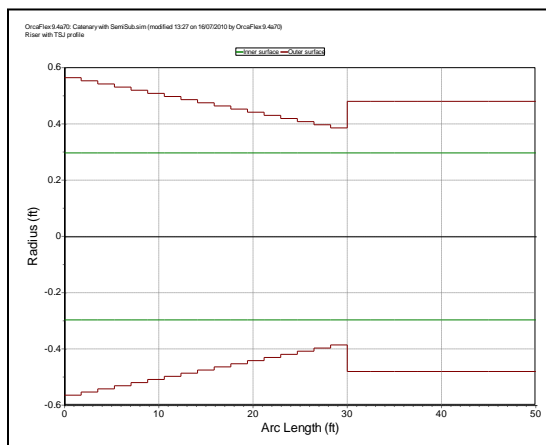
Select the *profile graph* button at the bottom of the line data form to see resulting riser diameter variation. Double-click on the graph and set the X-axis range to minimum and maximum values of 0ft and 50ft respectively. This shows the TSJ region more clearly.

The green line is the inner diameter while the red line is the outer diameter. The taper is modelled by a series of segments with reducing outer diameters. The refinement is controlled by the number of segments in that section.

The increase in the outer diameter, further along the line, is from the coatings and strakes on the riser.

The semisub is modelled as a vessel. Its visual representation could be generated by attaching shapes that have no stiffness. Instead, this example illustrates using the vessel type wire frame drawing.

Open the *Semi Displacement RAOs* vessel type data form from the *model browser* and move to the *drawing* page (there are many pages to this form so you may need to scroll along with the arrow buttons to find the *drawing* page). The semisub image has been produced by vertices connected by edges. If you have a mesh file for the vessel (as used in the diffraction analysis that generated the RAO data) then you can import this mesh into the drawing page, via the *import wire frame...* button.

For further details on this and the types of meshes that are supported, see the [User interface | 3D views | Importing wire frame drawing data](#) page of the OrcaFlex help.

In the shaded graphics mode, OrcaFlex will draw a solid shape based on the wire frame vertices and edges. When making use of this facility you need specify the wire frame with some care, as described in the [Modelling, data and results | Vessels | Vessel types | Shaded drawing](#) help topic.

In essence, OrcaFlex partitions the vertices into sets of connected vertices. Two vertices are deemed to be connected if there exists a path of edges between the two vertices. Then, for each set of connected vertices, the smallest convex shape enclosing the set is drawn. If you have concave shapes, then you need to subdivide your vertices into groups with no sharing of vertices.

Each 'box' of the semisub shape has therefore been given its own unique set of 8 vertices and 12 edges to produce the required shape in both wire frame and shaded view. The figures below show the shaded shape results with and without sharing.

Alternatively, you can again make use of the vessel's mesh file, if you have it, and import the panel mesh to create the shaded image. Shaded images in .obj and .x format are also supported. See the above-mentioned help file topic for full details.
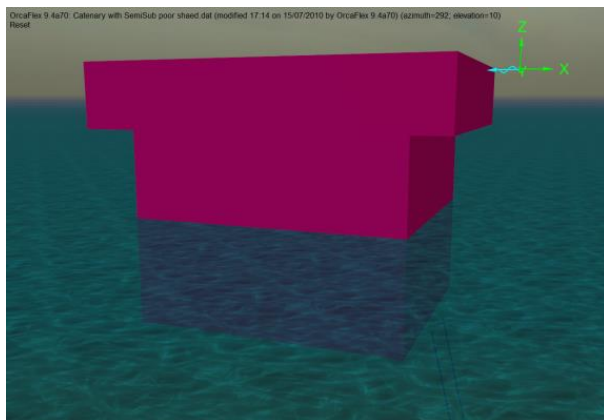

Results

Two results plots are generated by the default workspace when the simulation file is opened. These are API RP 2RD utilisation range graphs. They show the variation through the latest wave of utilisation along the first 100ft of riser extending from the hang-off.

The upper plot is for the *Riser with Flexjoint* line, the lower for the *Riser with TSJ*. They show different utilisation distributions, because of the way the devices work, but neither exceed the allowable limit of 1.0.

Lazy Wave with FPSO
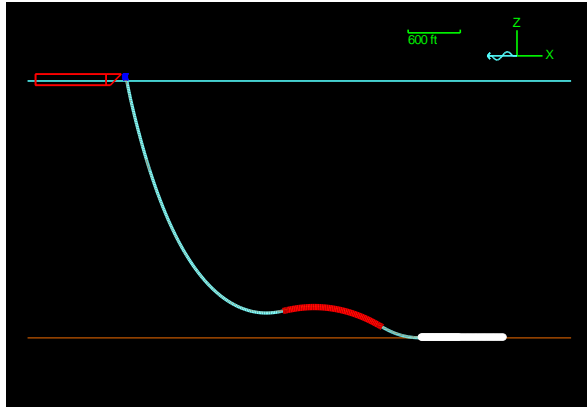
A steel riser descends from the bottom of an FPSO external turret in a lazy wave configuration. Buoyancy is applied to the riser length from 3515ft to 4705ft and a flexjoint provides local stress relief at the turret connection.

When you open the simulation file, you will see a wire frame elevation view of the configuration and two results plots.

The red section of riser shows the buoyancy location. The white section is seabed contact.

Building the model

Discussion on SCR properties and flexjoint modelling can be found in the *Catenary with spar* example earlier in this document. A detailed discussion on building a lazy wave configuration can be found in the *Lazy wave riser* example of A01 Catenary and wave systems.

The FPSO is modelled using OrcaFlex default vessel properties. The default vessel properties are for a 338ft long ship, while the vessel in this example is 984ft long. The vessel response has therefore been increased using Froude scaling. This facility is discussed in the example A04 Disconnecting turret system. In an actual project, these properties should be replaced by the real properties of the required FPSO, or the system response would be incorrect.

This example does not include a detailed turret that will rotate in relation to the vessel. It is simply modelled for visualisation as a *drawing* type shape.
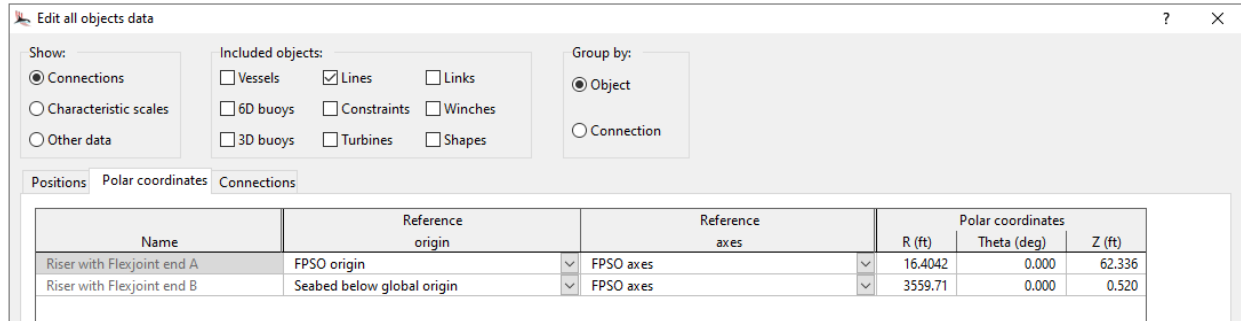
In reality, the turret will rotate about its vertical axis so that its orientation will remain reasonably constant when the vessel weathervanes. Weathervaning turrets can be produced in OrcaFlex and might be desirable for a mooring analysis, however for a riser analysis, this simplified model is usually sufficient. Consequently, in this model, the hang-off positions and orientations will need to be modified whenever the vessel changes heading between load cases. For further details about modelling a weathervaning turret, see example C03 Turret moored FPSO.

Turret hang-off locations are generally supplied in polar coordinates (radius and heading) relative to the turret centreline. It is therefore more convenient if OrcaFlex allows connections to be specified in polar coordinates too.

The vessel origin has therefore been moved to the turret centre at keel level. All FPSO data and connections have been adjusted to also be relative to this new origin. If the local axes are not already displayed, switch them on via the *view* dropdown menu, or using the keyboard shortcut *Ctrl + Y*.
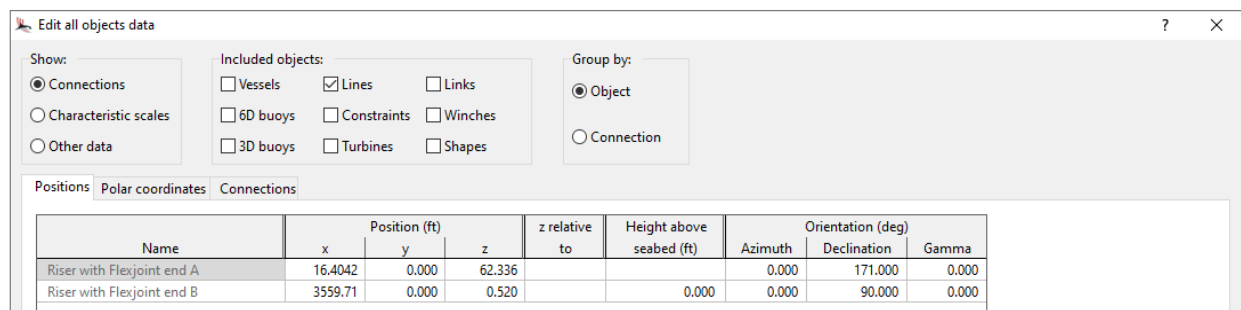
Polar coordinates are accessed in reset mode only so reset the model – via the *calculation* dropdown menu or by pressing *F12* – then open the *all objects data* form via the *model browser*. Make sure it is set up to *show connections* and *included objects* has *lines* selected. Now move to the *polar coordinates*

page. The connection position can now be specified as a radius (R), heading (Theta) and height (Z) from the vessel origin:

The resulting Cartesian coordinate positions are shown on the *positions* page.

Results

If you have reset the simulation, then reload the simulation file. The default workspace for this file will load the wire frame view and two range graph plots for the latest wave responses.

The plots show the distribution of API RP 2RD utilisation. The distribution over the range from hang-off to seabed is shown, together with a more detailed plot covering just the first 500ft from hang-off. The wave shape gives additional compliance to the system, distributing the utilisation over more of the length and so relieving the hang-off location.