A02 Midwater arch systems

Introduction

In this set of examples, three types of riser system are shown: a lazy S (both simple and detailed), a steep S and a pliant S.

Generally, an S configuration is similar to a catenary but has support provided by an arch structure at a midwater depth. The examples also show:

- Modelling of midwater arches.
- Contact between lines and with shapes, including friction.
- Modelling bridles and tethers.
- Using supports to model arch gutters.
- Setting the model North pointer.

Upon opening each simulation file, the default workspace will present views of the system in wire frame and/or shaded graphics and, in some cases, certain results of interest.

Lazy S simple

A lazy S configuration is similar to a catenary but has additional support provided by an arch structure at a midwater depth. The term 'Lazy' means that the riser centreline is near parallel with the seabed on contact, while 'S' describes the line shape as a result of the arch.

When you open the simulation file (*A02 Lazy S simple.sim*) you will see four shaded views. These show the midwater arch in plan, side and elevation views as well as a side elevation of the whole system. This example has three risers and an umbilical extending from a vessel to the seabed. All four lines pass over the same midwater arch.

There is a range of arch designs used in the industry. This example represents a structure comprising two buoyancy tanks with four gutters passing over the top. The example shows the basic modelling principles so the analyst can apply them to alternative arch structures too.
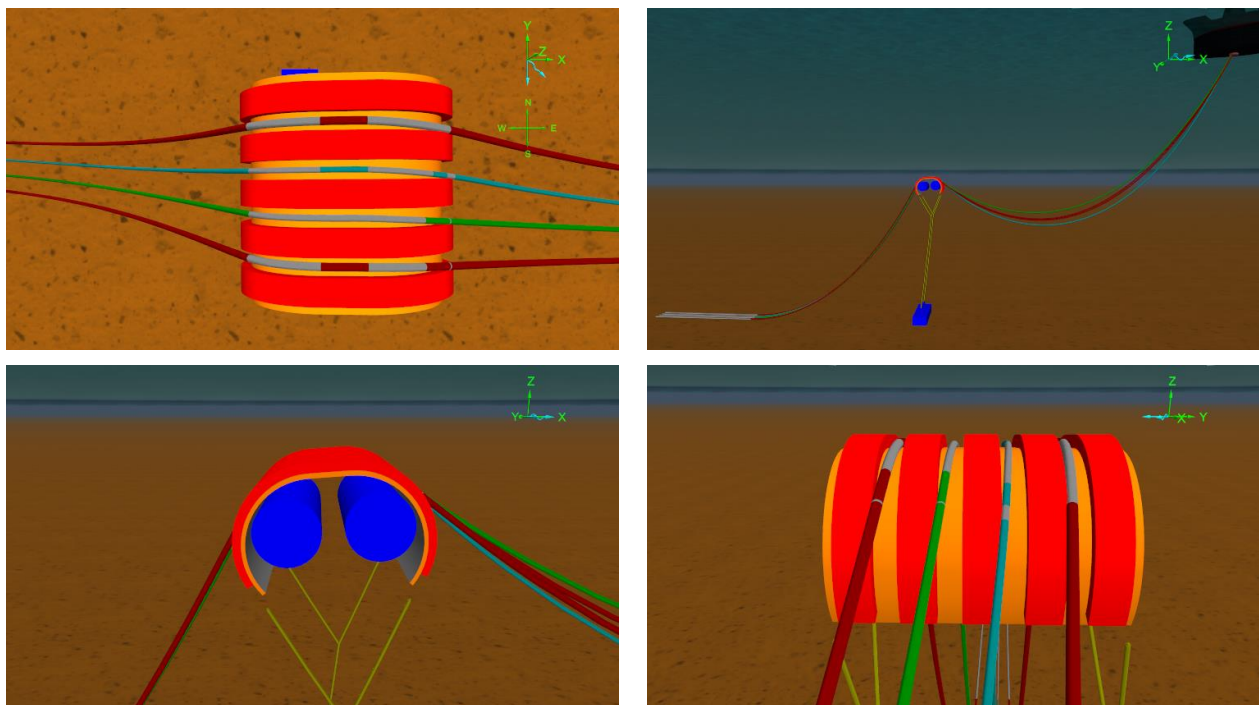
Building the model

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.
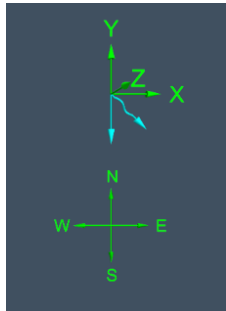
The arch plan view shows a compass. The settings for this can be found on the *drawing* page of the *general* data form. As there are a significant number of similar objects in this model, the object names are identified relative to these compass directions.

The plan view also shows the weather directions; current is out of the riser plane and waves are 45° off current. The vessel is head to waves with an offset collinear to waves. This offset is applied by moving the vessel initial position.

Riser clamp modelling

Each of the flexibles is connected to the arch using *mid-line connections*. This method means that each riser can be modelled using a single line, running from the vessel hang-off down to the seabed. Each line is clamped to the arch, in this case by a clamp that is 1m long and mounted to the top of the arch structure.

Open the data form for the line *10" riser N* and look at the *mid-line connections* page. You will see that the line has two mid-line connections, one at each end of the clamp, which effectively fixes a 1m long section of line in place on the arch.

On the *structure* page, the line has been split into multiple sections so that we can ensure that nodes are positioned exactly at the places where we want to apply connections (arc lengths of 135m and 136m). Splitting the line into multiple sections also allows us to consider shorter segmentation in the sections of riser that need it e.g. where it forms over the arch, and in its sagbend regions. Note that neighbouring risers have been given different upper catenary lengths to reduce the risk of contact when loading is out of the riser plane.

The *mid-line connections* could be made directly to the *Arch* 6D buoy, however in this instance we have chosen to connect both connections to a constraint object called *10" Riser N clamp*. We have done this because the constraint's connection load results will be the total of the loads acting on the clamp, and therefore we get useful design loads directly from the constraint, rather than having to sum the loads, from the line connection points, in some way.

A more detailed alternative is to use another 6D buoy to model the clamp instead of a constraint, which would enable you to include the physical and hydrodynamic properties for the clamp itself. You would then make the *mid-line connections* to the clamp buoy and use the buoy's connection load results to report the total clamp loads.

In either case, the use of *mid-line connections* allows moments to be transferred to the arch from the flexibles.

Arch modelling

The *Arch structure* group of the *model browser* includes several different model objects representing the arch, with many of the objects having been organised into sub-groups:

- *Arch* – a 6D *lumped buoy* which contains the physical and hydrodynamic properties for the whole arch structure.
- *Vertical support* – a sub-group of *elastic solid* type shapes that provide vertical restraint to the risers.
- *Horizontal support* – a sub-group of *elastic solid* type shapes that provide horizontal restraint to the risers.
- *Arch drawing* – a sub-group of *drawing* type shapes that provide the image of the arch structure.

The *Arch* buoy is currently hidden in this model (as explained below). The image here shows what the buoy would look like if it were visible in the wire frame view, on its own.

Open the *Arch* 6D buoy data form and look at the properties page. The buoy origin is at the top of the arch (clamp height) and its lateral position is mid-way along the arch. This is a convenient location when connecting the risers to the arch because their location is typically given relative to this point.
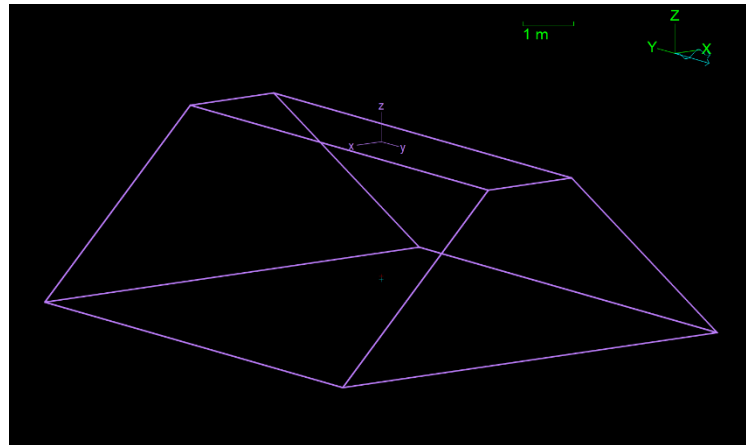
The *centre of mass* and *centre of volume* have been specified relative to this location too. Note that the position of the buoy origin and the centres of mass and volume can be visualised in the wire frame view. The buoy origin is shown by the axis system at the top of the 6D buoy. The *centre of mass* is just visible here as a small red cross, and the *centre of volume* is a small blue cross.

This provides a good check that the origins are positioned where you expect them to be.

On the *properties* page of the *Arch* 6D buoy, note that the hydrodynamic terms for *ROTATION* have not been set. For a large and well restrained structure, rotational terms tend to have little effect on the system response, so these data items remain at their default value of zero.

Note that the actual *mass*, *mass moments of inertia* and *volume* of the arch have been specified here too. It is important to apply the correct values here because they will contribute to the structure's net buoyancy, as well as its physical and hydrodynamic inertia in the dynamic analysis.

If the structure will trap water as it moves, then this trapped water should be included in the total *mass* and *volume* of the structure so that the physical and hydrodynamic inertia are correct.

Also, note that arches typically have complex shapes. Generally, reference shapes considered for hydrodynamic values / coefficients are simple and so the analyst needs to make assumptions during

selection. The overall hydrodynamic properties are therefore uncertain. It is recommended that sensitivity studies are carried out on the hydrodynamic terms used. This might include the rotational terms, if they are considered to be significant for a particular system.

The *Arch* and most of the *Support* shapes are hidden in the views. What you are seeing is a group of shapes in the *Arch drawing* group that give the appearance of the actual structure. These *drawing* type shapes are for visualisation only. This is a useful feature when presenting to 3rd parties who may not be analysts themselves, as it allows them to see a familiar visual representation of the system.

To see any hidden object (or group of objects) right-click on it in the *model browser* then select *show* from the popup menu. The same process can be followed to hide any visible objects / groups by selecting *hide* from the popup menu instead. Note that *hide* / *show* will only affect the model *view*; therefore, the object is still present in the mathematical model and will still affect the system response.

The *model browser* sub-group, named *Vertical support*, contains a series of shapes representing the bottom of the gutters. These are *elastic solid* type shapes with the appropriate stiffnesses assigned to them; meaning they will produce a reaction force on any line nodes which contact them. In the *Horizontal support* sub-group, some additional *elastic solid* shapes represent the side walls of the gutters that the risers pass through.

Shapes interact with the nodes of lines and so the segmentation, for each riser, is refined in the regions where the risers contact those shapes. Further details about the contact model, used by OrcaFlex for line node interactions with shapes, can be found on the [Theory | Shape theory](#) page of the OrcaFlex help.

The images below show the model with the *Vertical support* shapes (coloured blue) and *Horizontal support* shapes (coloured red) made visible in the model view. Note that the *Arch drawing* group has been hidden from view here.
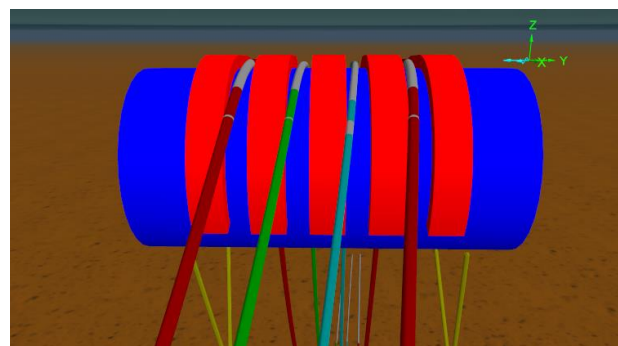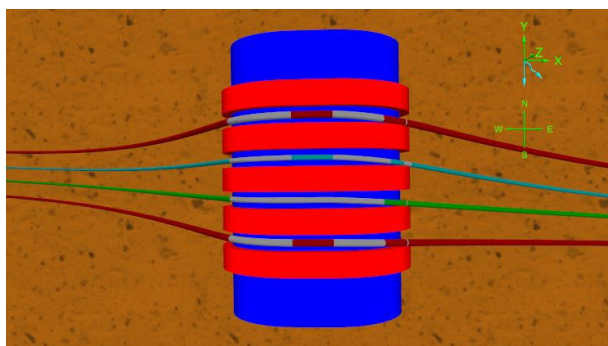
For the vertical support shapes, *Gutter W* provides a curved support to the catenary (extending from the arch down to the seabed). This is shown by the left-hand image below. The shape *Gutter E* provides support to the upper catenary (see the right-hand image) and the *Clamp* shape provides the flat part in between (middle image).

For this example, the bottom of each gutter has the same contact radius, so this group of three *elastic solid* shapes has been extended beneath all the gutters instead of a group being required for each one.


| *Gutter W* | *Clamp* | *Gutter E* |
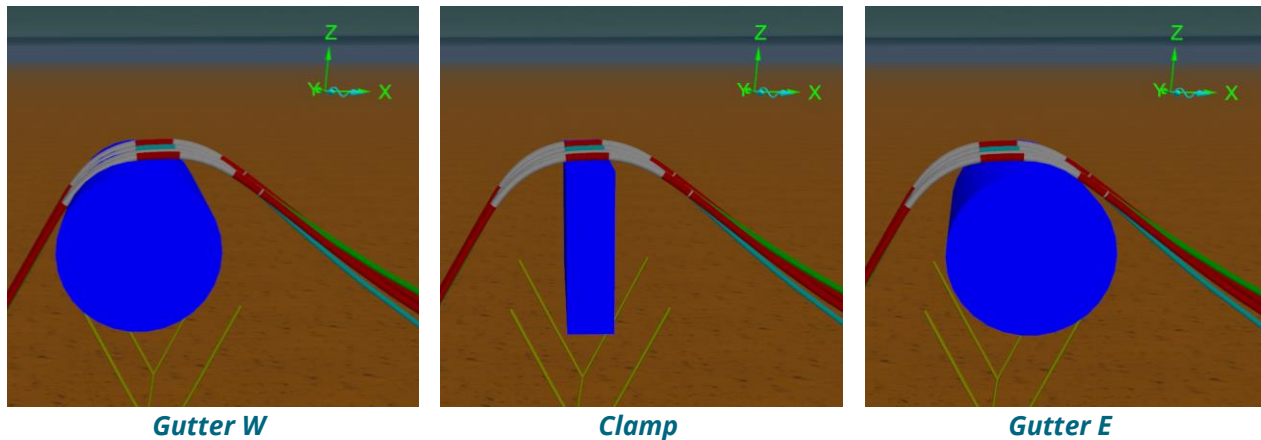
A very simple arch model might contain just these vertical support shapes; however, each riser would still be able to roll off the arch. This would mean that the only lateral restraint would be offered by friction and the bend stiffness of the riser. Riser curvature at the clamp location would therefore be highly over-estimated. Furthermore, any lateral moments would be transferred at the clamp connection(s) only, rather than being distributed along a gutter wall. This could produce errors in the 'yaw' response of the arch. Therefore, it is considered sensible to ensure the gutter walls are represented in some way.

Again, the *model browser* group, named *Horizontal support*, contains the shapes to represent the gutter walls. The walls in this example are very simple but should be sufficient for at least the first design iteration.

Each wall has its own *model browser* sub-group (*Wall 1*, *Wall 2*, etc.), with each containing shapes for the upper catenary, lower catenary and clamp regions; named *Wall up*…, *Wall dn*… and *Wall mid*… respectively.

The *Wall mid*… shapes use block shapes to represent their geometry. The curved gutter walls (*Wall up*… and *Wall dn*…) use *curved plate* shapes with a *radius* (4m) greater than the vertical supports (3.5m). The shapes also have an *included angle of revolution* (sweep) of about 130º. This is important because, if they swept a full 360º (to form a full cylinder) then they might interact with the line nodes on the tethers beneath the arch. This could provide additional, and unrealistic, restraint to the arch.
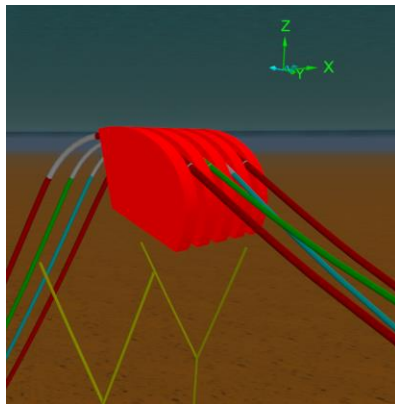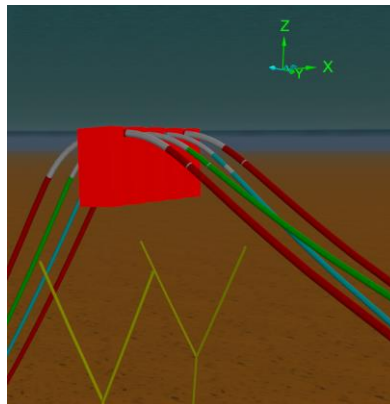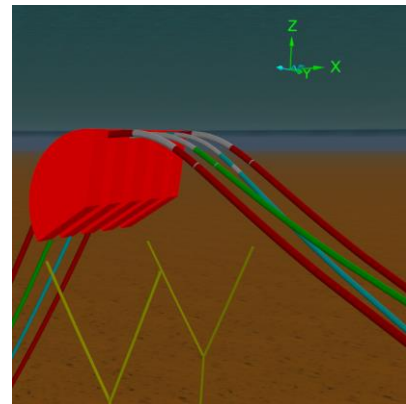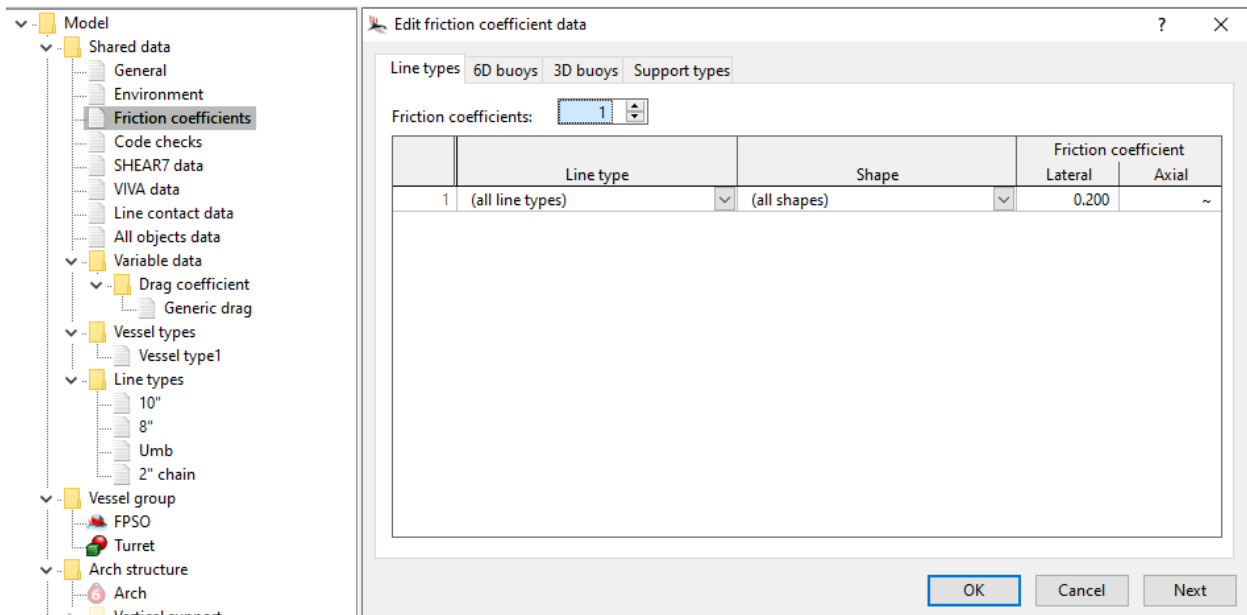
Wall up...   **Wall mid...**   **Wall dn...**

Friction has also been applied between the risers and these support shapes. Friction between lines and shapes is not set on the *line types* data form, but rather the *friction coefficients* data form; accessible from the *model browser*. On the *line types* page, a *lateral* coefficient of 0.2 has been applied between *all line types* and *all shapes*. The *axial* coefficient has been set to '~', which means the *lateral* friction coefficient will be used for all directions of motion.

Note that shape-to-line friction is only active in the dynamic simulation.

Generally, the purpose of the arch support shapes is to generate effective boundaries to the movement of the risers on the arch; meaning the shapes are not intended to accurately represent the true steel work of the structure itself. For example, the bottoms of the gutters might be made of pipework or steel plate which could be relatively thin components. Yet, the shapes here have significant thickness.

This approach is intentional because, if any parts of the actual structure have thin walls, it can be dangerous to model them like so in OrcaFlex. For example, if we used a very thin shape (say 2mm)

then a line node penetrating slightly more than 1mm into that shape would be pushed to the other side of the shape. This can lead to an unrealistic model response which, in turn, can lead to unrealistic results.

So, to stop this from happening, it is advisable to make the shape as thick as possible, without unexpectedly interfering with any other lines and / or buoys. This leads to a more robust contact model, particularly where loading may be applied to the system in a range of different directions.

Modelling the arch tethers and gravity base

The arch is restrained by a number of bridles and tethers; contained in the *Tethers and bridles* group.

The bridles are modelled using line objects, attached to the arch at their top connections (end A). The tethers (*ArchTie_N* and *ArchTie_S*) – also modelled using lines – are anchored to the seabed at their bottom connections (end B).

The tethers and bridles can be connected directly to each other. In this case the top end (end A) of each tether is *free* and the bottom ends of the two bridle lines are connected to it.

Some additional *drawing type* shapes have been used to represent the gravity base and the tether posts. Again, these are used for visualisation only.

The line type properties used to model the bridles and tethers represent 2" chain. Actual chain data should be used wherever possible but, if such information is unavailable, then generic chain properties can be produced using the OrcaFlex *line type wizard*. If you *reset* the simulation file (*F12*), then you can then see the wizard in action.

First, open the *2" chain* line type from the *model browser* and, on the *line types* data form, click the *wizard…* button on the top right-hand corner of the form. Following the instructions and stepping through the form will provide some visibility on how the *2" chain* properties were created for this model. Note that, unlike the risers, the *2" chain* line type has the *compression is limited* option ticked. That is because, unlike flexibles / umbilicals, chains are unable to support compression i.e. they simply go slack.

Further details about the *line type wizard* can be found on the <u>Modelling, data and results | Lines | Line type wizard</u> page of the OrcaFlex help.

Results

If your model is in the *reset* state, then reload the simulation file (*A02 Lazy S simple.sim*). Load the *A02 Lazy S results.wrk* workspace via the *workspace* dropdown menu at the top of the OrcaFlex window. This will automatically open four different time histories.

The top left-hand plot shows the surge (*X*) motion of the *Arch* 6D buoy. This has settled into a cyclic response; however, the *Arch* yaw motion (*rotation 3*) shows a more complex response with a double oscillation per wave cycle, instead of the single oscillation seen in the surge response.

This is not unusual because the arch is responding to transverse loads from the flexible upper and lower catenaries. These have different configurations in the water column so will have different load magnitudes and phases. This complex response is also evident in the *Arch* pitch response (*rotation 2*) (top right) and the *ArchTie_S* tether tension at its anchored end (bottom right).

Next, load the workspace *A02 Lazy S results#2.wrk*. Again, this can be done from the *workspace* menu, or you can simply drag-and-drop the workspace file from windows explorer into OrcaFlex.

In this workspace the *static state* clamp loads for each riser are presented in table format at the top of the screen. These are the 'pull-out' forces acting on each clamp constraint. We have used the *connection Lx-force* result to estimate this, which gives the connection force along the longitudinal axis of each riser.

Note, when showing *static state* results, you can add multiple results to the same table (as we have done here) by holding down the *Ctrl* key whilst pressing the *show* button on the *select results* form.

Also plotted is a *time history* of the *10" Riser S clamp* constraint *connection Lx-force*, which shows the maximum 'pull-out' force taking place in dynamics. This could be used as input to the clamp design.

Lazy S detailed

The more detailed version of the Lazy S system (*A02 Lazy S detailed.sim*) has the following additional complexity:

- Arch properties from a composite structure.
- Refined gutter wall modelling.

Discussion of the overall system model is largely covered by the *Lazy S simple* section earlier in this document. Only the changes to the system will be discussed here.

Open the *A02 Lazy S simple.sim* model in a separate instance of OrcaFlex on the same machine (the OrcaFlex licence will allow this). Having both open together will allow comparison of the two modelling methods. Make sure your model browser is in *view by groups* mode, as before.
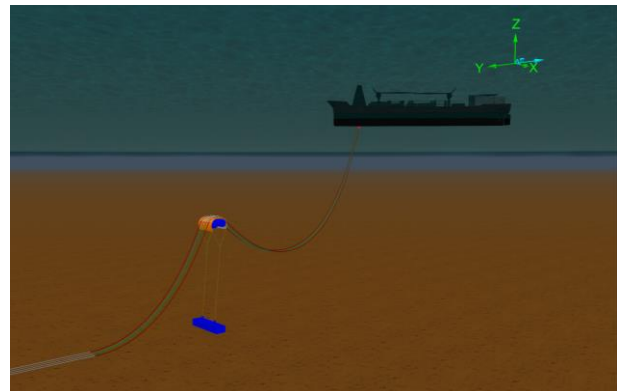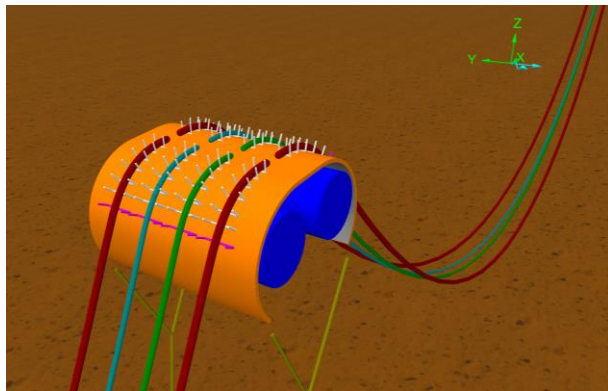
Building the model

From the first example model, we have seen how the physical and hydrodynamic properties of the arch can be represented by a single *lumped buoy*, with the overall properties calculated and specified by the user.

An alternative option is for the components of the arch to be represented by other objects connected to a common buoy. For this approach, the user specifies the properties for these individual components and OrcaFlex then calculates the overall properties of the entire structure. We refer to this modelling approach as structure discretisation, which is covered in further detail by the F01 Lowering hydrodynamics example.

This more detailed approach can be helpful for investigation of specific problems, for example having hydrodynamic loads calculated for each major object means that, if one end of the arch experiences very different flow from the other, then the resulting moments on the arch will be captured. However, adding components to the arch increases the size, complexity, and runtime of a model, so care should be taken not to add unnecessary detail.

Buoyancy tank modelling

When it comes to building a model of a midwater arch, in this way, it can often be convenient to model any buoyancy tanks using one-or-more line objects. For example, if you wanted to consider some cases with the tank flooded, this option saves having to replace the overall buoy properties for the empty condition with those for the flooded condition. All you need to do is change the line contents *density*, on the *contents* page of its data form.

In this example, only the two buoyancy tanks have been separated out. Each has been modelled as a series of single segment lines connected (at both ends) to the *Arch* 6D buoy. The lines are contained in the *Tanks* sub-group of the *Arch structure* group.

The advantage of using a series of single segment lines is that loads are calculated for each line and applied at its end connections, distributing the loads along the tank length. This will capture any resulting moments from change of flow across the length. With each line only having one segment, and with their ends connected to the 6D buoy in this way, they are unable to bend, twist and extend; meaning their stiffness properties are not important.

Note the line objects have been hidden from view and *drawing* type shapes have been used for visualisation in this example. Hiding the *Arch drawings* and *Gutters* groups, via the *model browser*, and showing the *Arch* 6D buoy and *Tanks* group will reveal the objects used to model the underlying physical properties of the midwater arch (pictured below). For more information on hiding and showing objects, see the *Lazy S simple* model discussion earlier in this document.
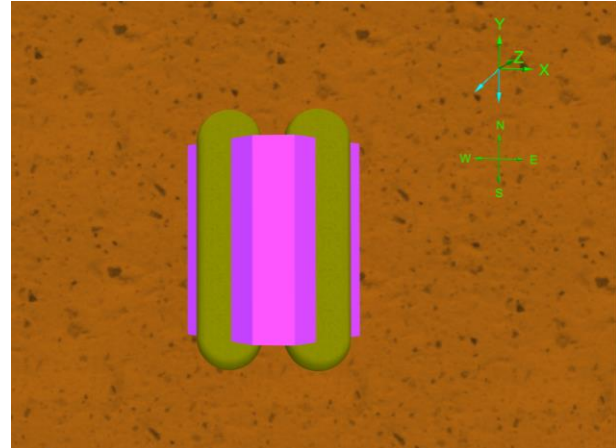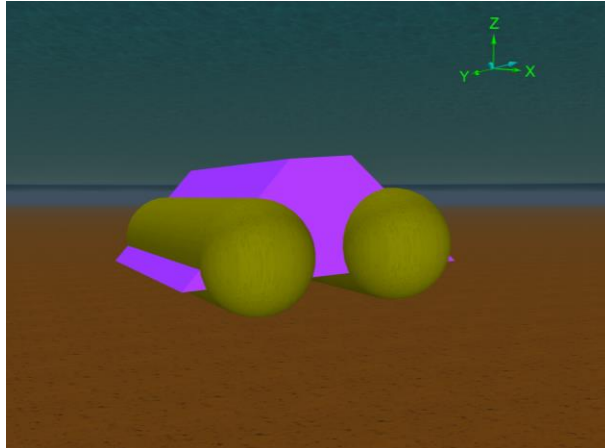
On the *line types* data form, you can see the properties of the tanks in *Buoyancy tank* type. It is important to be careful about setting the line *drag coefficients (Cd)* values for the line type, on the *drag & lift* page. Lines use the surface area in the axial drag formula, not the end (cross-sectional) area. The line axial Cd therefore needs to be calculated so it gives the correct resultant drag.

In this example, the two tanks are also side-by-side so one will shield the other when flow acts perpendicular to the longitudinal axes of the tanks. OrcaFlex does not automatically account for shielding, so both tanks will experience undisturbed drag. However, the downstream tank will see reduced flow in reality. The options for obtaining the correct horizontal normal drag are therefore:

- Divide the horizontal drag between the two tanks by halving the horizontal flow Cd, therefore applying non-isotropic normal Cd. Resulting pitch moments will not be exactly right, because both tanks see drag, but the differences will typically be small, and the arch is unlikely to be sensitive to them.

- Model wake effects for the lines (via the *fluid loads* page) so the downstream tank sees reduced flow. However, wake reduction will not include the effects of neighbouring structures such as supports & gutters. This approach also loads to more calculations so simulation run times will be slower.

The first of these options has been considered for this model.

In the vertical direction (line local x), both tanks see full flow, so the normal Cd is 0.7 (the full amount). In the horizontal direction (line local y), the normal Cd has been halved to 0.35 so each tank will produce half the drag.

Open the *Buoyancy tank E 1* line data form from the *model browser*. Note that torsion has been switched on – using the *include torsion* option at the top of the data form – and the ends connections are built in (stiffness of *infinity*). This is not necessary for a line with both ends connected to a rigid object, because there can be no rotation along its length. However, it does remove the warning message that OrcaFlex will produce when non-isotropic properties are applied, and torsion is switched off.

The remainder of the arch properties are then calculated and applied to the 6D buoy so the resultant physical and hydrodynamic properties match those used in the simple model.

Compare the data for the *Arch* 6D buoy between the two models. Taking away the tank contributions has changed the *mass*, *volume*, *mass moments of inertia*, *centre of mass* and the *drag areas*. In this simplistic example, the *mass* has reduced by approximately a factor of ten, therefore we have reduced the *mass moments of inertia* by the same factor.

Modelling the gutter walls

There are several options available for modelling the gutter walls. In this example, we show a method that uses the *supports* feature.

Make sure the *Gutters* sub-group (within the *Arch structure* group) is shown, via the *model browser*. The sub-group contains eight 6D buoys, with each buoy carrying a series of *U shaped* supports. Each support represents the struts of the gutter, and each *set* of supports has a *supported line* allocated to it, to indicate which riser is supported by which gutter.

Look at the *supports* and *support coordinates* pages of any of the *Arch supports*… 6D buoys. These pages are where the *supported lines* are specified, along with the *position* and *orientation* of each support. Click on the *Support types…* button at the bottom of the data form to open the *support types* form. You will see that seven different widths of support have been created here in order to model the radius of the arch gutter. Full details of the supports feature can be found in the Modelling, data and results | Supports | Modelling page of the OrcaFlex help.
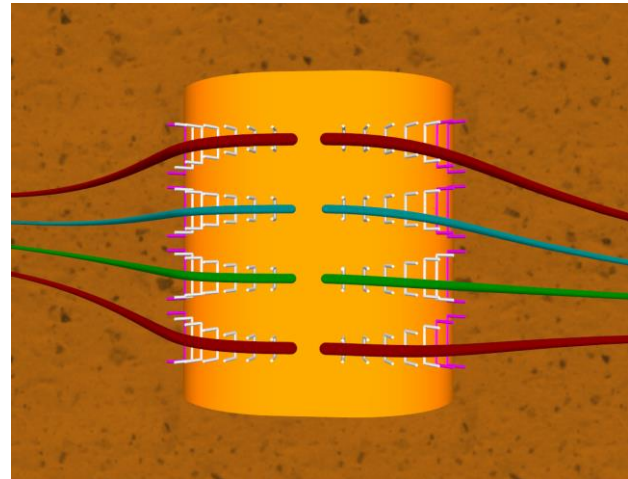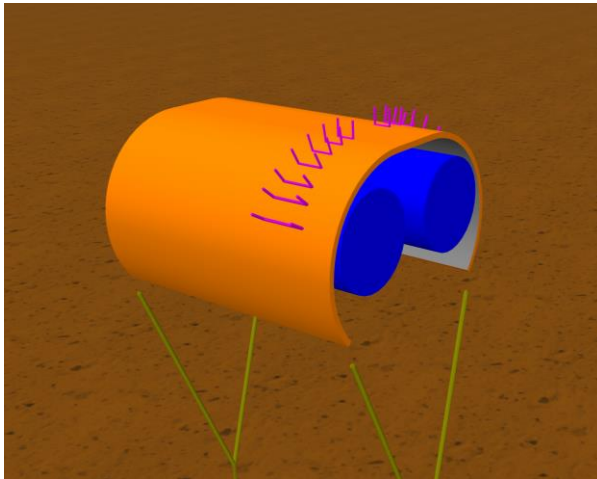
The *Arch support* buoys all have negligible properties in this example, so they have no effect on the arch response other than that generated by the contact of the riser and umbilicals. You could, if you wish, assign actual properties to these objects (i.e. give them the gutter's *mass*, *volume* etc.) and then remember to remove their contribution from the *Arch* 6D buoy properties.

This wall modelling method allows detailed control of the gutter wall and base shape, and greatly simplifies the static solution because OrcaFlex knows that each supported line must lie within the set of supports to which it is assigned. To demonstrate this, reset the model (press *F12*) and then re-run the static calculation by pressing *F9*. You will see the eight riser and umbilical lines solve very quickly into place within the gutter boundaries.

Available results for the supports include ...*clearance* and ...*lift out* results, as well as ...*reaction force* results for individual supports, or for entire sets of supports. Open the workspace file *A02 Lazy S supports.wrk* to see the support force and lift out results for the *lower 10" N riser*. The example E01 Simple rigid stinger, and its supporting description document, provide further details about the definition of support contact clearance and lift out results.

If you prefer to model the gutter walls as smooth surfaces, then you can do so using *elastic solid* type shapes (shown by the images below). In this instance each wall is constructed from two *curved plate* shapes and one rectangular *block*. Elastic solids are also required to form the curved arch surface i.e. to provide vertical support to the risers and umbilicals (see the *A02 Lazy S simple* model for details of how to do this).
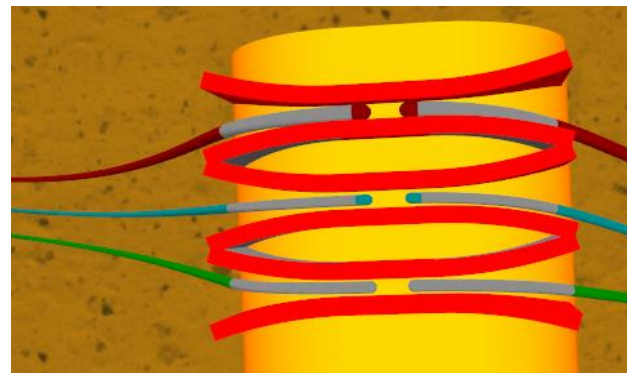
Results

Load the *A02 Lazy S results.wrk* workspace via the *workspace* menu. Make sure this workspace is also loaded for the *Lazy S simple* model so the results can be compared.

Some of the plots below have matching axes. To do this in OrcaFlex, double click on each graph to access and modify the plot ranges. From the *Axes* page, the ranges can be copied and pasted from one graph to another to get the same scale showing in different plots.

In the screen shots below, all *...simple* model plots are on the left and *...detailed* model plots are on the right.

The plots below show variation of tension at the bottom of a tether with time. The plot axes have the same scales. Tension variation is reduced in the more detailed model. The minimum tension is around 10te instead of 6te.
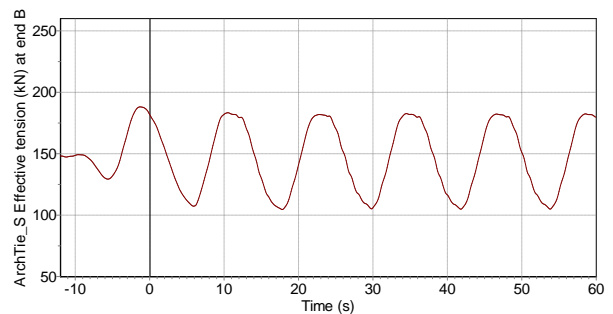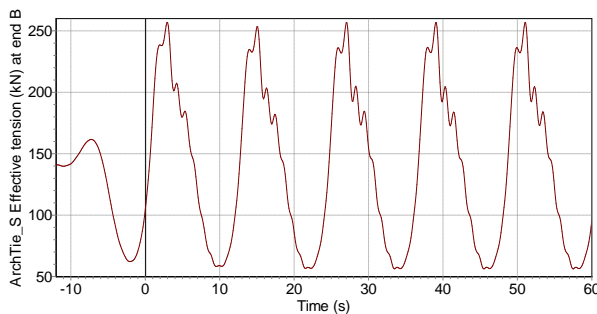
The time histories showing *Arch* yaw (*rotation 3*) have the same range but note that the *...simple* model has a negative range and the *...detailed* model a positive range. The *...simple* model has a yaw variation of between 179° and 181° (-181 and -179). The detailed model has a yaw variation between 177° and 183°. This is an increase from ±1° to ±3°. The increased yaw is due to the more accurate modelling of the moments on the arch from the riser contact with the gutter walls.
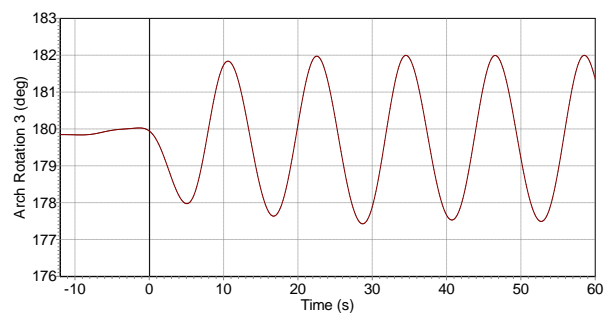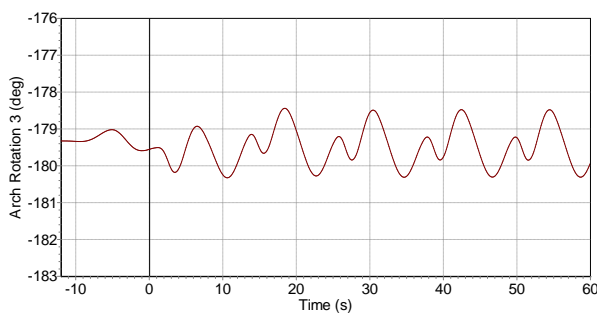
Arch systems have complex motions so a single change can vary more than one response. An example of this is shown by the plots below. The detailed model reduces the arch pitch variation (*rotation 2*) and shifts the mean arch surge (*X*).
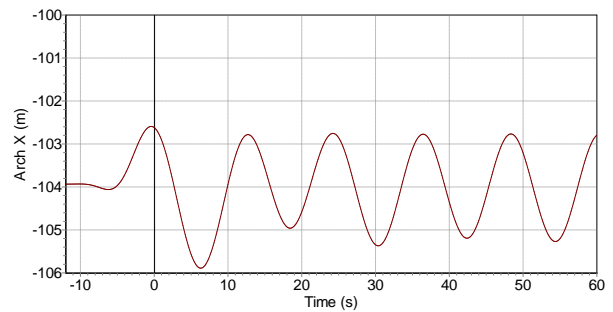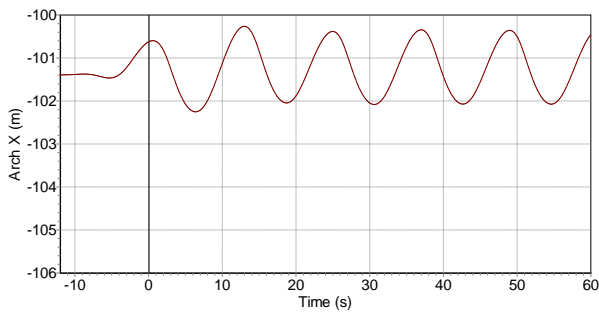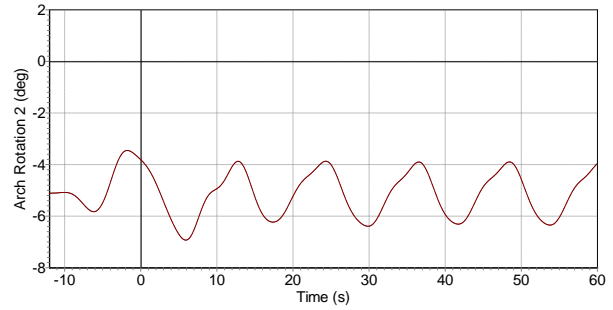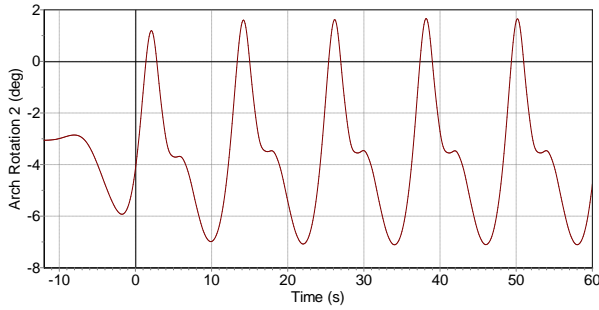
Steep S riser
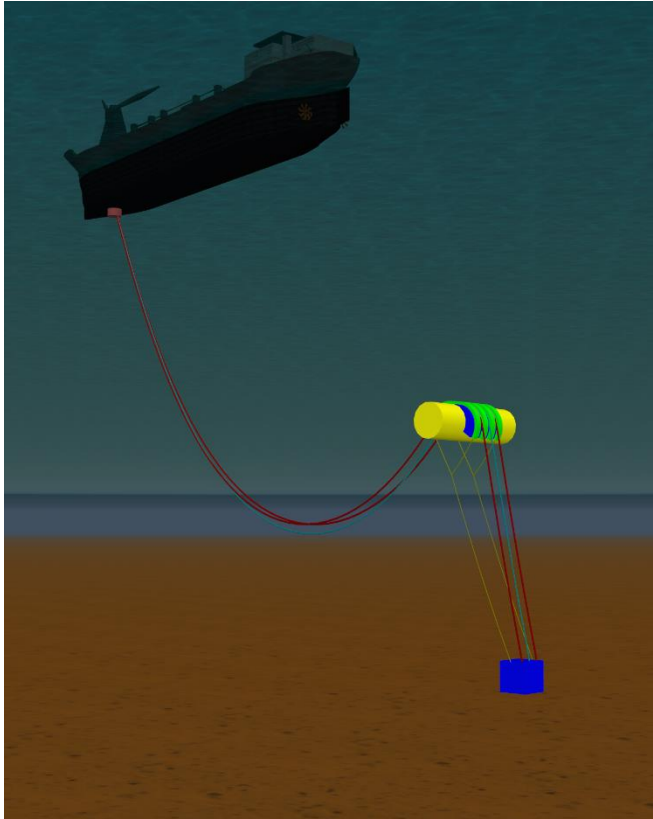
The Steep S system (*A02 Steep S.sim*) has a similar arrangement to that seen in the *Lazy S simple* example earlier.

However, instead of the lower catenary descending and lying on the seabed it terminates at a near vertical connection.

'Steep' means that the riser centreline is near vertical at the lowest end while 'S' describes the line shape as a result of the arch.

This example has two risers and an umbilical descending from an FPSO to the seabed. The central umbilical has a longer upper catenary to keep it out of the plane of the risers and reduce the risk of contact.

When you open the simulation file, you will see two views of the system and two plots. The upper is a wire frame plan view of the system. Note the North pointer in this case has been chosen to be in the riser plane and not aligned with a global axis. The lower view is a shaded graphics elevation of the system.

Building the model

Make sure your *model browser* is in *view by groups* mode, as before.

Various midwater arch modelling considerations have been discussed for the earlier lazy S systems, therefore only the seabed termination will be considered here. The lower catenary is vertical rather than horizontal when it reaches the arch gravity base. The *Gravity base* is represented by a *drawing type* shape, so it is used for visualisation only.

The *Risers and umbilicals* group contains all the lines representing the flexible products. End B of each lower catenary is anchored 5m above the seabed, on top of the arch gravity base. The connection stiffness is *infinity* so it is built in. Double click on any of the three '*lower*' lines in the *model browser* to see the settings.

The line end B *declination* in this example is 170º, indicating the line is heading down into an end connection that is 10º (180º–170º) from vertical. The *azimuth* is 10º because anchored connection orientations are relative to the global axis directions. The riser system is on a heading of 10º anti-clockwise from the global X direction.

For more information on line end connections, please refer to the A01 Catenary and wave systems example (*Catenary riser* example) and to the tutorial video Setting line end orientations, available on the Orcina website.

The lower catenary line lengths have to be optimised carefully. If the flexibles are too slack then they could enter compression as the arch moves. If too taut, then they could become the tethers for the arch system. Typically, the flexibles / umbilicals are more taut than those seen in the lazy systems.

A taut line will want to descend vertically during the static search. It will resist the shape restraint more than in the lazy system. This means more care is usually needed to achieve an acceptable static solution. It is easier for the static search to find a solution where the line is on the 'wrong' side of the shape ('wrong' being the side you don't want the line to end up on).

Temporary restraints – such as links or winches – can assist the convergence, as can using the *supports* method (described in the *Lazy S detailed* section earlier. Alternatively, a *spline* could be used to define an approximate shape for the line. This option can be selected from the *statics* table on a line's data form. When using this method, its settings are specified on the *spline starting shape* page of the same form.

The spline itself does not need to be very precise because the static analysis will eventually find the true equilibrium positions for the line nodes. The spline just needs to be 'good enough' to direct the line to the required side of the shape at the start of the static analysis.

Reset the model by selecting *reset* from the *calculation* menu, at the top of the OrcaFlex window, or using the keyboard shortcut *F12*. Switch the lower left-hand view to wire frame mode (click in the view window and press *Ctrl + G*). The splines will appear as grey lines, with the control points drawn as small, grey-coloured crosses. The control points can be dragged by holding down the left mouse button. The control points will move in the plane of the view and the spline will adjust accordingly. When configuring a spline, it is important to remember to check its shape in the plan view, as well as the elevation view.

If the lines leap around during the static convergence process, then the search may need a bit more damping. Open the *statics convergence* page of one of the line's *data forms*. The *min damping* has been increased from the default value (1.0) to 1.5 while *max damping* has been increased from 10 to 15. This will reduce overshoot but can make convergence slightly slower.

For more assistance with static convergence issues, please refer to the [Statics convergence guide](#) on the [Papers and technical notes](#) page of our website.

Results

If the model has been *reset*, then reload the simulation file. Two *instantaneous value* range graphs are presented to the right of the model views. The upper plot is for *10" upper East* riser and the lower is for *10" lower East* riser. They show the variation of *effective tension* along the length at any instant in time.

Run the replay (from the *replay* menu or using the keyboard shortcut *Ctrl + R*). The tension distributions will vary to match what is happening in the simulation replay. The tension varies steadily in this case.

These *instantaneous value* range graphs also help us to identify where a particular line response originated. They will also show axial waves in the line that might not be observed with other plots e.g. time histories.

Pliant S riser

A pliant S configuration (*A02 Pliant S.sim*) is a lazy S with the addition of a tether restraining the touchdown point. This example is the same as the *Steep S riser*, but the lower termination has been changed to the lazy style with restraining tethers.

When you open this model, you will see a shaded graphics view of the arch arrangement.

Building the model
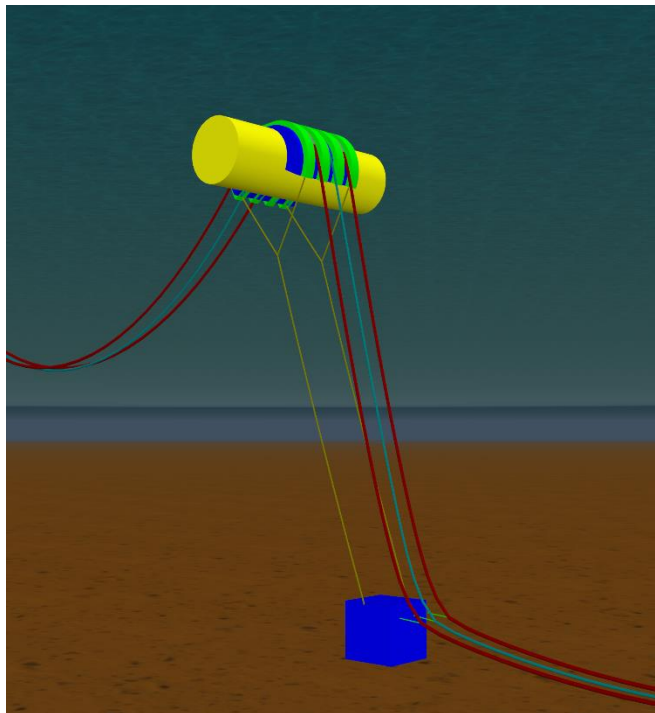
Make sure your model browser is in *view by groups* mode, as before.

The line is built in the same manner as for the previous *Lazy S* and *Steep S* models, therefore only the termination change will be discussed here.

The line is made pliant by the addition of a tether close to touchdown. In this example, the tether has been modelled as an OrcaFlex link attached to the riser. No clamp details have been included.

This is the simplest way of modelling the tether and is sufficient for most cases. For discussion of the alternatives, please refer to the example A01 Catenary and wave systems example (*Pliant wave*).

Again, the *spline* method has been used here to assist the statics convergence process.

Results

The default workspace shows two plots, along with the shaded graphics view. The top plot shows a range graph of *effective tension* for the umbilical lower catenary (*Umb lower*) for the *latest wave*. This shows the tension variation along the length of the line, through the final wave cycle. The step-change in tension takes place at an arc length of around 50m, which is where the tether is attached.

The lower plot is a time history of *Arch* motion in the global *Y* direction. This shows a settled, cyclic response.