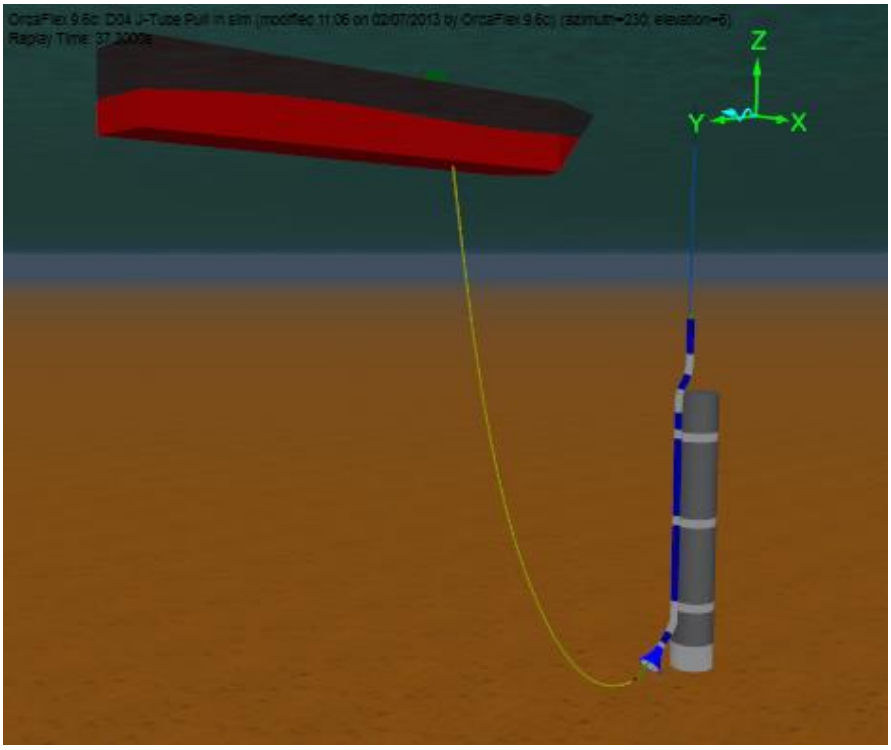
D04 J-tube pull-in

Introduction

This example shows how to model a typical J-tube pull-in operation. It includes pulling a line (a riser) out from a vessel over a chute whilst simultaneously pulling in the other end up a J-tube. It demonstrates the *line contact* model and introduces some tips for controlling the line position in the static solution. *Line feeding* is used at both ends of the riser to pay-out and haul-in the riser, and friction is included in the contact between the riser and the J-tube wall. Note that objects have been modelled 'not to scale', for clarity.

The areas of interest in this type of analysis would most likely be the seabed clearance, possible jamming of the pull-head in the J-tube and tension in the riser. This model also captures flexing of the J-tube in response to the pull-in loads and the forces placed on the clamps holding it to the main structure.

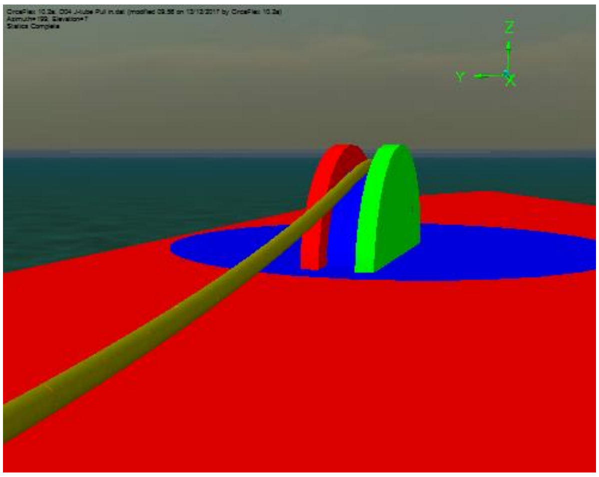
When you open the simulation file the default workspace is opened up which shows several views of the J-tube and vessel/chute arrangement. Replay the simulation and you will see the riser and pull head being pulled up into the J-tube. Notice that the riser and J-tube highlight white when contact occurs. The bottom right hand view shows the line being paid out over the chute. Turn on the node axes (Ctrl + Alt +Y) to see the nodes being created at the line end on the vessel deck, and disappearing as the nodes are being hauled in at the top of the J tube. Press Ctrl + Alt + Y again to turn the node axes off again.


OrcaFlex 9.6c: D04 J-Tube Pull in.sim (modified 11:06 on 02/07/2013 by OrcaFlex 9.6c) (azimuth=230; elevation=6)
Replay Time: 37.2000s

Building the model

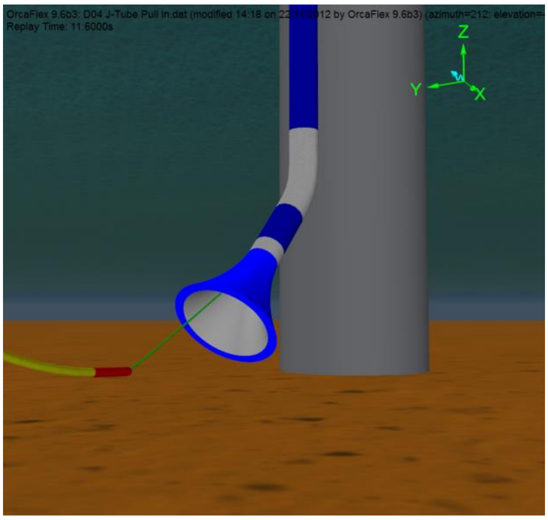
Ensure that *view by groups* is selected (right-mouse click in the model browser to find this option).

Select the workspace *D04 J-Tube Pull In vessel.wrk* from the menu bar. This shows two views of the vessel and the top end of the riser passing over the chute. An *elastic solid shape* is used to model the deck surface; this is named *Deck* and is currently hidden in the model. This elastic solid isn't actually necessary in this model, because the line isn't in contact with it, but if you wanted to model a longer length of line that was lying on the vessel deck, then this shape would provide the contact surface.


OrcaFlex 10.2a: D04 J-tube Pull in.dat (modified 09:56 on 13/12/2017 by OrcaFlex 10.2a)
Azimuth=199; Elevation=7
Statics Complete
Z
Y
X

The riser passes through a moonpool and will be shielded from the environmental loads in this region. This is modelled by adding a *trapped water* type shape to the vessel (named *Moonpool Shielding* in the model), which will modify the fluid motion within the shape's boundary to remove the environmental contribution.

The chute is modelled using *elastic solid shapes*, which form the contact surfaces for the riser to pass over. In this model we have also included some supports to help find the desired static solution. In the wire frame view, the supports are visible as four pink rollers, which are positioned so that they are inside the chute. The idea here is that the support's contact surface is below the chute's contact surface. When the static calculation starts, the supports do the initial work of picking up the riser and placing it on top of the supports. At this point the riser line's nodes are in contact with the elastic solid shapes modelling the chute surfaces, and therefore the node-to-shape contact takes over. So the supports are only used to get the riser on top of the chute initially, after that they are no longer used and the elastic solid models the contact surface. This is a good solution because the chute surface often really is a smooth continuous surface (just like the elastic solid shape), and you might want to include friction (which the shape to line contact model allows).



OrcaFlex 9.6b3: D04 J-Tube Pull In.dat (modified 14:18 on 22/11/2012 by OrcaFlex 9.6b3) (azimuth=212; elevation=
Replay Time: 11.6000s

The supports are part of the vessel object, and are defined on the *supports* page of the *Vessel1* data form. Four flat supports are used, and we have positioned them using a *simple geometry specification*, which allows us to easily position them around a radius.

The J-tube itself is modelled as a line with pre-bend. Open the *J-tube* line data form and look at the *pre-bend* page to see how this is done. Note that bending is given relative to the nodal x and y directions. Try turning nodal axes on when setting these so you can remind yourself if you rotate about x or y. Positive direction follows the right-hand rule. Note that the pre-bent shape can be previewed using the *view pre-bend shape…* button. The top end of the line is built-in (it has infinite connection stiffness) and therefore has its *end orientation* angles set appropriately (see *Modelling, data and results | Lines | Line data | End connections* of OrcaFlex help for more details).

End B of the J-tube is left 'Free' so that it is free to deflect in response to loads arising from the pull in operation.

The *line contact* model is used to specify that the riser line should pass through the bore of the J-tube. The J-tube is modelled as a line, so it is able to flex under loading as the riser is pulled through it. A second line contact relationship is used to model the clamps holding the J-tube to the structure.

Open the *line contact data form* from the *model browser*. On the *relationships* page there are two relationships defined, one between the *Riser* and *J-tube* lines and another between the *Tower* and *J-tube* lines. In the first relationship, the riser is the *penetrating line* and the J-tube is the *splined line*. The relationship is of type *inside*; this means that the J-tube is fitted with a smooth spline surface and the riser is fitted with penetrators that will contact with it. The penetrators are positioned at the nodes of the riser line.

Containment has been enabled; this means that the inner line (the riser) will be shielded from the environmental fluid forces by the outer line (the J-tube) wherever it is positioned inside it. The contents of the J-tube line are set to *free flooding* therefore the external pressure and buoyancy force acting on the inner line are as for a normal (non-contained) line.

The second relationship is of type *around*. This means that penetrators on the *Tower* line are placed around the *J tube* line at the positions and offset as defined on the *penetrator locations* page.

A friction coefficient has been assigned to apply axial friction between the riser and the J-tube.

Making the J-tube the splined line means that a spline tube is created which has the inner diameter of the line type as the bore diameter, and that has infinite wall thickness. This means that the static solution is found easily, as the section of the riser that is detected to be between the ends of the J-tube is pushed into the centre of the spline tube.

Select *D04 J-Tube pull in j-tube.wrk* from the menu bar. Reset the model (F12) and then run statics again (F9) and watch how the lines behave. Note that during *full statics* the lines do not interact with each other; the *line contact* model only comes into play during *whole system statics* and this can be seen clearly when the riser is suddenly pulled inside the J-tube.

An *elastic solid shape* has been positioned at the bottom end of the J-tube to model the bell mouth. Because the 'real' bell mouth will have a relatively thin wall thickness, a *drawing* type Shape has been used to show the bell mouth visually. A *drawing* type shape has no stiffness and therefore does not interact with other objects. A second bell mouth shape has been created with a much thicker wall to act as the structural boundary. This shape is an *elastic solid* type, therefore is has a stiffness and will interact with the line nodes that contact it. This item is hidden in the model (called *Bell mouth guide*). Making this boundary shape thicker than the physical structure removes the risk of line nodes that penetrate into the shape being pushed to the outside surface instead of the inside one. This happens if the nodes penetrate past the mid-point of the shape's thickness, which can happen if the shape is thin-walled; nodes are pushed to the nearest surface hence in this situation the outer surface becomes the nearer one.

Open the *Riser* line data form and look at the *feeding* page; both ends of the line have been given a non-zero payout rate. End A has a negative payout rate, which means that it will haul in line. Both ends have *apply ramp* selected, which means that the payout will be ramped up during the build-up stage (stage 0). This helps to avoid introducing discontinuities between statics and dynamics. An alternative would be to set up a variable payout rate, which can include a gradual transition between payout rates.

Note that the *initial arc length* of end B is 255m. The total riser line length is defined on the *structure* page as 297m (88m of wire, a 2m long pull head and 207m of riser). So by setting the initial arc length to 255m, there is 42m of line that is initially inactive, and that can be paid out during the simulation. See the help file *Modelling, Data and Results | Lines | Line Data | Feeding* for further details on these settings.

The riser line needs to find a solution in statics where it is on top of the vessel deck and over the chute, as well as being inside the J-tube. As already mentioned, we use some supports to lift the line up onto the vessel, however we can make the calculation easier still by using the step 1 statics method *spline*. This gives the line a better starting position for this application than the default *catenary* method. If you reset the model again (F12) you will see the spline visible as a white line.

OrcaFlex needs to know some information about the 'sense' of the riser line, because some of the calculations performed need to know which end of the line (end A or end B) is at the top and which is at the bottom. For example, if a line contacts the seabed, the touchdown results point is determined by starting at the top end and moving towards the bottom end until the first node in contact with the seabed is found. In this example, both ends of the line are lifted up, so there would be two touchdown points if the line contacted the seabed. To indicate that we want the touchdown point to be reported as the point closest to end B that touches the seabed, we need to set the *top end* data item on the line data form to *end B* (rather than the default setting of *end A*). Full details of the calculations that use this data can be found in the help file section *Modelling, data and results | Lines | Line data.*

The model also needs some damping applied during *whole system statics* to assist with finding the static solution. This is applied on the *general data form*, on the *statics* page.

Results

Re-open the simulation file and select *D04 J-tube pull in results.wrk*. A *range graph* for the tension in the *Riser* line is displayed (bottom right). Notice how range graph results are not available for the parts of the line that are not active in the model for the entire duration of the specified results period (in this case that's end A to arc length 29.5m and 253.5m through to end B).

The middle right hand graph shows the *time history* results for *effective tension* at end A – the tension results get a little noisier once the rigid pull head enters the J tube. In some cases, using the *around* line contact relationship instead of the *inside* relationship will give a smoother pull-in (and therefore a smoother tension result plot), due to the fact that with the *around* relationship you are pulling the smooth spline surface over the penetrator ring at the mouth of the J-tube, rather than the other way round. In this instance however, both relationships give similar results, so the *inside* option has been used here as it gives an easier static solution.

Also displayed are the Instantaneous *effective tension* result at end A. Run the replay and notice how this result changes with time i.e. it shows the instantaneous value.

Finally, there is a graph showing the force exerted on the clamps holding the J-tube to the structure (bottom left).