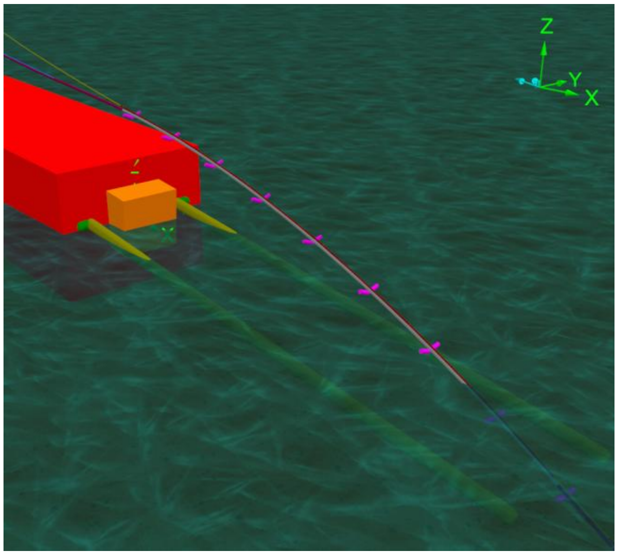
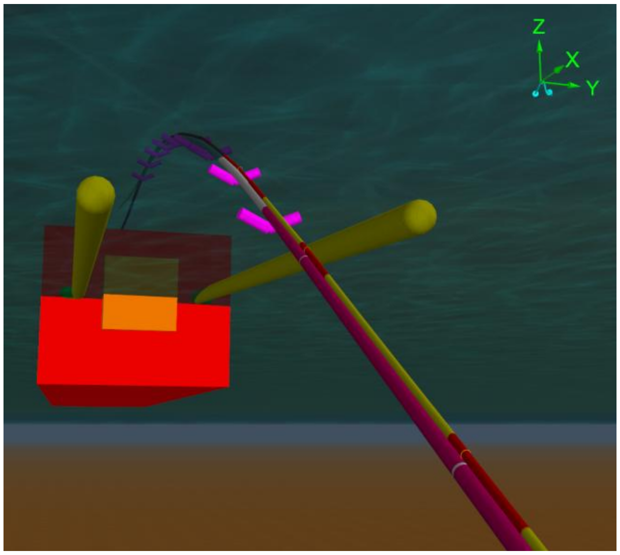
E02 Rigid hinged stinger

This model represents a rigid stinger hinged off the back of a lay vessel using the pipelay supports feature. In addition to the pipeline, another line is included to model a cable ‘piggybacked’ to the pipe.

As the interaction between the pipeline and the stinger can tolerate axial movement of the pipe, this model allows the pipe to move in and out of the two pipe tensioners. The model therefore represents the situation where welding operations have been suspended and the pipe is moving relative to the vessel to limit stresses. Modelling both tensioners is not generally necessary, but it has been done here to demonstrate how easily it can be done. A third tensioner is included on the piggyback line.

The pipe has also been assigned some code check data to allow a check against DNV OS F201 to be performed. This is discussed further in the results section of this document.





Building the model

The model consists of a lay vessel, the pipe being laid, a piggyback cable and a stinger. Open the workspace file *E02 Rigid hinged stinger chute exit.wrk* to see a view looking up at the point where the pipe exits the stinger.

The stinger is modelled using the supports feature, which is available on vessel and 6D buoy objects. In this model the supports are on a 6D buoy. The buoy is set up with negligible properties and has two sets of lines attached to it. The lines determine the stinger's hydrodynamic behaviour. A consequence of using a 6D buoy as a frame of reference is that the stinger is rigid and therefore this model provides no information on internal loads within the stinger. For a more detailed stinger arrangement, see the example E03 Detailed hinged stinger.

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 main groups called *LayBarge and Pipe* and *Stinger*.

The first group contains the lay vessel, the pipe being laid, and two constant tension winches used to tension the lay pipe. There is also the piggyback cable and a third constant tension winch to tension it.

The second group contains a 6D buoy (which holds the supports) with negligible properties, a constraint with one rotational degree of freedom (used to hinge the stinger to the vessel), and two further groups containing the sets of single segment lines representing the sides of the stinger. Each of these sides consists of thirteen such lines, each one 4m in length. The main reason for modelling these as individual lines, rather than individual segments in a single line, is that they can be independently ballasted more easily. This is achieved by simply changing the contents density on a line-by-line basis. An additional advantage is that, because the lines are connected to the 6D buoy at both ends and they have no intermediate nodes, they have no influence on the stability of the numerical integration. All line ends are connected to the buoy, so they all move as one unit.

Setting up groups of lines and buoys, for an arrangement like this, is made straightforward by a combination of the *all objects data* form and the duplication facilities available in the *model browser*. By placing a set of lines in a group, the user has the option of duplicating and moving groups using standard model browser facilities. Careful choice of object names at the start can make this process run much more smoothly.

For details on how to model the stinger supports see example E01 Simple rigid stinger.

The pipe is 'built in' at its connection to the vessel with an infinite *end connection stiffness* and a *declination* of 94° at end A. The *target segment length* is set to 2.5m over the upper section of the line, increasing to 10m in the suspended length. The seabed portion has 2.5m segments around the touchdown region, increasing to 20m towards the anchor (end B). Apart from the first section, the pipe is made of the line type named *Laypipe*, which models a steel pipeline with a 300mm outer diameter and 20mm wall thickness. The pipe properties have been derived using the *homogeneous pipe* line type *category*.

The first section of the pipe uses line type *Laypipe Slider* this is identical to the line type used for the rest of the pipe except that its *axial stiffness* is very low. This allows the upper part of the line to move in response to the tension applied by the two tensioner winches. The first winch (*Pipe Tensioner1*) is connected to the vessel at the same point as the pipe and to the pipe at an arclength of 20m (the end of section 1). The second tensioner (*Pipe Tensioner2*) is connected to the vessel at the same point and to the pipe at arclength 30m. Both tensioners apply a force of 240 kN. It is not strictly necessary to model two independent tensioners in this way, as one winch applying twice the force would produce very similar results. However, users often prefer their models to resemble the physical arrangement more closely, hence the approach taken here.

Note that the first section of line contains a single segment. This is necessary to ensure static convergence of the line, which for more segments would prove extremely difficult because of the low axial stiffness. Note also that the *homogeneous pipe* category does not allow the line type *axial stiffness* to be altered, so the *category* for *Laypipe Slider* is set to *general*.

The piggyback *Cable* is set up in the same way as the lay pipe, only this time using a single winch object to model the tensioner.

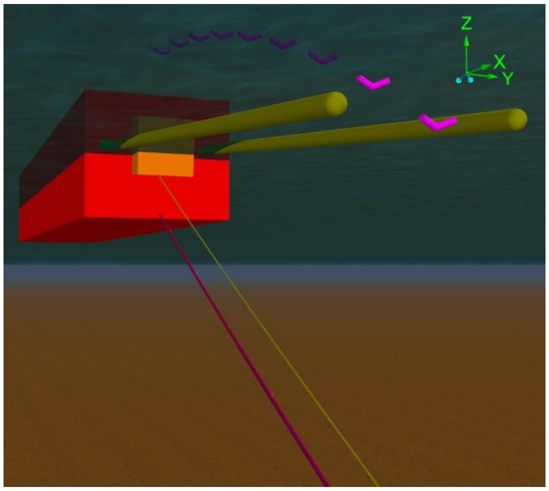
The vessel is the OrcaFlex default vessel. The pipeline and its supports are placed on the vessel's axis of symmetry but can easily be repositioned by using the move selected objects wizard available from the *model browser*.

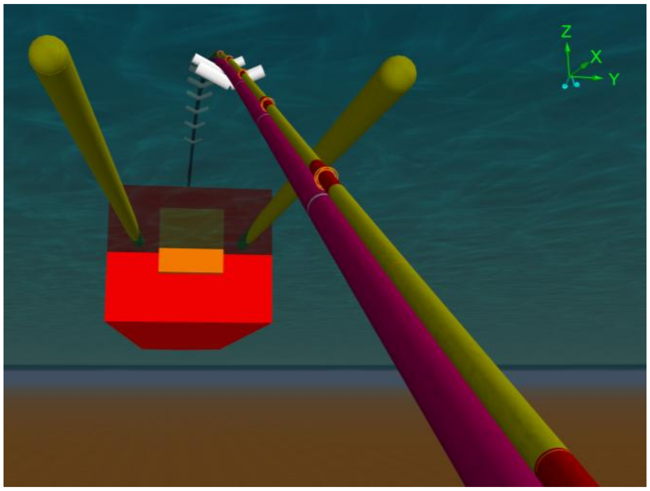
Initial setup

In general, a pipelay configuration will be set up to achieve a specific level of top tension. In this case, the tension is applied directly by the two tensioner winches each applying a tension of 240kN in still water to give a total tension of 480kN in the static calculation. In this model, we wish to model the situation where pipelay operations have been paused, and the tensioners are allowing the pipe to move relative to the vessel to minimise the dynamic loads.

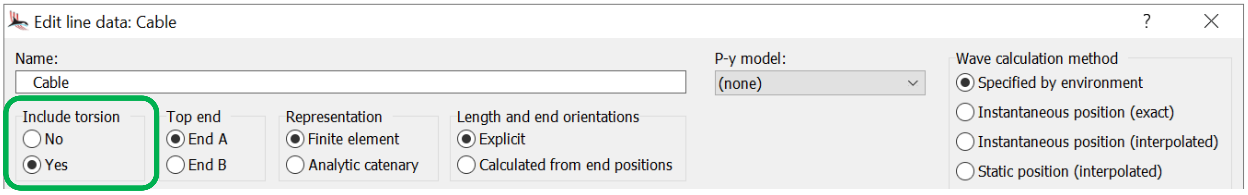
Line contact has been used here to piggyback the *Cable* onto the *Pipe*. Open the *line contact data* form from the *model browser*. There are two contact relationships, one of which defines an *around* relationship, with clamp positions that hold the cable to the pipe, and the other defines an *outside* relationship, which allows the outer surfaces of the two lines to contact each other. For more details on the line contact relationships, see the Modelling, data and results | Lines | Line contact | Data page of the OrcaFlex help.

The images below show the *Pipe* and *Cable* positions before and after running statics. Notice that the yellow cable has been pulled inside the cable clamps (assigned to the purple-coloured pipe). The clamps are represented by the orange rings. The red sections of the cable show where contact takes place between the cable and the clamps; this colour is set on the *drawing* page of the *Cable* data form.





It is important to ensure that torsion is enabled for both the *Pipe* and the *Cable*, due to the line contact relationship between the two (the cable has to stay in a fixed position offset from the pipe). So, the *include torsion* option has been set to *yes* for each line.



Edit line data: Cable

Name: Cable

P-y model: (none)

Include torsion
○ No
● Yes

Top end
● End A
○ End B

Representation
● Finite element
○ Analytic catenary

Length and end orientations
● Explicit
○ Calculated from end positions

Wave calculation method
● Specified by environment
○ Instantaneous position (exact)
○ Instantaneous position (interpolated)
○ Static position (interpolated)

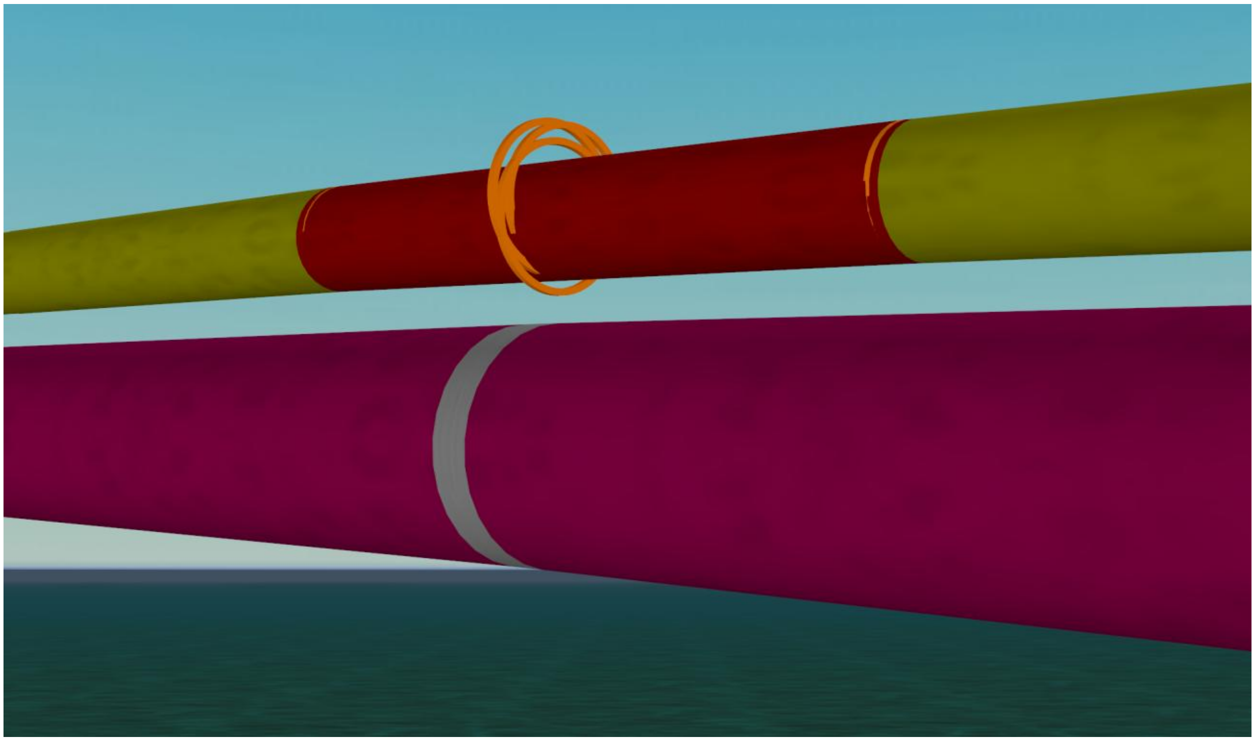
The static calculation in OrcaFlex is split into stages; the first stage is *line statics* where the equilibrium position of each line is found with the line ends fixed in position. This stage is further sub-divided into two steps, *step 1* and *step 2*, and this provides some control over the methods used in the line statics calculation. The final stage is *whole system statics* (*WSS*) in which all the objects are included, and the equilibrium position of the entire system is calculated. For further details about the static analysis process, please refer to the Theory | Static analysis page of the OrcaFlex help and to the Introduction to OrcaFlex training video, 7. Static analysis.

Line contact only becomes active during the WSS stage of the static calculation. In other words, the positions that the lines are in at the end of the *line statics* stage do not account for any contact with other lines. In this model, if the static convergence methods were left as the default settings, then the two lines would end *line statics* in roughly the same position i.e. lying on the supports, overlapping each other. When the *WSS* stage starts, the contact between the two lines is detected, and a large force appears that is trying to push the lines apart. This sudden application of a large contact force can lead to convergence difficulties, which can cause the static calculation to fail.

To avoid this, we have made sure that the lines are *not* overlapping at the end of the *line statics* stage. This has been achieved by setting the *step 1* statics method of the cable to *spline* and the *step 2* method to *None.* This means that the *Cable* is put into the shape set out by the spline profile, visible in the model as a white line when the model is in the *reset state*. The line is left there until *WSS* starts. If *step 2* were set to *full statics*, then the cable would find its equilibrium position, which would again be lying on the supports, therefore overlapping the pipeline. By removing this step, we ensure that the *Cable* starts *WSS* in a better position than it would have done otherwise.

In this model the clamps have a size-for-size fit with the cable that they are holding. When line contact penetrators have the same, or very similar, diameter to the line that they are contacting, it is necessary to use *penetrator discretisation*. That's because, in these situations, the true contact force occurs at all points round the circumference of the penetrator, simultaneously, whereas the OrcaFlex penetrator can only represent a single contact point. This means that the contact is ill-defined by a single penetrator, which results in a singularity in the force model. In turn, this can lead to noisy or unstable simulations.

By splitting the original penetrator into multiple smaller penetrators, placed around the original contact surface, the contact can be defined more robustly. Here we have applied a *penetrator discretisation count* of 3 on the *line contact* data form. For further details on this, with some sketches to help visualise the effect of penetrator scaling, see the Modelling, data and results | Lines | Line contact | Modelling page of the OrcaFlex help.



Reset the model (*F12*) and run statics (*F9*) to see the line contact model in action.

The environmental conditions are set as follows:

- 1.0 m/s slab current, 20° off the stern of the vessel.
- JONSWAP waves, Hs 1m, Tz 8s, 45° off the stern of the vessel.

The simulation has been set to run for 100s – post build-up – without searching for a particularly severe design wave.

Results

Re-load the simulation file to load the default workspace, so results summaries are automatically generated.

The range graph of *max von Mises stress* for the whole simulation shows the variation over the entire length of the line. Dynamic variations are small, and the highest stresses occur in the stinger region, as expected. The peak stress of around 290,000 kPa (290 MPa) would be acceptable for good quality steel.

The time history of *tension* for the *Pipe Tensioner1* winch shows the tension is held constant at 240kN. The corresponding plot for *length* (top left) shows that there is a small amount of pay out and haul in, but that the length remains close to 19 m throughout the simulation. Since this tensioner acts over the first 20 m of the pipe, this shows that the initial set-up of the line is acceptable and that the sliding section of line is not being unreasonably stretched or squashed. Generally, it is inadvisable to let the winch length get too small, or instability can occur.

Loading the workspace file *E02 Rigid hinged stinger results.wrk* opens a time history of in-plane rotation for the stinger buoy, indicating how it responds to the applied waves. It also opens time histories for *reaction force* and *contact clearance* of the pipe against the last support on the stinger (*support 9*). Note that the contact clearance result is a negative value – this indicates that the pipe is always in contact with the support. *Reaction force* and *contact clearance* results can be viewed for any of the supports. For a definition of support contact clearance, and the other available support results, see the example E01 Simple rigid stinger and its supporting document.

The pipe in this model has been given some code check data to enable a DNV OS F201 code check result to be reported. OrcaFlex has a number of built in code checks; open the *code checks* data form from the *model browser* to see what's included. In this case we've assigned data to the *DNV OS F201* page of this form. Note that most of this data is specific to the line type, hence you see a list of the available line types in the model, with data assigned to the *Laypipe* and *Laypipe Slider* line types only. All the data items refer to parameters specified in the code. Select one of the data items on this form and then press F1 to open the relevant page of the OrcaFlex help (Modelling, data and results | Lines | Code checks | DNV OS F201). This page lists which table or section in the code each of the data items correspond to.

To facilitate the requirement in the DNV codes to separate the functional load from the environmental load, the *DNV functional load* page of the *code checks* data form allows you to choose either *current model* or *simulation file* to specify a functional load case.

The functional load case should only include contributions from functional loads, i.e. loads arising from the physical existence of the system, so it should not include any form of environmental loading, such as drag from wind and current, or a static vessel initial position representing a mean offset due to wave, wind and current.

If you choose *current model* as the functional load case, then you should ensure that the static state of the current model does not include any of the above-mentioned environmental loading. In this model this has been done by excluding the current from the static analysis, by ticking the *ramped* box on the *current* page of the *environment* data form.

If you choose *simulation file*, then a separate model file can be used to identify the functional load, but this model must be consistent with the currently active model. Any line, for which a DNV-OS-F101 or DNV-OS-F201 code check result will be requested, must exist, with the same name, in both the currently active model and the functional load case, and have the same segmentation in both models.

Open the workspace *E02 Rigid hinged stinger code check.wrk* to view the *WSD* (working stress design) and *LRFD* (load and resistance factored design) code check results for this simulation.

OrcaFlex simulation files can sometimes become very large due to the amount of results data that they contain. One option available to help reduce the file size is to increase the *sample interval* (i.e. the frequency with which the results are recorded). This setting is available on the *dynamics* page of the *general* data form. In this case, we have increased the *target sample interval* to 0.5s, which is sufficient for the rate of change of loading in this model because there are no high frequency results to log.

Further details about results logging can be found on the Modelling, data and results | General data | Logging page of the OrcaFlex help.