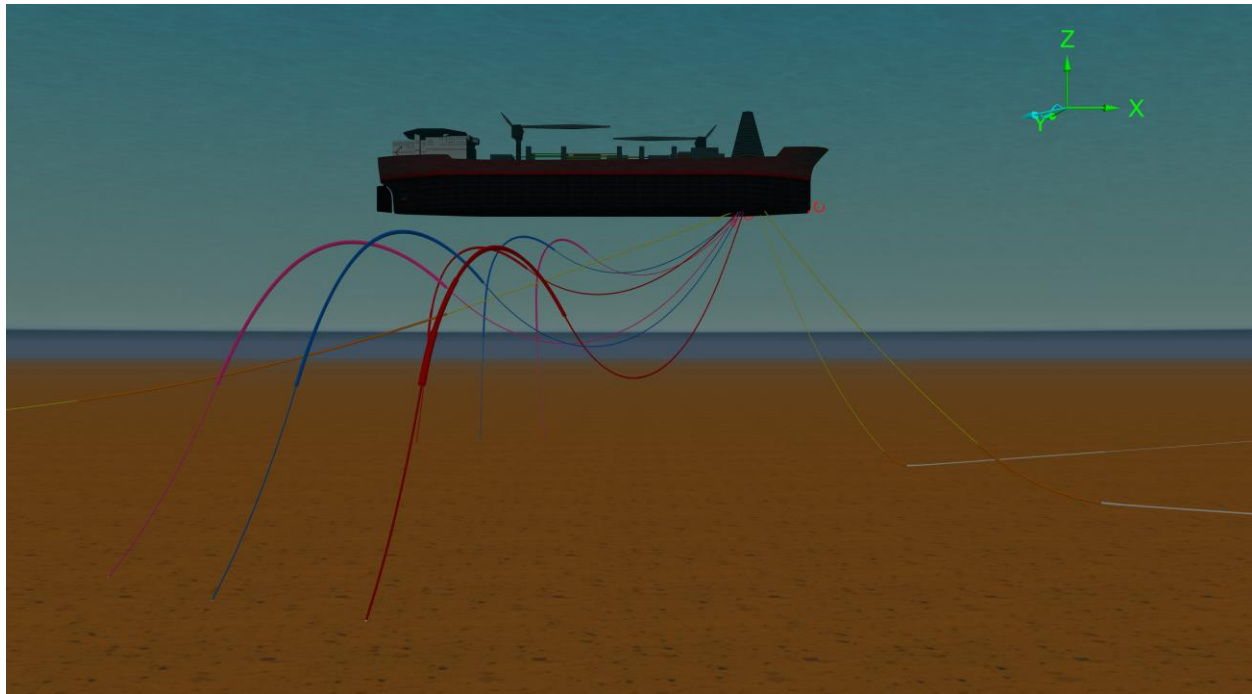# C03 Turret moored FPSO

In this example, an FPSO weathervanes about the turret in order to adjust its heading to the incident weather. The subsea system includes three moorings along with a number of risers and umbilicals. Friction between the vessel and turret is modelled.

Building the model

Turret moored FPSO systems have the moorings, risers and umbilicals connected to the turret. The turret is free to rotate relative to the FPSO, and so this leaves the FPSO free to weathervane so that it turns to face the weather, without forcing the moorings or risers to rotate with it.

In OrcaFlex, the connection between the turret and the FPSO is modelled using a *constraint* object. Constraints give you the ability to enable or disable individual degrees of freedom, therefore in this situation we require a constraint with a single rotational degree of freedom (DOF).

First, open the model, run the replay (*Ctrl+R*) and make sure that the *view axes* and *environment axes* are ticked on the *View | Axes and labels* menu item in OrcaFlex. During the simulation, you will see that the vessel gradually rotates around until it is approximately head-on to the weather, while the turret orientation (and therefore the moorings connected to it) remain largely unchanged. Note that the weather direction is shown by the blue-coloured arrows in the top right-hand corner of the model view.

The two left-hand results graphs show this. Here, we plot the *dynamic Rz* results for both the vessel and the turret. Generally, 'dynamic' position and rotation results report the change in position or rotation from the static state. The results for the vessel show that it rotated by almost 100° initially, before returning to oscillate slowly at around 40° from its initial orientation. This matches the

weather direction. Note that the simulation would need to be run for a much longer duration for the vessel to settle to a steady orientation.

In the model browser, double click on the object called *Turret rotation* to open its data form; this is the constraint object that allows the turret to rotate. On the *degrees of freedom* page, notice that the *Rz* box is the only DOF enabled (ticked). The *initial value* indicates the initial position or rotation of the constraint, relative to its connection point. To see the effect this has, close the data for, reset the model (*F12*) and switch to the wire frame view (*Ctrl+G*). If you now change the *initial value* to something other than zero, you will see the turret, and the ends of the lines connected to it, rotate about the connection point.
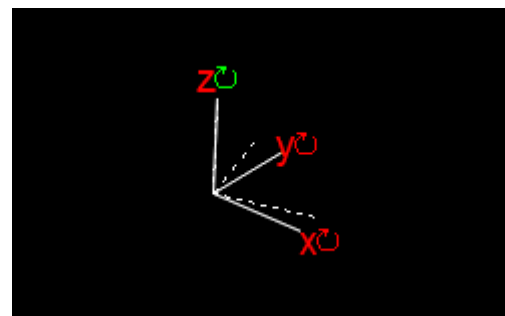
Apply an *initial value* of 30° and then run statics (*F9*). Note that although the turret starts the calculation in the 30° rotation, it quickly solves back to its expected equilibrium orientation. The constraint itself currently offers no resistance to rotation, so the connection loads from the moorings and risers are acting to rotate the turret back to this orientation.

Any translation or rotation allowed by a constraint can have a stiffness and a damping term associated with it. If you wish to model physical stops or limits on the rotation, then you can do so by applying a non-linear rotational stiffness (note that both the stiffness and damping terms can be either linear or non-linear).

In this model, a non-linear stiffness profile has already been set up. To see the effect of this, reset the model again (*F12*) and on the *stiffness & damping* page of the *Turret Rotation* object select the variable data *Rotation Stiffness* via the dropdown arrow on the *rotational stiffness* box. To see what this variable data represents, click the dropdown arrow again and select *<variable data…>.* This will show you a table of *angular displacement* vs *moment*. The data here imposes a high moment if a rotation of +/-30° is reached; therefore it represents hard physical stops at the rotation limits. Click the *profile* button to see this relationship graphically.

Next, close the data forms by clicking the *OK* button and re-run the simulation (press *F10*). The three results graphs will re-draw as the simulation runs. The results plotted in the top right graph are particularly useful here – this is the *Rz* result for the constraint.

Constraints have two axes, the *in-frame* and the *out-frame*. These are visible in the model if you switch to wireframe view (*Ctrl+G*) and zoom in on the turret. The *in-frame* (solid white lines) shows the initial position and orientation of the constraint, while the *out-frame* (dashed white lines) shows the position and orientation of the constraint at any point in time. So, the *Rz* result reports the amount that the constraint has rotated about its z-axis, relative to its initial position. Notice that the turret is now prevented from rotating beyond 30°, which is governed by the non-linear rotational stiffness for the constraint.
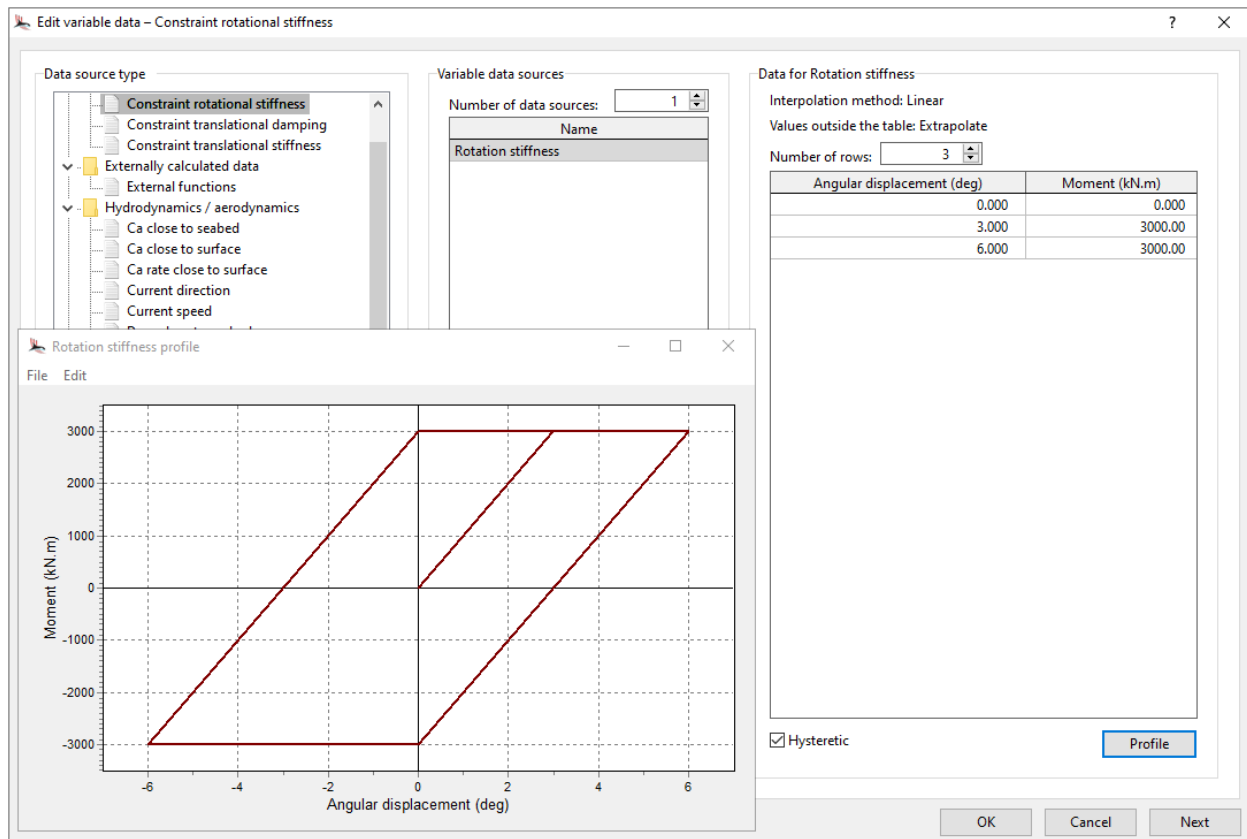
Note that the vessel doesn't start dynamics head to weather. That is because we have deliberately excluded the vessel from the static calculation. This means we are forcing the vessel to start dynamics from its initial position (a heading of 0°), and hence you see it gradually rotate to face the weather over time.

If you include the vessel in the statics calculation, then the loads acting on it will be included in the calculation, and as a result the static solution will be with the vessel already rotated head to weather. To see this, reset the model (*F12*), open the *Vessel* data form and on the *calculation* page, select *6 DOF* as the *included in static analysis* choice. Then run statics (*F9*) to see the vessel solve to the head to weather position.

In this example, the turret buoy does not have its hydrodynamic coefficients specified, because it is constrained to move with the vessel. The constraint holding the turret will generate a connection load on the vessel that may be of interest; but this is largely due to the loads on the buoy from the moorings and attached risers.

To model friction effects between the turret and the vessel, an *elastic solid* type shape, called *Buoy hull*, is connected to the turret buoy along with four 'friction lines' (connected to the vessel). These four lines penetrate into the shape, and a friction coefficient has been assigned to the interaction. Friction coefficients between lines and elastic solids are defined on the *friction coefficients* data form, accessible from the *model browser*. Friction effects applied in this way (line to elastic solid) are not active during statics, but will be applied throughout dynamics.

As an alternative friction model, it is also possible to apply a non-linear *hysteretic* rotational stiffness to the constraint i.e. where the friction means that the turret and vessel rotate together ('stick') up until the point where the mooring restoring moment is greater than the friction moment, at which point the vessel starts to rotate ('slip') relative to the turret. Although this is not included in this model, this type of stiffness profile would look something like that shown by the figure below.

Results

Re-open the original simulation file and open the workspace file *C03 Turret moored FPSO results.wrk*.

Here we show some time histories of the local z (*Lz*) component of the *connection force* on the constraint and the *effective tension* at the top end of one of the moorings, both for the last 100 seconds of the simulation. The mooring tension is also shown for the entire simulation, where you can clearly see that it is highest before the vessel has turned into the weather, and then reduces once the loads on the vessel have stabilised.