

C03 Turret moored FPSO



In this example, an FPSO weathervanes about the turret in order to turn head to weather. The moorings and several risers and umbilicals are included, and 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 rotation degree of freedom.

First, open the model and run the replay. You will see that during the course of the simulation the vessel gradually rotates around until it is approximately head to weather, while the turret orientation remains largely unchanged. The two results graphs shown at the left hand side of the screen show this. Here, we plot the *dynamic Rz* results for both the vessel and the turret. 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 80° initially, before returning to oscillate slowly at about ~40° from its initial orientation, which matches the weather direction. Note that the

simulation would have to run for a much longer period of time for the vessel to settle into a steady orientation.

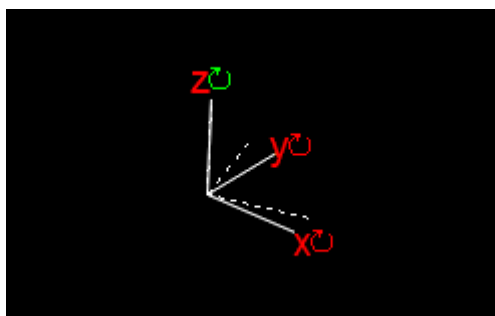
In the model browser, double click on the object called *Turret rotation* to open its data form; this is the constraint object that allows the turret to rotate. On the *degrees of freedom* page, notice that the *Rz* box is the only DOF enabled (ticked). The *initial value* indicates the initial position or rotation of the constraint, relative to its connection point. To see the effect of this, close the data form and reset the model (*F12*) and switch to wire frame view (*Ctrl + G*). As we are looking at the system from above, if you now change the *initial value* to something other than zero, you will see the turret, and the ends of the lines that are 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 the equilibrium position. The constraint itself currently offers no resistance to rotation, so the connection loads from the moorings and risers rotate the turret to its equilibrium position.

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 and 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.

Then 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* (dashed white lines) shows the initial position and orientation of the constraint, while the *out-frame* (solid white lines) shows the position and orientation of the constraint at any point in time. So this particular result reports the amount that the constraint has rotated about its z-axis, relative to its initial position. It's clear to see the effect of the non-linear rotational stiffness, which locks the turret at a rotation of 30°.

Note that the vessel doesn't start dynamics head to weather. This is because we have deliberately excluded the vessel from the static calculation. So we are forcing it 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.

To model a friction effect between the turret and the vessel, a shape called *buoy hull* was attached to the turret buoy, and four friction lines attached 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, which is accessible from the model browser.

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.

Results

Some further results can be displayed by opening the workspace file *C03 Turret moored FPSO results.wrk*. (Note first re-open the sim file if the model is currently in the *reset* or *statics complete* state).

Here we show 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.