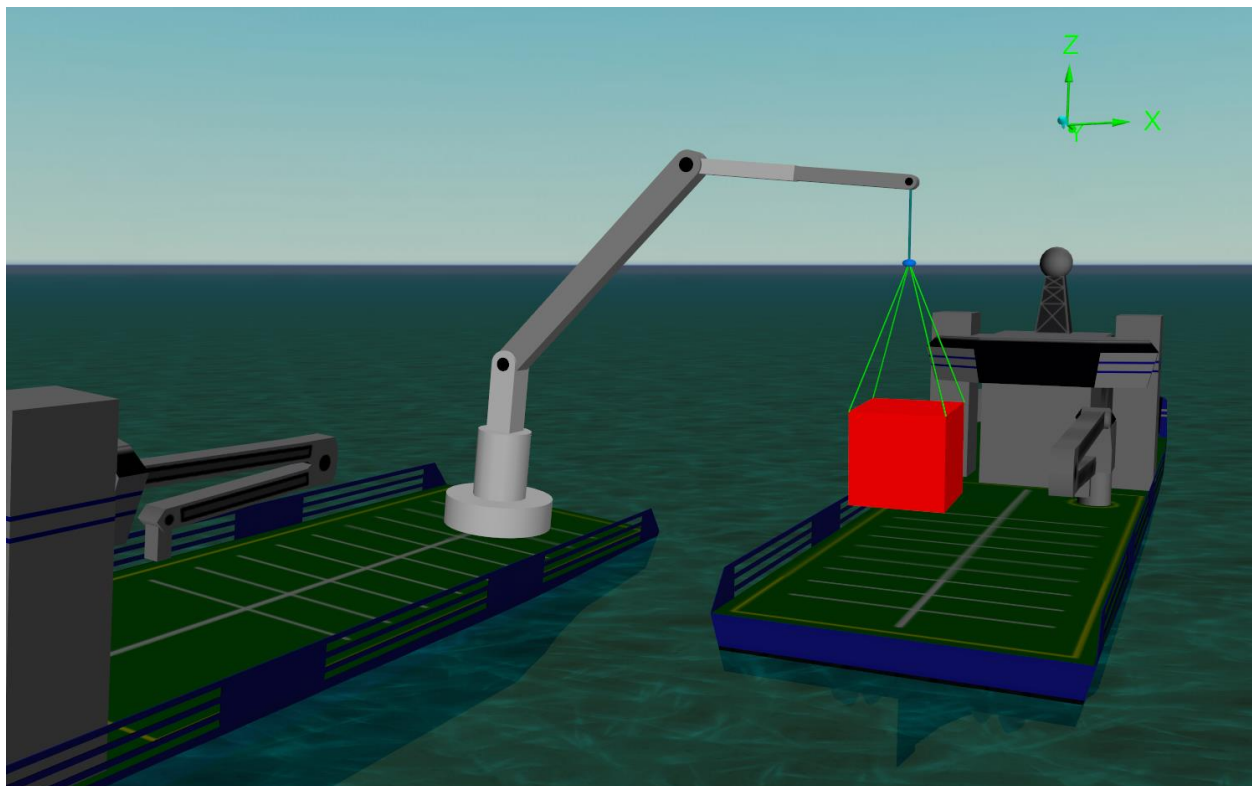


## F04 Payload transfer

This example shows a large payload being transferred from the deck of one vessel to the deck of another. Constraint objects are used to model the movements of the crane. The response of both vessels is fully calculated and therefore captures the effect of the shifting payload.

The crane hoists the payload off the deck of the offloading vessel. The crane then slews the payload across to the support vessel. At the same time it luffs the arm downwards. The arm then extends to move the payload further over the support vessel's deck and finally the payload is lowered to rest it on the deck.

Note that we have deliberately sized the payload so that the roll of the offloading vessel is exaggerated, to make it easy to see the changing response.



## Building the model

The model has two vessels: *Offloading Vessel* and *Support Vessel*. The payload is transferred between the two via a crane. The decks are made up of rectangular elastic solid shapes attached to the vessels.

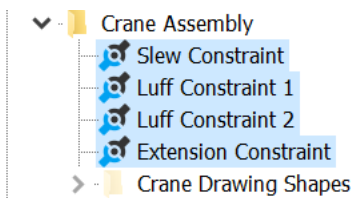
From the *model browser*, open the data form for one of the vessels. Selecting the *calculation* page shows that both vessels use calculated motion however, in this instance, we have chosen not to include the mooring or dynamic positioning systems. This means there are reduced restraints on the vessel, and so we have decided not to include the vessels in the static calculation.

Included in the dynamic calculations are 1<sup>st</sup> order wave loads and added mass and damping loads; ticked as *included effects* on the vessel's data form. The vessels will therefore respond not only to the waves applied, but to the shifting weight of the payload. Current, wind and 2<sup>nd</sup> order wave load calculations have been switched off. Because the vessels are subject to only wave frequency excitation, the corresponding option has been selected for the *primary motion is treated as* data item.

Further guidance on these settings can be found on the [Modelling, data and results | Vessels | Vessel modelling overview](#) page of the OrcaFlex help.

### Crane motion

The crane motion is modelled using four constraints. The crane also uses a number of *drawing* type shapes to give it a realistic visual representation. Make sure the *model browser* is in *view by groups* mode (right mouse click in the browser and make sure this option is ticked), then hide the sub-group named *Crane Drawing Shapes* (*Ctrl + H* on the group). Next, show the four constraints currently hidden from view (multi-select and *Ctrl + H*):



Watching the simulation replay (*Ctrl + R*) shows the axes system of each constraint moving through time. *Imposed motion* type constraints are used here, with the motion specified as a time history, the data for which can be in either an *internal* or *external* source.

Open the data form for the *Slew Constraint* and look at the *time history source* page. In this case we have used an *internal* source, and therefore the data is listed on the *time history data* page. Note that the slew constraint is only controlling the rotation of the crane about the crane's vertical axis, therefore only the *time* and *Rz* columns have non-zero data.

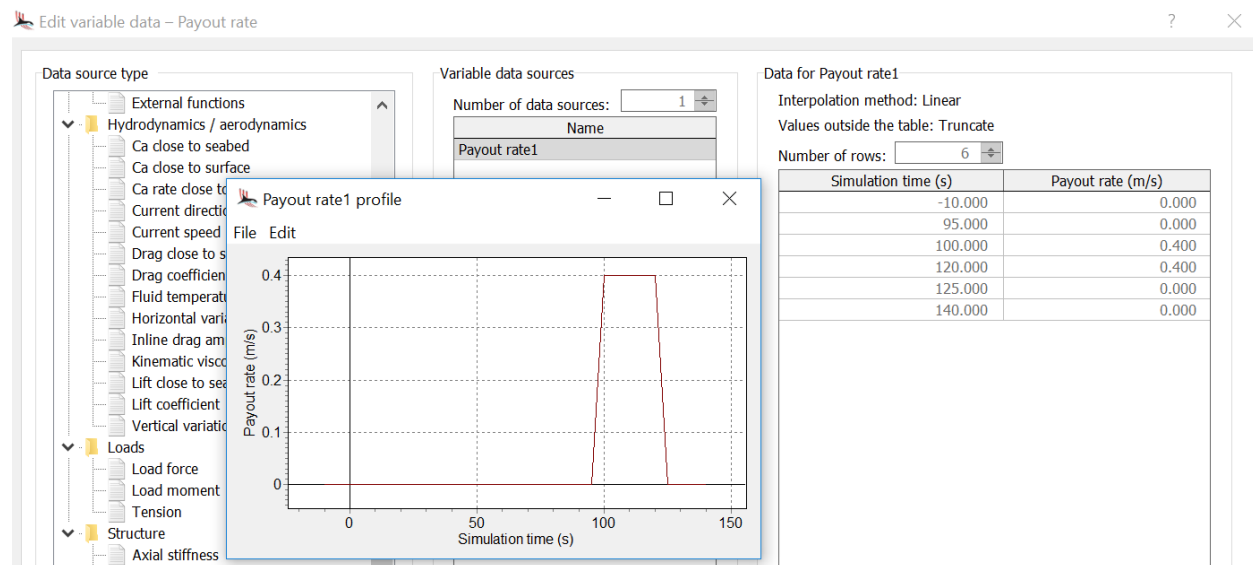
The four constraints are connected in series, so as the slew constraint rotates, both luff constraints and the extension constraint rotate with it. Similarly, as the first luff constraint rotates, the 2<sup>nd</sup> luff and the extension constraints will rotate with it, and so on. As the vessel moves, the constraints move with it, so using a constraint to model each individual 'joint' on the crane makes it easy to set the crane motions.

## Crane wire modelling

The crane wire is modelled as a line with steel wire rope properties. *Line feeding* is used to pay the line out, so that the payload is lowered on to the support vessel's deck. A variable *payout rate* is used so that (i) the payout can be started and stopped at appropriate times in the simulation, and (ii) the payout can be ramped up / down gradually, rather than applying an instantaneous change in payout rate (this would introduce shock loading to the system).

The line payout is set up on the *feeding* page of the *Crane Wire* line data form. End A is connected to the crane tip, so we want to apply the feeding at that end. The *initial arc length* at end A is set to 20m, the line's total length (on the *structure* page) is set to 30m. This means that, at the start of the simulation, there is only 10m (i.e. 30–20) of line present in the model, leaving 20m hidden away ready to be paid out during the simulation.

The variable payout rate, shown by the image below, accelerates over a period of 5 seconds, then maintains a constant payout of 0.4m/s for 20 seconds, before decelerating over a period of 5 seconds. This results in a total payout of 10m, sufficient in this case to land the payload on the support vessel's deck.



## Payload modelling

The payload itself is modelled using a simple 6D *lumped buoy*. It is attached to the crane via a bridle arrangement of four links (representing slings) and a 3D buoy (for the master link). Note that the axial *stiffness* of each sling has been given an unrealistically low value. This is intentional because we are moving the crane / payload around very quickly in this example (to save on simulation time and therefore file size). A stiff bridle will result in noisy tension results, due to the sudden changes in motion of the crane.

A model of a real crane operation would of course need to include realistic properties for the slings, payload, crane wire and vessels.

When the payload is lowered onto the support vessel's deck, the drawing vertices of the 6D buoy generate a reaction force when they contact the deck shape (*Support Deck*). It is also possible to include friction as part of this interaction. Open the *friction coefficients* data form from the *model browser*. This allows us to control friction between lines (or buoys) and elastic solid shapes, and also between lines and supports.

In this case, on the *6D buoys* page, a *friction coefficient* of 0.2 has been applied between *all 6D buoys* and *all shapes*. The payload is a 6 x 6 x 6m box, so the area of the payload that would contact the deck would be 36m<sup>2</sup>. The payload will land on the deck with four vertices in contact with the deck shape, therefore we need those four vertices to each have a contact area of 9m<sup>2</sup> (36 / 4).

The buoy contains a total of eight vertices and the *total contact area* on the buoy's *contact* page is divided equally among the total number of vertices. Therefore, setting the *total contact area* to 72m<sup>2</sup> means that each vertex will represent 9m<sup>2</sup> of contact area.

## Results

Opening the simulation file opens the default workspace. The two results graphs show how the two vessels roll in response to the moving payload position. The top right-hand window shows the instantaneous time history results for tension in the four slings, and in the crane wire.

Run the replay (*Ctrl + R*) and notice that the values change to show the instantaneous result.

