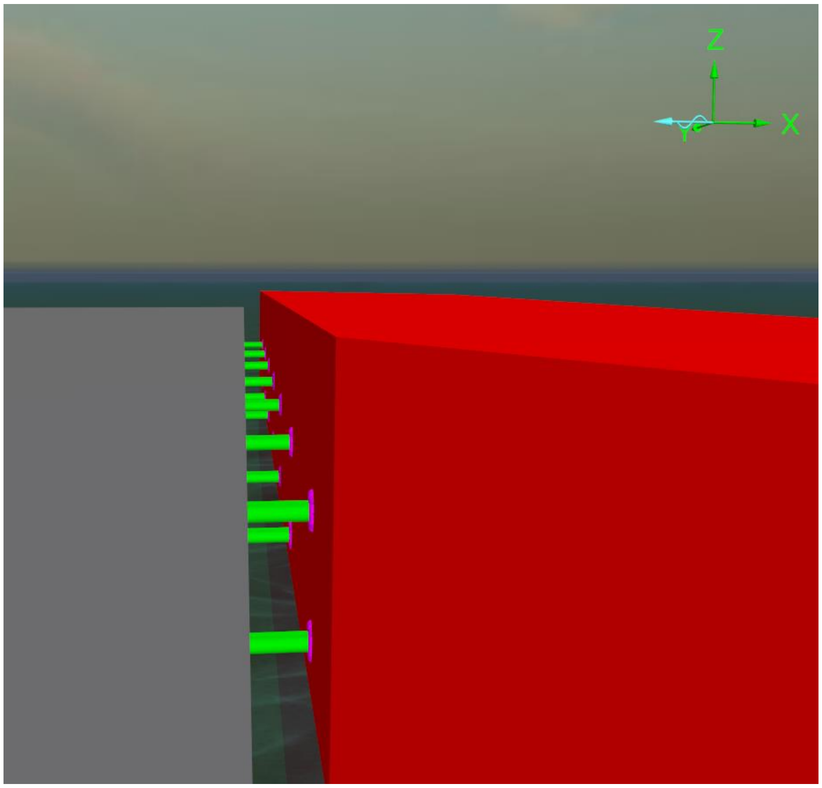
C09 Fenders



Introduction

In this example a method of modelling a quayside fender is presented. The method uses a *constraint* object to model the non-linear deflection and damping properties of the fender. The type of fender considered is a cell fender, but a similar approach could be used for other types of fenders (e.g pneumatic fenders etc).

Building the model

The fender has been simply modelled using two objects, a *constraint* object and a lumped mass *6D buoy* object. The constraint object has been used to model the non-linear stiffness and damping characteristics of the fender, and also to limit the fender's motion so that it only moves in its axial direction.

If we open the data form for one of the constraint objects, we can see that the only degree of freedom (DOF) that is set to be free on the constraint is the local x direction. This ensures that the fender will only move forwards and backwards in translation in the local x direction only. Motion in all other degrees of freedom is not permitted.

If we then examine the *stiffness and damping* tab of the constraint object we can see that the non-linear deflection and damping properties of the fender are accounted for by the variable data items, *Fender non-linear stiffness* and *Fender non-linear damping* respectively. The data used here is for demonstration purposes and is not representative of any particular model of fender.

If we look at the stiffness profile we can see that the reaction force is quite low for initial compression of the fender, before rising more steeply as the fender is compressed further. In reality only the negative displacement values need to be modelled, but it is good practice when using non-linear stiffness tables to include both positive and negative displacement values.

Contact between the vessel and fender is modelled via the 6D buoys (e.g. *Fender Contact Face Upper...*) attached to the constraint objects, and the elastic solid shape (*Vessel Contact*) attached to the vessel object. Contact occurs between the 6D buoy vertices, and the elastic solid face. The *lumped buoy* type of 6D buoy allows you to position the vertices wherever you like, so it is ideal for modelling the contact face of the fender. The *total contact area* is specified on the *contact* page of the buoy data form, and this area is divided equally among the vertices.

In this model the *total contact area* is $0.3m^2$ and there are 28 vertices, therefore each vertex represents $0.01m^2$ of contact area. All other properties assigned to the buoy are negligible.

Some *drawing* shapes have been attached to the 6D buoy objects to visually represent the fender body. Two shapes have been used here (one connected to the buoy and the other fixed to global) so that in shaded graphics view it appears that the fender is contracted when contact occurs between the vessel and the fender.

The mooring lines between the vessel and quayside have been simply modelled as constant tension winch objects, but lines could be used instead if more detail is required in the moorings.

The quayside has been represented by a drawing shape, for visualisation purposes.

The vessel object has been modelled using the default vessel type. For its calculation the vessel object must be included in the static calculation, and also have its *primary motion* set to *calculated (6 DOF)* in order for the contact events between the fender buoys and elastic solid shape attached to the vessel to have an impact on the vessel motion and position.

Results

Opening up the simulation file opens up the default workspace file, which displays a shaded graphics view and a wire frame view of the fender and vessel and some important results.

A time history of fender deflection and reaction force is shown for one of the lower fender objects.