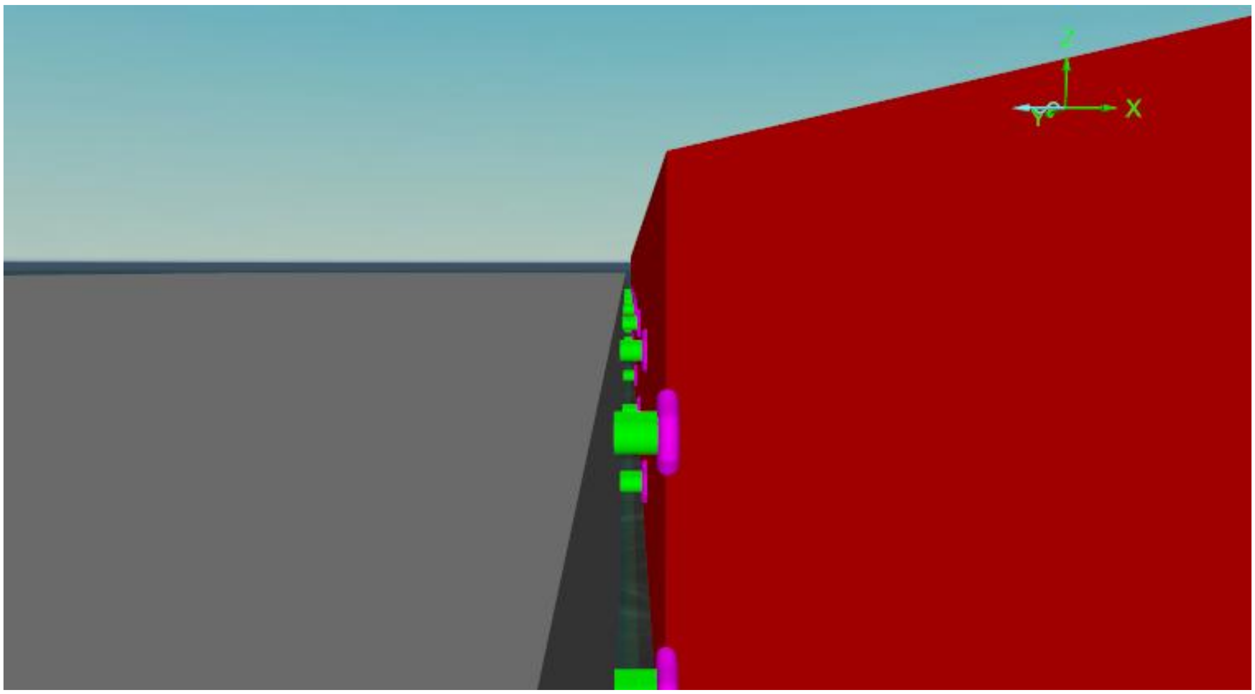
C09 Fenders

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 a cell type fender. A similar approach could be used to model other types of fender e.g. pneumatic fenders.



Building the model

Each fender has been modelled using two objects: a *constraint* and a lumped mass *6D buoy*. The constraint object has been used to model the non-linear stiffness and damping characteristics of the fender, and 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 only one *degree of freedom* (DOF) is free i.e. in the local *x* direction. This ensures that the fender will only translate forward and backwards in the x-direction. Motion is not permitted in the other DOFs.

If we then examine the *stiffness and damping* page of the constraint data form, we can see that the non-linear deflection and damping properties of the fender are captured by the variable data sets named *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 fender specification.

Right-click on the *translational stiffness* data item and select *edit variable data*. Press the *profile* button to open a graph of the *Fender non-linear stiffness* profile. Here 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 to include both positive and negative values when using non-linear stiffness tables.

Contact between the vessel and fender is modelled via the 6D buoys named *Fender Contact Face Upper…*. The 6D buoys are connected to the constraint objects, and an *elastic solid* type shape (*Vessel Contact*) is connected to the vessel object (*Barge*). Contact occurs between the 6D buoy vertices, and the elastic solid face. The *lumped buoy* type 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* type 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 modelled as *winch* objects with constant tension, but lines could be used instead if more detail is required in the moorings.

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

The *Barge* object has been modelled using the default *vessel type*. For its calculation, the vessel object must be included in the static calculation (*included in static analysis* set to *6 DOF*) and have its *primary motion* set to *calculated (6 DOF)*. This ensures that the contact between the fender buoys and *Vessel Contact* shape will affect the vessel motion and position.

Results

Opening the simulation file, the default workspace displays separate shaded graphics and wire frame views of the fender & vessel. Also shown are a couple of *time history* graphs, displaying the fender deflection and reaction force for one of the lower fender arrangements.