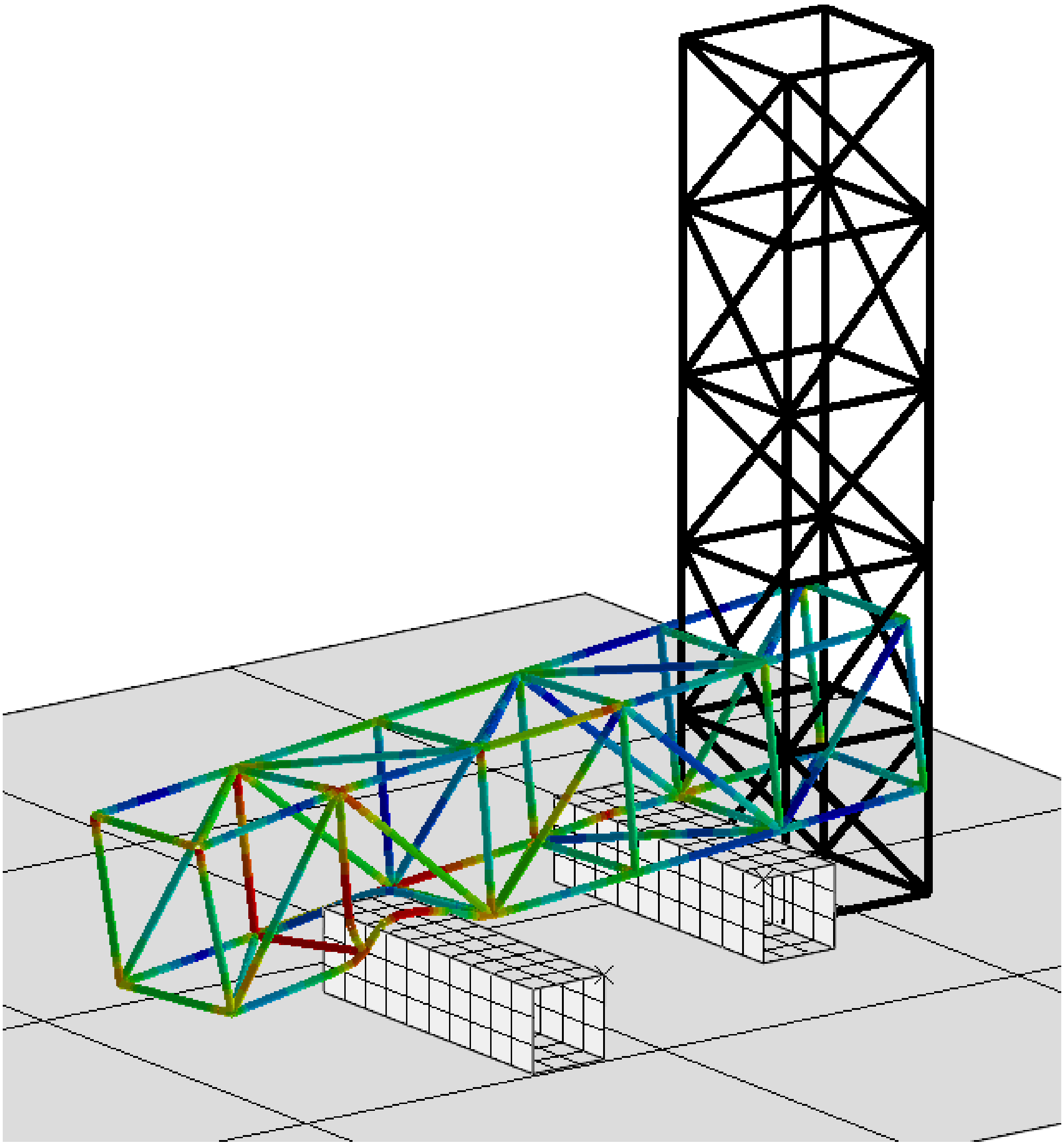
Tutorial 32:

Tower fall: beam contact



1.1 INTRODUCTION

This exercise involves the use of beam elements to model a **tower falling**. Contact with two objects on the floor will deform the tower.

Contact between **beam elements** and the surrounding environment is defined via general contact algorithm.

Preliminaries

- **Double click** on the file *tutorial32.cae*, this will open an Abaqus database where you will already find a model called *Model-TOWER* including the geometric parts, the instance positioning in the assembly and the discretization of the parts. Material properties and beam section assignment to the tower have already been defined as well.

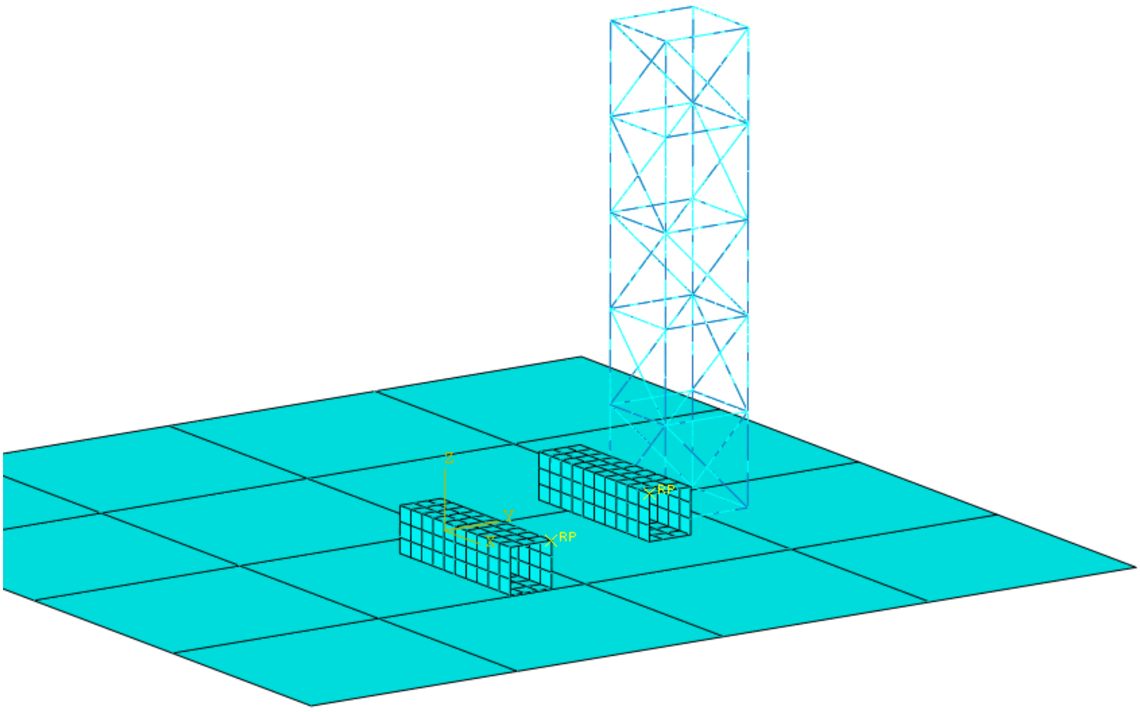


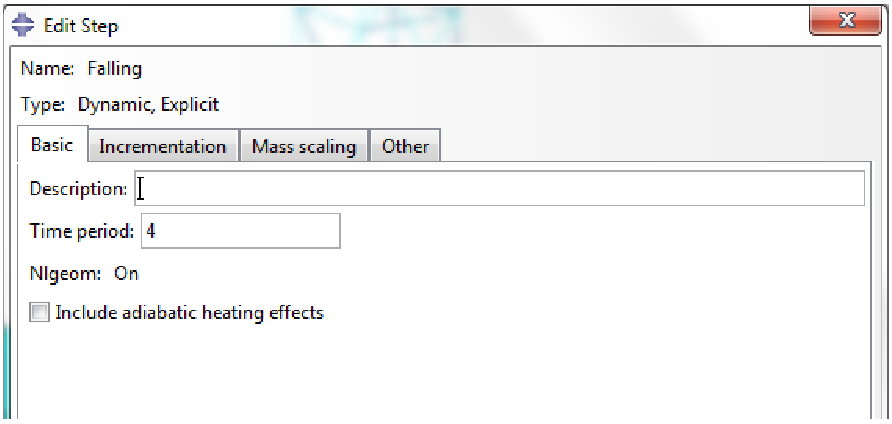
Figure 1. Assembly of the tower model.

1.2 MATERIAL AND SECTION PROPERTIES

Check the material properties definition. In particular, notice:
- The **plastic behaviour** of steel. This will results in permanent deformations after the impact of the tower with the floor.
- The **profile definition** used for the beam section properties. A pipe profile characterized by an internal diameter and a thickness is used in this model.

1.3 DEFINE THE ANALYSIS STEPS

- Double-click on **Steps** in the model tree and create a new *Dynamic, Explicit* step called *Falling*. Set **Nlgeom** parameter as on and enter 4 as Time Period. Click OK.



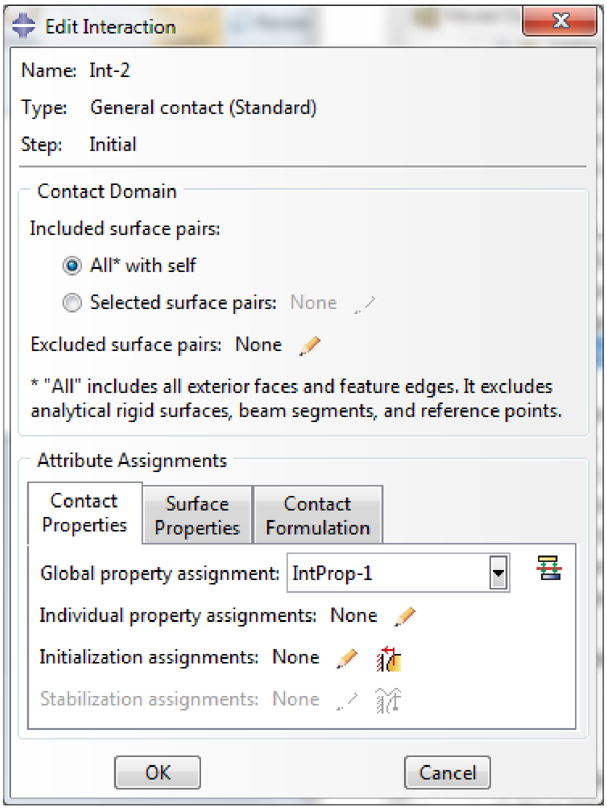
- Enter the field output requests manager by right-clicking the **Field Output Requests** container in the model tree and selecting manager. Edit the Field outpit request in the Step Connection entering 200 as number intervals.

1.4 DEFINE INTERACTIONS USING GENERAL CONTACT APPROACH

- Create new Interaction Property by double-clicking **Interaction Properties** in the model tree, call it *IntProp-1* and select *Contact* as Type. Select Hard contact for **Mechanical→ Normal**

Behaviour. Select **Mechanical→Tangential Behaviour**, select Penalty as Friction formulation and enter 0.1 as friction coefficient.

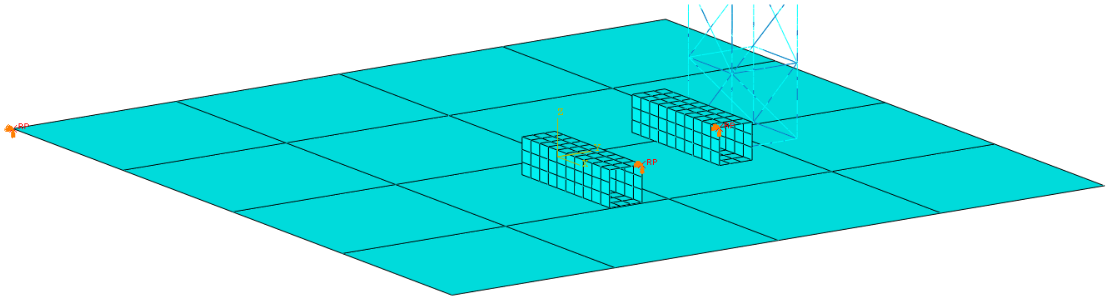
- Create now a new General Contact interaction. Double click on Interactions, select Initial as Step, call the new interaction as *INT-GC* and choose General Contact as type. Keep **All with self** as contact domain to say that you want all the parts of your system in contact with everything. Select *IntProp-1* as global interaction property.



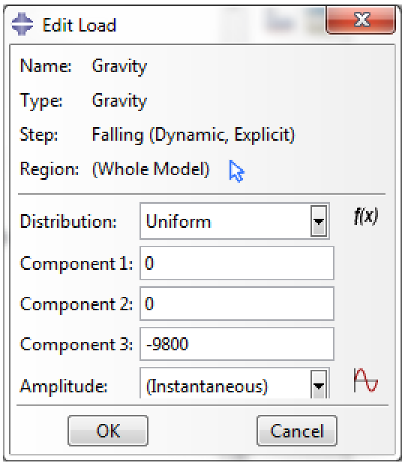
1.5 DEFINE BOUNDARY AND LOADING CONDITIONS

One boundary condition and two loadings are applied to this model.

- Create now a new Boundary condition called **BC-Encastre** to encastre the lower rigid tool. Select Initial as step and Symmetry/Antisymmetry/Encastre as type. Select the set called Set-ENCASTRE as region and then **ENCASTRE** as type. This boundary condition keeps the terrain and the two objects fixed at their initial position.

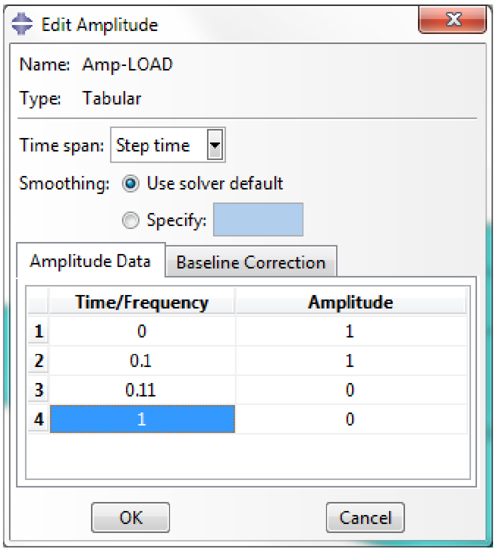


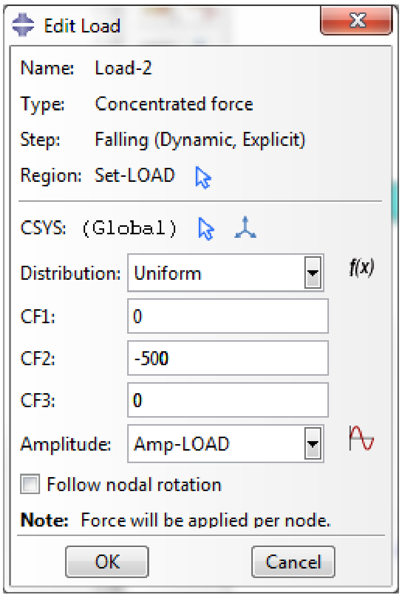
- Define Gravity. Double-click on **Loads** in the model tree. Select *Falling* as step, **Gravity** as type, select the whole model as region (default). Enter -9800 in the Component 3 field and 0 in the others. Click OK.



- Create a new Amplitude by double-clicking on **Amplitudes** in the model tree. Select **Tabular** as type, and enter the values shown in the following picture. Click OK.

- Create a new Load by double-clicking on **Loads** in the model tree. Select Step-1 as step, **Concentrated Force** as type, select the set Set-LOAD as region and enter -500 in the as CF" field ond 0 in the others. Select Amp-LOAD as amplitude. Click OK.

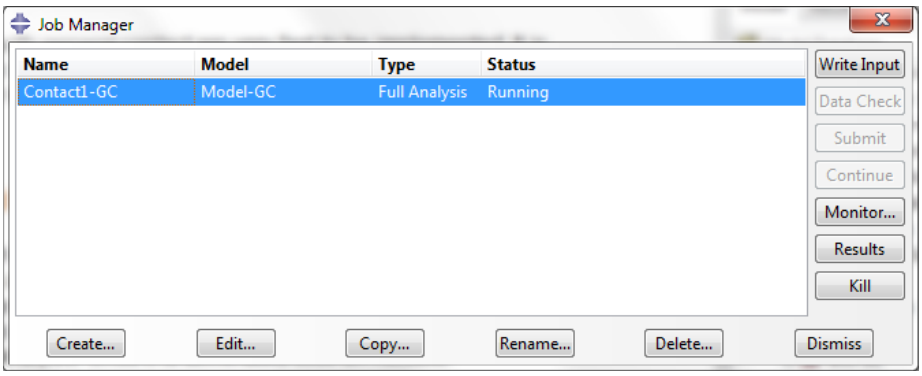




- Save the model. **File → Save**.

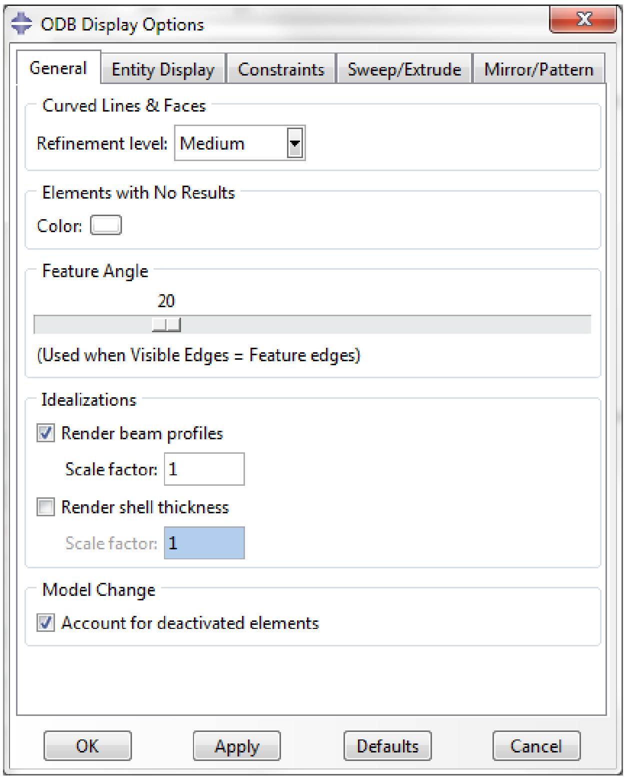
1.6 CREATE AND SUBMIT A JOB

- Double-click **Jobs** in the model tree and create a new Job called *Contact8-TOWER*. Select the Model-TOWER and click Continue and then ok. Now, right-click on Jobs and open the Job-Manager. Highlight the job *Contact8-TOWER* previously created and click on *Submit* to start your analysis and on *Monitor* to monitor the advancement of the analysis. Once it is terminated click on Results.



1.7 VIEW THE RESULTS

- In the main toolbar, click on View→ ODB display options and tick the **Render beam profiles** options so that the real profile is shown in the Results viewport. Click OK.



- Visualize the analysis results and animate the simulation.