Lab 2a

Spheres falling into a rigid box

Introduction

In this lab, we will be performing dynamic analysis of problems in which inertia effects are considered. We will examine the behavior of rubber spheres falling under gravity into a rigid box. General contact conditions will be defined. We will be using an explicit dynamic procedure to model the problem. Since it is a dynamic problem, inertial effects are taken into account and hence we need to define mass or density.

Goals

1. Define a rigid body constraint.
2. Define a general contact interaction.
3. Apply gravity loading.
4. Use Abaqus/Viewer to view results.

Recall that Abaqus /CAE does not use specific units, but the units must be consistent throughout the model.
Note that SI units are used in this lab: N, mm, tonne, and sec. With this choice of units, stresses are expressed in MPa.

Preliminaries

1. In the **Start Session** dialog box, underneath **Create Model Database**, click **With Standard/Explicit Model**.
2. To create a model, select **Model→Create** from the main menu bar and enter the name **`SPHERESINBOX`** in the **Edit Model Attributes** dialog box. Click **OK**. (you also could rename Model-1 to **`SPHERESINBOX)`**
3. To save the model database, select **File→Save As** from the main menu bar and type the file name **`SPHERESINBOX`** in the **Save Model Database As** dialog box. Click **OK**.

   The **`.cae`** extension is added to the file name automatically.

Creation of Parts

In this section, you create two parts, a sphere and an open box.

Creation of the sphere

1. First, create a three-dimensional, deformable solid body of the type revolution by sketching the two-dimensional profile of the sphere (a semi-circle of radius 7.5 mm) and revolving it 360 degrees.

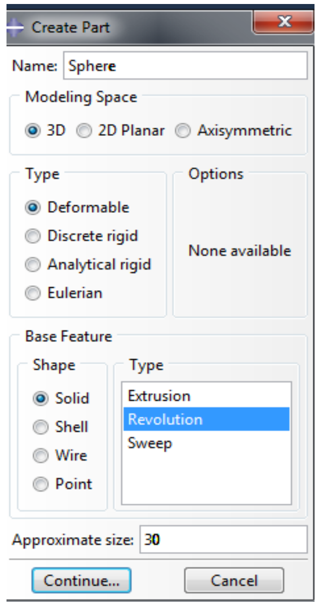


Figure L1a–1. Create part dialog box.

2. From the main menu bar, select **Part→Create** to create a new part.
   In the **Create Part** dialog box that appears, Figure L1a–1, name the part **`Sphere`**, and specify an approximate size of **`30`**. Accept the settings of a three-dimensional, deformable body with a solid, revolution base feature. Click **Continue**.

3. To sketch the profile of the sphere, you need to select the arc drawing tool,  which draws an arc based on the center and 2 endpoints. Close the two open ends of the semi-circle using the line drawing tool . Dimension the semi-circle as shown in Figure L1a–2.

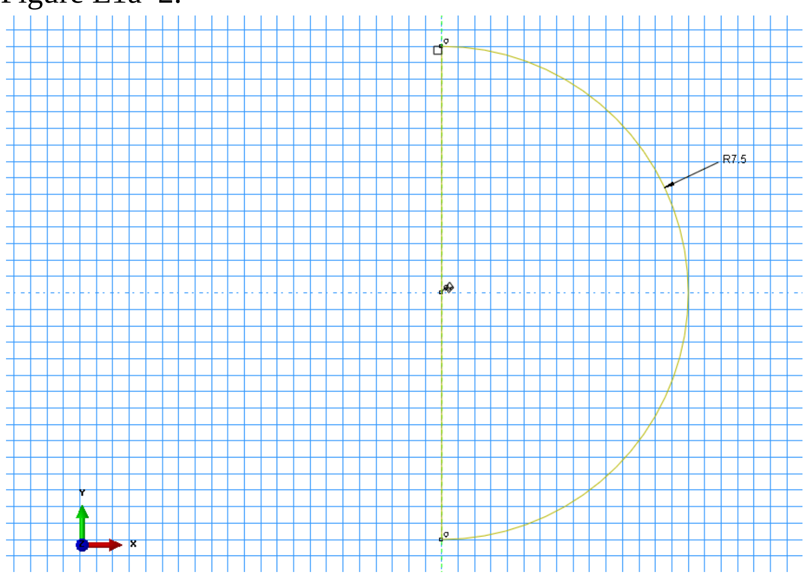


Figure L1a–2. Sketch of the semi-circle.

4. Click **Done** in the prompt area to exit the sketcher.
5. Enter a revolution angle of 360 degree and click **OK**. The revolved part is shown in Figure L1a-3.

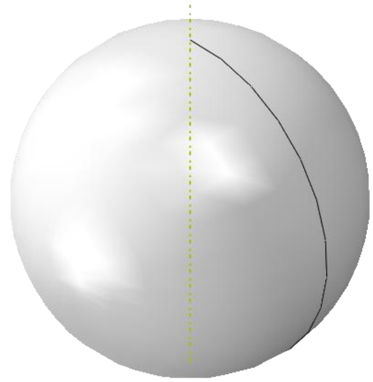
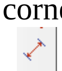


Figure L1a–3. Revolved part.

6. Once the part has been created, from the main menu bar, select **Tools→Surface→Create** to represent the outer surface of the sphere. Name the surface as **`sphere-outer`** and hit continue. Select the sphere by clicking in the viewport and hit Done.

Creation of the box

1. Create a three-dimensional, discrete rigid shell base feature of the type planar to create the base for the open box and name it **`Base`**. Choose an approximate size = 100.
2. In the sketcher, select the create lines: rectangle drawing tool,  to sketch a rectangle by choosing the position of 2 opposite corners of the rectangle. Dimension the rectangle using the dimensioning tool  such that both sides measure 50 mm. Click **Done** in the prompt area to exit the sketcher.
3. Now, create a three-dimensional, discrete rigid shell base feature of the type extrusion to create the extruded part for the open box and name it **`Extruded`**. Choose an approximate size = 100.
4. Select the create lines: rectangle drawing tool,  to sketch a rectangle and dimension the rectangle such that both sides measure 50 mm. Click **Done** in the prompt area to exit the sketcher. Enter an extrusion depth of 50 mm.

Material and Section Properties

The revolved sphere is to be made of rubber and is modeled with general **hyperelastic** material properties as a Mooney-Rivlin material with the constants $C_{10} =$ 0.690 MPa, $C_{01} =$0.173 MPa and $D_1 =$0.0145 MPa^{-1}. The density of the tennis ball is 1.068 x 10^{-6} kg/mm^3.

1. From the main menu bar, select **Material→Create** to create a new material.
2. In the **Edit Material** dialog box that appears, name the material **`Rubber`**.

3. From the material editor's menu bar, select **General→Density,** and enter a value for density as `1.068E-6`.
4. Go to **Mechanical→Elasticity→Hyperelastic.** Select **Mooney Rivlin** as the strain energy potential function and the input source as **Coefficients**. Then enter the value for coefficients of the Mooney Rivlin model as shown in Figure L1a-4.

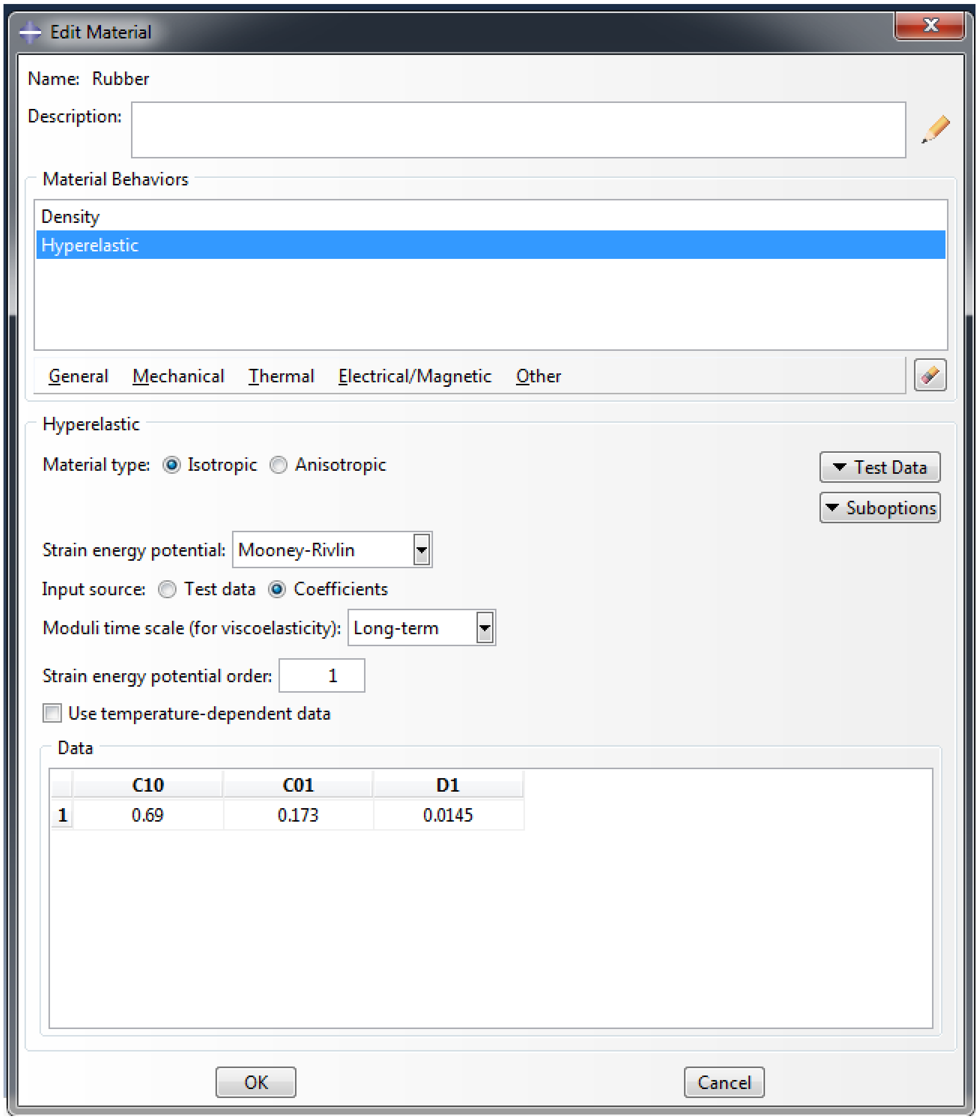


Figure L1a–4. Material editor.

5. Click **OK** to exit the material editor.
6. From the main menu bar, select **Section→Create**.
7. In the **Create Section** dialog box that appears:
    a. Name the section `SolidSection`.
    b. Accept the default category **Solid** and the default type **Homogeneous**.
    c. Click **Continue**.
8. In the **Edit Section** dialog box that appears:

a. Accept the default selection of **Rubber** for the **Material** associated with the section.

b. Click **OK**.

9. From the main menu bar, select **Assign→Section**.

   a. Click anywhere on the sphere to select it as the region to which the section will be assigned.

   b. Click **Done** in the prompt area to accept the selected geometry.

   c. In the **Edit Section Assignment** dialog box, accept the default selection of **SolidSection** as the section definition, and click **OK**.

Assembling the model

4. In the **Module** list located in the context bar, select **Assembly** to enter the Assembly module.
5. From the main menu bar, select **Instance→Create**.
6. In the **Create Instance** dialog box, select the parts **`Base and Extruded`** (hold the shift key and select) and click **OK**.
7. From the main menu bar, select **Instance→Translate,** and select the **`Base-1`** instance from the viewport by clicking on it. Select A as the start point of translation and B as the end point of translation as shown in Figure L1a-5.

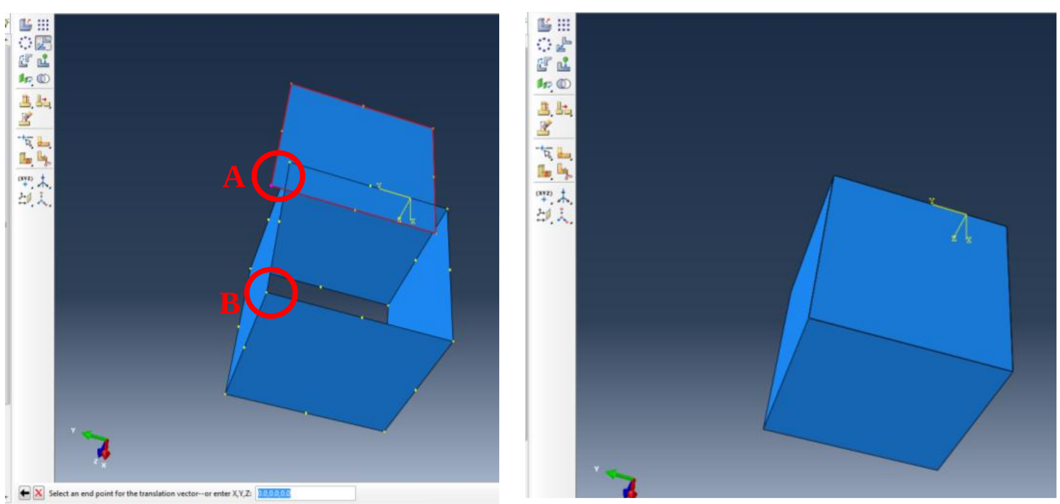


Figure L1a–5. Translating instance from one point to another.

8. From the main menu bar, select **Instance→Merge/Cut,** name the part as **`open box`** and select to merge geometry and accept all other defaults as shown in Figure L1a-6 and hit continue. Select the two instances **`Base-1 and Extruded-1`** and hit Done. This operation merges the two instances to a single part **`open box`** and creates an instance for the same named as **`open box-1.`** It also suppresses the original instances **`Base-1`** and **`Extruded-1`**.

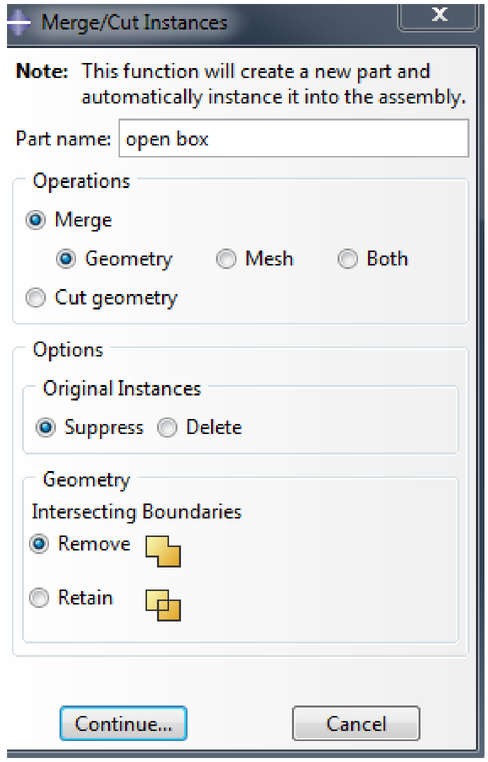


Figure L1a–6. Assembly operations performed on instances.

9. Return back to the Part module and select the open box part that was just created. To define a surface on the inside of the box, from the main menu bar, select **Tools→Surface→Create.** Name the surface as **`box-inner`** and hit continue. Select all the inner faces of the box by holding the shift key while selecting and hit Done. When prompted on choosing a side for the internal faces, select the color (Brown/Purple) that corresponds to the inner faces (in this case, see Figure L1a-7, Brown). Again select **Tools→Reference Point** and select any vertex of the open box to act as a reference point.

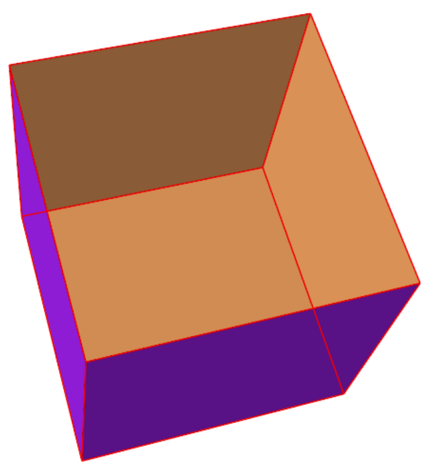
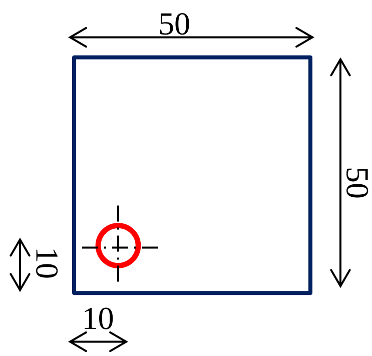
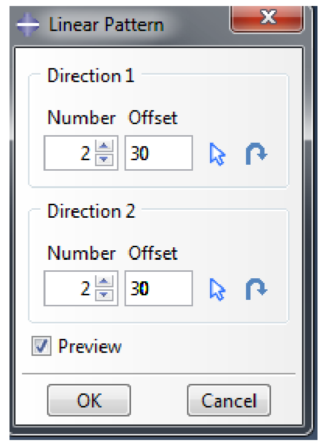


Figure L1a–7. Choosing a side for a surface.

10. Create an instance for the part **`Sphere`**. Translate the instance inside the box by using the translation tool  such that it is positioned in the box as shown in Figure L1a-8a. Then select **Instance→Linear Pattern** to create a linear pattern for the instance **`Sphere-1.`** Fill the Linear Pattern dialog box as shown in Figure L1a-8b. The final assembly looks like the Figure L1a-9.





(a) (b)

Figure L1a–8. a) Positioning of the sphere instance inside the box, b) Assembly operation: linear pattern

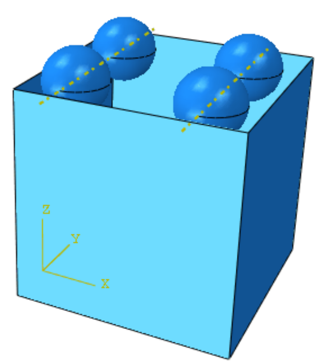
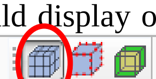
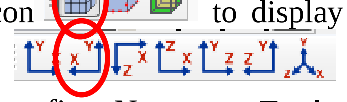


Figure L1a-9. Assembly consisting of spheres inside the open box.

11. In order to track the displacement of the spheres in the box, we will create two node sets. From the main menu bar go to **Tools→Display Group→Create.** Select **Part Instances** from the item selection list and choose instance **`openbox-1.`** Select remove from the Boolean operations. This would display only the four spheres in the viewport. Switch on the show native mesh icon  to display the mesh on the sphere instances. Switch to the X-Y view  to view the faces of the spheres that would contact the rigid box first. Now go to **Tools→Set→Create**. Name the

set as **`bottomnodes`** and select the bottom nodes from each sphere as shown in Figure La-10. Click on **Replace All**  to resume the display of all instances.

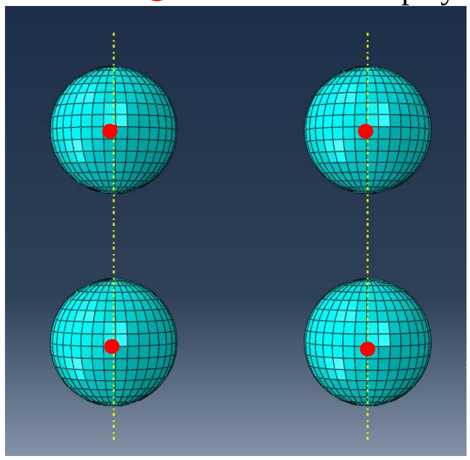


Figure L1a–10. Creating node sets.

Configuring the analysis

In this simulation, we will be performing dynamic analysis to examine the behavior of rubbers spheres falling under gravity loading. The explicit dynamics procedure performs a large number of small time increments efficiently. A point to remember here is that as the element size gets smaller, a smaller time increment is used and longer is the time required to solve a problem at hand with this procedure.

To create a dynamic, explicit analysis step:

1. Enter the **Step** Module and from the main menu bar, select **Step→Create** to create a step.
2. In the **Create Step** dialog box that appears:
   a. Name the step **`Gravityload`**.
   b. From the list of available general procedures in the **Create Step** dialog box, select **Dynamic, Explicit**.
   c. Click **Continue**.
3. In the **Edit Step** dialog box that appears:
   a. In the **Description** field of the **Basic** tabbed page, enter **`falling of spheres under gravity.`** Enter a time period of **`0.2 sec`**.
   b. Accept all other defaults and click **OK** to create the step and to exit the step editor.

Defining contact interaction for the model

1. In the Model Tree, double-click **Interaction Properties**.
2. In the **Create Interaction Property** dialog box, name it **`contact`**, accept **Contact** as the interaction type and click **Continue**.
3. In the **Edit Contact Property** dialog box, select **Mechanical→Normal Behavior** and accept the defaults. Click **OK** to close the dialog box.
4. In the Model Tree, double-click **Interactions**.

5. In the **Create Interaction** dialog box, accept **Gravityload** as the step in which the interaction will be created and **General contact (Explicit)** as the interaction type.
6. Click **Continue**.
7. In the **Edit Interaction** dialog box, accept the **All* with self** in the contact domain.
8. Choose the contact property **contact** that you just defined under global property assignment and click **OK** to close the dialog box.

Defining rigid body constraint

The rigid body constraint allows you to constrain the motion of regions of the assembly to the motion of a **reference point**. This is used to model parts which are much stiffer compared to other bodies in the assembly.

1. From the main menu bar, select **Tools→Display Group→Create.** In the create display group dialog box, select **Part Instances** in the item selection, and in the right hand side of the dialog box select `open box-1`. Perform the **Replace** Boolean by clicking on it. This operation shows only the open box instance in the viewport. Click dismiss to come back to the viewport.
2. In the Model Tree, double-click **Constraints**.
3. In the **Create Constraint** dialog box, select **Rigid body** as the constraint type and click **Continue**.
4. In the **Edit Constraint** dialog box, select the region type **Body (elements)** and select the entire open-box instance (using the select tool) using a rectangular drag shape across it. See Figure L1a-11.
5. Similarly, select the reference point **RP** in the viewport as the rigid body reference point.
6. In the **Edit Constraint** dialog box, click **OK** to apply the constraint.
7. Click the **Replace All**  icon in the panel under the main menu bar in order to display all instances back in the viewport.

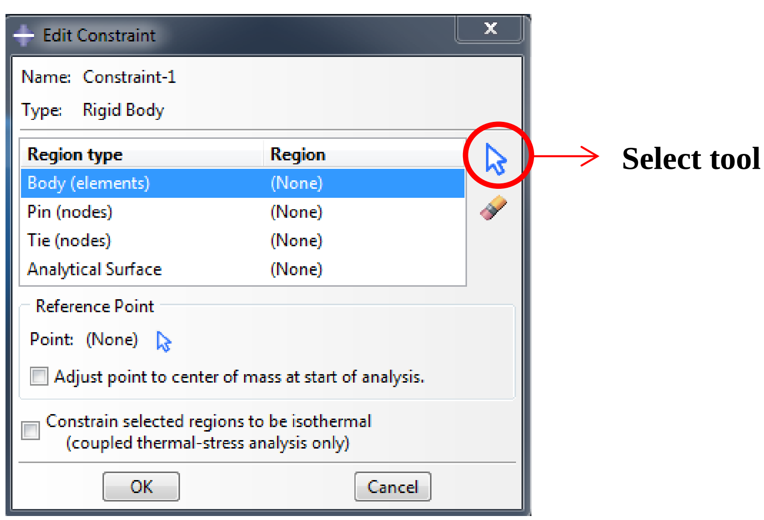


Figure L1a–11. Defining rigid body constaints.

Applying a boundary condition and a load to the model

Defining a fixed boundary condition for the base of the open box

Next, you will define the boundary condition and loading that will be active during the `GravityLoad` step.

1. In the **Module** list located in the context bar, select **Load** to enter the Load module.
2. From the main menu bar, select **BC→Create**.
3. In the **Create Boundary Condition** dialog box that appears:
   a. Name the boundary condition `Fixed`.
   b. Select **Initial** as the step in which the boundary condition will be activated.
   c. In the **Category** list, accept the default category selection **Mechanical**.
   d. In the **Types for Selected Step** list, select **Displacement/Rotation** as the type.
   e. Click **Continue**.
4. Since this part is a rigid body, all the loading and boundary conditions for this part can be imposed on the reference point, as shown in Figure L1a–12.

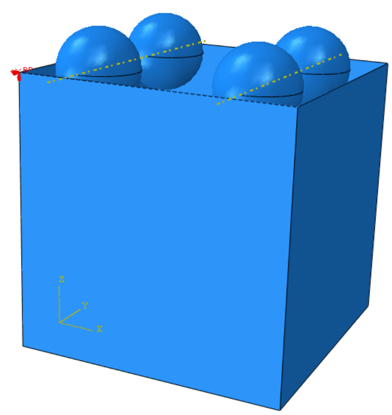


Figure L1a–12. Fix the reference point.

5. Click **Done** in the prompt area to accept the selected geometry.

   The **Edit Boundary Condition** dialog box appears. In the **Edit Boundary Condition** dialog box:

   a. Toggle on **U1**, **U2**, **U3, UR1, UR2, and UR3** to constrain all degrees of freedom.
   b. Click **OK** to create the boundary condition definition and to exit the editor.

Application of gravity load:

1. From the main menu bar, select **Load→Create**.
2. In the **Create Load** dialog box that appears:
   a. Name the load `Gravity`.

   b. Select **`Gravityload`** as the step in which the load will be applied.
   c. In the **Category** list, accept the default category selection **Mechanical**.
   d. In the **Types for Selected Step** list, select **Gravity**.
   e. Click **Continue**.
3. By default the Whole Model will be selected for the gravity load application. Fill in the components of the gravity load as shown in Figure L1a-13.

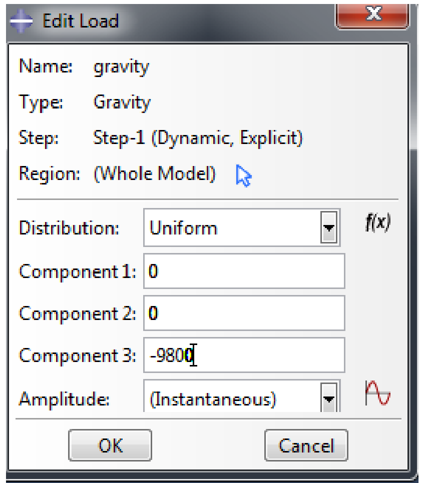


Figure L1a–13. Definition of gravity load.

4. Make sure that negative z direction corresponds to the vertically downward direction in your assembly. Click **OK** to create the load definition and to exit the editor.

Meshing the model

Meshing the open box

To assign the mesh controls:

1. In the **Module** list located in the context bar, select **Mesh** to enter the Mesh module.
2. In the context bar, select **Part** as the displayed object and select **`open box`** as the part.
3. From the main menu bar, select **Mesh→Controls**.
4. When prompted to select the region to be assigned mesh controls, select the entire open box by dragging a rectangular shape across it and click Done. In the **Mesh Controls** dialog box that appears, accept **Quad** as the default **Element Shape** selection.
5. Accept **Structured** as the default **Technique** selection.
6. Click **OK** to assign the mesh controls and to close the dialog box. The part will turn green signifying that a structured mesh can be generated on this part.

To assign an Abaqus element type:

1. From the main menu bar, select **Mesh→Element Type** and select the entire part for the region to be assigned element type as before.
2. In the **Element Type** dialog box, accept the following default selections:
   - **Explicit** is the default **Element Library** selection.
   - **Linear** is the default **Geometric Order**.
   - **Discrete Rigid Element** is the default **Family** of elements.
3. Click **OK** to assign the element type and to close the dialog box.

Seeding and meshing the part:

1. From the main menu bar, select **Seed→Part** to seed the part.

   The **Global Seeds** dialog box appears. In the **Global Seeds** dialog box, enter an approximate global size of **5** and click **OK**.
2. From the main menu bar, select **Mesh→Part** to mesh the part.

Meshing the sphere

To assign the mesh controls:

1. In the context bar, select **Part** as the displayed object and select `Sphere` as the part.
2. From the main menu bar, select **Mesh→Controls**.
3. In the **Mesh Controls** dialog box that appears, accept **Hex-dominated** as the default **Element Shape** selection.
4. Accept **Sweep** as the default **Technique** selection.
5. Click **OK** to assign the mesh controls and to close the dialog box. The part will turn yellow signifying that a sweep mesh can be generated on this part.

To assign an Abaqus element type:

1. From the main menu bar, select **Mesh→Element Type** and select the entire part for the region to be assigned element type as before.
2. In the **Element Type** dialog box, accept the following default selections:
   - **Explicit** is the default **Element Library** selection.
   - **Linear** is the default **Geometric Order**.
   - **3D Stress** is the default **Family** of elements.
3. In the lower portion of the dialog box, examine the element shape options. A brief description of the default element selection is available at the bottom of each tabbed page.
4. In the **Hex** tabbed page, select **Reduced Integration**.

   A description of the element type C3D8R appears at the bottom of the dialog box. Abaqus/CAE will now mesh the part with C3D8R elements.
5. Click **OK** to assign the element type and to close the dialog box.

Seeding and meshing the part:

1. From the main menu bar, select **Seed→Part** to seed the part.

   The **Global Seeds** dialog box appears. In the **Global Seeds** dialog box, enter an approximate global size of `1.5` and click **OK.**
2. From the main menu bar, select **Mesh→Part** to mesh the part.

   Switch to assembly view in the mesh module to view the resulting mesh, as shown in Figure L1a–14.

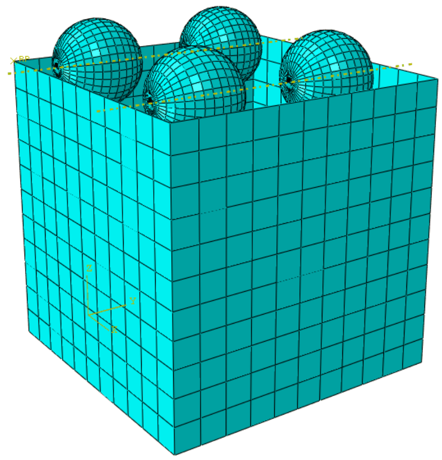


Figure L1a–14. Assembly showing the resulting mesh.

Creating and submitting an analysis job

The definition of the model **`SPHERESINBOX`** is now complete. Next, you will create and submit an analysis job to analyze the model.

To create and submit an analysis job:

1. In the **Module** list located in the context bar, select **Job** to enter the Job module.
2. From the main menu bar, select **Job→Manager**.
3. From the buttons on the bottom of the **Job Manager**, click **Create** to create a job.
4. In the **Create Job** dialog box that appears, name the job **`gravityfall`** and select the model **`SPHERESINBOX`**. Click **Continue**.

   The job editor appears.
5. Click the tabs to see the contents of each folder of the job editor and to review the default settings. Click **OK** to accept the default job settings.
6. Set the working directory of your job by selecting **File→Set Work Directory,** and select a directory where you would want this analysis files to be saved.
7. Right click the job that you just created and click **Submit** to submit your job for analysis. The status field will now show **Running**. You can monitor the progress of your analysis by right clicking on the job and selecting **Monitor**.
8. When the job completes successfully, the status field will change to **Completed**. You are now ready to view the results of the analysis in the Visualization

module. (It might take 3-5 minutes to complete depending on the speed of your workstation).

Viewing the analysis results

You are now ready to view the results of the analysis in the Visualization module.

1. Once the job completes, right click on the job and click **Results** to enter the Visualization module.

   Abaqus/CAE switches to the Visualization module, opens the output database created by the job (**`gravityfall.odb`**), and displays the undeformed shape of the model, as shown in Figure L1a–15.

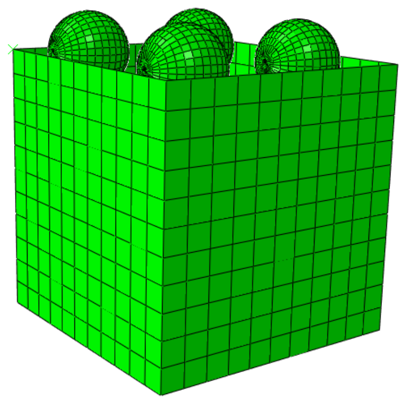


Figure L1a–15. Undeformed model shape.

2. In the toolbox, click  to animate the time history plot for the spheres falling under gravity and impacting the inside surface of the open box. See Figure La1-16. Click on the animation options tool  to set the speed of the animation. Select the field output to be displayed on the top panel bar as Primary S Mises stress and set the global translucency level to be able to view the motion of the spheres inside the open box. See Figure La1-17.

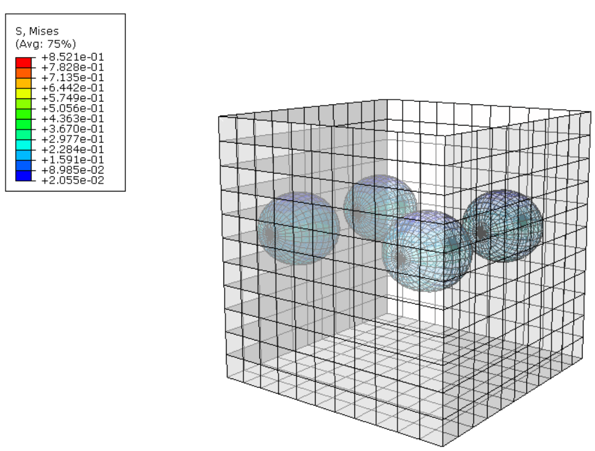


Figure L1a–16. Time history animation showing the bouncing of spheres.

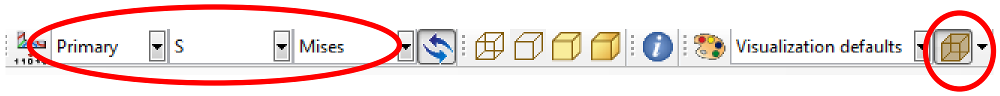


Figure L1a–17. Viewer options.

3. In the toolbox, click  (or select **Tools→X-Y Data →Create)** to plot X-Y data for the contact pressure on the contacting surfaces. Select **ODB field output** and hit Continue. Toggle the position to **Unique Nodal** and select **CPRESS** under variables. Move to the **element/nodes** tab, switch to the internal sets method selection and select all the General contact faces as shown in Figure L1a-18. Then hit plot. This shows the contact stresses on the region of the assembly that comes into contact when the spheres hits the rigid box and bounce back. This gives us information about the time instant at which the contact initiated, when the contact was lost, how much of contact stress was encountered between the two surfaces etc. You would notice that the contact takes place between 0.09-0.12 seconds. Save the plot as an X-Y Data so that you can access that later as well.

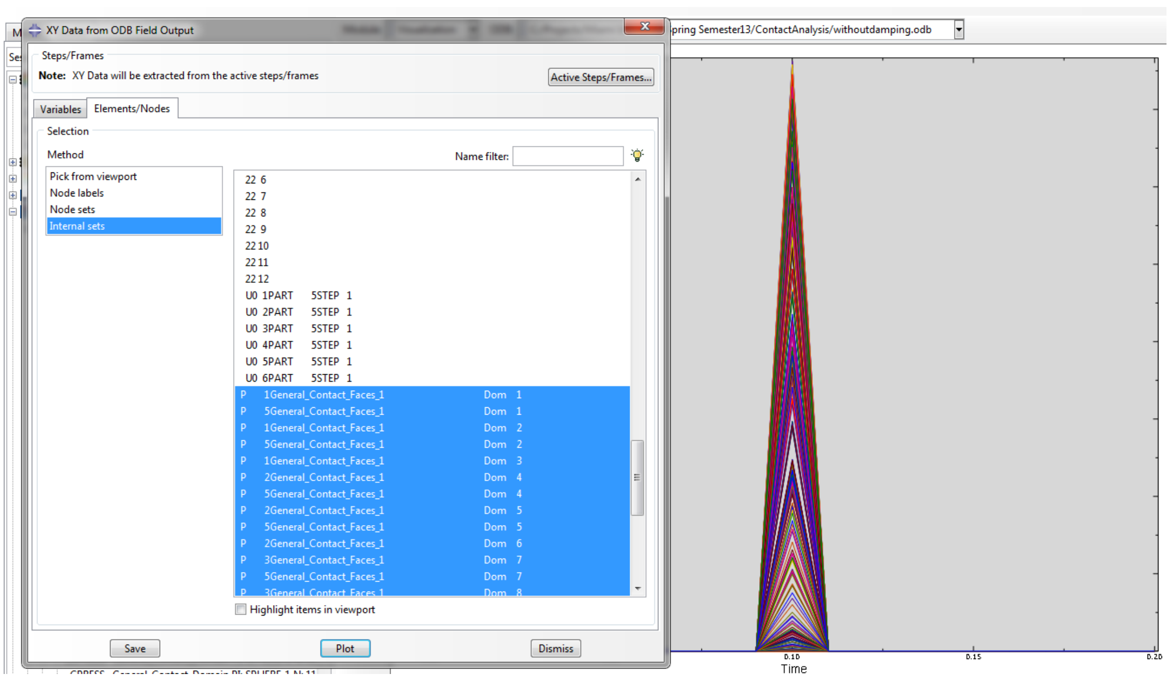


Figure L1a–18. Contact pressure plot.

4. From the list of available output variables in the center of the toolbar, select output variable **U3** (spatial displacement at nodes in the z direction), and in the element/nodes tab, pick the Node set named **`bottomnodes`** to track the displacement of the spheres. Figure L1a-19 shows the displacement plot.

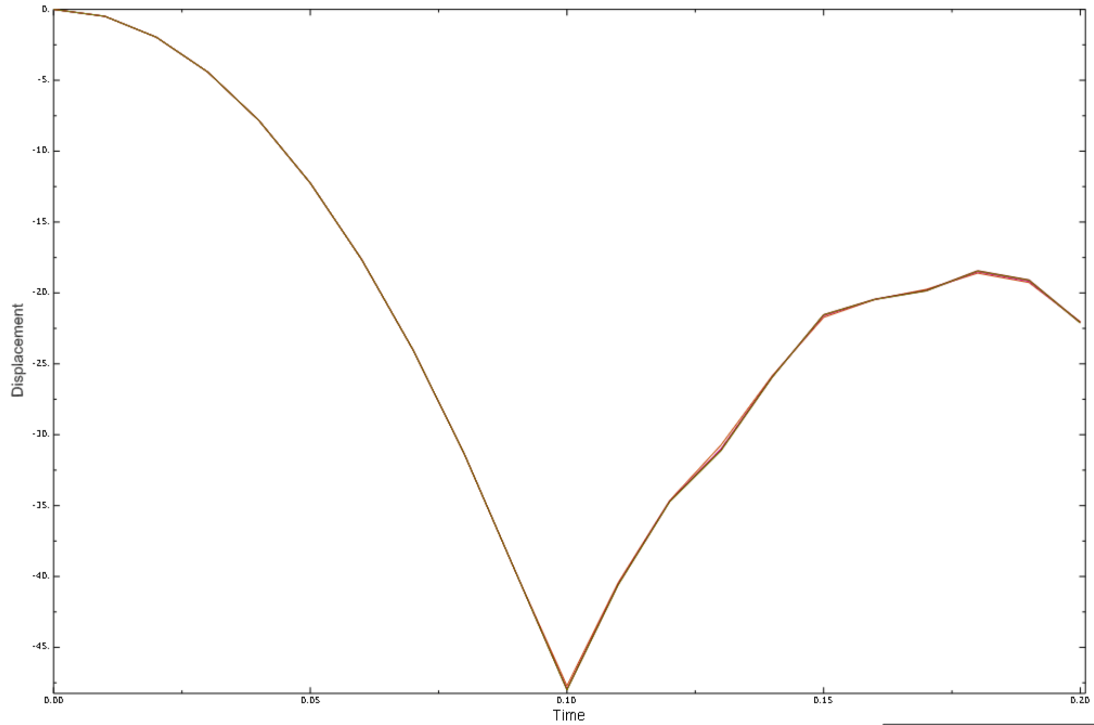
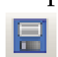


Figure L1a–19. Displacement vs time X-Y plot.

Save this X-Y data plot (you will need this plot later for comparison purposes). This shows that the spheres touches the base of the box around time t=0.09 sec, and due to inertia and impact it travels back upwards.

5. In the toolbar, click  (or select **File→Save** from the main menu bar) to save your model in a model database file.

Exercise:

Try running the analysis for a greater time period now, say t=0.5 seconds* and observe the behavior of the spheres in their displacement time graphs.

**Note : This analysis may take a long time to run (approx~ 1 hr)*