



Lab 1

Bolted Flange Assembly

Introduction

In this lab you will model two rectangular plates that sandwich a gasket and are tightened against each other using 2 bolts and nuts. The plates are made of brass, the gasket is made of rubber, and the bolts and nuts are made of steel.

Recall that Abaqus/CAE does not use specific units, but the units must be consistent throughout the model.

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 **Bolt tightening** in the **Edit Model Attributes** dialog box. Click **OK**. (you also could rename Model-1 to **Bolt tightening**)
3. To save the model database, select **File**→**Save As** from the main menu bar and type the file name **Bolt tightening** in the **Save Model Database As** dialog box. Click **OK**.

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

Creating the parts

In this section you will create the following parts – Plate, gasket, bolt and nut. Abaqus/CAE automatically loads the Part module. Any other module can be accessed from the **Module** list located in the context bar, as shown in Figure L1–1

To create the plate:

1. From the main menu bar, select **Part**→**Create** to create a new part. In the **Create Part** dialog box that appears, name the part **Plate**, and specify an approximate size of **20**.

Note: The size that you enter is used by ABAQUS/CAE to calculate the size of the Sketcher sheet and the spacing of its grid. You should choose this value to be on the order of the largest dimension of your finished part.

Accept the settings of a **3D, Deformable** type part with a **Solid** shape, **Extrusion** type base feature. Click **Continue**.

Abaqus/CAE displays text in the prompt area near the bottom of the window to guide you through the procedure, as shown in Figure L1–2. Click the cancel

button to cancel the current task; click the backup button to cancel the current step in the task and return to the previous step.

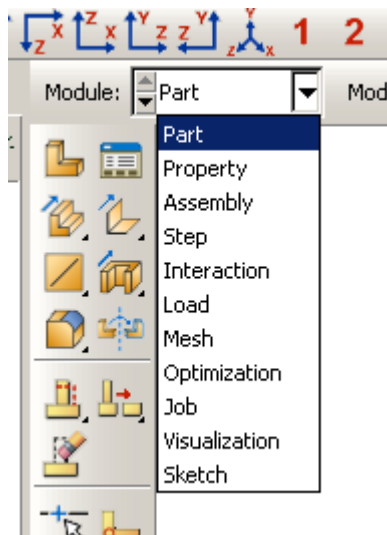


Figure L1-1. Module list

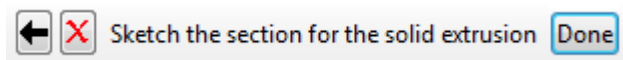


Figure L1-2. Prompt area.

The Sketcher toolbox appears in the left side of the main window, and the Sketcher grid appears in the viewport.

2. To sketch the rectangular profile of the plate, you need to select the **Create Lines: Rectangle** drawing tool, as shown in Figure L1-3.

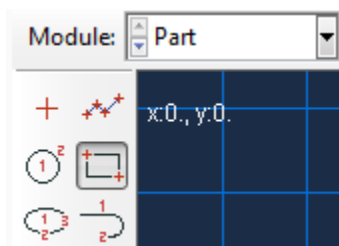




Figure L1-3. Rectangle sketch tool.

3. You will first sketch a rough approximation of the rectangle and then use dimension tool to refine the sketch. In the viewport, sketch rectangle as follows:
 - a. Select any two points on the grid as the diagonally opposite corners of the rectangle.
 - b. Click mouse button 2 anywhere in the viewport to exit the rectangle tool.

- c. Use the dimension tool  to dimension the radius of the circle to vertical side to 2.5 in. and the horizontal side to 10 in. When dimensioning, simply select the line, click mouse button 1 to position the dimension text, and then enter the new dimension in the prompt area.
4. Select the **Create circle: Center and Perimeter** drawing tool from the sketcher toolbox to sketch the two bolt holes.
5. You will first sketch a rough approximation of the hole and then use dimension tool to refine the sketch. In the viewport, sketch the hole as follows:
 1. Select any two points in the rectangle as the center and a point on the perimeter of the hole.
 2. Use the dimension tool  to dimension the radius of the circle to 0.25 in., the horizontal distance of the center of the circle from the left side of the rectangle to 2.5 in., and the vertical distance of the center of the circle from the top side of the rectangle to 1.25 in.
 3. Sketch a similar circle with its center 2.5 in. from the right side of the rectangle.
 4. The final sketch is shown in Figure L1–4.

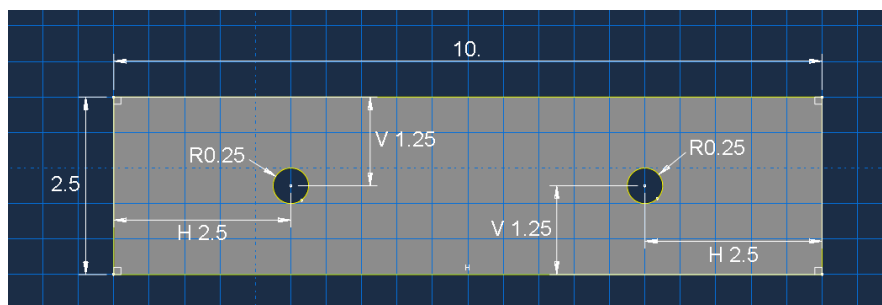


Figure L1–4. Sketch of the rectangular profile of the plate.

5. Click mouse button 2 and then click **Done** in the prompt area.
6. In the **Edit Base Extrusion** dialog box, enter a value of **0.5** for the **Depth** and click **OK**.
7. Abaqus/CAE displays the new part, as shown in Figure L1-5.
6. You will now create datum planes to help you create partitions which will allow proper meshing later. To create the datum planes:
 - a. Select **Tools**→**Datum** from the main menu bar.
 - b. In the **Create Datum** dialog box, select **Plane** type and **Offset from plane** as the method.

Abaqus/CAE displays text in the prompt area near the bottom of the window to guide you through the procedure.

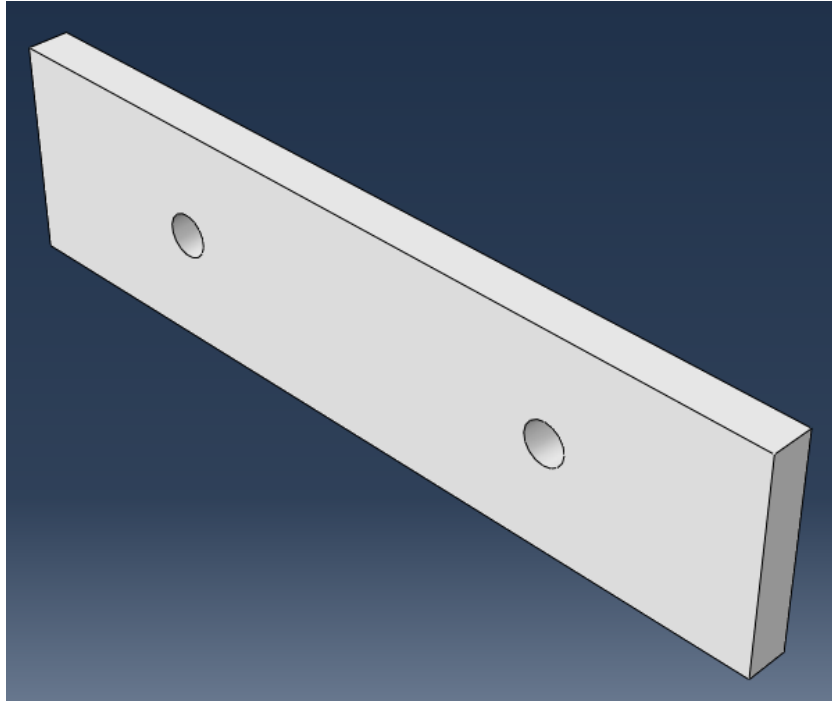


Figure L1–5. Plate.

- c. Select the vertical plane on the left side of the plate and click on **Enter Value** in the prompt area.
- d. Click on **Flip** in the prompt area if the arrow displayed in the viewport points in a direction away from the plate, then click **OK**.
- e. Enter an offset value of 1.9 in.
- f. Repeat steps a. to e. with offset values of 2.5, 3.1, 6.9, 7.5 and 8.1 in. See Figure L1-6.

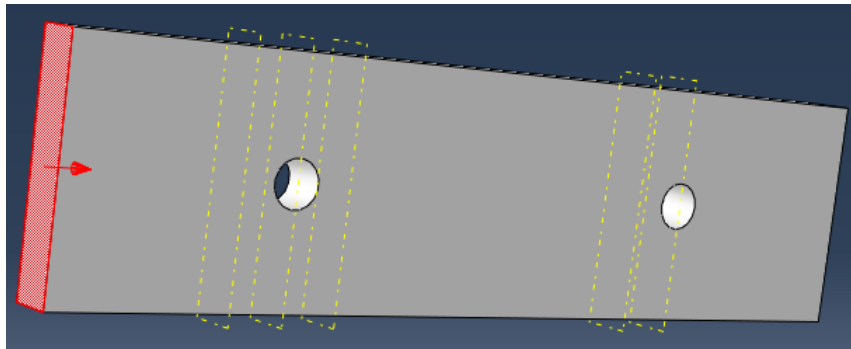


Figure L1–6. Datum planes.

- g. Now select the horizontal plane at the top of the plate and repeat similar procedure as above for creation of a single plane with the arrow pointing towards the plate and an offset value of 1.25 in.
7. Next, you will partition the plate using the datum planes you just created. To create the partitions:
 - a. Select **Tools**→**Partition** from the main menu bar.
 - b. In the **Create Partition** dialog box, select **Cell** type and **Use datum plane** as the method.
 - c. Select any vertical datum plane in the viewport and click **Create Partition** in the prompt area.
 - d. Repeat this for all the vertical datum planes you created. Note that once the plate has been partitioned, you will first have to select the cell or cells through which the desired partition will pass.
 - e. For the horizontal partitions, select the four small cells that enclose the two bolt holes. You can select multiple cells while holding the shift key. Then select the horizontal datum plane you created earlier and click **Create Partition** in the prompt area.
 - f. Click **Done** in the viewport. The final part is displayed in Figure L1-7.

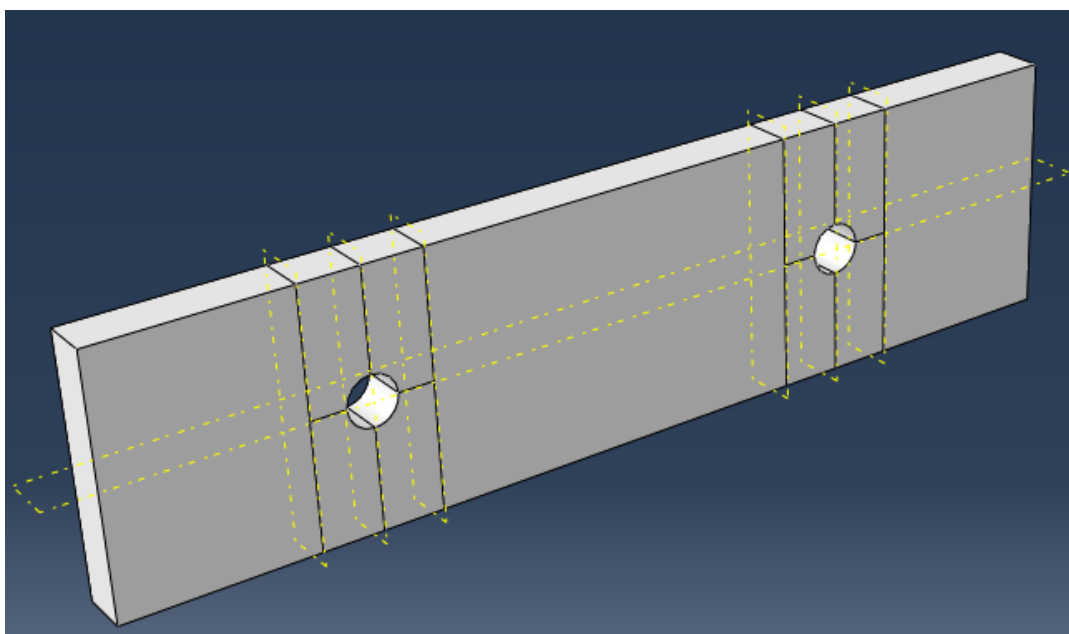


Figure L1–7. Plate with partitions.

To create the gasket:

The gasket is 0.125 in. thick and has the same rectangular profile as the plate.

1. In the model tree, click mouse button 3 on the **Plate** part and select **Copy**. Name the part **Gasket** and click **OK**.

- Expand **Gasket** in the model tree and then expand its **Features**. Double click on **Solid extrude-1** and change the depth to **0.125** in. Make sure that **Regenerate on OK** is toggled on. Click **OK**. The final part is displayed in Figure L1-8.

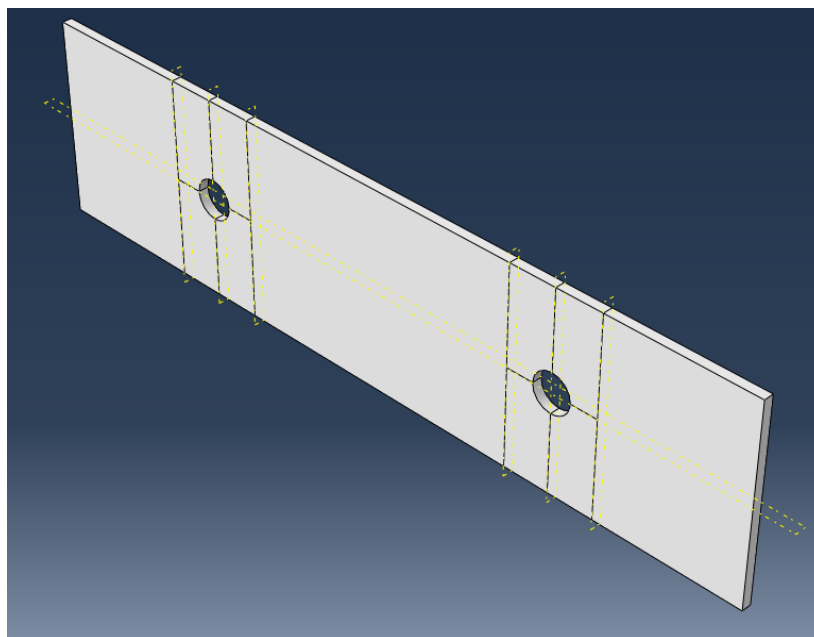



Figure L1–8. Gasket.

- To facilitate the definition of interaction properties later, you will define a surface that includes all the outer surfaces of the gasket. In the model tree expand **Parts**→**Gasket** and double click on **Surfaces**.
- In the **Create Surface** dialog box, name the surface **GasketAll**, and click **Continue**.
- In the viewport, click mouse button 1 and drag to create a rectangle that includes the entire gasket, then click **Done** in the prompt area.

To create the bolt:

- From the main menu bar, select **Part**→**Create** to create a new part. In the **Create Part** dialog box that appears, name the part **Bolt**, and specify an approximate size of **3**.

Accept the settings of a 3D, **Deformable** type part with a **Solid** shape, **Revolution** type base feature. Click **Continue**. The vertical green dashed line in the viewport grid is the line about which your sketch will be revolved.

- To sketch the bolt profile, select the **Create Lines: Connected**  drawing tool from the toolbox. Dimension the sketch as shown in Figure L1-9.
- Click mouse button 2 and then click **Done** in the prompt area.
- In the **Edit Revolution** dialog box, enter a value of **360** for the **Angle** and click **OK**. The part is displayed in Figure L1-10.

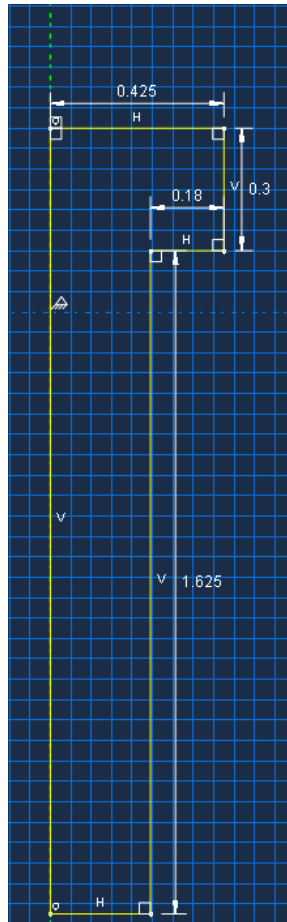


Figure L1–9. Bolt sketch.

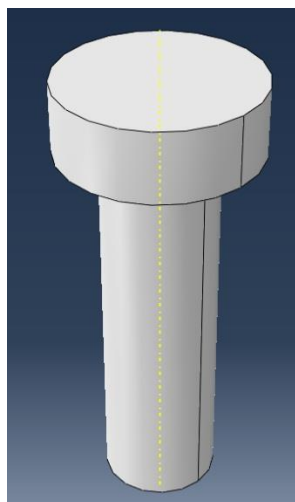








Figure L1–10. Bolt part after revolution.

5. Now you will sketch a profile that will be cut extruded into the bolt head to give it a hexagonal shape. From the toolbox, select the **Create Cut: Extrude**  tool.
6. Select the top surface of the bolt head as the plane for the extruded cut, and select the perimeter of that surface as the edge that will appear **vertical and on the right**.
7. Using the **Create Circle: Center and Perimeter**  tool and the Add Dimension  tool in the toolbox, sketch a circle of radius 0.6 in. centered at the center of the bolt head.
8. Using the **Create Construction: Oblique Lines Thru 2 Points**  tool in the toolbox, sketch three construction lines passing through the center of the bolt head. Make the first line horizontal, and use the **Add Dimension**  tool to orient the second line at angle of 60 degree to the first and the third line at an angle of 60 degree to the second. To do this, click the two lines one after the other and then enter the value of the desired angle after picking a location to place the dimension text in the viewport.
9. Now using the **Create Lines: Connected**  drawing tool from the toolbox, sketch a hexagon that passes through the six points at which the three construction lines intersect the perimeter of the bolt head. The final sketch is shown in Figure L1-11.

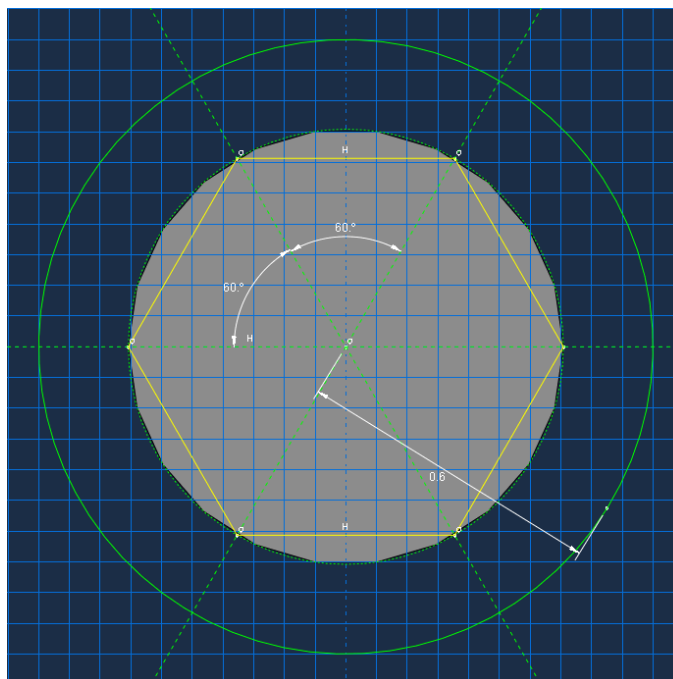






Figure L1–10. Sketch for the profile that will be cut extruded into the bolt head to give it a hexagonal shape.

10. Click mouse button 2 and then click **Done** in the prompt area.
11. In the **Edit Cut Extrusion** dialog box, select the **Through All** type, and click on  to reverse the **Extrude direction** if the arrow displayed in the viewport is pointing away from the bolt. Click **OK**.
12. To facilitate the application of bolt load later, a surface perpendicular to the axis of the bolt needs to be defined at distance of 1.25 in. from the tip of the bolt. You need to create a partition to do this. To create the partition, you need a datum plane. Select **Tools→Datum** from the main menu bar. Select **Plane** as the **Type**, and **Offset from plane** as the **Method**.
13. Select the plane perpendicular to the bolt axis at the bolt tip as the plane from which to offset, and select **Enter Value** in the prompt area. Make sure that the arrow displayed in the viewport points towards the bolt, click **OK** and enter a value of 1.25 in.
14. Now to create a partition, select **Tools→Partition** from the main menu bar. Select **Cell** as the **type** and **Use datum plane** as the **Method**.
15. Select the datum plane you just created and click **Create Partition** in the prompt area. Click **Done**.
16. In the model tree expand **Parts→Bolt** and double click on **Surfaces**.
17. In the **Create Surface** dialog box, name the surface **BoltLoad**, and click **Continue**.
18. From the selection toolbar at the top, click and hold , and then select the **Select From Interior Entities**  tool. Now select the six surfaces making up the entire partition you just created, and then click **Done** in the prompt area.
19. In the prompt area, click **Brown**.
20. More partitions need to be created to allow proper meshing later. Select **Tools→Partition** from the main menu bar. Select **Cell** as the **type** and **Extend face** as the **Method**. Select the cell with the bolt head as the cell to partition and click **Done**. Select the curved surface of the bolt as the face to be extended as the partition tool, and then click **Create Partition** in the prompt area.
21. Now select the entire bolt, and click **Done**. Select the **Select From Exterior Entities**  tool and select any surface on the bottom of the bolt head, then click **Create Partition**.
22. For partitioning the bolt head, select **Cell** as the **type** and **Define cutting plane** as the **Method**. Select the entire bolt as the cells to partition and click **Done**. Select **3 Points** as the way to specify the plane, and select the endpoints of any outer edge of the bolt head that is parallel to the bolt axis as the first and the second point, and any point on the similar diagonally opposite edge. Click **Create Partition** in the prompt area. Do this for all the six bolt head edges.

The final part is displayed in Figure L1-11.

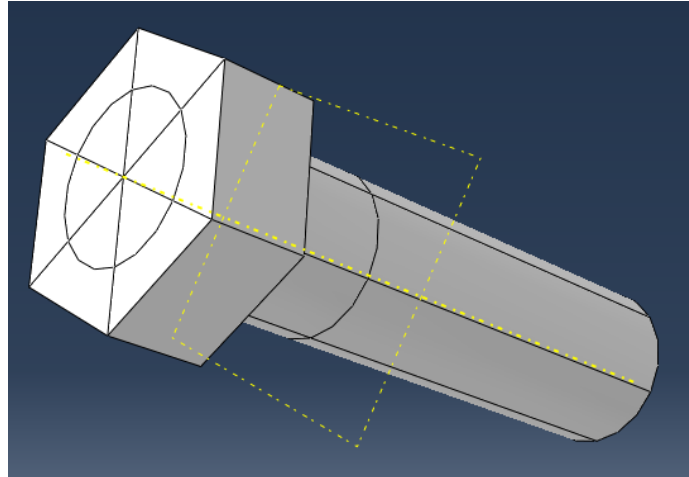






Figure L1–11. Final bolt part.

To create the nut:

1. From the main menu bar, select **Part→Create** to create a new part. In the **Create Part** dialog box that appears, name the part **Nut**, and specify an approximate size of **2**. Accept the settings of a **3D, Deformable** type part with a **Solid** shape, **Extrusion** type base feature. Click **Continue**.
2. Using the **Create Circle: Center and Perimeter**  tool and the **Add Dimension**  tool in the toolbox, sketch a circle with a radius of 0.25 in.
3. In the toolbox, click and hold the **Create Construction: Oblique Lines Thru 2 Points**  tool, and select the **Create Construction: Circle** tool  to sketch a circle with a radius of 0.42 in. centered at the center of the previous circle.
4. As done while creating the bolt part, sketch three construction lines passing through the center of the circles at an angle of 60 degree to each other, and then sketch a hexagon by joining the six points where these lines intersect the outer circle. The final sketch is shown in Figure L1-12.

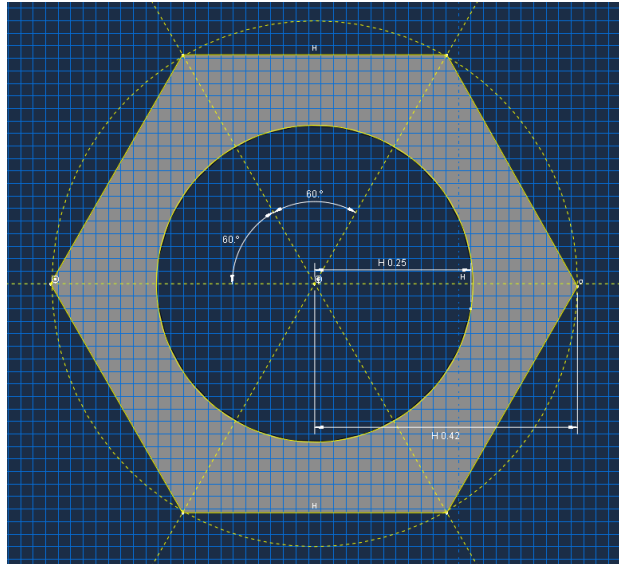


Figure L1-12. Nut profile before extrusion.

5. Click mouse button 2 and then click **Done** in the prompt area.
6. In the **Edit Base Extrusion** dialog box, enter a value of **0.325** for the **Depth**. Click **OK**.
7. Now you will create partitions in this part to allow for proper meshing later. From the main menu bar, select **Tools**→**Partition**. In the **Create Partition** dialog box, select **Cell** as the type, and **Define cutting plane** as the Method.
8. Click on **3 Points** in the prompt area, and select the end points of any outer edge of the bolt that is parallel to the bolt axis, and any one endpoint of the diagonally opposite edge as shown in Figure L1-13. Click **Create Partition** in the prompt area.

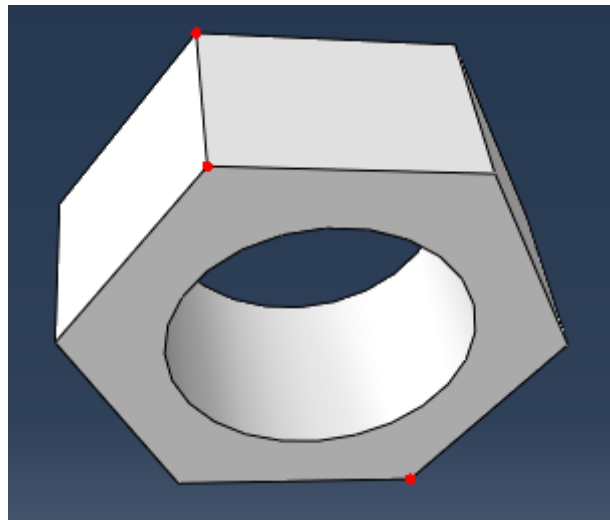


Figure L1-13. Selection of 3 points for partitioning the nut.

9. Repeat the above step to create partitions along all the outer edges that are parallel to the bolt axis. Note that once the nut has been partitioned, you will first have to select the cell through which the desired partition will pass. The final part is displayed in Figure L1-14.

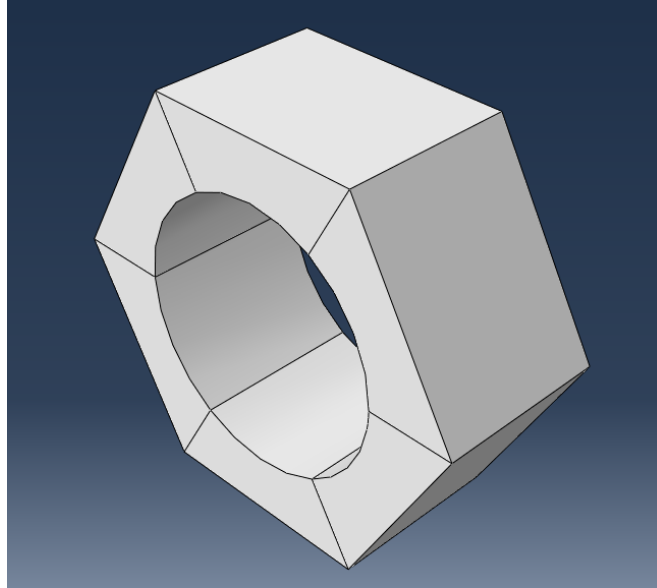


Figure L1–14. Nut part.

Creating a material definition

You will now define three materials – brass, rubber and steel that will be assigned to the metal plate, the gasket, and the nut and bolt respectively.

Note: Make sure that the model titled **Bolt tightening** is underlined in the model tree as you continue to work.

To define the brass material:

1. In the **Module** list located in the context bar, select **Property** to enter the Property module.
2. From the main menu bar, select **Material**→**Create** to create a new material.
3. In the **Edit Material** dialog box that appears, name the material **Brass**. Notice the various options available in this dialog box.

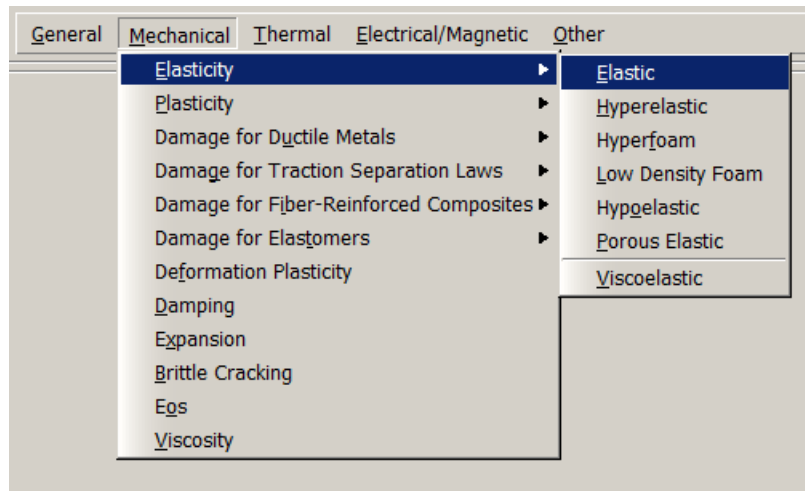


Figure L1-15. Material pull-down menu.

4. From the material editor's menu bar, select **Mechanical**→**Elasticity**→**Elastic**, as shown in Figure L1-15.
Abaqus/CAE displays the **Elastic** data form.
5. Enter a value of **14935000** for Young's modulus and a value of **0.3** for Poisson's ratio in the respective fields, as shown in Figure L1-16. Use **[Tab]** to move between cells, or use the mouse to select a cell for data entry.

Data		
	Young's Modulus	Poisson's Ratio
1	14935000	0.3

Figure L1-16. Material editor - Elastic data for brass.

6. Next, from the material editor's menu bar, select **Mechanical**→ **Plasticity**→ **Plastic**. Fill the **Plastic** data form as shown in Figure L1-17, then click **OK**.

Data		
	Yield Stress	Plastic Strain
1	28000	0
2	30000	0.1

Figure L1-17. Material editor - Plastic data for brass.

To define the steel material:

Define the steel material the same way as so defined brass above. Name the material **Steel** and use the values displayed in Figure L1–18 to fill the elastic and plastic data forms.

Data		
	Young's Modulus	Poisson's Ratio
1	30000000	0.3

Data		
	Yield Stress	Plastic Strain
1	52000	0
2	65000	0.1

Figure L1–18. Material editor – Elastic and Plastic data for steel.

To define the rubber material:

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**. Notice the various options available in this dialog box.
3. From the material editor's menu bar, select **Mechanical** →**Elasticity** →**Hyperelastic**.
4. Select **Mooney-Rivlin** as the Strain energy potential, and toggle **Coefficients** as the input source.

Abaqus/CAE displays the **Hyperelastic** data form. Use the values displayed in Figure L1–19 to fill this form, and then click **OK**.

Data			
	C10	C01	D1
1	556	139	2.8E-005

Figure L1–19. Material editor – Hyperelastic data for rubber.

Defining and assigning section properties

Next, you will define and assign sections to all the parts. The plate section will refer to the brass material, the gasket section will refer to the rubber material, and the Nut-Bolt section will refer to the steel material.

To define the plate section:

1. From the main menu bar, select **Section**→**Create**.
2. In the **Create Section** dialog box that appears:
 1. Name the section **PlateSection**.
 2. Accept the category **Solid** and the type **Homogeneous**.
 3. Click **Continue**.
3. In the **Edit Section** dialog box that appears:

4. Select **Brass** as the **Material**.
5. Click **OK**.

To assign the plate section to the plate:

1. Double click on **Plate** under **Parts** in the model tree.
2. From the main menu bar, select **Assign**→**Section**.
Abaqus/CAE displays prompts in the prompt area to guide you through the procedure.
3. Select the entire plate in the viewport as the region to be assigned the plate section. Click mouse button 1 and drag to create a rectangle that encloses the entire plate to do this.
4. Click mouse button 2 in the viewport or click **Done** in the prompt area to accept the selected geometry.
The section assignment editor appears.
5. In the **Edit Section Assignment** dialog box, select **PlateSection** as the **Section**, and click **OK**.

Abaqus/CAE colors the part green to indicate that the section has been assigned.


Defining and assigning the gasket and nut-bolt sections:

Following the same procedure as used above for the plate section, define the gasket and nut-bolt sections. Name them **GasketSection** and **Nut-boltSection**, and select **Rubber** and **Steel** respectively as the associated material. Then assign **GasketSection** to **Gasket** part, and **Nut-boltSection** to **Nut** and **Bolt** part.

Assembling the model

The assembly for this analysis consists of a single instance of the part **Gasket**, and double instances of the parts **Plate**, **Nut** and **Bolt**.

To assemble the model:

1. In the **Module** list located in the context bar, select **Assembly** to enter the Assembly module.
2. From the main menu bar, select **Instance**→**Create**.
3. In the **Create Instance** dialog box, select **Gasket** as the **Part**. Toggle on **Auto-offset from other instances** (Not doing so might lead the instances that you will create later to overlap) and click **OK**. Abaqus/CAE displays the new part instance in the viewport.
4. Repeat step 2 and 3, except chose **Plate** as the **Part** this time.
5. You will now translate the **Gasket-1** instance so that it lies right next to the **Plate-1** instance. Use the **Translate Instance**  tool in the toolbox and select **Gasket-1** in the viewport as the instance to be translated. Click **Done**.

6. Select a corner point of **Gasket-1** as the start point of the translation vector. Select a corner point of **Plate-1** instance as the end point for the translation vector. Choose the points so that the two instances align with each. Click **OK**.
7. Now create the second instance of the plate (repeat step 4) and translate it to lie on the other side of the gasket. Follow the same procedure as in step 5 and 6. Chose the plate as the instance to be translated this time. The assembly is shown in Figure L1–20.

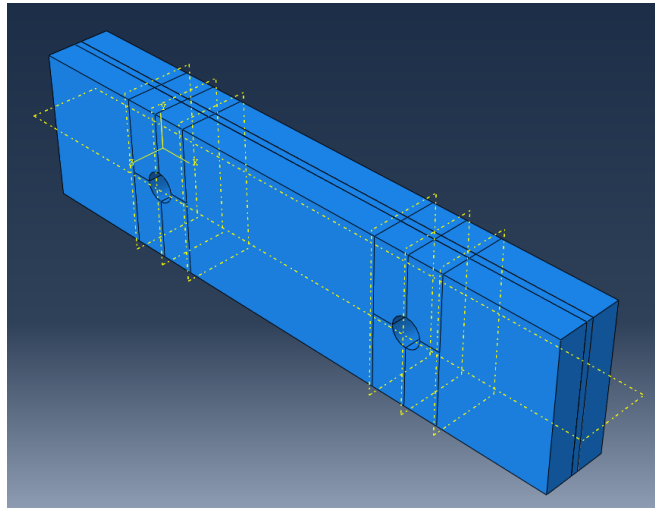







Figure L1–20. Arrangement of the plates and the gasket.

Next, you will create the bolt instances and place them in position.

8. Create the first instance of the bolt. To align the bolt with the bolt hole, use the **Create Constraint: Coaxial**  tool (click and hold ) in the toolbox.
 9. Select the cylindrical face of the bolt as the movable instance, and the cylindrical face of any bolt hole as the fixed instance. Make sure that the arrows displayed in the viewport are such that the bolt aligns in the desired direction after the constraint is applied. Click **OK**.
 10. To make the bolt head flush with the plate surface, use the **Create Constraint: Face to Face**  tool in the toolbox. Select the base of the bolt head as the planar face of the movable instance, and the plate surface as the planar face of the fixed instance. Make sure that the two arrows displayed in the viewport align in the same direction, and enter a value of **0.0** for the distance from the fixed plane along its normal. Press **Enter**. Repeat the same process to place the second instance of the bolt into the other bolt hole.
- Next, you will create the nut instances and place them in position.
11. Create the first instance of the nut. To align the nut with the bolt, use the **Create Constraint: Coaxial**  tool in the toolbox.

12. Select the cylindrical face of the nut as the movable instance, and the cylindrical face of any bolt as the fixed instance. Click **OK**.
13. To make the nut flux with the plate surface, use the **Create Constraint: Face to**

Face  tool in the toolbox. Select the base of the nut as the planar face of the movable instance (right click on plate and gasket instances in the model tree and select hide/show to allow this selection), and the plate surface as the planar face of the fixed instance. Make sure that the two arrows displayed in the viewport align in the same direction, and enter a value of **0.0** for the distance from the fixed plane along its normal. Press **Enter**. Repeat the same process to place the second instance of the nut around the other bolt. The final assembly is displayed in Figure L1–21.

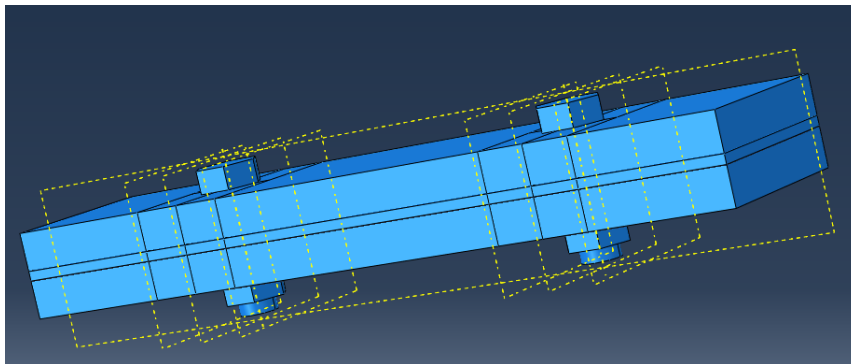



Figure L1–21. Final assembly.

14. To facilitate the application of boundary conditions later, define a set bottom surface of the bottom plate. In the model tree expand **Assembly** and double click on **Sets**.
15. Name the set **Bottom**, and click **Continue**. Select the **Select From Exterior** **Entities**  tool and select the entire bottom surface (the plate surface in contact with the nut). Click Done.

Configuring the analysis

In this simulation we want to study the response of the assembly to the application of bolt loads. Consequently, this model will consist of two steps:

- An initial step, in which you will apply the interactions and boundary conditions.
- A general, static step, in which you will apply the bolt load.

Abaqus/CAE generates the initial step automatically, but you must create the analysis step yourself.

To create a general, static step:

1. In the **Module** list located in the context bar, select **Step** to enter the Step module.
2. From the main menu bar, select **Step**→**Create** to create a step.
3. In the **Create Step** dialog box that appears:



- a. Name the step **BoltLoad**.
 - b. From the list of available **General** procedures in the **Create Step** dialog box, select **Static, General**.
 - c. Click **Continue**.
4. In the **Edit Step** dialog box that appears:
 - a. In the **Description** field of the **Basic** tabbed page, enter **Apply bolt loads**, and toggle **Nlgeom On**.
 - b. Click on the **Incrementation** tab, and change the **Initial Increment size** to **0.25**. Click **OK** to create the step and to exit the step editor.

Defining the interaction properties

Next, you will define the interaction properties. The friction coefficient for interaction between the gasket and the plate will be 0.4, rest all interactions will have a friction coefficient of 0.1.

To define the interaction property:

1. In the **Module** list located in the context bar, select **Interaction** to enter the Interaction module.
2. From the main menu bar, select **Interaction→Property→Create**.
3. In the **Create Interaction Property** dialog box that appears:
 - a. Name the interaction property **Friction1**.
 - b. Select **Contact** as the **Type**, and click **OK**.
4. In the **Edit Contact Property** dialog box that appears:
 - a. Select **Mechanical→Tangential Behavior**.
 - b. Select **Penalty** as the **Friction formulation**
 - c. Enter a **Friction Coeff** value of **0.1**.
 - d. Select **Mechanical→Normal Behavior**.
 - e. Select **Penalty (Standard)** as the **Constraint enforcement method**.
 - f. Click **OK**.

Repeat steps 2, 3 and 4 to create another interaction property named **Friction2** with a Friction Coefficient of 0.4.
5. From the main menu bar, select **Interaction→Create**.
6. Name the interaction **Contact**, select **Initial** as the **Step**, and select **General contact (Standard)** as the **Types for Selected Step**. Click **Continue**.
7. In the **Edit Interaction** dialog box, select **Friction1** as the **Global property assignment**.
8. Click on the **Edit** tool  next to **Individual property assignments**.
9. In the **Edit Individual Contact Property Assignments** dialog box, select **(Global)** as the **First Surface**, **Gasket-1.GasketAll** as the **Second Surface**, and **Friction2** as the **Property Assigned**. Click , and then click **OK**.
10. Click **OK** in the **Edit Interaction** dialog box.

Applying the constraints

Now you will apply the constraints that hold the bolt and the nut together.

To apply the constraints:

1. From the main menu bar, select **Constraint**→**Find contact pairs**.
2. In the **Find Contact Pairs** dialog box, enter a value of 0.1 for **Include pairs within separation tolerance**, then click on **Find Contact Pairs**.
3. In the list of contact pairs that populates, delete all the pairs except **Bolt-1-Nut-1** and **Bolt-2-Nut-2**.
4. In the column titled **Type**, change the contents of the two cells from **Interaction** to **Tie**.
5. Click **OK**.

Applying the boundary condition to the model

Next, you will define the boundary condition that involves fixing the bottom surface of the assembly (the surface in contact with the nut).

To apply boundary conditions to the bottom surface of the assembly:

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 **FixBase**.
 - 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 **Symmetry/ Antisymmetry/ Encastre** as the type.
 - e. Click **Continue**.
4. Click on **Sets** in the prompt area, and in the **Region Selection** dialog box, select **Bottom**. Click **Continue**.
5. In the **Edit Boundary Condition** dialog box, toggle on **ENCASTRE**, then click **OK**.

Applying the bolt loads

In Abaqus, assembly bolt loads are applied across user-defined pre-tension sections in the bolt. When modeling the bolt with solid elements, the pre-tension section is defined as a surface in the bolt shank that effectively partitions the bolt into two regions. To apply the bolt load to the first bolt:

1. From the main menu bar, select **Load**→**Create**.
2. In the **Create Load** dialog box that appears:
 - a. Name the load **BoltLoad1**.
 - b. Select **BoltLoad** as the **Step** in which this load will be activated.

- c. In the **Category** list, accept the default category selection **Mechanical**.
- d. In the **Types for Selected Step** list, select **Bolt load** as the type.
- e. Click **Continue**.
3. Click on **Surfaces** in the prompt area.
4. In the **Region Selection** dialog box, select **Bolt-1.BoltLoad** as the eligible surface, then click **Continue**.
5. In the viewport, select the datum axis that runs along the length of the Bolt-1 as the **datum axis that is aligned with the bolt centerline**.
6. In the Edit Load dialog box, enter a magnitude of **270**.

To apply the bolt load to the second bolt:

Repeat the same procedure as for the first bolt. Name the load **BoltLoad2**, select **Bolt-2.BoltLoad** as the eligible surface in the **Region Selection** dialog box, and select the datum axis that runs along the length of the Bolt-2.

Meshing the model

You will now mesh all the parts. In the **Module** list located in the context bar, select **Mesh** to enter the Mesh module.

To assign a global part seed and create the mesh for the bolt:

1. In the context bar, toggle on **Part** as the **Object**, and select **Bolt** as the part from the list.
2. From the main menu bar, select **Seed**→**Part** to seed the part.
The **Global Seeds** dialog box appears. The default global element size is based on the size of the part.
3. In the **Global Seeds** dialog box, enter an approximate global size of **0.1** and click **Apply**.
Abaqus/CAE applies the seeds to the part, as shown in Figure L1–22.

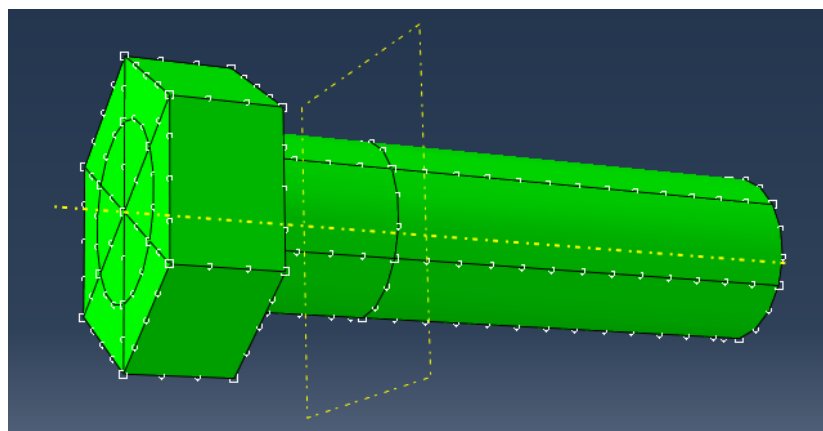


Figure L1–22. Seeded part instance.

4. Click **OK**.

5. From the main menu bar, select **Mesh→Part** to mesh the part.
 6. Click **Yes** in the prompt area or click mouse button 2 in the viewport to confirm that you want to mesh the part instance.
- Abaqus/CAE meshes the part instance and displays the resulting mesh, as shown in Figure L1–23.

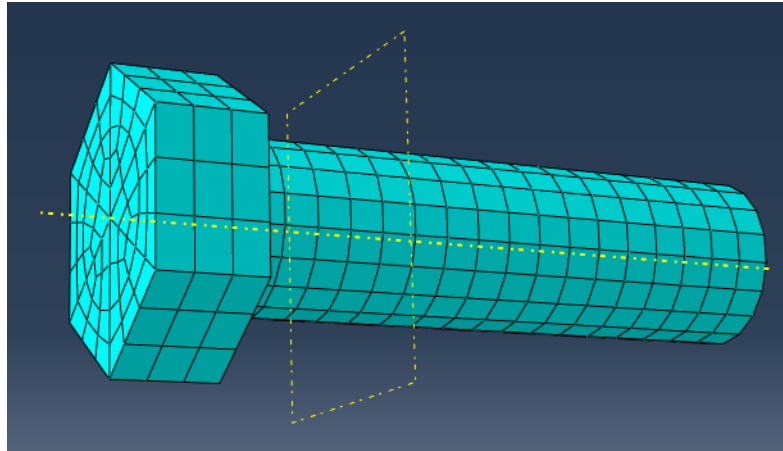


Figure L1–23. Bolt part mesh.

Follow the same procedure to mesh the nut and plate part instances with global seed sizes of 0.05 and 0.15 respectively. **C3D8R** elements are used by default for the bolt, nut and plates.

The gasket too will be meshed following the same procedure with a global seed size of 0.15, except that you will use **C3D8H** elements for it. To change the default element type, follow the steps below before you mesh the gasket.

1. Select **Mesh→Element Type** from the main menu bar.
2. Select the entire gasket in the viewport as the region to be assigned the element type and click done in the prompt area.
3. In the **Element Type** dialog box that appears, under the **Hex** tab, toggle on **Hybrid formulation** and toggle off **Reduced integration**.

Creating and submitting the analysis job

The definition of the model **Bolt tightening** 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 **BoltTightening** and select the model **Bolt tightening**. Click **Continue**.

The job editor appears.

5. In the **Description** field of the **Edit Job** dialog box, enter **Tightening of bolts**.
6. 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.
7. From the buttons on the right side of the **Job Manager**, click **Submit** to submit your job for analysis. The status field will show **Running**.
When the job completes successfully (this might take a while), the status field will change to **Completed**. You are now ready to view the results of the analysis in the Visualization module.

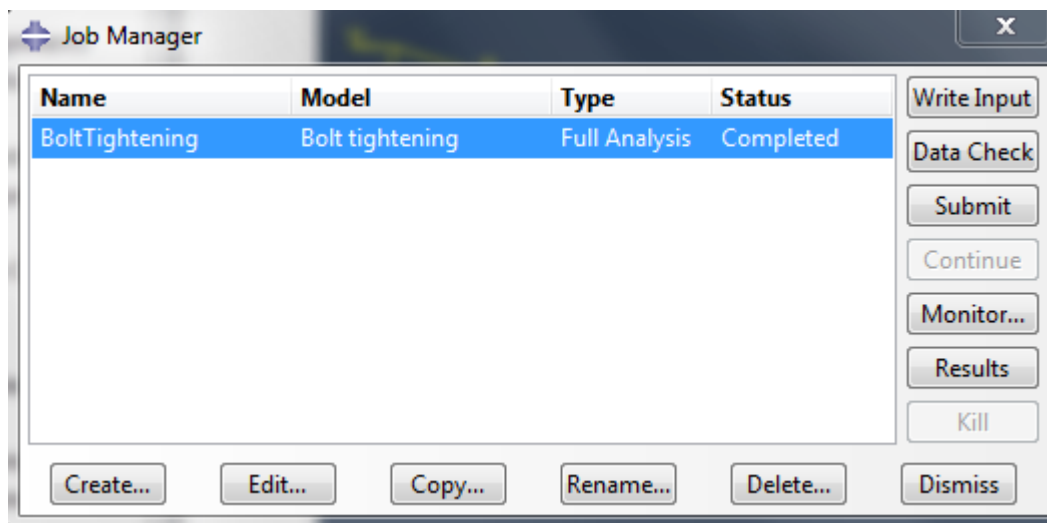



Figure L1–24. Job status in the **Job Manager**.

Viewing the analysis results

You are now ready to view the results of the analysis in the Visualization module. Click **Results** in the **Job Manager** to enter the Visualization module. Abaqus/CAE switches to the Visualization module, opens the output database created by the job (**BoltTightening.odb**), and displays the undeformed shape of the model.

You will now plot the contact stress for the gasket.

1. From the **Display Group** toolbar at the top, select the **Replace Selected**  tool, and select **Part instances** as the entity to replace in the prompt area. Select the gasket in the viewport and click **Done**.
2. Plot contact pressures on the gasket (use the **Field Output** toolbar to change the selected output variable to **CPRESS**). The plot appears as shown in Figure L1–25. Firm contact is established between the gasket and the plates, resulting in a tight seal.

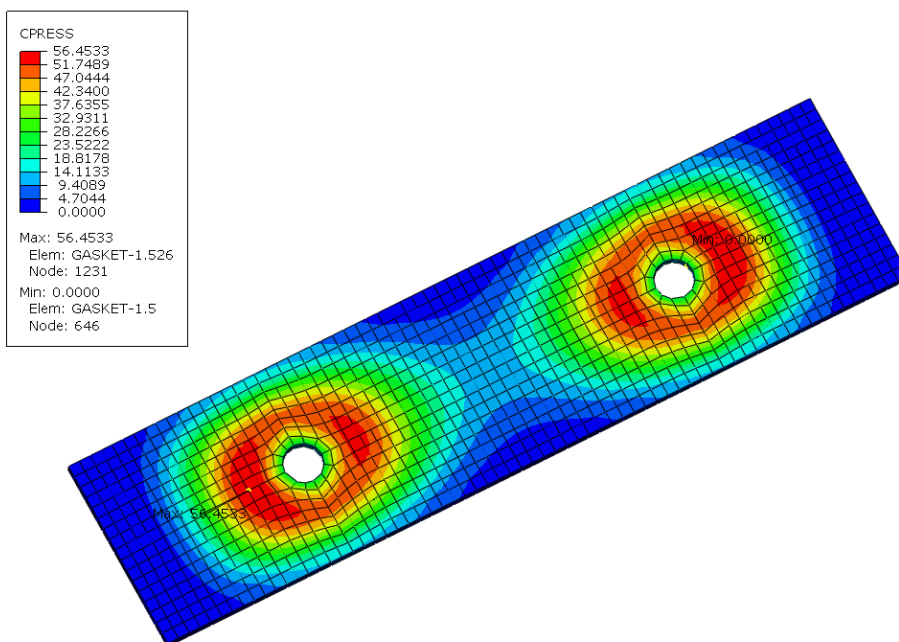



Figure L1-25. Gasket contact pressure distribution.

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