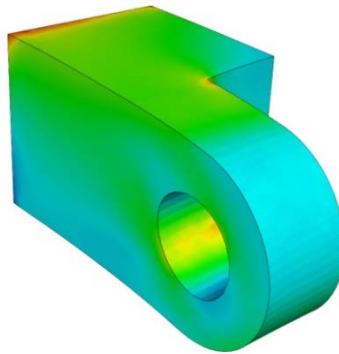


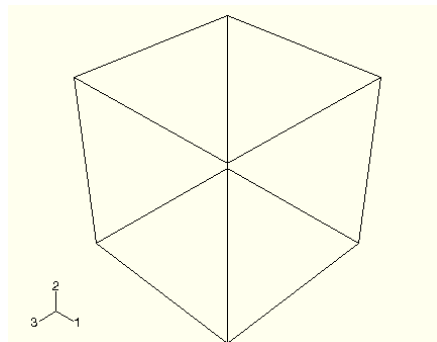
# ABAQUS - Tutorial

## Part module




### 1 Creating the cube

To create the cube (the base feature), you create a solid, three-dimensional, extruded part and name it. You then sketch its profile ( $0.04 \text{ m} \times 0.04 \text{ m}$ ) and extrude the profile over a specified distance ( $0.04 \text{ m}$ ) to produce the base feature of the first half of the hinge.

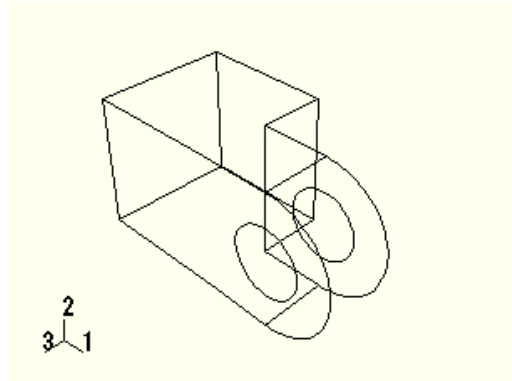


#### To create the cube:

1. Start Abaqus/CAE, and create a new model database.
2. In the Model Tree, double-click the **Parts** container to create a new part.
3. Name the part `Hinge-hole`. Accept the following default settings:
  - A three-dimensional, deformable body
  - A solid extrusion base feature
4. In the **Approximate size** text field, type `0.2`. You will be modeling the hinge using meters for the unit of length, and its overall length is `0.14` meters; therefore, `0.2` meters is a sufficiently large approximate size for the part. Click **Continue** to create the part.
5. From the Sketcher toolbox, select the rectangle tool .
6. Sketch an arbitrary rectangle, and click mouse button 2 in the viewport to exit the rectangle tool.
7. Dimension the top and left edges so that each is `0.04` m long.
8. Click mouse button 2 to exit the Sketcher.
9. In the dialog box, type a **Depth** of `0.04` and press **[Enter]**.

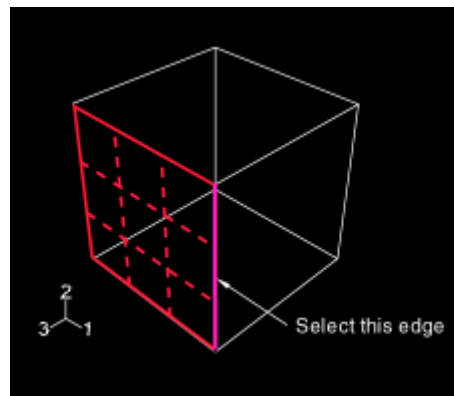
## 2. Adding the flange to the base feature

You will now add a solid feature—the flange—to the base feature. You select one face of the cube to define the sketching plane and extrude the sketched profile through half the depth of the cube. The cube and flange are shown below.

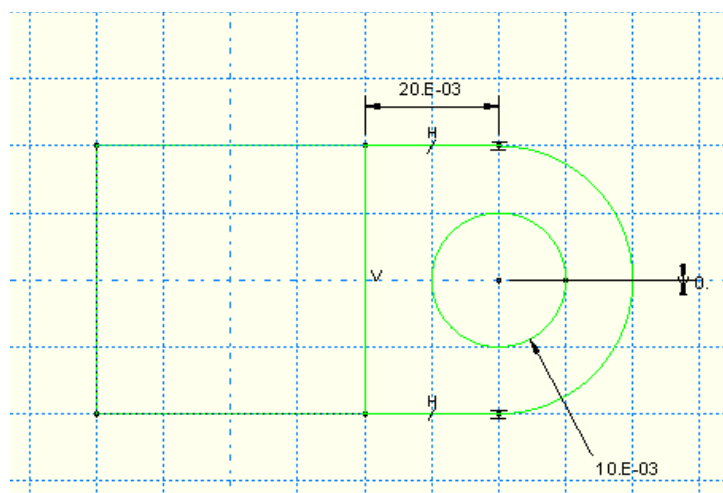



**To add the flange to the base feature:**

1. From the main menu bar, select **Shape** → **Solid** → **Extrude**.
2. Select a face to define the sketching plane. Select the face at the front of the cube below.




3. Select an edge that will appear vertical and on the right side of the sketch, as shown below.
4. Zoom out to view the area where you will sketch the flange:



5. As before, the approximate shape of the new feature will be sketched first. From the Sketcher toolbox, select the connected lines tool .
6. Sketch the rectangular portion of the flange by drawing three lines as follows:
  - a. Starting at any point to the right of the cube, connect the line to the top-right corner of the cube.
  - b. Continue the next line to the bottom-right corner of the cube. This line is automatically assigned a vertical constraint.
  - c. The final line extends from the bottom-right corner of the cube to any point to the right of the cube.


Click mouse button 2 in the viewport to exit the connected lines tool.


Refine the sketch by defining the following constraints and dimension:

Close the profile by adding a semicircular arc using the 3-points circle tool .

- a. Select the two vertices at the open end of the rectangle as the endpoints of the arc, starting with the top one. Select any point to the right of the sketch as a point that lies on the arc.
- b. Define tangent constraints between the ends of the arc and the horizontal lines to refine the sketch.

Click mouse button 2 in the viewport to exit the 3-points circle tool.

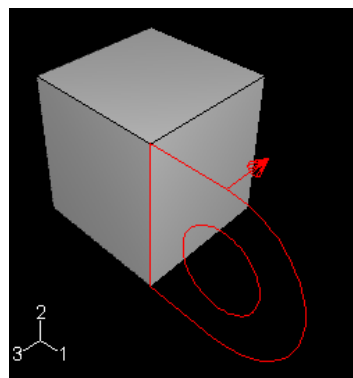
From the Sketcher toolbox, select the center-perimeter circle tool  to sketch the flange hole.

- a. The center of the circle should coincide with the center of the arc created previously. The perimeter point should be placed to the right of the center point.
- b. Use the dimension tool  to change the value of the radius to 0.01 m.
- c. Dimension the vertical distance between the center of the circle and the perimeter point. Edit this dimension so that the distance is 0. (If the distance is already 0, you cannot add a vertical dimension.) This will adjust the location of the perimeter point so that it is on the same horizontal plane as the center point.

Click mouse button 2 to exit the Sketcher.

In the **Edit Extrusion** dialog box:

- a. Accept the default **Type** selection of **Blind** to indicate that you will provide the depth of the extrusion.
- b. In the **Depth** field, type an extrusion depth of 0.02.
- c. Click **Flip** to reverse the extrusion direction, as shown below.



## Property Module

1. Create Material, Elastic, Young's modulus = 210GPa, Poisson's ratio = 0.3.
2. Create Section, Homogeneous, Solid Section.
3. Assign Section, Assign section to solid.

## Assembly Module

1. Instance Part, Choose Independent (Mesh on Instance) => OK

## Step Module

1. Create Step, Static General => OK

## Load Module

1. Create Load, Surface traction.
2. Select :
  - a. Traction: General
  - b. Vector: Edit ... (Give a vector pointing in the negative y-direction)
  - c. Magnitude: 1 MPa
3. Point to the surface in the hole => OK.
4. Create Boundary Condition, Symmetry/Antisymmetry/Encastre,
  - a. Select back plane
  - b. Choose Pinned ( $U_1=U_2=U_3=0$ ) => OK

## Mesh Module

1. Assign Mesh Control, Choose:
  - a. Element Shape => Tet
  - b. Technique => Free
2. Seed Part Instance, Approximate Global Size = 0.0045
3. Mesh Part Instance => Yes

## Job Module

1. Create Job => Continue => OK
2. Job Manager => Submit => OK
3. When the analysis has completed, In the Job Manager => Results

## Visualization Module

1. Plot Contours on deformed Shape.
2. In the Pull Down Menus choose: Result => Field Output => Choose a variable to visualize
3. Animate Scale Factor