ENGN 1750: Advanced Mechanics of Solids
Three-dimensional problems in Abaqus

1 Objectives

In the previous exercises, we investigated plane problems in Abaqus, which took advantage of simplicities in the problem geometry and loading to reduce the analysis to two dimensions. These types of simplifications are incredibly useful and should be exploited whenever possible. However, many problems you will encounter are inherently three-dimensional and must be analyzed accordingly. One challenge of three-dimensional modeling is accounting for the geometry. In the Abaqus part module, there are three basic tools for creating three-dimensional features or cuts: (i) extrude, (ii) revolve, and (iii) sweep, which we will explore in these exercises. The part generating tools built into Abaqus are not extensive, but the program does allow for complex geometries, generated in a program like SolidWorks, to be imported in the Part module. A second challenge of three-dimensional modeling is creating a mesh, which can be more difficult in 3D than in planar problems, requiring partitioning the part into smaller, simpler regions. We will consider meshes of both hexahedral (brick) elements and tetrahedral elements. In these exercises, we will consider

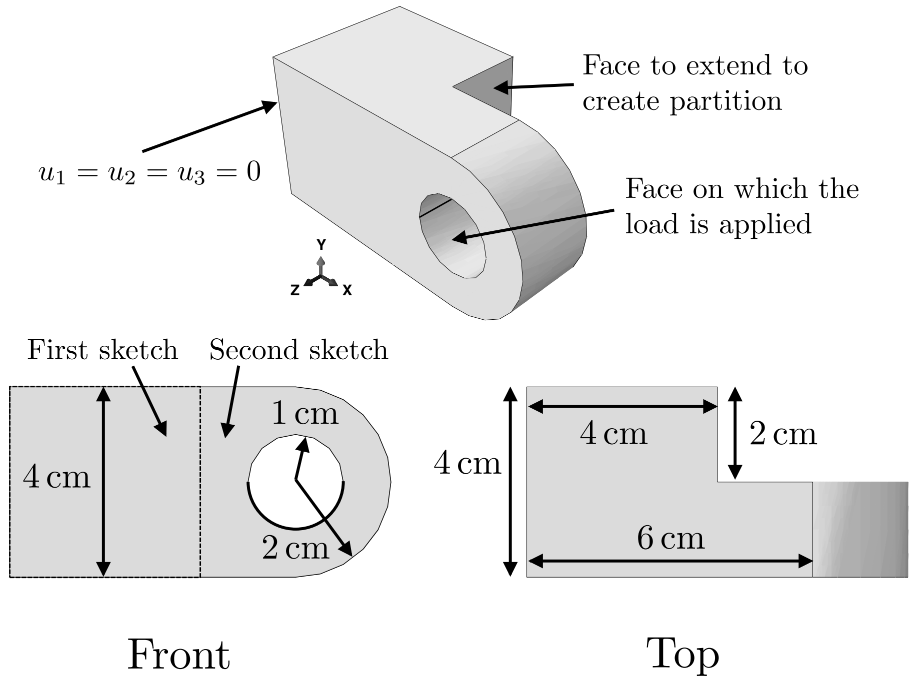
1. A bracket under a distributed load and
2. The stress concentration arising around a transverse circular hole in a round bar under tension.

2 Exercises

2.1 Bracket under a distributed load

We first analyze a bracket-like structure subjected to a load. This exercise will familiarize ourselves with using the extrude tool, partitioning geometries, and creating a mesh with hexahedral (brick) elements. The bracket is made of steel, and the geometry is shown on the next page. Below is an outline of the steps for performing the analysis in Abaqus/CAE:

- Part:
    - Part ⇒ Create
    - Select 3D, Deformable, Solid, Type: Extrusion, and Approximate size: 0.1 m.
    - We will first create the block part of the bracket. Sketch a 4 cm by 4 cm square, and click Done. Input an extrusion depth of 4 cm, and click OK.
    - Next, to create the bracket part of the geometry, click the "Create Solid: Extrude" button on the left side menu. Select the front face in the xy-plane and select the edge on the right side of that face.



- Sketch the bracket part of the geometry relative to the existing block and click Done. Note that the sketch geometry will need to be closed. Input an extrusion depth of 2 cm, and click OK. Make sure that the extrude direction is in the correct direction. If it is not, click the flip button.
- For future use in applying boundary conditions and meshing, we will create several partitions. First, to help us mesh the part, we will partition the solid into the block and bracket sections. Find and click the “Partition Cell: Extend Face” button on the left side menu. Select the 2 cm by 4 cm stepped face, noted in the above figure, and click “Create Partition.”
- Next, we wish to partition the face on which the load will be applied into top and bottom sections, requiring us to create two face partitions, dividing the hole face into top and bottom surfaces. Find and click the “Partition Face: Use Shortest Path Between 2 Points” button, select the hole face, and click Done. Select two corresponding points on the front and back on one side of the hole to create a horizontal partition and click “Create Partition.” Repeat the process to create another partition on the opposite side of the hole face.

- Property:
  - Material ⇒ Create
  - Mechanical ⇒ Elasticity ⇒ Elastic
  - Enter the material properties for steel and click OK
  - Section ⇒ Create

- Solid ⇒ Homogeneous ⇒ Continue
- Make sure your material is selected, and click OK
- Assign ⇒ Section
- Select the entire part and click Done/OK.

- Assembly:
  - Instance ⇒ Create ⇒ OK

- Step:
  - Step ⇒ Create ⇒ Static/General ⇒ Continue ⇒ OK

- Load:
  - BC ⇒ Create ⇒ Mechanical ⇒ Displacement/Rotation ⇒ Continue ⇒ Select left face and click Done ⇒ Enter U1=U2=U3=0 and click OK
  - Here, we will apply a downward distributed load on the bottom half of the hole face. We will use the surface-traction-type of load, rather than pressure-type. Load ⇒ Create ⇒ Mechanical ⇒ Surface Traction ⇒ Continue ⇒ Select the bottom half of the hole face and click Done ⇒ Change traction type from shear to general ⇒ Click the arrow beside "Vector" ⇒ Input the first and second points for the direction vector as $(0, 0, 0)$ and $(0, -1, 0)$, respectively ⇒ Input the magnitude as 10 MPa ⇒ Click OK.

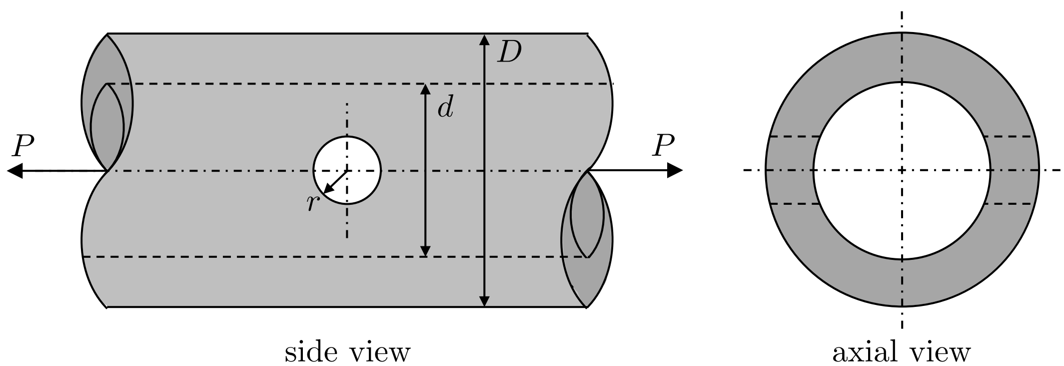
- Mesh:
  - Make sure Object is set to Part.
  - Mesh ⇒ Element Type ⇒ Select the entire part and click Done ⇒ Family: 3D Stress ⇒ Click OK
  - Mesh ⇒ Controls ⇒ Select the block section and click Done ⇒ Element Shape: Hex ⇒ Technique: Structured ⇒ Click OK
  - Mesh ⇒ Controls ⇒ Select the bracket section and click Done ⇒ Element Shape: Hex ⇒ Technique: Sweep ⇒ Algorithm: Advancing Front ⇒ Click OK
  - Seed ⇒ Part ⇒ Approximate global size: 2.5 mm ⇒ Click OK
  - Mesh ⇒ Part ⇒ Yes.

- Job:
  - Job ⇒ Create ⇒ Continue/OK
  - Job ⇒ Submit ⇒ Job-1
  - When the job successfully completes: Job ⇒ Results ⇒ Job-1

- Visualization:
  - Examine contour plots of displacement, stress, and strain.
  - Probe quantitative results at specific points of the model:
    Tools ⇒ Query ⇒ Probe values
  - Where does maximum Mises stress occur?
  - To view calculation results in the interior: Tools ⇒ View Cut ⇒ Manager ⇒ Select one of the three planes and drag the position bar to move the cut plane.

2.2 Stress concentration for a transverse circular hole in a round bar under tension

In previous exercises, we considered stress concentrations in planar problems. Stress concentrations, of course, also occur in three-dimensional problems. As an example, consider a round tubular bar with outer diameter D and inner diameter d. The bar contains a transverse circular hole of radius r, as shown below.

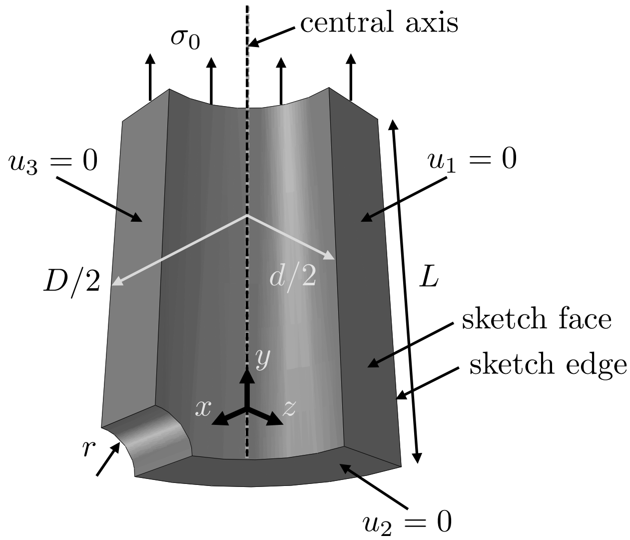


Defining the nominal stress in this geometry as $\sigma_0 = 4P/[\pi(D^2 - d^2)]$, the stress is raised in the region of the transverse hole, quantified by the stress concentration factor

$$K_{\mathrm{t}} = \frac{\sigma_{1,\max}}{\sigma_0},$$

where $\sigma_{1,\max}$ is the maximum value of σ_1 in the region of the transverse hole. We will determine K_{t} for the following geometry: $D = 10\,\mathrm{cm}$, $d = 6\,\mathrm{cm}$, and $r = 1\,\mathrm{cm}$. As in prior stress concentration problems, since the problem is linear and stress-controlled, the numerical values for P, E, and ν are arbitrary in determining K_{t}. For concreteness, we take $\sigma_0 = 100\,\mathrm{MPa}$ and material properties for steel.

We exploit the symmetries of the problem and only model one eighth of the bar, as shown on the next page. The three resulting symmetry boundary conditions are denoted on the schematic. The bar must be taken to be sufficiently long so as not to affect the determination of the stress concentration factor. To this end, we take $L = 10\,\mathrm{cm}$. As before, we apply the axial load through a constant negative pressure ($= -\sigma_0$) on the top surface.



Below is an outline of the steps for performing the analysis in Abaqus/CAE:

- Part:
  - Part ⇒ Create
  - Select 3D, Deformable, Solid, Type: Revolution, and Approximate size: 0.5 m.
  - We will first create the tubular bar without the transverse circular hole. Sketch a rectangle with opposing corners at $(3, 0)$ cm and $(5, 10)$ cm, and click Done. Input a revolution angle of 90°, and click OK.
  - Next, to create the transverse hole, click the "Create Cut: Extrude" button on the left side menu. Select the face parallel to the yz-plane (noted in the figure above) and select the edge on the right side of that face (also noted in the figure).
  - Sketch the quarter hole relative to the existing geometry (at the bottom left) and click Done. Note that the sketch geometry will need to be closed. Set the extrusion type to "Through All", and click OK. Make sure that the cut direction is in the correct direction. If it is not, click the "Flip Direction" button.
- Property:
  - Material ⇒ Create
  - Mechanical ⇒ Elasticity ⇒ Elastic
  - Enter the material properties for steel and click OK
  - Section ⇒ Create
  - Solid ⇒ Homogeneous ⇒ Continue
  - Make sure your material is selected, and click OK
  - Assign ⇒ Section
  - Select the entire part and click Done/OK.

- Assembly:
    - Instance ⇒ Create ⇒ OK
- Step:
    - Step ⇒ Create ⇒ Static/General ⇒ Continue ⇒ OK
- Load:
    - Here, we will specify the three symmetry boundary conditions, which are noted in the above figure. BC ⇒ Create ⇒ Mechanical ⇒ Displacement/Rotation ⇒ Continue ⇒ Select one of the faces and click Done ⇒ Set the appropriate displacement component to zero and click OK
    - Load ⇒ Create ⇒ Mechanical ⇒ Pressure ⇒ Continue ⇒ Select the top face and click Done ⇒ Enter Magnitude ($= -\sigma_0$) and click OK
- Mesh:
    - Make sure Object is set to Part.
    - Mesh ⇒ Element Type ⇒ Select the entire part and click Done ⇒ Family: 3D Stress ⇒ Click OK
    - Mesh ⇒ Controls ⇒ Element Shape: Tet ⇒ Technique: Free ⇒ Click OK
    - Seed ⇒ Part ⇒ Approximate global size: 5 mm ⇒ Click OK
    - We wish to refine the mesh in the region of the transverse circular hole.
      Seed ⇒ Edges ⇒ Select the four edges which border the face of the circular hole and click Done ⇒ Approximate element size: 2 mm ⇒ Click OK
    - Seed ⇒ Edges ⇒ Check the "Use single-bias picking," select the four edges which end on the face of the circular hole and click Done ⇒ Minimum size: 2 mm and Maximum size: 5 mm ⇒ Apply (Make sure the fine seeds are on the ends of the edges near the hole. If not, flip bias on the appropriate edges) ⇒ OK
    - Mesh ⇒ Part ⇒ Yes.
- Job:
    - Job ⇒ Create ⇒ Continue/OK
    - Job ⇒ Submit ⇒ Job-2
    - When the job successfully completes: Job ⇒ Results ⇒ Job-2
- Visualization:
    - Examine contour plots of displacement, stress, and strain, especially in the region of the stress concentration.
    - Set the output field to Max. Principal stress and probe quantitative results at specific points of the model: Tools ⇒ Query ⇒ Probe values
      Specifically find the maximum value of the Max. Principal stress, $\sigma_{1,\mathrm{max}}$, and calculate the stress concentration factor, $K_\mathrm{t} = \sigma_{1,\mathrm{max}}/\sigma_0$, for this geometry.