

# ENGN 1750: Advanced Mechanics of Solids

## Three-dimensional problems in Abaqus

### 1 Objectives

In the previous session, we investigated plane problems in Abaqus, which took advantage of simplicities in the problem geometry 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. For example, consider the complex geometry of the brain and head/helmet assembly that must be accounted for in order to model the process of blast-induced traumatic brain injury, shown below.

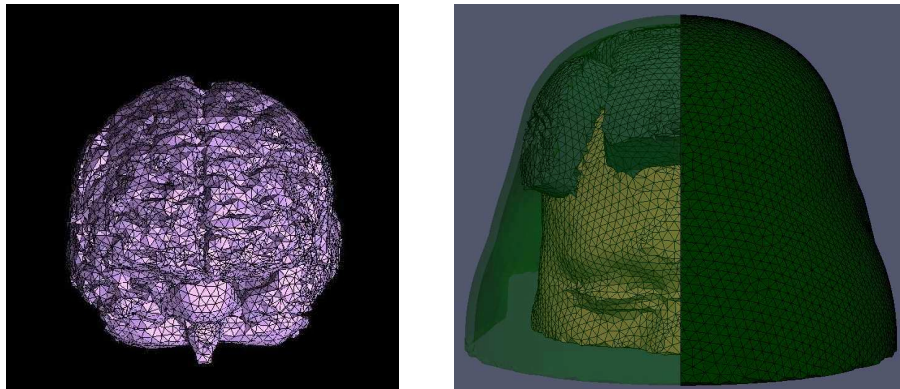


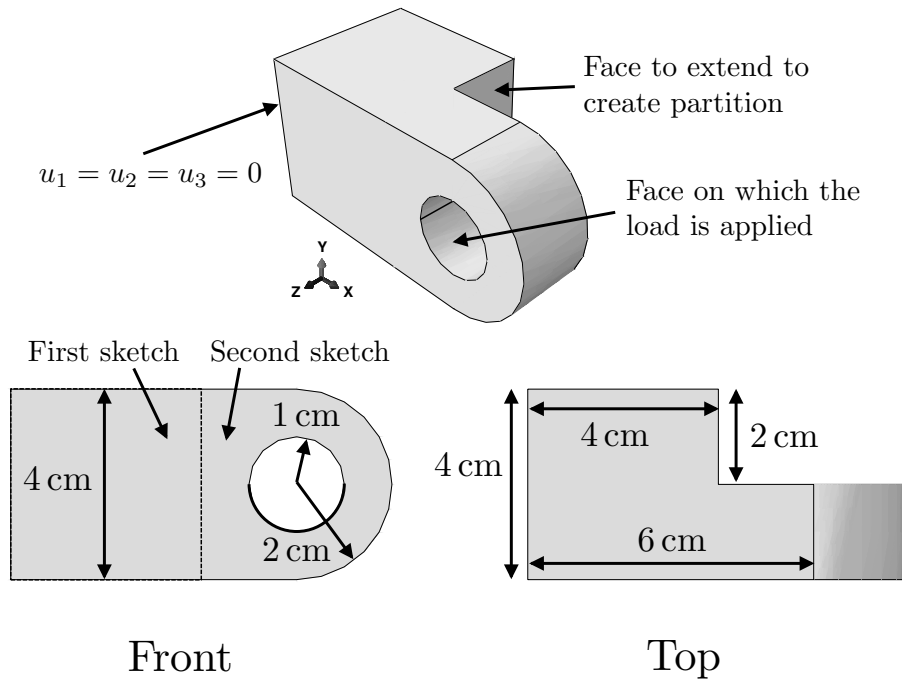
Figure 1: Finite element mesh of the brain (left) and head/padding/helmet assembly (right) for analysis of traumatic brain injury (TBI). (Radovitzky et al., *PNAS* (2010))

Such a problem precludes planar simplifications, requiring a three-dimensional model. The major 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 this session. Moreover, creating a mesh in 3D can be more challenging than in planar problems, requiring partitioning the part into smaller, simpler sections. As a final comment, the part generating tools in 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.

### 2 In-class exercises

#### 2.1 Bracket under a distributed load

We first analyze a bracket-like structure subjected to a load. This exercise will familiarize ourselves to the extrude tool. The bracket is made of steel, and the geometry is shown below:



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

- Part:
  - Part  $\Rightarrow$  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

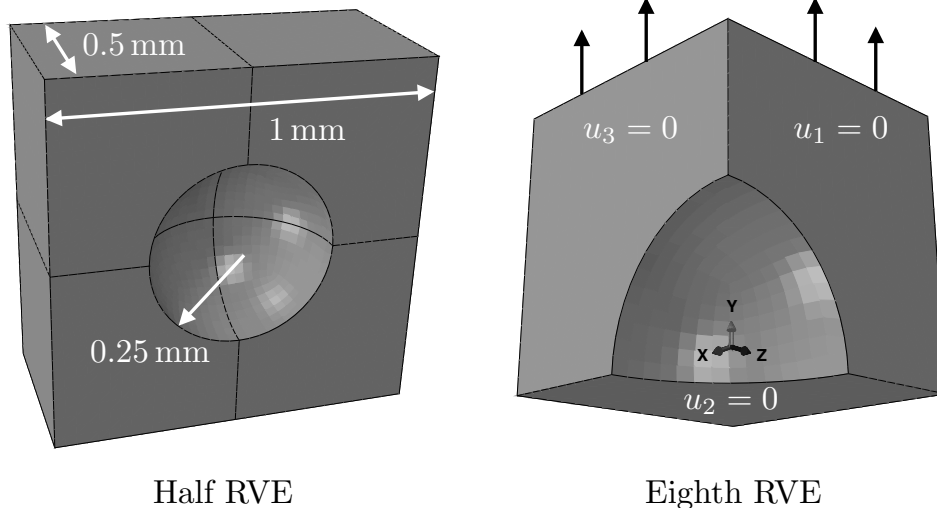
Path Between 2 Points” button, select the hole face, and click Done. Select two corresponding points on the front and back on one side of the hole to create a horizontal partition and click “Create Partition.” Repeat the process to create another partition on the opposite side of the hole face.

- Property:
  - Material  $\Rightarrow$  Create
  - Mechanical  $\Rightarrow$  Elasticity  $\Rightarrow$  Elastic
  - Enter the material properties for steel and click OK
  - Section  $\Rightarrow$  Create
  - Solid  $\Rightarrow$  Homogeneous  $\Rightarrow$  Continue
  - Make sure your material is selected, and click OK
  - Assign  $\Rightarrow$  Section
  - Select the entire part and click Done/OK.
- Assembly:
  - Instance  $\Rightarrow$  Create  $\Rightarrow$  OK
- Step:
  - Step  $\Rightarrow$  Create  $\Rightarrow$  Static/General  $\Rightarrow$  Continue  $\Rightarrow$  OK
- Load:
  - BC  $\Rightarrow$  Create  $\Rightarrow$  Mechanical  $\Rightarrow$  Displacement/Rotation  $\Rightarrow$  Continue  $\Rightarrow$  Select left face and click Done  $\Rightarrow$  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  $\Rightarrow$  Create  $\Rightarrow$  Mechanical  $\Rightarrow$  Surface Traction  $\Rightarrow$  Continue  $\Rightarrow$  Select the bottom half of the hole face and click Done  $\Rightarrow$  Change traction type from shear to general  $\Rightarrow$  Click the arrow beside “Vector”  $\Rightarrow$  Input the first and second points for the direction vector as  $(0, 0, 0)$  and  $(0, -1, 0)$ , respectively  $\Rightarrow$  Input the magnitude as 10 MPa  $\Rightarrow$  Click OK.
- Mesh:
  - Make sure Object is set to Part.
  - Mesh  $\Rightarrow$  Element Type  $\Rightarrow$  Select the entire part and click Done  $\Rightarrow$  Family: 3D Stress  $\Rightarrow$  Click OK
  - Mesh  $\Rightarrow$  Controls  $\Rightarrow$  Select the block section and click Done  $\Rightarrow$  Element Shape: Hex  $\Rightarrow$  Technique: Structured  $\Rightarrow$  Click OK
  - Mesh  $\Rightarrow$  Controls  $\Rightarrow$  Select the bracket section and click Done  $\Rightarrow$  Element Shape: Hex  $\Rightarrow$  Technique: Sweep  $\Rightarrow$  Algorithm: Advancing Front  $\Rightarrow$  Click OK

- Seed  $\Rightarrow$  Part  $\Rightarrow$  Approximate global size: 2.5 mm  $\Rightarrow$  Click OK
- Mesh  $\Rightarrow$  Part  $\Rightarrow$  Yes.
- Job:
  - Job  $\Rightarrow$  Create  $\Rightarrow$  Continue/OK
  - Job  $\Rightarrow$  Submit  $\Rightarrow$  Job-1
  - When the job successfully completes: Job  $\Rightarrow$  Results  $\Rightarrow$  Job-1
- Visualization:
  - Examine contour plots of displacement, stress, and strain.
  - Probe quantitative results at specific points of the model:  
Tools  $\Rightarrow$  Query  $\Rightarrow$  Probe values
  - Where does maximum Mises stress occur?
  - To view calculation results in the interior: Tools  $\Rightarrow$  View Cut  $\Rightarrow$  Manager  $\Rightarrow$   
Select one of the three planes and drag the position bar to move the cut plane.

## 2.2 Micro-mechanical modeling of a porous metal

Next, we will determine the elastic properties of porous aluminum. As a rudimentary micro-mechanical model of a porous material, consider a cubic array of spherical voids in an aluminum matrix. To this end, we consider a 1 mm cubic representative volume element (RVE), or unit cell, with a spherical void located at the center, shown below:



The RVE is subjected to a state of uniaxial stress in the 2-direction, and measuring the displacement response of the RVE will allow us to calculate the Young's modulus and Poisson's ratio of the porous material.

We exploit symmetry and only model one eighth of the RVE shown above. In order to maintain the periodic nature of the RVE (the ability to be tiled in all directions), we wish

to ensure that the outer (non-symmetry) faces remain planar. To do so, we will use an Equation constraint in the Abaqus Interaction module.

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

- Part:

- Part  $\Rightarrow$  Create
- Select 3D, Deformable, Solid, Type: Extrusion, and Approximate size: 1 mm.
- We will first create a cube with side lengths of 0.5 mm. Sketch a 0.5 mm by 0.5 mm square with bottom left corner at (0,0), and click Done. Input an extrusion depth of 0.5 mm, and click OK.
- Find and select the “Create Cut: Revolve” button  $\Rightarrow$  Select the face corresponding to  $z = 0 \Rightarrow$  Select the edge of the cube corresponding to the  $y$ -axis  $\Rightarrow$  Sketch a quarter-circle with center located at the bottom right corner of the face and radius of 0.25 mm (The sketch should be closed.)  $\Rightarrow$  Create a vertical construction line along the right edge of the face, and click Done  $\Rightarrow$  Input a revolution and angle of 90 degrees, make sure the revolution direction is into the part, and click OK.
- For use in applying constraints, create a reference point at (0.5, 0.5, 0.5) mm. Tools  $\Rightarrow$  Reference Point  $\Rightarrow$  Input coordinates.
- Additionally, we will need to create sets for defining constraints.
  - \* Tools  $\Rightarrow$  Set  $\Rightarrow$  Create  $\Rightarrow$  Name: ref, Type: geometry, Continue  $\Rightarrow$  Select the Reference point (RP) and click Done
  - \* Tools  $\Rightarrow$  Set  $\Rightarrow$  Create  $\Rightarrow$  Name: xface, Type: geometry, Continue  $\Rightarrow$  Select the face corresponding to  $x = 0.5$  mm, and click Done.
  - \* Tools  $\Rightarrow$  Set  $\Rightarrow$  Create  $\Rightarrow$  Name: yface, Type: geometry, Continue  $\Rightarrow$  Select the face corresponding to  $y = 0.5$  mm, and click Done.
  - \* Tools  $\Rightarrow$  Set  $\Rightarrow$  Create  $\Rightarrow$  Name: zface, Type: geometry, Continue  $\Rightarrow$  Select the face corresponding to  $z = 0.5$  mm, and click Done.

- Property:

- Material  $\Rightarrow$  Create
- Mechanical  $\Rightarrow$  Elasticity  $\Rightarrow$  Elastic
- Enter the material properties for aluminum, and click OK.
- Section  $\Rightarrow$  Create
- Solid  $\Rightarrow$  Homogeneous  $\Rightarrow$  Continue
- Make sure your material is selected, and click OK
- Assign  $\Rightarrow$  Section
- Select the entire part, and click Done/OK.

- Assembly:

- Instance  $\Rightarrow$  Create  $\Rightarrow$  OK

- Step:

- Step  $\Rightarrow$  Create  $\Rightarrow$  Static/General  $\Rightarrow$  Continue  $\Rightarrow$  OK

- Interaction: In order to keep the outer faces planar, we apply Equation constraints.

- Constraint  $\Rightarrow$  Create  $\Rightarrow$  Equation  $\Rightarrow$  Continue  $\Rightarrow$  Enter the following information and click OK.

Coefficient	Set Name	DOF
1	Part-1-1.xface	1
-1	Part-1-1.ref	1

This will constrain all nodes on the face “xface” to move together in the 1-direction.

- Constraint  $\Rightarrow$  Create  $\Rightarrow$  Equation  $\Rightarrow$  Continue  $\Rightarrow$  Enter the following information and click OK.

Coefficient	Set Name	DOF
1	Part-1-1.yface	2
-1	Part-1-1.ref	2

This will constrain all nodes on the face “yface” to move together in the 2-direction.

- Constraint  $\Rightarrow$  Create  $\Rightarrow$  Equation  $\Rightarrow$  Continue  $\Rightarrow$  Enter the following information and click OK.

Coefficient	Set Name	DOF
1	Part-1-1.zface	3
-1	Part-1-1.ref	3

This will constrain all nodes on the face “zface” to move together in the 3-direction.

- Load:

- BC  $\Rightarrow$  Create  $\Rightarrow$  Mechanical  $\Rightarrow$  Displacement/Rotation  $\Rightarrow$  Continue  $\Rightarrow$  Select  $x = 0$  plane and click Done  $\Rightarrow$  Enter  $U1=0$  and click OK.
- BC  $\Rightarrow$  Create  $\Rightarrow$  Mechanical  $\Rightarrow$  Displacement/Rotation  $\Rightarrow$  Continue  $\Rightarrow$  Select  $y = 0$  plane and click Done  $\Rightarrow$  Enter  $U2=0$  and click OK.
- BC  $\Rightarrow$  Create  $\Rightarrow$  Mechanical  $\Rightarrow$  Displacement/Rotation  $\Rightarrow$  Continue  $\Rightarrow$  Select  $z = 0$  plane and click Done  $\Rightarrow$  Enter  $U3=0$  and click OK.
- Load  $\Rightarrow$  Create  $\Rightarrow$  Mechanical  $\Rightarrow$  Pressure  $\Rightarrow$  Continue  $\Rightarrow$  Select top face (“yface”) and click Done  $\Rightarrow$  Enter Magnitude ( $= -1$  MPa) and click OK.

- Mesh:

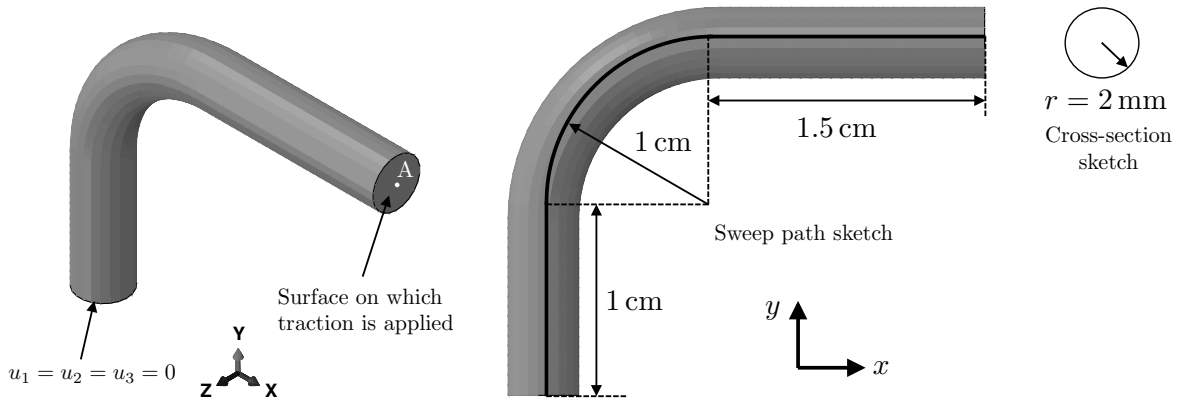
- Make sure Object is set to Part.

- Mesh  $\Rightarrow$  Element Type  $\Rightarrow$  Select the entire part and click Done  $\Rightarrow$  Family: 3D Stress  $\Rightarrow$  Click OK
- Mesh  $\Rightarrow$  Controls  $\Rightarrow$  Element Shape: Hex  $\Rightarrow$  Technique: Structured  $\Rightarrow$  Click OK
- Seed  $\Rightarrow$  Part  $\Rightarrow$  Approximate global size: 0.03 mm  $\Rightarrow$  Click OK
- Mesh  $\Rightarrow$  Part  $\Rightarrow$  Yes
- Job:
  - Job  $\Rightarrow$  Create  $\Rightarrow$  Continue/OK
  - Job  $\Rightarrow$  Submit  $\Rightarrow$  Job-2
  - When the job successfully completes: Job  $\Rightarrow$  Results  $\Rightarrow$  Job-2
- Visualization:
  - Examine contour plots of displacement, stress, and strain.
  - Probe quantitative results at specific points of the model:  
Tools  $\Rightarrow$  Query  $\Rightarrow$  Probe values
  - Obtain the 2-component of displacement ( $u_2$ ) on the top face (“yface”). The axial strain of the porous material is then  $u_2/(0.5 \text{ mm})$  (since 0.5 mm is the original length in the 2-direction). Use this value of strain to calculate the Young’s modulus. (You should get  $E \approx 62 \text{ GPa}$ .) How does it compare to that of aluminum?
  - Obtain the 1-component of displacement ( $u_1$ ) on the face “xface”. The lateral strain of the porous material is then  $u_1/(0.5 \text{ mm})$  (since 0.5 mm is also the original length in the 1-direction). Use this value of the strain along with the previously calculated axial strain to calculate the Poisson’s ratio. (You should get  $\nu \approx 0.29$ .) How does it compare to that of aluminum?

# ENGN 1750: Advanced Mechanics of Solids Abaqus Assignment 4

Due: Monday, November 10, 2014 OR Thursday, November 13, 2014 (in class)

1. Consider the porous solid examined in class. We define the void fraction  $\phi$  as the ratio of the void volume to the total volume. You are to calculate the Young's modulus  $E$  and Poisson's ratio  $\nu$  of the porous solid as a function of the void fraction. Consider void fractions of 0.05, 0.10, 0.15, and 0.2. Submit plots of (a)  $E/E_{Al}$  versus void fraction and (b) Poisson's ratio versus void fraction.
2. Next, consider the curved bar with circular cross-section, shown below. The bar is made of steel.



The bar is anchored at one end, and a traction is applied on the other. Consider three traction vectors with respect to the global coordinate system:

- (a)  $\mathbf{t} = (1, 0, 0) \text{ MPa}$ ,
- (b)  $\mathbf{t} = (0, 1, 0) \text{ MPa}$ , and
- (c)  $\mathbf{t} = (0, 0, 1) \text{ MPa}$ .

For each traction vector, submit a contour plot of the Mises stress on the deformed shape. The mesh should be visible. Further, query the displacement of the center point on the face on which the traction is applied (denoted as point A above). Report, the following stiffnesses for each case enumerated above:

$$(a) \ t_1/(u_1^A/r), \quad (b) \ t_2/(u_2^A/r), \quad (c) \ t_3/(u_3^A/r).$$

Your answer should be in MPa. A guide for creating and meshing the part is provided on the next page.



## Guide for creating and meshing the curved bar geometry:

- Part:
  - Part  $\Rightarrow$  Create
  - Select 3D, Deformable, Solid, Type: Sweep, and Approximate size: 5 cm.
  - Sketch the sweep path as shown in the figure on the previous page and click Done.
  - Maximum scale for the section sketch: 5 mm.
  - Sketch the cross-section as shown in the figure on the previous page and click Done. Be sure that the circle is centered about the origin.
- Mesh:
  - Make sure Object is set to Part.
  - Mesh  $\Rightarrow$  Element Type  $\Rightarrow$  Select the entire part and click Done  $\Rightarrow$  Family: 3D Stress  $\Rightarrow$  Click OK
  - Mesh  $\Rightarrow$  Controls  $\Rightarrow$  Note that the geometry in its current state does not allow for Free, Structured, or Swept mesh controls. We will have to partition the part.
  - Find and click the “Create Datum Plane: Offset From Plane” button in the left menu  $\Rightarrow$  Select the fixed face  $\Rightarrow$  Select Point  $\Rightarrow$  Select a point on the edge at the beginning of the curved section.
  - Find and click the “Create Datum Plane: Offset From Plane” button in the left menu  $\Rightarrow$  Select the loaded face  $\Rightarrow$  Select Point  $\Rightarrow$  Select a point on the edge at the end of the curved section.
  - Find and click the “Partition Cell: Use Datum Plane” button in the left menu  $\Rightarrow$  Select the first datum plane  $\Rightarrow$  Create partition.
  - Find and click the “Partition Cell: Use Datum Plane” button in the left menu  $\Rightarrow$  Select the appropriate cell and click Done  $\Rightarrow$  Select the second datum plane  $\Rightarrow$  Create partition.
  - Mesh  $\Rightarrow$  Controls  $\Rightarrow$  Select the whole part and click Done  $\Rightarrow$  Element Shape: Hex  $\Rightarrow$  Technique: Sweep  $\Rightarrow$  Algorithm: Advancing Front  $\Rightarrow$  Click OK
  - Seed  $\Rightarrow$  Part  $\Rightarrow$  Approximate global size: 0.5 mm  $\Rightarrow$  Click OK
  - Mesh  $\Rightarrow$  Part  $\Rightarrow$  Yes