ENGN 1750: Advanced Mechanics of Solids
Contact modeling in Abaqus

1 Objectives

To this point, we have considered problems involving a single part, to which we apply appropriate loads and displacement boundary conditions. In applications, situations in which two or more parts contact and interact are common. In fact, there is entire branch of mechanics devoted to interacting surfaces, called tribology, which is the study of friction, wear, and lubrication. Important contact problems include:

- interacting bones in a joint,
- braking systems,
- tires interacting with the road,
- trains interacting with rail,
- sheet metal forming processes, and
- machining processes.

Modeling of contact is not as simple as applying a traction or specifying the displacement at the interface. It requires specialized techniques, which are inherently nonlinear – the first nonlinearity we will consider in this class. The purpose of this Abaqus session is to introduce the process of modeling contact in Abaqus. We will not get into numerical specifics of how Abaqus handles contact, as these techniques are quite advanced, rather focusing on using these tools in Abaqus.

2 In-class exercises

2.1 Contact between a rigid sphere and a flat substrate

We first analyze the problem of a flat elastic substrate being indented by a rigid sphere, shown schematically in Fig. 1(a). In addition to introducing contact in Abaqus, we will use this exercise to introduce the use of rigid bodies. The substrate is made of aluminum with $E = 70\,\mathrm{GPa}$ and $\nu = 0.3$, and the spherical indenter has a radius of $R = 1\,\mathrm{mm}$. For the case in which the substrate is an infinite half-space, the analytical solution is given by

$$F = \frac{4}{3}\frac{E}{1-\nu^2}R^{1/2}d^{3/2}, \tag{1}$$

where F and d are the force applied to and the displacement of the rigid sphere, respectively. We will model this problem with an axisymmetric approach, shown in Fig. 1(b). We aim to take the substrate large enough so as to approximate an infinite half space. The substrate shown in Fig. 1(b) represents a disk of radius 4 mm and thickness 4 mm when revolved.

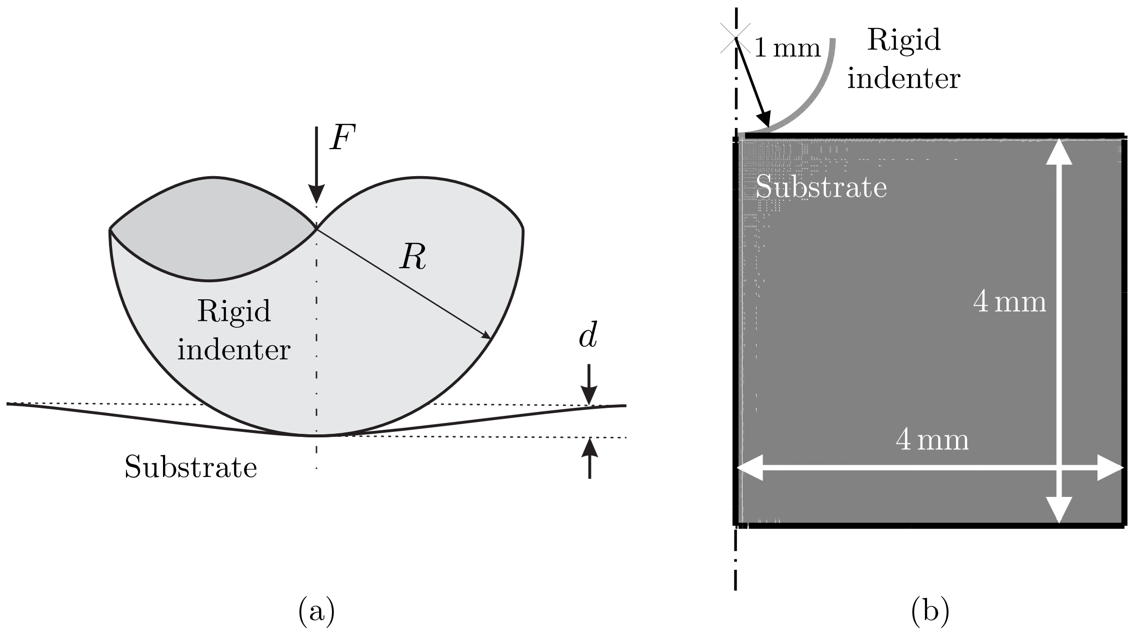


Figure 1: (a) Schematic of indentation of a flat substrate by a rigid sphere. (b) Axisymmetric Abaqus configuration.

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

- Part:
    - Part ⇒ Create
    - Select Axisymmetric, Deformable, Shell, and Approximate size: 10 mm ⇒ Continue
    - We will first create the substrate. Sketch a 4 mm by 4 mm square. Make the top of the square align with the $y = 0$ plane. Click Done.
    - Next we will create the rigid sphere. Part ⇒ Create
    - Select Axisymmetric, Analytical rigid, and Approximate size: 10 mm ⇒ Continue
    - Sketch a quarter of a circle with radius of 1 mm. Make the bottom of the quarter circle align with the $y = 0$ plane. Click Done.
    - All rigid bodies need to be associated with a reference point. Tools ⇒ Reference Point ⇒ Select the center of the quarter circle.
    - For use in requesting output information, we will create a set for the reference point. Tools ⇒ Set ⇒ Create ⇒ Continue ⇒ Select the reference point and click Done.
- Property:
    - Material ⇒ Create

  - Mechanical ⇒ Elasticity ⇒ Elastic
  - Enter the material properties for aluminum and click OK
  - Section ⇒ Create
  - Solid ⇒ Homogeneous ⇒ Continue
  - Make sure your material is selected and click OK
  - Assign ⇒ Section
  - Select Part-1 and click Done/OK. Rigid parts cannot be assigned material properties.

- Assembly:
  - Instance ⇒ Create ⇒ Select both parts ⇒ OK
  - If you sketched the parts as suggested, they will appear in the appropriate relative positions and not overlap. If not, you will need to move one of the parts with the "Instance ⇒ Translate" tool.

- Step:
  - Step ⇒ Create ⇒ Static/General ⇒ Continue
  - Contact is a nonlinear problem, so it will be necessary to solve it in a series of smaller load increments, rather than all at once. In the Incrementation tab, input 0.05 for the initial and maximum increment sizes, and click OK.
  - We wish to obtain non-default output, relating the force applied to and displacement of the rigid sphere. Output ⇒ History Output Requests ⇒ Create ⇒ Continue ⇒ Change Domain to Set: Part-2-1.Set-1 ⇒ In the Displacement family of Output Variables, find U2 and select ⇒ In the Forces family of Output Variables, find RF2 and select ⇒ Click OK

- Interaction:
  - We will first specify that contact is frictionless. Interaction ⇒ Property ⇒ Create ⇒ Contact/Continue ⇒ Mechanical/Tangential Behavior/Frictionless ⇒ Mechanical/Normal Behavior/"Hard" Contact ⇒ Click OK
  - Interaction ⇒ Create ⇒ Surface-to-surface contact/Continue ⇒ Select the rigid surface and click Done ⇒ Select the color corresponding to side of the rigid surface facing the substrate ⇒ Select Surface ⇒ Select the top surface of the substrate and click Done ⇒ Click OK

- Load:
  - BC ⇒ Create ⇒ Mechanical ⇒ Displacement/Rotation ⇒ Continue ⇒ Select left face and click Done ⇒ Enter U1=0 and click OK
  - BC ⇒ Create ⇒ Mechanical ⇒ Displacement/Rotation ⇒ Continue ⇒ Select right face and click Done ⇒ Enter U1=0 and click OK

- BC ⇒ Create ⇒ Mechanical ⇒ Displacement/Rotation ⇒ Continue ⇒ Select bottom face and click Done ⇒ Enter U2=0 and click OK
- BC ⇒ Create ⇒ Mechanical ⇒ Displacement/Rotation ⇒ Select reference point and click Done ⇒ Enter U1=UR3 =0, U2=-0.1 mm and click OK

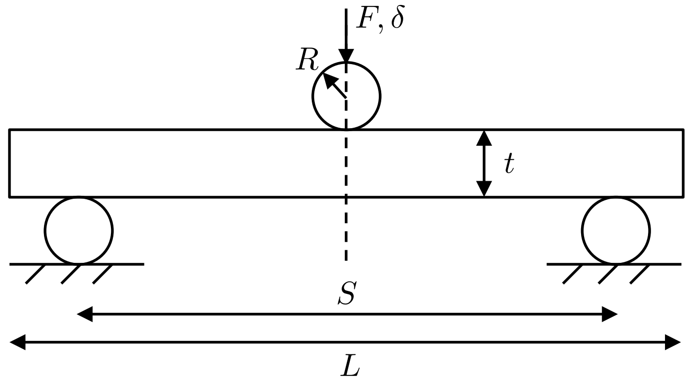
- Mesh:
  - Make sure Object is set to Part: Part-1. The rigid body cannot be meshed.
  - Mesh ⇒ Element Type ⇒ Select the entire part and click Done ⇒ Family: Axisymmetric Stress ⇒ Click OK
  - Mesh ⇒ Controls ⇒ Element Shape: Quad ⇒ Technique: Structured ⇒ Click OK
  - We will seed each edge individually so as to concentrate elements near the point of contact. Seed ⇒ Edges ⇒ Select "Use single-bias picking," select the top edge, and click Done ⇒ Method: By size, Minimum size: 0.01 mm, Maximum size: 0.1 mm ⇒ Click Apply and ensure that the finer seeds are at the top left point ⇒ Click OK
  - Repeat this process for the other three edges, ensuring that the finer seeds are always near the top left point.
  - Mesh ⇒ Part ⇒ Yes

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

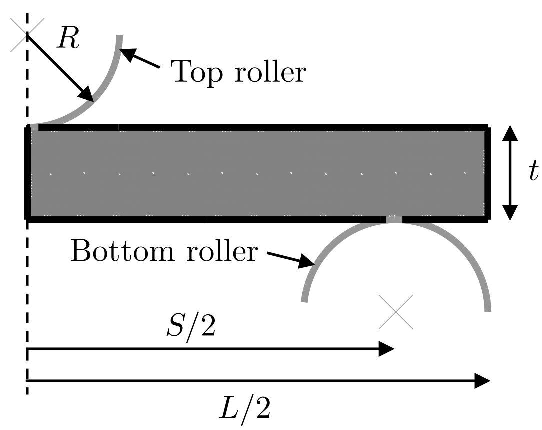
- Visualization:
  - The Deformation Scale Factor may need to be set to Uniform: 1 under Common Plot Options.
  - Examine contour plots of displacement, stress, and strain. Where is the Mises stress maximum?
  - Result ⇒ History Output... ⇒ Select RF2 for the reference point and click Plot. Is it linear?
  - The force applied to the rigid sphere RF2 and the displacement of the sphere U2 may be saved and reported as XY Data to be analyzed in another program.

2.2 Three point bending

Next we will consider the problem of three point bending – a technique commonly used in materials testing, shown schematically in Fig. 2(a). A beam of length L, thickness t, and depth b (into the page) is bent using rollers of radius R. The distance between the two



(a)



(b)

Figure 2: (a) Schematic of the three point bending configuration. (b) Abaqus configuration.

bottom rollers is called the span and denoted as S. We will assume the roller material to be much stiffer than the beam material and treat the rollers as rigid. In this case, the relation between the force applied to the top roller and its displacement may be obtain through beam theory as

$$\frac{F}{b} = \frac{4Et^3}{S^3}\delta. \tag{2}$$

Due to symmetry, we will consider half of the three point bending configuration in our modeling, as shown in Fig. 2(b). We will take $t = 1\,\mathrm{mm}$, $L = 10\,\mathrm{mm}$, and $S = 8\,\mathrm{mm}$. The consider a beam made from aluminum so that $E = 70\,\mathrm{GPa}$ and $\nu = 0.3$.
Below is an outline of the steps for performing the analysis in Abaqus/CAE:

- Part:
    - Part $\Rightarrow$ Create
    - Select 2D Planar, Deformable, Shell, and Approximate size: 10 mm $\Rightarrow$ Continue

- We will first create the beam. Sketch a 5 mm by 1 mm rectangle. Make the bottom of the rectangle align with the $y = 0$ plane. Click Done.
- Next we will create the top roller. Part ⇒ Create
- Select 2D Planar, Analytical rigid, and Approximate size: 10 mm ⇒ Continue
- Sketch a quarter of a circle with radius of 1 mm. Place the center of the quarter circle at the point (0,2) mm. Click Done.
- Create a reference point for the top roller. Tools ⇒ Reference Point ⇒ Select the center of the top roller.
- For use in requesting output information, we will create a set for this reference point. Tools ⇒ Set ⇒ Create ⇒ Continue ⇒ Select the reference point and click Done.
- Next we will create the bottom roller. Part ⇒ Create
- Select 2D Planar, Analytical rigid, and Approximate size: 10 mm.
- Sketch slightly less than a half circle with radius of 1 mm. (Abaqus doesn't allow rigid parts with arcs of 180 degrees or greater.) Place the center of the center of the roller at the point (4,-1) mm. Click Done.
- Create a reference point for the bottom roller. Tools ⇒ Reference Point ⇒ Select the center of the bottom roller.

• Property:

  - Material ⇒ Create
  - Mechanical ⇒ Elasticity ⇒ Elastic
  - Enter the material properties for aluminum and click OK
  - Section ⇒ Create
  - Solid ⇒ Homogeneous ⇒ Continue
  - Make sure your material is selected and click OK
  - Assign ⇒ Section
  - Select Part-1 and click Done/OK. Rigid parts cannot be assigned material properties.

• Assembly:

  - Instance ⇒ Create ⇒ Select all parts ⇒ OK
  - If you sketched the parts as suggested, they will appear in the appropriate relative positions and not overlap. If not, you will need to move one of the parts with the "Instance ⇒ Translate" tool.

• Step:

  - Step ⇒ Create ⇒ Static/General ⇒ Continue

- In the Incrementation tab, input 0.05 for the initial and maximum increment sizes, and click OK.
- Output ⇒ History Output Requests ⇒ Create ⇒ Continue ⇒ Change Domain to Set: Part-2-1.Set-1 ⇒ In the Displacement family of Output Variables, find U2 and select ⇒ In the Forces family of Output Variables, find RF2 and select ⇒ Click OK

- Interaction:

  - We will first specify that contact is frictionless. Interaction ⇒ Property ⇒ Create ⇒ Contact/Continue ⇒ Mechanical/Tangential Behavior/Frictionless ⇒ Mechanical/Normal Behavior/"Hard" Contact ⇒ Click OK
  - Interaction ⇒ Create ⇒ Surface-to-surface contact/Continue ⇒ Select the top roller surface and click Done ⇒ Select the color corresponding to side of the rigid surface facing the beam ⇒ Select Surface ⇒ Select the top surface of the beam and click Done ⇒ Click OK
  - Interaction ⇒ Create ⇒ Surface-to-surface contact/Continue ⇒ Select the bottom roller surface and click Done ⇒ Select the color corresponding to side of the rigid surface facing the beam ⇒ Select Surface ⇒ Select the bottom surface of the beam and click Done ⇒ Click OK

- Load:

  - BC ⇒ Create ⇒ Mechanical ⇒ Displacement/Rotation ⇒ Continue ⇒ Select left face of the beam and click Done ⇒ Enter U1=0 and click OK
  - BC ⇒ Create ⇒ Mechanical ⇒ Displacement/Rotation ⇒ Continue ⇒ Select top roller reference point and click Done⇒ Enter U1=UR3 =0, U2=-0.1 mm and click OK
  - BC ⇒ Create ⇒ Mechanical ⇒ Displacement/Rotation ⇒ Continue ⇒ Select bottom roller reference point and click Done ⇒ Enter U1=U2=UR3 =0 and click OK

- Mesh:

  - Make sure Object is set to Part: Part-1. The rigid body cannot be meshed.
  - Mesh ⇒ Element Type ⇒ Select the entire part and click Done ⇒ Family: Plane Stress ⇒ Click OK
  - Mesh ⇒ Controls ⇒ Element Shape: Quad ⇒ Technique: Structured ⇒ Click OK
  - Seed ⇒ Part ⇒ Approximate global size: 0.03 mm ⇒ Click OK
  - Mesh ⇒ Part ⇒ Yes

- Job:

  - Job ⇒ Create ⇒ Continue/OK

- Job ⇒ Submit ⇒ Job-2
- When the job successfully completes: Job ⇒ Results ⇒ Job-2

- Visualization:
  - The Deformation Scale Factor may need to be set to Uniform: 1 under Common Plot Options.
  - Examine contour plots of displacement, stress, and strain.
  - Result ⇒ History Output... ⇒ Select RF2 for the reference part and Plot.
  - The force applied to the rigid sphere RF2 and the displacement of the sphere U2 may be saved and reported as XY Data to be analyzed in another program. Note that the reaction force will be half of the total force since we are only considering one half of the beam.

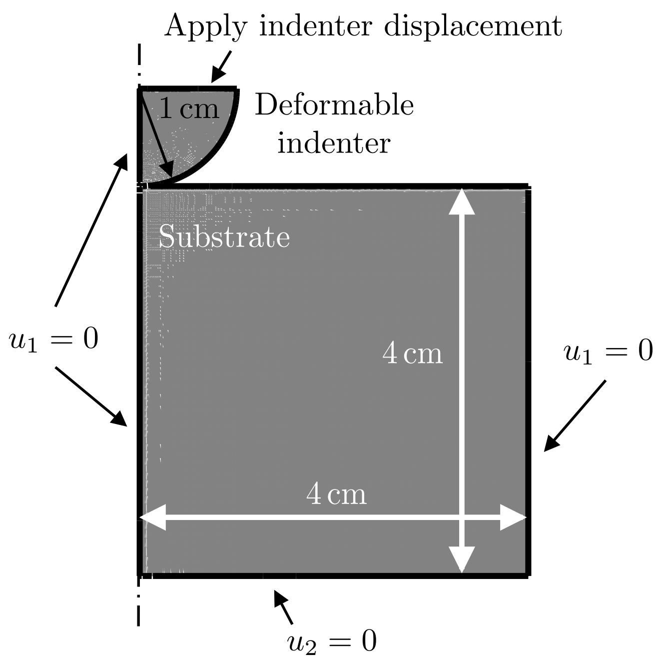
ENGN 1750: Advanced Mechanics of Solids
Abaqus Assignment 5

Due: Tuesday, November 25, 2014

1. Consider the indentation of a flat substrate, discussed in class.

   (a) For the case of the rigid indenter, submit a plot of the calculated force/displacement relation along with the analytical relation (1). Comment on how they compare. What might be the source of any discrepancy?

   (b) Now consider the case of a deformable indenter. In your analysis, make your quarter circle indenter deformable, and make it of the same material as the substrate, as shown below. (You will need to create a new model to do this.)



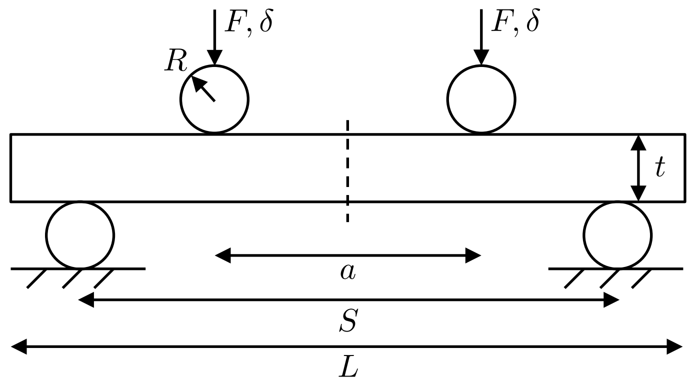
Apply the downward displacement to the top surface of the indenter. You will need to create a set for this top surface in order to obtain RF2 output for all nodes on this surface, which you will then sum to obtain the total applied force. Submit a plot of the calculated force/displacement relation along with the analytical relation for this case,

$$F = \frac{2}{3}\frac{E}{1-\nu^2}R^{1/2}d^{3/2}. \tag{3}$$

How does this compare to that which you calculated for part (a)? Does this make sense? Also submit a contour plot of the Mises stress on the deformed shape.

2. (a) For the case of three point bending, submit a plot of the calculated force/displacement relation (F/b vs. δ) along with the analytical relation (2). Comment on how they compare. What might be the source of any discrepancy?

   (b) Next, consider the case of four point bending, shown schematically below.



The beam is made from aluminum, and you may idealize the rollers as rigid. Take $L = 10\,\text{mm}$, $S = 8\,\text{mm}$, $a = 4\,\text{mm}$, and $t = 1\,\text{mm}$. Submit a plot of the calculated force/displacement relation for four point bending. Also submit a contour plot of the Mises stress on the deformed shape.