

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, one of the most common ways to load a solid is through contact with another solid. Important problems involving contact 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 since the region where the two solids come into contact is generally not known from the outset and must be determined as part of the solution. The purpose of these exercises 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. Several important points are noted below (quoted from Bower Chapter 7.1).

Modeling a stiff solid as a rigid surface: In many cases of practical interest one of the two contacting solids is much more compliant than the other. As long as the stresses in the stiff or hard solid are not important, its deformation can be neglected. In this case the stiffer of the two solids may be idealized as a rigid surface. A rigid surface cannot change its shape, but it can move about and rotate. Its motion is defined using a *reference point* on the solid, which behaves like a node. To move the solid around during an analysis, you can define displacement and rotational degrees of freedom at this node. Alternatively, you can apply forces and moments to the reference point.

Defining a contact pair – master/slave surfaces: Whenever you set up a finite element calculation that involves contact, you need to specify pairs of surfaces that may come into contact during the analysis. One of each pair must be designated the master surface, the other must be designated the slave surface. This (very outdated) finite element terminology refers to the way that contact constraints are actually applied during a computation. The geometry of the master surface will be interpolated as a smooth curve in some way (usually

by interpolating between nodes); however, the slave surface is not interpolated. Instead, each individual node on the slave surface is constrained so as not to penetrate into the master surface. For a sufficiently fine mesh, the results should not be affected by your choice of master and slave surface. However, it is good practice to choose the more stiff of the two surfaces to be the master surface. For example, if a rigid surface participates in a contact pair, it must be the master surface. If you don't know which surface is more rigid, just make a random choice. If you run into convergence problems later, try switching them.

Contact parameters: It is necessary to define several parameters that describe the behavior of two contacting surfaces. First, you will need to specify the relationship between the contact pressure and separation between the contacting surfaces. Alternatively, you can assume the contact is “hard” this means the interface cannot withstand any tension, and the two contacting surfaces cannot inter-penetrate. Second, you will need to specify the tangential behavior of the interface – for example by giving the friction coefficient or specifying frictionless contact.

We will consider two representative contact problems:

1. Contact between a rigid sphere and a flat substrate and
2. The stress concentration for a plate loaded in tension by a pin through a hole.

2 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 $E = 70$ GPa and $\nu = 0.3$, and the spherical indenter has a radius of $R = 1$ mm. For the case in which the substrate is an infinite half-space, the analytical solution – derived by Heinrich Hertz in 1882 – is given by

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

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.

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

- Part:
 - Part \Rightarrow Create
 - Select Axisymmetric, Deformable, Shell, and Approximate size: 10 mm \Rightarrow Continue

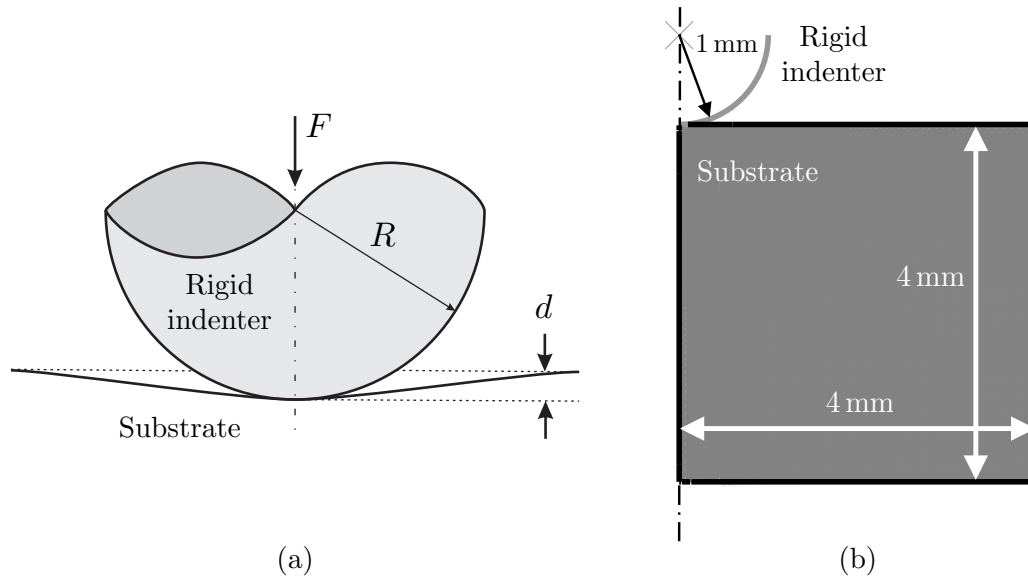


Figure 1: (a) Schematic of indentation of a flat substrate by a rigid sphere. (b) Axisymmetric Abaqus configuration.

- 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 \Rightarrow Create
- Select Axisymmetric, Analytical rigid, and Approximate size: 10 mm \Rightarrow 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 \Rightarrow Reference Point \Rightarrow Select the center of the quarter circle.
- For use in requesting output information, we will create a set for the reference point. Tools \Rightarrow Set \Rightarrow Create \Rightarrow Continue \Rightarrow Select the reference point 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 Part-1 and click Done/OK. Rigid parts cannot be assigned material properties.
- Assembly:
 - Instance \Rightarrow Create \Rightarrow Select both parts \Rightarrow 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 \Rightarrow Translate” tool.
- Step:
 - Step \Rightarrow Create \Rightarrow Static/General \Rightarrow 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 \Rightarrow History Output Requests \Rightarrow Create \Rightarrow Continue \Rightarrow Change Domain to Set: Part-2-1.Set-1 \Rightarrow In the Displacement family of Output Variables, find U2 and select \Rightarrow In the Forces family of Output Variables, find RF2 and select \Rightarrow Click OK
- Interaction:
 - We will first specify that contact is frictionless. Interaction \Rightarrow Property \Rightarrow Create \Rightarrow Contact/Continue \Rightarrow Mechanical/Tangential Behavior/Frictionless \Rightarrow Mechanical/Normal Behavior/“Hard” Contact \Rightarrow Click OK
 - Interaction \Rightarrow Create \Rightarrow Surface-to-surface contact/Continue \Rightarrow Select the rigid surface and click Done \Rightarrow Select the color corresponding to side of the rigid surface facing the substrate \Rightarrow Select Surface \Rightarrow Select the top surface of the substrate and click Done \Rightarrow Click OK
- Load:
 - BC \Rightarrow Create \Rightarrow Mechanical \Rightarrow Displacement/Rotation \Rightarrow Continue \Rightarrow Select left face and click Done \Rightarrow Enter U1=0 and click OK
 - BC \Rightarrow Create \Rightarrow Mechanical \Rightarrow Displacement/Rotation \Rightarrow Continue \Rightarrow Select right face and click Done \Rightarrow Enter U1=0 and click OK
 - BC \Rightarrow Create \Rightarrow Mechanical \Rightarrow Displacement/Rotation \Rightarrow Continue \Rightarrow Select bottom face and click Done \Rightarrow Enter U2=0 and click OK
 - BC \Rightarrow Create \Rightarrow Mechanical \Rightarrow Displacement/Rotation \Rightarrow Select reference point and click Done \Rightarrow 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 \Rightarrow Element Type \Rightarrow Select the entire part and click Done \Rightarrow Family: Axisymmetric Stress \Rightarrow Click OK
- Mesh \Rightarrow Controls \Rightarrow Element Shape: Quad \Rightarrow Technique: Structured \Rightarrow Click OK
- We will seed each edge individually so as to concentrate elements near the point of contact. Seed \Rightarrow Edges \Rightarrow Select “Use single-bias picking,” select the top edge, and click Done \Rightarrow Method: By size, Minimum size: 0.01 mm, Maximum size: 0.1 mm \Rightarrow Click Apply and ensure that the finer seeds are at the top left point \Rightarrow Click OK
- Repeat this process for the other three edges, ensuring that the finer seeds are always near the top left point.
- 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:
 - 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 \Rightarrow History Output... \Rightarrow 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 Stress concentration for a plate loaded in tension by a rigid pin through a hole

In previous exercises, we considered stress concentrations in planar and 3D problems. Stress concentrations can also arise in problems involving contact. We will consider the stress concentration in a plate loaded in tension by a rigid pin through a hole, shown in Fig. 2(a). Defining the nominal stress in this geometry based upon the reduced area of the plate, $\sigma_0 = P/[(W - d)t]$, the stress is raised in the region of the pin, quantified by the stress concentration factor

$$K_t = \frac{\sigma_{1,\max}}{\sigma_0},$$

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: $W = 10$ cm, $H = 5$ cm, and $d = 4$ cm. We idealize

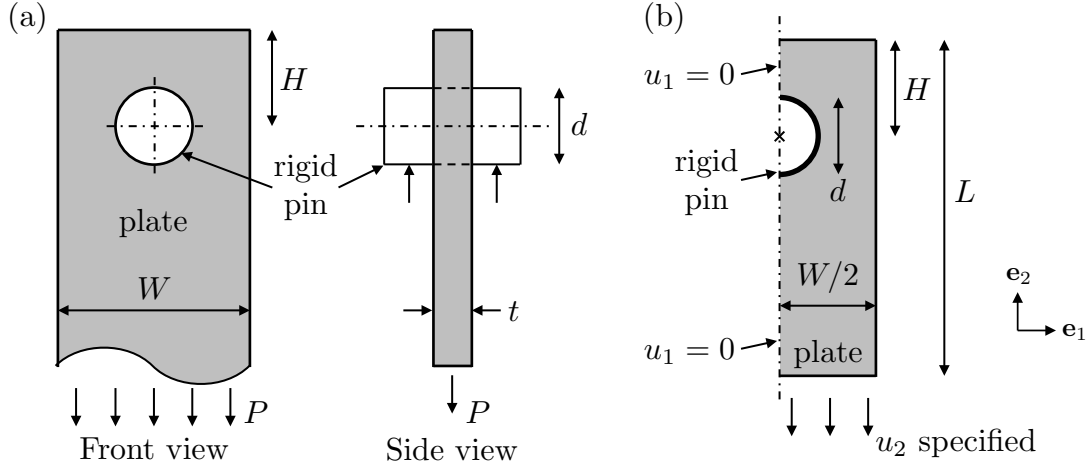


Figure 2: (a) Schematic of a plate loaded in tension by a rigid pin through a hole. (b) Plane stress Abaqus configuration.

the plate as being in a state of *plane stress*, model the pin using a rigid body, and exploit the symmetry of the problem, only modeling one half of the plate and pin, as shown in Fig. 2(b). The plate must be taken to be sufficiently long so as not to affect the determination of the stress concentration factor, and to this end, we take $L = 25$ cm. For concreteness, we use material properties for aluminum. In prior stress concentration problems, we applied the far-field load using a traction-type boundary condition; however, it is difficult during a finite-element analysis to establish contact under load-control. For this reason, we apply the far-field load by specifying a downward displacement of $u_2 = 0.8$ mm on the bottom face, as shown in Fig. 2(b).

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

- Part:
 - We will first create the plate with a hole. Part \Rightarrow Create
 - Select 2D Planar, Deformable, Shell, and Approximate size: 1 m \Rightarrow Continue
 - Sketch the plate geometry as shown in Fig. 2(b). Make the center of the circle coincide with the origin. Click Done.
 - Next we will create the pin. Part \Rightarrow Create
 - Select 2D Planar, Analytical rigid, and Approximate size: 1 m \Rightarrow Continue
 - Sketch slightly less than a half circle with radius of 2 cm. (Abaqus doesn't allow rigid parts with arcs of 180 degrees or greater.) Place the center of the pin at the origin. Click Done.
 - Create a reference point for the pin. Tools \Rightarrow Reference Point \Rightarrow Select the center of the pin.
 - For use in requesting output information, we will create a set for this reference point. Tools \Rightarrow Set \Rightarrow Create \Rightarrow Continue \Rightarrow Select the reference point 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 Part-1 and click Done/OK. Rigid parts cannot be assigned material properties.
- Assembly:
 - Instance \Rightarrow Create \Rightarrow Select all parts \Rightarrow 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 \Rightarrow Translate” tool.
- Step:
 - Step \Rightarrow Create \Rightarrow Static/General \Rightarrow Continue
 - In the Incrementation tab, input 0.05 for the initial and maximum increment sizes, and click OK.
 - Output \Rightarrow History Output Requests \Rightarrow Create \Rightarrow Continue \Rightarrow Change Domain to Set: Part-2-1.Set-1 \Rightarrow In the Forces/Reactions family of Output Variables, find RF2 and select \Rightarrow Click OK
- Interaction:
 - We will first specify that contact is frictionless. Interaction \Rightarrow Property \Rightarrow Create \Rightarrow Contact/Continue \Rightarrow Mechanical/Tangential Behavior/Frictionless \Rightarrow Mechanical/Normal Behavior/“Hard” Contact \Rightarrow Click OK
 - Interaction \Rightarrow Create \Rightarrow Surface-to-surface contact/Continue \Rightarrow Select the pin surface and click Done \Rightarrow Select the color corresponding to side of the rigid surface facing up \Rightarrow Select Surface \Rightarrow Select the surface of the hole and click Done \Rightarrow Click OK
- Load:
 - BC \Rightarrow Create \Rightarrow Mechanical \Rightarrow Displacement/Rotation \Rightarrow Continue \Rightarrow Select left faces of the plate, i.e., the symmetry plane, and click Done \Rightarrow Enter U1=0 and click OK

- BC \Rightarrow Create \Rightarrow Mechanical \Rightarrow Displacement/Rotation \Rightarrow Continue \Rightarrow Select the pin reference point and click Done \Rightarrow Enter $U1=U2 = UR3 =0$ and click OK
- BC \Rightarrow Create \Rightarrow Mechanical \Rightarrow Displacement/Rotation \Rightarrow Continue \Rightarrow Select the bottom face of the plate and click Done \Rightarrow Enter $U2=-0.8$ mm and click OK
- Mesh:
 - Make sure Object is set to Part: Part-1. The rigid body cannot be meshed.
 - Mesh \Rightarrow Element Type \Rightarrow Select the part and click Done \Rightarrow Family: Plane Stress \Rightarrow Geometric Order: Quadratic \Rightarrow Click OK
 - Mesh \Rightarrow Controls \Rightarrow Element Shape: Quad \Rightarrow Technique: Free \Rightarrow Click OK
 - We will give the coarse seed size through the global seeds and then specify fine edge seeds where appropriate. Seed \Rightarrow Part \Rightarrow Approximate global size: 3 mm \Rightarrow Click OK
 - We wish to refine the mesh along the hole face. Seed \Rightarrow Edges \Rightarrow Select the hole edge and click Done \Rightarrow Approximate element size: 0.4 mm \Rightarrow Click OK
 - Seed \Rightarrow Edges \Rightarrow Check the “Use single-bias picking,” and select the edges on the top and bottom left side of the plate and click Done \Rightarrow Minimum size: 0.4 mm and Maximum size: 3 mm \Rightarrow Apply (Make sure the fine seeds are on the ends of the edges near the hole. If not, flip bias on the appropriate edges) \Rightarrow 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:
 - 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 \Rightarrow History Output... \Rightarrow Select RF2 for the reference point and Plot.
 - The reaction force RF2 represents half of the total force, P , applied to the pin since we are only considering one half of the plate. The nominal stress, σ_0 , may then be obtained by dividing the total force, P , by the reduced cross-sectional area, $(W - d)t$ with reference to Fig. 2.
 - Set the output field to Max. Principal stress and probe quantitative results at specific points of the model: Tools \Rightarrow Query \Rightarrow Probe values
Specifically find the maximum value of the Max. Principal stress, $\sigma_{1,\max}$, and calculate the stress concentration factor, $K_t = \sigma_{1,\max}/\sigma_0$, for this geometry.