

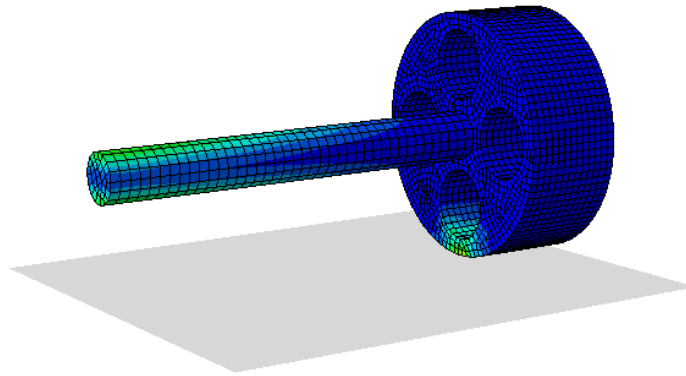


School of Engineering
Brown University

EN1750: Advanced Mechanics of Solids

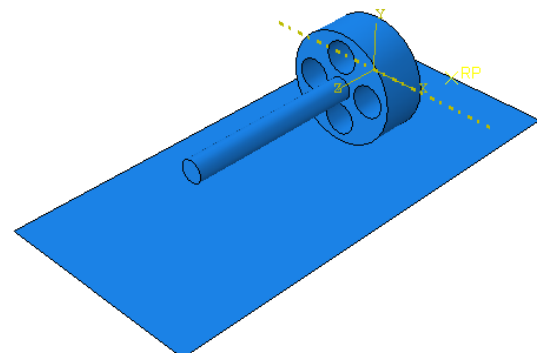
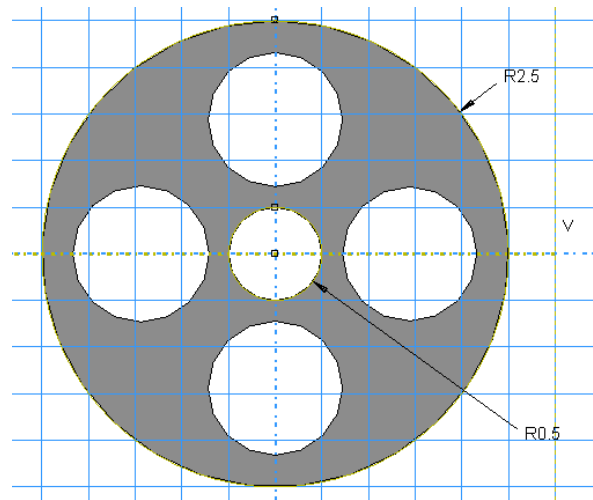
Homework 5: Analyzing Assemblies in ABAQUS

Due Friday Oct 18, 2019



1. Analyzing Assemblies in ABAQUS/CAE. Two problems arise when you analyze an assembly: (i) you will need to define properties for contacts between the parts in the assembly; and (ii) you need to find a way to define boundary conditions for your analysis that ensure that none of the parts have unconstrained rigid body modes. To see how to deal with these problems, you will calculate stresses in the simple wheel-axle-ground assembly shown in the figure.

- 1.1 Start by creating the wheel in the **Part Module**: you can do this by creating a 3D deformable part; then sketch an annular region with outer radius 2.5cm and inner radius 0.5cm. It is helpful to add a vertical construction line to the sketch so you can define the orientation of the part for the cut-extrude. Finally extrude the part with an extrusion depth of 2cm. You can add the four cutout circles (which have radius 0.75cm) by using a cut-extrude operation.
- 1.2 Next, create the axle. You can do this by creating a new 3D deformable part; sketch a circle with radius 0.5 centered at the origin, and extrude it by 10cm
- 1.3 Finally, create the ground. This is a 3D rigid analytic shell, with width of about 10cm and extrusion depth of 20cm (the exact dimensions are not important). It is helpful to create the part so it will be just tangential to the wheel; but if you put it in the wrong place you can always translate it to the correct position in the assembly. Add a reference point (Tools > Reference Point) to the surface that you can use to define boundary conditions during the analysis.
- 1.4 Next, in the **Material Module**, create an elastic material with Young's Modulus $210 \times 10^5 \text{ N/cm}^2$ and Poisson's ratio 0.3. Create a homogeneous solid

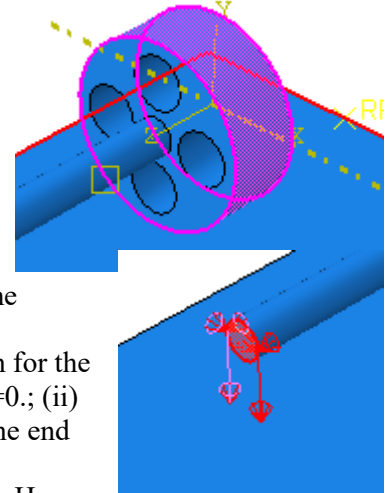


section, with the material properties you just created, and assign it to both the wheel and the axle.

1.5 In the **Assembly Module** create an instance of the wheel, the axle, and the ground. If necessary, translate the ground so that it is just tangent to the wheel as shown in the figure.

1.6 In the **Step** module create a static general step with NLGEOM on and all the default options

1.7 In the **Interaction** module create an Interaction Property with Hard Contact for normal behavior and Frictionless for tangential behavior. Use Interaction > Find Contact Pairs to create contact pairs between the axle and wheel, and (if you are lucky) between the wheel and ground. The Find Contact Pairs algorithm does not always detect the contact between wheel and ground. If this happens create it manually, using Interaction > Create; select Surface to Surface contact from the list of options; in the viewport select the ground for the master surface (select the upper side) and then select Surface for the slave, and select the circumferential surface of the wheel to complete the contact definition.

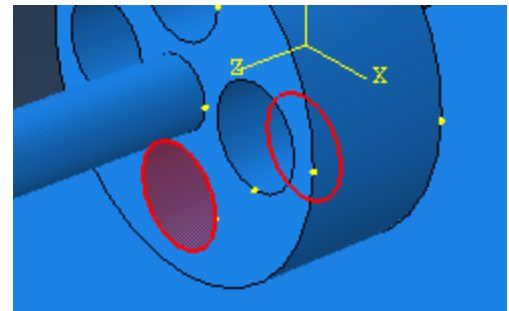


1.8 In the **Load Module** create (i) a BC with Displacement/Rotation option for the Reference Point on the ground that sets $U1=U2=U3=UR1=UR2=UR3=0$.; (ii) create a boundary condition that sets $U1=U3=0$ and $U2= - 0.01\text{cm}$ on the end of the axle

1.9 In the **Mesh Module** (i) Mesh the wheel – use Mesh Controls... to set a Hex-dominated mesh using Swept meshing with the Advancing Front algorithm. Seed the wheel with a mesh size of 0.25cm; and then generate the mesh.; (ii) Mesh the axle – use the same meshing algorithm; and seed the part with a 0.2cm global size; then generate the mesh.

1.10 In the **Job Module** create a job with all the default options. Submit the job, and use Monitor... to track its progress. You will see a warning, which you can ignore. But the analysis will fail – ABAQUS will keep restarting the computation with progressively smaller time-steps, and will terminate once the time-step has been reduced to the minimum value.

1.11 ABAQUS fails because the wheel is not properly constrained – with frictionless contacts at both the ground and the axle, the wheel can rotate freely about the axle, and can also slide up the axle, so there equation system ABAQUS is trying to solve does not have a unique solution. Solutions do exist, of course, and in principle ABAQUS should be able to find them, but it is not coded to do so. To demonstrate that this is the cause of the problem, in the **Load module** create a new BC that will constrain the displacements on the inner surface of the lowest hole in the wheel (see the figure). Prescribe a displacement $U1=U3=0$, $U2= - 0.005$. Then return to the **Job module** and resubmit the analysis. This time the analysis will complete. In the **Visualization module** open the ODB and plot contours of Mises stress.



Hand in a contour plot of the Mises stress distribution in the wheel/axle assembly as a solution to this problem.

1.12 Friction between the wheel and axle and/or between the wheel and ground should eliminate the rigid body mode for the wheel. To check this, (i) in the **Load Module** delete the BC created in part 1.11. Then go to the **Interaction Module** and use Interaction > Property > Edit to edit the Tangential Behavior of the contact – select ‘Penalty’ for the friction formulation, and enter a friction coeff of 0.1. Then return to the **Job module** and resubmit the job. This time it should complete successfully, but ABAQUS will need to reduce the time-step for the analysis to converge.

1.13 The choice of friction algorithm can have a strong influence on convergence of an analysis with frictional contacts in an assembly. To see this, return to the **interaction module**, use Interaction > Property > Edit to edit the Tangential Behavior and select the Lagrange Multiplier formulation for friction. Then return to the **Job Module** and resubmit the job. It will complete successfully, but will take many more increments than were necessary with the penalty formulation

1.14 Even if friction does eliminate rigid body modes of all the parts, some boundary conditions may still cause the analysis to fail. To see this (i) first reset the friction algorithm to the ‘penalty’ formulation in the interaction module, then in the **Load** module use BC > Manager to edit the 2nd boundary condition to change the value of U1 from 0 to 0.2 (to try to roll the wheel). Then return to the job module and resubmit the analysis. It will fail again.

1.15 You can sometimes help ABAQUS solve problems with challenging boundary conditions by breaking the analysis up into several steps. To do this, (i) In the **Step** module create a new general static step; then (ii) in the **Load** module use BC >

Boundary Condition Manager				
	Name	Initial	Step-1	Step-2
✓	BC-1		Created	Propagated
✓	BC-2		Created	Inactive

manager to deactivate BC 2 in Step-2. Also edit BC-2 to apply $U_1=0$, $U_2= - 0.01$, $U_3=0$ (this is the same as in part 1.8). Then create a new BC that is active only in Step-2 (this will be the default when you create the BC), and constrains $U_1=0.2$, $U_2= - 0.01$, $U_3=0$ on the end of the shaft (see the figure for part 1.8) Then return to the Job module and resubmit the analysis. This time the analysis should complete successfully. If you like, you could make the solution look more interesting by editing BC-3 to apply a larger value for U1 (you could try $U_1=1$ or maybe greater) – then you will see the wheel rolling along the ground....

Hand in a contour plot of the Mises stress distribution in the wheel/axle assembly at the end of Step 2 as a solution to this problem.

2. Use some of the procedures suggested in Problem 1 to design and analyze the stresses in a simple assembly of your choice. Those of you on car team may have a piece of car you’d like to analyze. Otherwise look around for anything you would find interesting. Setting up an analysis like this is often quite tricky so

- (i) Keep the problem reasonably simple – don’t try to do an automatic transmission system with thousands of different parts in it!
- (ii) Try to break up the analysis into stages – for example, add the parts one at a time; apply suitable loads to them (which may not necessarily be the loads you are interested in; just something similar to check the simulation) and run a simplified analysis. This will help you check that your part/property/contact/boundary condition definition is working. If you set up a big complex simulation all at once it will probably fail, and with a huge number of parts it can be very difficult to trace the cause of the problem.

We will grade this problem based partly on your creativity in devising an interesting analysis; and the level of sophistication of the analysis.

Hand in a 1-2 page description of your analysis, including:

- Include a figure showing the assembly with relevant dimensions
- State properties used for materials, and any contact interactions
- Describe the boundary conditions
- Include some figures showing results, e.g. relevant stress measures at important locations, along with a description of conclusions you could draw from the results – eg a safety factor for the assembly