# ES 128: Computer Assignment #4

Due in class on Monday, 12 April 2010

Task 1. Study an elastic-plastic indentation problem. This problem combines plasticity with contact mechanics and has many rich aspects. Industry has long relied on indentation techniques for measuring the hardness and stiffness of engineering materials, include metals and ceramics. Indentation tests can be performed quickly without sample preparation. In this assignment, we focus on the hardness measurement during penetration (loading). The material stiffness can be obtained from the unloading curves after the indentation experiment is finished.
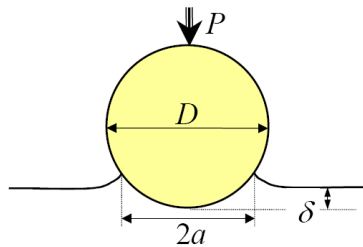
The most commonly used indenter shapes are pyramid, cone, and spherical. Spherical indentation is studied in this assignment such that the problem can be reduced to axisymmetric. The general findings in this assignment are also applicable to other shapes of indenters. As illustrated in the figure, a rigid spherical indenter (typically made of diamond) of diameter D is pushed into a large substrate by a static load P. The contact radius is a and the indent depth is δ. The substrate material is taken to be elastic-perfectly plastic with tensile yield stress σ_y, Young's modulus E and Poisson's ratio v. We assume the material is isotropic and homogeneous. Depending on the yield strain of the material, the substrate will either produce a plastic pile-up or sink-in at the crater. The hardness of the material is defined as $H=P/\pi a^2$. Classic plasticity analysis show that $H=3\sigma_y$. This formula enables people to measure the material yield strength σ_y from the indentation experiment. We will examine the accuracy of this equation by using the finite element method, with 8-node quadratic block elements.

In this assignment, we take D=3mm. Displacement-controlled indentation is used where δ=0.1mm . The force P is obtained as a reaction force. Our sample (substrate), which is much larger than the indenter, is a cylindrical plate with radius 5cm and thickness 3cm. The sample is made of steel and its E=200GPa , σ_y=200 MPa ,and v=0.3. You are asked to develop an ABAQUS input file for this problem with simple axisymmetric geometry, by following the guidelines outlined below.

This problem contains 2 parts: the substrate, which is a rectangular deformable region within the axisymmetric content, and the indenter, which is a rigid analytical surface. First, we create the nodes and elements within the substrate. We set the origin of the cylindrical coordinate system at the initial point of contact. For axisymmetric problems, the first coordinate is in the r direction and the second is in the z direction.

We first create 3 nodes along the top surface of the substrate:

*HEADING
CONTACT PROBLEM
**
** DEFINE NODE MESH
**
*NODE,NSET=NODE1
1,0.0,0.0
*NODE,NSET=NODE2
61,0.0008,0.0
*NODE,NSET=NODE3
101,0.05,0.0

Next, we use the *NFILL keyword to generate all nodes between 1 and 101 (consult ABAQUS/Keywords manual and ABAQUS/Standard User's manual I to understand and complete the following data lines):
*NFILL,NSET=TOP,BIAS=1.0,TWO STEP
NODE1,NODE2,60,1
*NFILL,NSET=TOP,BIAS=0.8,TWO STEP
NODE2,NODE3,?,?

Next, we copy the node set TOP by 2 times, and fill the nodes in between to create all nodes in the substrate:
*NCOPY,OLD SET=TOP,NEW SET=MIDDLE,CHANGE NUMBER=3030,SHIFT
0.0,-0.0004,0.0
0.,0.,0.
*NCOPY,OLD SET=TOP,NEW SET=BOTTOM,CHANGE NUMBER=6060,SHIFT
0.0,-0.03,0.0
0.,0.,0.
*NFILL,NSET=NALL,BIAS=1.0,TWO STEP
TOP,MIDDLE,30,101
*NFILL,NSET=NALL,BIAS=0.8,TWO STEP
MIDDLE,BOTTOM,30,101

A total of 61 rows of nodes are generated and all nodes are continuously numbered. The node difference between two neighboring rows is 101. Next, we define a node set SYMM which includes all nodes on the symmetry line ($r = 0$):
*NSET,NSET=SYMM,GENERATE
?,?,?

Now we define the elements. First, we define a master element (1) and then we generate all elements. Pay attention to the sequence of nodes when the master element is defined, as it will

decide the face number identifiers. This is covered in the class, and more information can be obtained from the ABAQUS/Standard user's manual II and ABAQUS/Keywords manual:

```
**
** DEFINE ELEMENTS
**
*ELEMENT,TYPE=CAX8R,ELSET=ELALL
1,1,203,205,3,102,204,104,2
*ELGEN,ELSET=ELALL
1,50,2,1,30,202,50
*ELSET,ELSET=TOPLINE,GENERATE
1,50,1
```

Here, ELALL is the element set of all substrate elements, and TOPLINE is the element set of the first (top) row of elements in the substrate. This row of elements will be in contact with the indenter, so they are separately defined. The elements are continuously numbered, from 1-50 in the first row, and 51-100 in the 2nd row, and so forth. There are 30 rows of elements in total.

Next, we define the rigid indenter. We first define a reference node 99999, and a rigid analytical surface BSURF. Then we create a rigid body that is associated with the rigid surface and the reference node. Finally, we define the top surface of the substrate, ASURF, which is our slave surface:

```
** DEFINE RIGID SPHERE WITH D=3MM
**
*NODE,NSET=BALL
99999,0.0,0.0015
*SURFACE,TYPE=SEGMENTS,NAME=BSURF
START,0.0015,0.0015
CIRCL,0.0,0.0,0.0,0.0015
*RIGID BODY,REF NODE=99999, ANALYTICAL SURFACE=BSURF
*SURFACE,TYPE=ELEMENT,NAME=ASURF
?,?
```

Now define the contact pairs and boundary conditions:

```
**
** DEFINE CONTACT PAIRS
**
*CONTACT PAIR,INTERACTION=WITHFRIC
?,?
*SURFACE INTERACTION,NAME=WITHFRIC
*FRICTION
0.1
```

```
*BOUNDARY
SYMM,1,1,0.0
BOTTOM,2,2,0.0
BALL,1,1,0.0
BALL,6,6,0.0
```

We use a friction coefficient of 0.1 in this study. It is very important to enforce the r-displacement component on the symmetric axis to be zero (such that the problem remains axisymmetric), and to restrain the indenter movement such that it is only able to move in the z direction without any rotations. Now define materials:

```
**
** Define Ideal Plastic Material
**
*SOLID SECTION,ELSET=ELALL,MATERIAL=STEEL
*MATERIAL,NAME=STEEL
*ELASTIC
?,?
*PLASTIC
200.0E+06,0.0
200.0E+06,0.5
```

Read ABAQUS/Standard user's manual II (inelastic behavior) to understand how to define elastic-perfectly plastic properties in the 2 data lines of the *PLASTIC keyword. Finally, we indent the sample by applying a non-zero z-displacement on the reference node of the indenter:
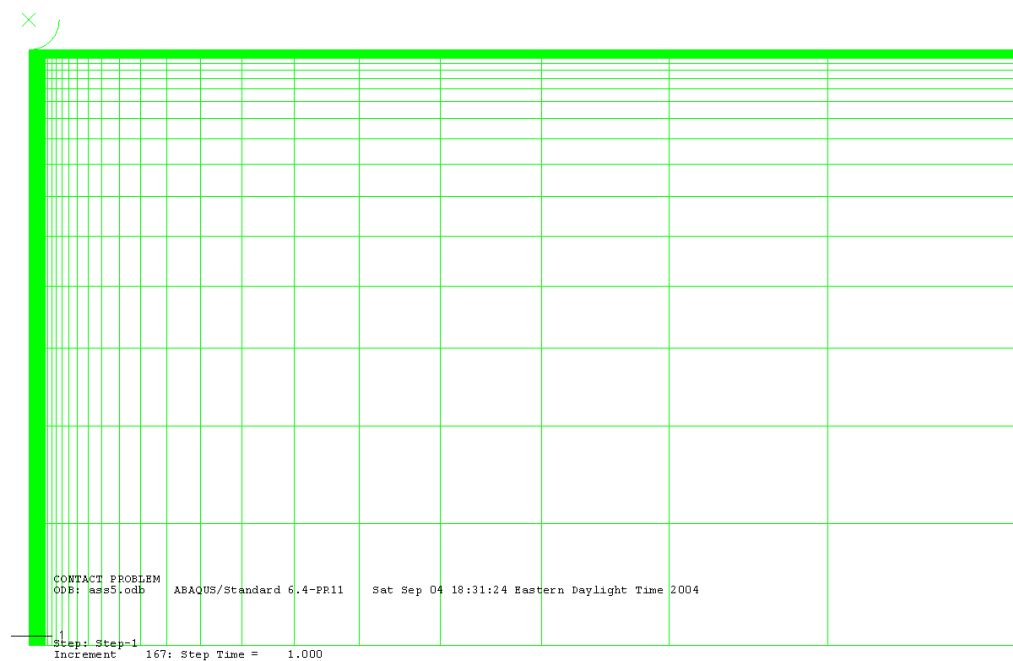
```
**
** Apply Indentation
**
*STEP,NLGEM,INC=1000
*STATIC
0.001,1.0,1E-9,1.0
*BOUNDARY
99999,2,2,-0.0001
*NODE PRINT,NSET=BALL,freq=1000
RF2
*NODE PRINT,NSET=TOP,FREQ=1000
COORD
*EL PRINT,ELSET=TOPLINE,FREQ=1000
S
*CONTACT PRINT,SLAVE=ASURF,FREQ=1000
*OUTPUT, FIELD, VARIABLE=PRESELECT
*OUTPUT, HISTORY, VARIABLE=PRESELECT
```
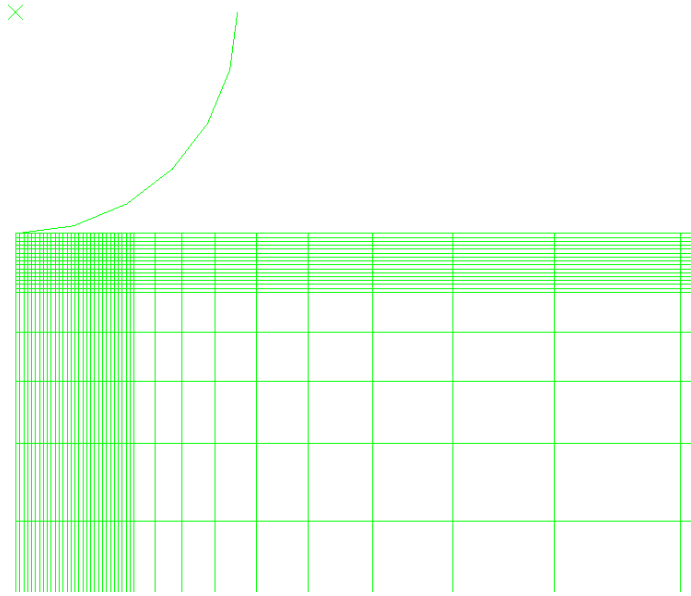
*END STEP

Remember that all contact analyses are nonlinear, moreover, plasticity is also involved, thus, nonlinear geometry must be enforced in the *STEP keyword. In the input file shown above, the mesh density and initial time increment have all been preselected, so you do not have to worry about the issue of convergence. This example illustrates that in order to get converged contact solution, very refined mesh in the contact area should be used and tiny initial increment is a must. When a contact analysis diverges, further refinement of the mesh within the contact area is always helpful (this can be easily done by adjusting the coordinates of characteristic nodes). There are many other rich aspects of the contact analysis that this assignment cannot cover.

It will take a much longer time to run compared with your previous homeworks. Print your input file and load your .odb file in ABAQUS/View. Print the deformed mesh. How much is the plastic-pile-up amount? Print the contour plot of the Mises stress. The Mises stress is a measurement of plasticity. The region where its Mises stress is above the yield stress is likely to yield plastically. Can you plot/print the plastic region? (Hint: adjust the "limits" of your contour plots from "Options" "Contour", set the upper limit to the yield stress value. Then, adjust the color spectrum, such that the color is red for all regions above the upper limit.)

Finally, let us verify the accuracy of the formula $H \approx 3\sigma_y$, How much is P (through the reaction force on the rigid indenter)? How large is a ? To determine the contact radius, look for the table "contact output" in the .dat file. Find out which substrate surface node is the last node that is still in contact with the indenter. Then, look into the "node output" table to obtain the r-coordinate of that node, and that is your contact radius. How much is H, and how accurate is $H=3\sigma_y$?

CONTACT PROBLEM
ODB: ass5.odb   ABAQUS/Standard 6.4-PR11   Sat Sep 04 18:31:24 Eastern Daylight Time 2004

Step: Step-1
Increment   167: Step Time =   1.000

The input file is sent available on the course website. Of course, these parameters are omitted in that file.

Task 2. Create in ABAQUS CAE the input file to solve the problem described above