2.31 Assignment 12 (a.k.a. The Never-ending Story)

Due Wed, Dec 5 at 9:30 am

In sheet metal forming operations, a sheet of metal is shaped into a product by plastic deformation. Typically, the sheet is deformed by forcing it to conform to the shape of a rigid die. If the final curvature of the part is not too severe, a simple process that is often selected is the stretch-forming process. In stretch-forming, a sheet is clamped by the grips of a machine, stretched, and wrapped around the die. Because of the elastic-plastic behavior of the sheet material, the deformation imposed on the sheet when it is forced to conform to the die has both an elastic and a plastic component. The elastic part of the deformation is recovered when the sheet is unloaded. This is known as springback. In this assignment we will model a stretch forming operation of an aluminum sheet over a cylindrical die. The aluminum sheet in its undeformed configuration (the blank) is a \( L=20 \text{ cm} \times W=2 \text{ cm} \) strip in 0.063” gage. The material can be modeled as elastic-perfectly plastic with elastic properties \( E=70 \text{ GPa} \), \( \nu=0.3 \), and a yield stress \( \sigma_y=300 \text{ MPa} \). The radius of the die is \( R_{\text{die}}=7 \text{ cm} \). The sheet is first stretched along its length by an axial force \( F=3.2 \text{ KN} \), then wrapped around the die by a \( \theta_L=\pi/4 \), and finally unloaded to obtain a final part with a rounded bend of radius \( R_U \) (@ the midplane), and flat edges opening up to an angle \( \theta_U \).

1) **Pen and Paper**

   a) Neglecting the effect of the axial force \( F \) (i.e., for \( F=0 \)), estimate the bending moment \( M_{0,L} \) necessary to force the sheet around a radius of curvature (@ the midplane) of 7cm: draw the corresponding strain profile through the thickness, and the stress profile through the thickness, indicating the actual magnitude of strain at the skins of the sheet, and the extension of the elastic core. If you do not remember how to do this, you can find help in the pdf file on elastic-plastic beam bending in the Announcement folder, and/or you can dig up your old 2.002 lab notes. (aaagh!)
b) Now think about the effect of the pre-stretching force \( F \). What is the strain distribution through the thickness at the end of the pre-stretching step (1)? Sketch it with its numerical value.

c) Now when the sheet is wrapped around the die as in (2), with the pulling force \( F \), what happens to the sheet will be a nonlinear superposition of the situation in (a) and (b) \( \rightarrow \) you cannot simply obtain the strain and stress profile by summing up the profiles in (a), (b). Try to come up with a reasonable sketch of the actual strain profile and the corresponding stress profile. You do not need to put in actual numbers if you cannot figure out how to obtain them. Remember that plane sections remain plane (i.e. the strain remains linear through the thickness), and that the integral of the axial stress should give you \( F \). Do you expect the actual bending moment in the loaded bent part, \( M_{F,L} \), to be higher or lower than \( M_{0,L} \)?

2) **FE model**

Create an FE model of the forming process. Due to symmetry of the geometry and loading conditions, you can limit your model to \( \frac{1}{4} \) of the actual geometry, imposing the appropriate boundary conditions along the 1-2 and 2-3 symmetry planes.

Things to keep in mind as you set up the model:

**PART:**

For the sheet metal, you want to create a 3D Deformable Shell Part using Extrusion (Approx size : 0.3). You will extrude in the depth direction, so in the sketch plane the blank profile is simply a horizontal line of length 0.1. Your life will be easier later on if you sketch your part so as to have the symmetry plane coinciding with the 2-axis and the part profile going from (0,0) to (0.1,0). Extrude the part by a depth of 0.01.

For the die, you want to create a 3D Analytical Rigid Extruded Shell. Because of the symmetry boundary conditions along the 2-3 plane, you do not need to worry about “nodes falling off the edge of the die” along that plane, and you can actually limit the die to a single quadrant. In the sketch plane the profile of the cylindrical die is \( \frac{1}{4} \) of a circle that you can center for now at (0,-0.07) and then reposition later when you assemble the model. Give it an extrusion depth of 0.02; note that this value is just used for visualization (ABAQUS thinks of this as an infinite cylinder in the extrusion direction). Create a reference point at the center of curvature (Part \( \rightarrow \) Reference Point).
PROPERTY: In material properties you have to input the Mechanical props for the sheet. Remember to input both the elastic ($E$, $\nu$) and the plastic ($\sigma_y$) properties. In section property you want to create a *shell homogeneous* section of thickness 0.0016m (0.063”). The section properties will have to be integrated during the analysis. In the *integration*... option, change the number of integration points to 11 (Simpson rule), so as to better capture the elastic-plastic stress profile as the sheet goes plastic. Assign the section to the entire shell (Part-1). Now you want to assign local material orientations, so the stress output tensor rotates with the sheet. If you want the local 1 axis ($\sigma_{11}$) along the loading direction, you first need to define a rectangular datum coordinate system: use the ToolÆDatumÆCSYSÆdefaultÆrectangular tool to create a datum coordinate system. Now you can use the AssignÆmaterial orientation tool: select the entire part, and choose the newly created CSYS. Select the Y direction (Axis-2) as the direction that defines the approximate shell normal, with 90 degrees additional rotation, and click OK to confirm the change. Use the QueryÆMaterial Orientation tool to check the material directions.

ASSEMBLY: Instance the sheet and the die. With the current geometry, the two parts are coincident along the 3-axis. To start a simulation with parts in contact is a sure way to get in trouble. To avoid problems, create a small clearance between the sheet and the die by translating the die instance down by (0,-0.001,0). Use the InstanceÆTranslate tool to reposition the die.

STEP: For this assignment, we are going only to model the loading sequence. (The unloading will be part of the weekly project). We will model the loading history through three loading steps: in Step 1 we will apply the pre-stretch load $F$. In Step 2 we will bring the sheet down to touch the die. In Step 3 we will wrap the sheet around the die. So you have to create three steps. For each step choose General Static steps, with a step time of 1, and make sure you check the nonlinear geometry (NLGEOM) option!. Under the incrementation tab choose for each step an initial increment size of 0.1. Use the Field Output Request Manager to edit the requests and change the default for the quantities written to the .odb file: add Section Forces and Moments to the required forces/reactions output quantities.
INTERACTION: Here is where you define the contact conditions. Create two surfaces using ToolÆSurfaceÆCreate. Pick as the first surface the top die surface, and as the second surface the bottom surface of the sheet. Now create a contact interaction between these two surfaces using InteractionÆCreateÆSurface to Surface Contact: make sure to create the interaction at Step 1. Select the master surface (die) and the slave surface (sheet) from the surface list available if you click the Surfaces… button next to the prompt line. In the Edit Interaction box that pops up, click the Create… button to create interaction properties. Choose type contact, and under the Mechanical tab set normal behavior to hard contact and the tangential behavior to frictionless. You are done. You should get something that looks like this.

LOAD: First thing, in the initial step encastre the reference point for the Die, and apply XSYMM boundary conditions to the edge of the sheet in the 2-3 plane and ZSYMM boundary conditions on the edge of the sheet in the 1-2 plane. These boundary conditions will be propagated to all three steps. Then apply the axial load. You will have to apply the load in step 1 as two concentrated forces in the 1-direction at the two corner points.

Make sure you click on the follow nodal rotation option, which will let the axial load continue to pull the sheet as it gets wrapped on the die. Now to set up the movement of the sheet, you need to add additional boundary conditions: In step 1 the sheet is not touching the die and it would be free to move in the vertical (2) direction. To prevent this, in the Boundary condition manager create a displacement boundary condition for step 1 on the right edge of the sheet, imposing U2=0 for Step 1. By default this BC gets propagated to steps 2 and 3. You want to change this by clicking on the corresponding boxes in steps 2 and 3. For step 2, choose to edit this condition to impose a vertical displacement of the edge toward the die to bring the sheet and the die in contact: Edit U2 to read –0.002. For step 3, simply click on the Deactivate tab so as to make this boundary condition inactive. Finally, to force the wrap, create an additional boundary condition on this same edge at Step 3, imposing a rotation about the 3 axis UR3=–π/4. Done.
MESH: Follow what you get by default: (0.01 seed, SR4 elements). Mesh the sheet.

Submit the job: note the CPU time from the .dat file. Give a look to the .msg file to see if you have some clue of what’s going on.

Look at the results in the VISUALIZATION module of ABAQUS/CAE:
Plot and print contour plots of the following quantities at the end of step 3: S11, SF1, SM1, UR3, CPRESS.

[Very good advice: By default the visualization module shows an extruded rigid cylinder (the die) that goes from here to New Zealand. You should click off the default extrude elements tab (so that you see only the die profile in the sketch plane) that you find under View ➔ ODB Display Option ➔ Sweep and Extrude]

S11: For what surface is S11 plotted? Is what you see consistent with what you were expecting? Why is the stress higher than the yield stress?
SF1 and SM1: are the section forces and moments consistent with expectation? Is SM1 in the right ball park based on the pen and paper calculations?
UR3: did the BCs work to obtain a flat flap at 45 degrees in the loaded configuration?
CPRESS: Why does the contact stress decrease from the 1-2 symmetry plane to the outer edge of the sheet?

© LIFE THE UNIVERSE AND EVERYTHING: what were you thinking when you signed up for this class?