A Technique for Elastomer Extrusion Bending to Detect the Onset of Kinks

Frank Pfeiffer, FEA Engineer
Cooper-Standard Automotive Division
Cooper Tire and Rubber Company
Farmington Hills, Michigan

ABSTRACT

A major failure mode of extruded vehicle seals is a waving or puckering of the sealing lip or bulb after the seal has been installed around a tight radius. Of the many available techniques for modeling the bending of an extrusion, most constrain the bend to one prescribed radius. This technique incrementally steps the extrusion through a decreasing radius, allowing us to detect and thus predict the onset of a failure in the part.

Why

As little as five years ago, finite element analysis was a luxury in the tier-one automotive supplier market. Hardware was slow, algorithms were long and wall time was rounded to days and hours. Today, FEA is on the critical path; nothing gets out of the design centers before being blessed by FEA engineers. Teams of design engineers gather around the FEA tube to tweak designs, demanding results in minutes. Managers want to see the next step, and 3D is that uncharted territory. A simple 2D analysis will not show the same failure modes that occur in 3D, and now that computing power is catching up with our imagination, we can explore techniques that used to be time-prohibitive. The advantage this has over other techniques is that we can see a failure that occurs just beyond our service radius, something we would have missed using conventional techniques.

What

This is one technique for bending an extrusion of constant cross-section to show the onset of kinks. What sets it apart from most bending techniques is that it allows a controlled, incremental bending of the entire part, without requiring any advanced math. To do this we will first bend the part to a large, uniform radius, then decrease that radius incrementally until the part fails.

How

The first step in making this or any bending model is to determine the neutral bending axis. In many cases this axis can actually float around, depending on the design of the carrier holding the part onto a flange, and the severity of the bend. Here we want to fix the bending axis, just to keep things simple. At the neutral bending axis we will want to draw a line. The model will be built up from this line, so that the axis is preserved as a flat area of the part itself. In meshing the 2D model, make sure that the mesh captures any features that might affect the bending, but don't make the mesh too coarse. Once we extrude into 3-space, a coarse mesh can become very expensive.

To prove out our technique we will make up a simplified 5-cube extrusion. Draw a 4-node quad that measures 10mm by 10mm. On the side where we will assign the bending axis, add one 2-node element. When we expand the elements, this will give us a layer of quads at the neutral bending axis. In the expand menu, we want to go 10mm in the z-direction, and repeat 5 times. We now have a row of cubes, with one surface of quads.

The point of adding a surface of quads at the axis is that we can assign them to have a very stiff material property and no hinge degree of freedom. This will create a thin membrane that will resist compression/extension while allowing uniform bending to occur.

The real trick to this technique comes next, when we select all of the nodes of the quad elements and expand them down 100mm in length. These one-dimensional elements will be extremely stiff "pins" which will create the initial radius when gathered together at their free ends. We will build a curved, rigid "scoop" to catch the end nodes of the pins. A half circle expanded to create a surface will do.

Three different boundary conditions should be set up: one each for zero displacement in x, y, and z. Anchor the pins to keep them from going out of their bending plane, and anchor one end of the brick elements at the bending axis in x, y, and z. Anchor the rest of the nodes on that end of the part to keep them in their plane. This will eliminate edge effects at this end of the part, while not leaving the part completely cantilevered there. The other end of the part will serve as a free end, where we will be able to observe edge effects.

We really have three distinct materials here that we need to model. The first is the rubber for the bricks. For that we can use a simple Neo-Hookean model with a C10 of 6. The second material will be for the membrane elements at the neutral axis. For that we'll use steel: isotropic with Young's modulus of 190,000 and Poisson's ratio of 0.3. The third material will be for the actuator pins: isotropic with Young's modulus of 1E10 and Poisson's ratio of 0.3. This will keep them very stiff and ensure that they keep the length we want them to have.

Next we need to assign special geometric properties to these groups of elements, to get us the degrees of freedom we need. Our neutral bending axis will not work very well if the elements are free to rotate around their shared nodes. Create a new geometric type and label it "axis." From Mechanical Elements, 3-D, choose Membrane and give it a thickness of 1, normal to the plane. Add all of the 4-node quads.

Create a new geometry type and call it "actuator." From Mechanical Elements, 3-D, choose Truss and assign a cross-sectional area of 0 and turn on the actuator toggle. Give the actuator a length of 100, and we will write a table to assign to this length. Back out and get to a tables menu. Name the new table "act." This table will be a combination step and ramp table that will tell the actuator pins to change length. Add points {0,1}, {1,1}, {6, 0.75}. This should be sufficient to see the actuators do their stuff. Be sure not to begin your table with {0,0} or things won't turn out quite as we would like. Go back into the truss menu and assign this table to the actuator length, then select all of the 2-node elements for this geometry.

Contact bodies will only need to be the scoop and the pins, as the part itself never contacts anything else. Cbody1 will be deformable, and will be all of the 2-node elements. Cbody2 will be the rigid surface for our scoop, assigned an initial velocity of 1 [in z, the direction of the pins] and an overall velocity of 50 in the z-direction and 5 in the y-direction. We will make tables to turn these directions on and off when needed. The table for the z-direction should step to one for the first second and then stay off for the remainder of the solution. The gathering of the pins will thus take place in one second. Again, be sure not to use {0,0}, but give this table points {0,1}, {1,1}, {1,0} and {6,0}. The y-direction table should do just the opposite. Give it points {0,0}, {1,0}, {1,1) and {6,1}. Assign these tables to the appropriate velocities. Write a contact table so that the two bodies will glue to one another with a separation force of 1E10 and a distance tolerance of 0.1.

Everything should be about set to run, but first we have to give loadcases. We will need two loadcases, so we can turn off the y-fix boundary condition when we want to shrink the actuators. Loadcase 1 should be a static loadcase with total time of 1 second, and give it 50 steps to get there. Select loads x-fix, y-fix, and z-fix to be on during this loadcase. In Convergence Testing Criteria, change the default to Displacement. In Contact, call up the contact table ctable1.

Loadcase 2 will be static with time of 5 seconds and 5 steps. The actuators will shrink by 5mm during each step of this loadcase, as prescribed by both the actuator length table “act” and the y-direction table controlling the position of our scoop. For this loadcase, turn off y-fix, and again select the Displacement testing and ctable1.

In Jobs, Mechanical, select lcase1 and lcase2. Choose Large Displacement under Analysis Options, and make sure that all three boundary conditions are set under Initial Loads. In Contact Control, Initial Contact, choose ctable1. For the membrane, we will use element type 18, and the bricks can be element type 120. The actuators should use element 9.

The results are very easy to interpret. The length of the actuators are the length of the radius of the bend at any given time. If the model runs fairly well while scooping up the actuator ends, but crashes during the shrinking phase, check your motion and length tables. If some of the actuators fail to shrink as prescribed, check to see that they have a zero cross-sectional area.

Acknowledgements

I would like to acknowledge Mr. Felipe Hüls of Henniges Elastomerfabrik, Gmbh. in Rehburg, Germany for helping me develop this technique three years ago. Herr Hüls is an outstanding MSC.Marc user and a brilliant engineer.