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Unwrapping Round Structural Members with SolidWorks (Rev 1)

This applies to a frame that has been modeled using weldments and trimmed with the trim/extend feature. Each member is then saved into its own part by right-clicked the tube in the feature manager and selecting “insert into new part”.

1. If you open up the tube that you want to unwrap you will notice that it is in the same coordinate system as your main frame file. We need to add a plane going through the tube centerline (the orientation is not important). To do this, add an axis through the centerline of the tube by going to *Insert→Reference Geometry→Axis* and selecting the outer surface of the tube. To create the plane we need an axis and a point. Go to *Insert→Reference Geometry→Plane* to create the plane through the centerline. Select the axis and select a point on the end of the tube to make the plane. If you have no sharp corners where your tube is trimmed you may have to go back and add a point *Insert→Reference Geometry→Point* and selecting the curved trim line.

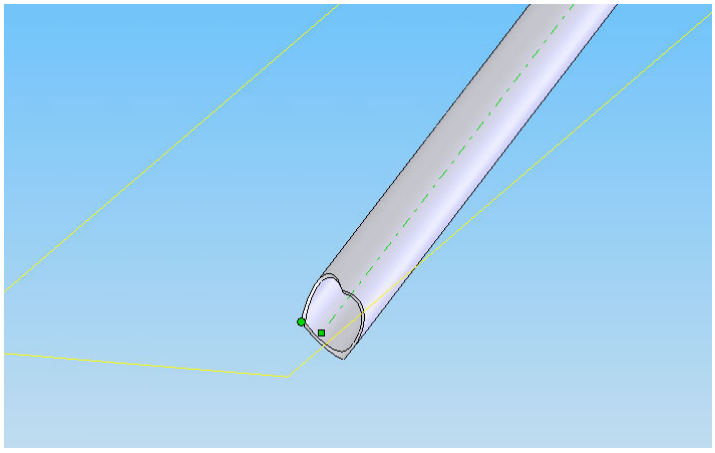

Figure 1.

2. Next create a sketch on the plane just created. Draw a line approximately through the centerline. Hold the Ctrl key and select the line and the axis. In the properties dialog under add relations select collinear. You can also do this by going to *Tools→Relations→Add*. Finally drag the endpoints of the line so they extend past the tube on both sides. Figure 2 shows how this should look.

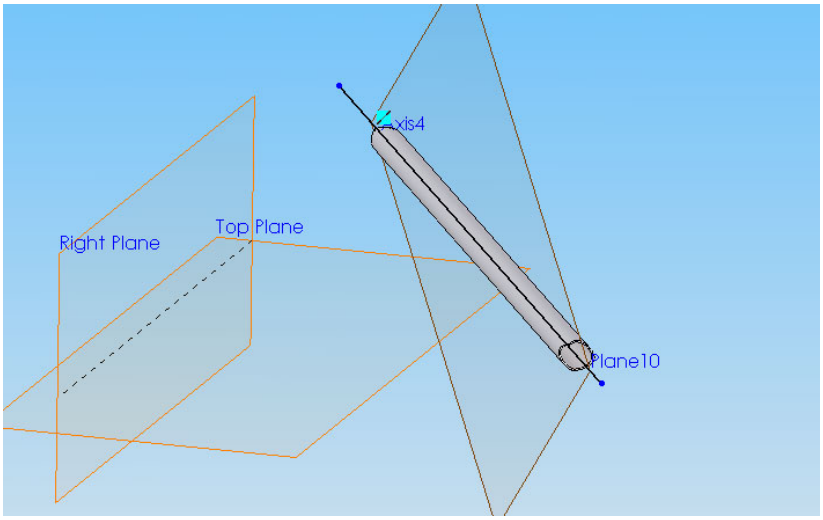

Figure 2.

3. Under Features select Extruded Cut. Select the line previously drawn. Mine defaulted to extruding both directions but just have it go in one direction. Finally select *Thin Feature* in the properties dialog and enter a thickness of 0.001in to slice the tube. Figure 3 shows how this should look before accepting. This will leave you with a small slit on one side of the tube.

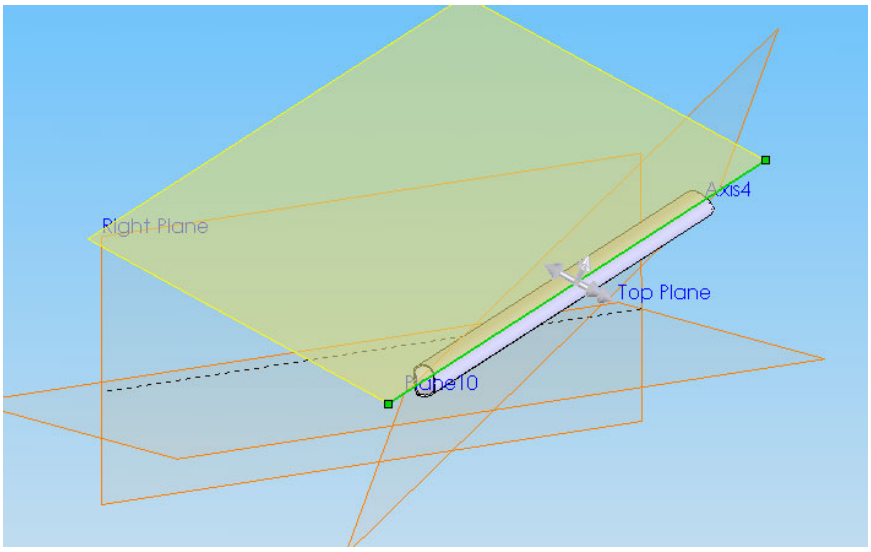

Figure 3.

4. Now to unwrap the tube. Go to *Insert→Sheet Metal→Bends* and select the edge of the slit just created from the inside of the tube as shown in figure 4. It is <u>important</u> to use the inside edge as the outside will lead to tubes that are too long as you are not accounting for the wall thickness. Under *Bend Radius* enter the OD of your tube, and under *Bend Allowance* use *K-Factor* set to 1.

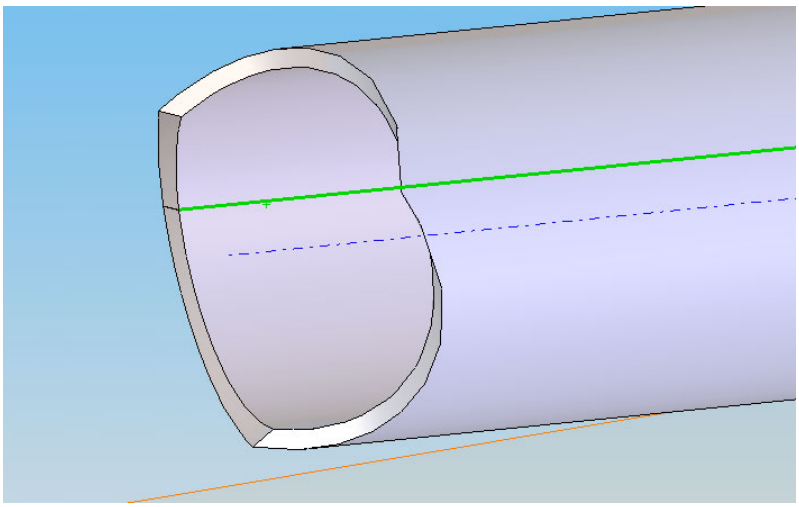

Figure 4.

5. To unwrap your tube, simply suppress Process-Bends in your feature manager. Create a sketch like the one shown in Figure 5 to allow you to orient your ends properly. Now under *Standard Views* select *Normal To* and select the surface of your tube. Make sure you're plane is underneath the unwrapped tube. Use the arrow keys if you need to rotate

to verify this, this allows you to go back to the same spot. As a tip you can hold the shift key and use the arrows to make 90 degree rotations.

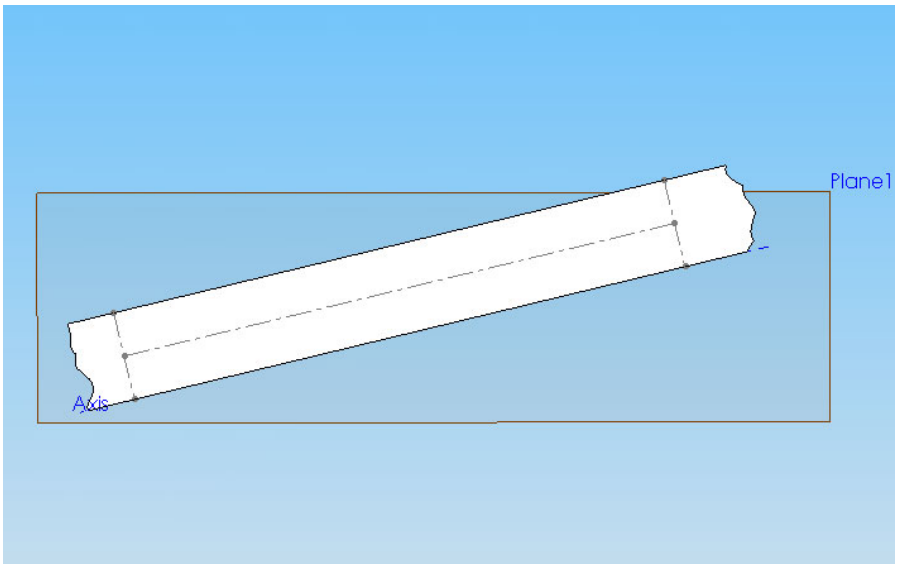

Figure 5.

6. Make a drawing from your part and use the current model view. Make sure it is scaled 1:1, and add a second one. Throw in a note on each side about the distance between the tubes and you are done.

Figure 6.

TIP: If your tube is symmetric side to side then go back into the part change your view so you are looking normal to the inside of the tube and create another drawing. Slicing is only necessary once.

Also this process can be automated by recording a macro.

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7. Cut out the templates and wrap them around the tube. Line up the template centerlines and space the lines perpendicular to the tubes centerline the correct distance apart. Using a piece of bar-stock, placed up against the tubing, with ½ the diameter of the tubing makes lining up the templates very easy. A bench grinder was used to grind away the tubing until it matches the template. An angle grinder was used for anything the bench grinder could not get to.