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Suggested Technique for Creating a Helical Coil Around a 3D Pipe

Introduction

This technique will allow a helical coil to be placed around a pipe which bends in three dimensions. First the center of the pipe will be located with a regular swept surface, the helical shaped surface will be created using a Variable Section Sweep with the trajpar parameter, and finally the helical coil will be created using a regular sweep referencing edge of the helical surface. This technique can be applied to any geometry requiring a 3D helical coil.

Procedure

1. Create a new part in the piping assembly for the coil part. With the assembly window active, select **Component, Create, Part, Solid**, and enter a name (such as "coil"). Then select Empty and press OK. Select **Done/Return** from the COMPONENT menu, **Modify, Mod Part**, and select the coil part from the Model Tree.
2. Create a surface copy of one half of the pipe geometry. Select **Feature, Create, Data Sharing, CopyGeom** and define the Surface Refs element. Then pick surface one half of the solid pipe. Select **OK** from the COPY GEOMETRY dialog box when complete. The edges of the surface are yellow in the image below.

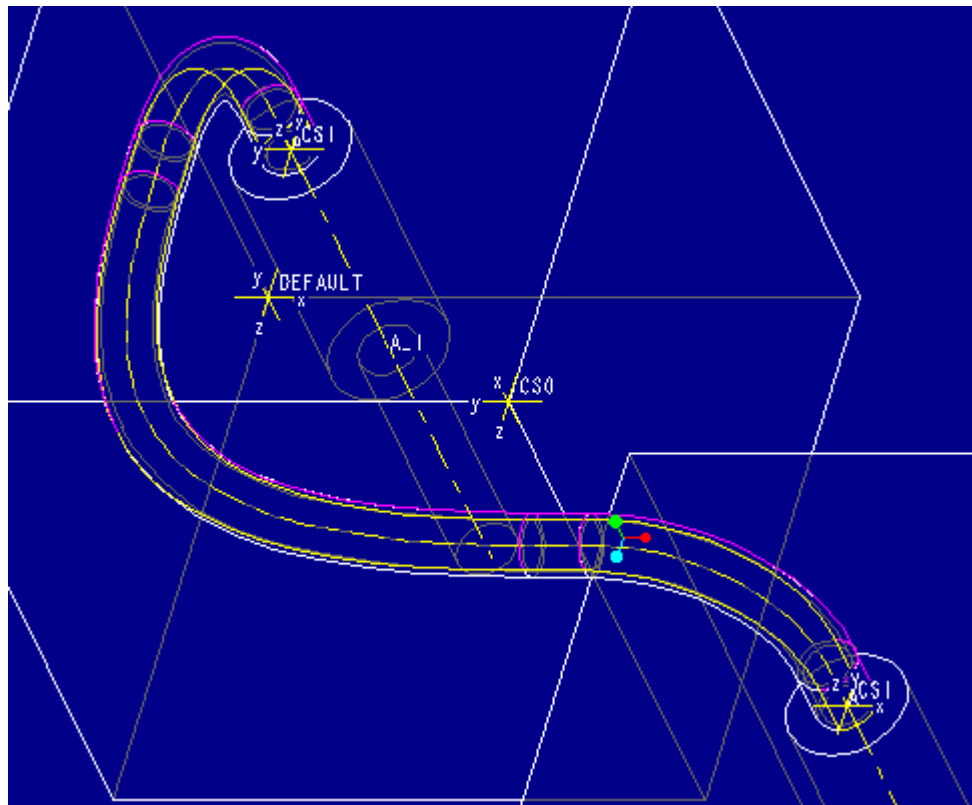


Figure 1

3. Open the coil part into its own window, then create a surface using a regular sweep which will locate the center of the pipe. Select **Feature, Create, Surface, New, Sweep, Done, Select Traj**. The trajectory should be selected from the edge of surfaces created in the previous step as shown in the image below.

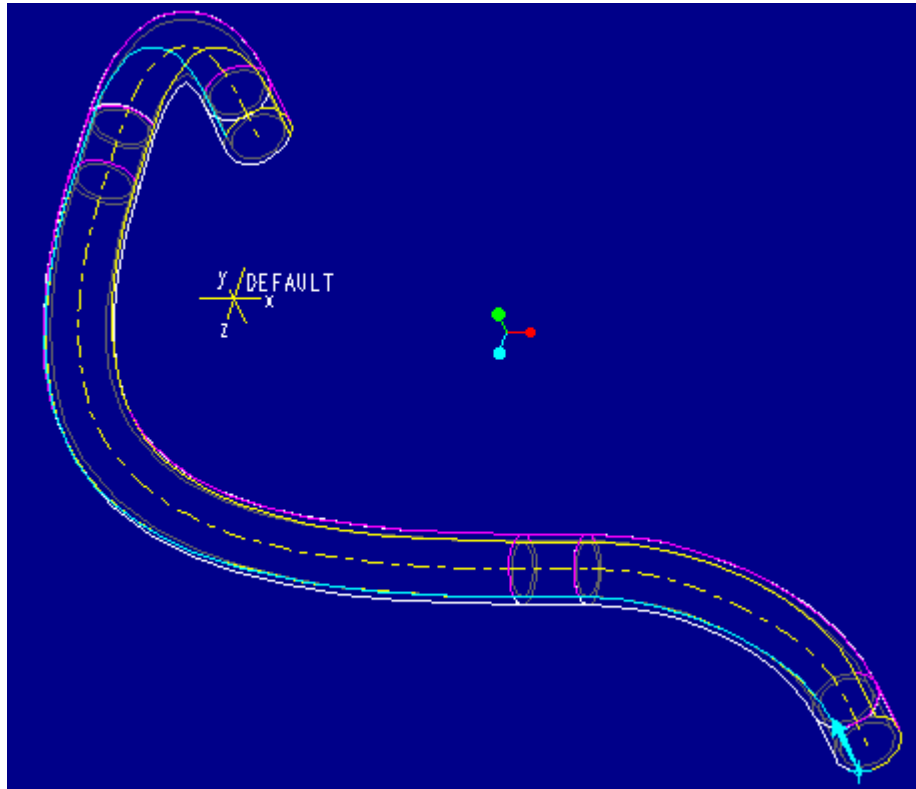


Figure 2

4. Select **Okay** when prompted "Select upward direction of the horizontal plane for sweep section." and then **No Join, Done, Open Ends, Done**. The sketch view will rotate into view. For the sketch, create a straight line starting at the cross-hairs and extending half way into the pipe. Dimension the length of the line to be the same value as the radius of the pipe. There are no references or alignments required since the line will automatically snap to the cross-hairs.

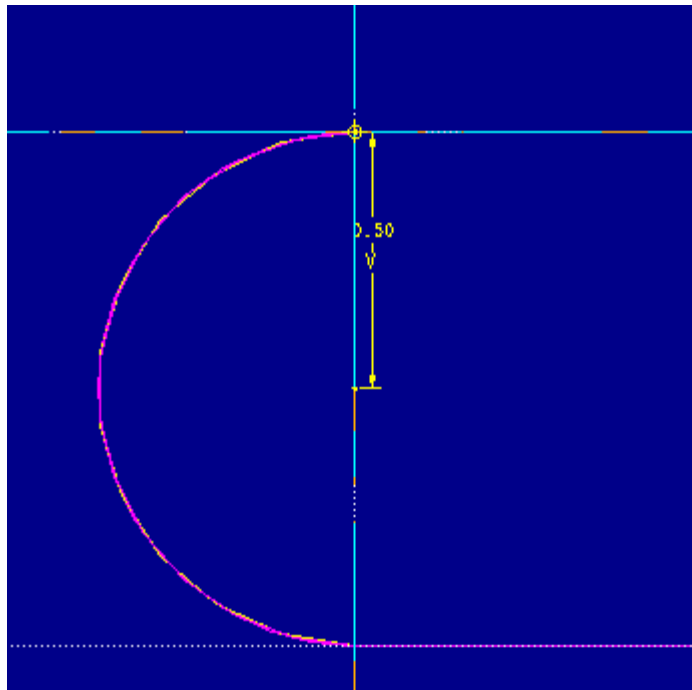


Figure 3

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5. After successfully regenerating the sketch, select **Done**, then press the OK button in the SURFACE:Sweep dialog box. There should now be surface whose edge is the centerline of the pipe.
 6. Before creating the helical sweep, a datum point will be created on the trajectory to limit the length of the coil. First create a datum curve which the point will be placed. Select **Insert, Datum, Curve, Composite, Done, Exact, Done, Tangent Chain**, and select the edge of the surface locating the center of the pipe. Complete the curve pressing OK from the CURVE:Datum Curve:Composite dialog box:
-

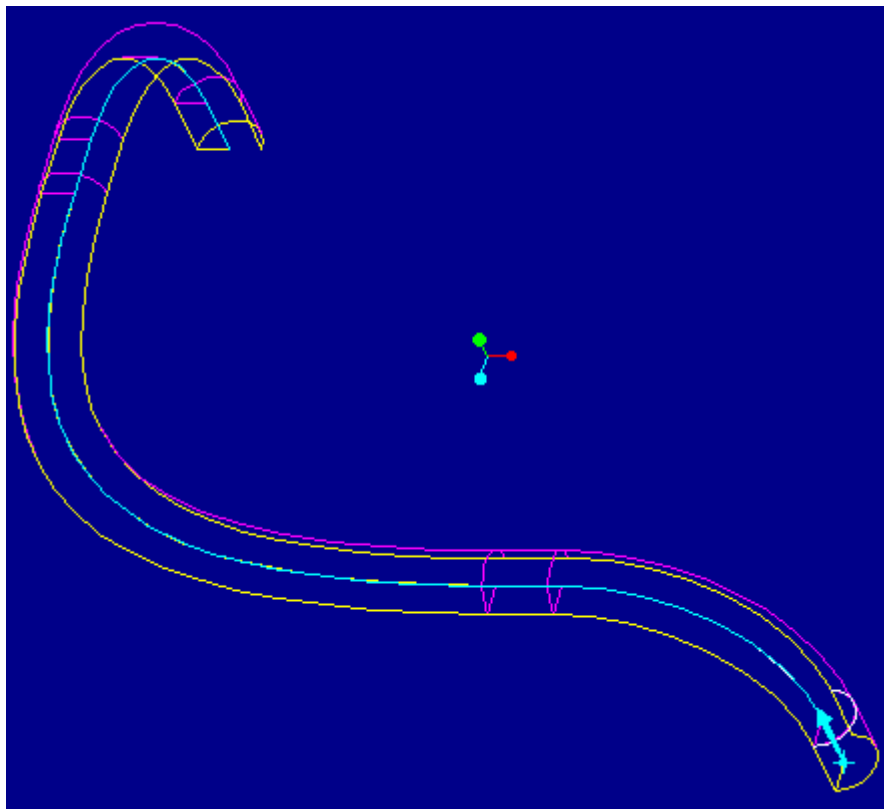


Figure 4

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7. Place the swept surface on a layer and blank the layer to prevent it from becoming selected for later features.
-
8. Next create the point on the composite curve defining the length of the coil. Select **Insert, Datum, Point, On Curve, Actual Len.** and use **Query Sel** to select the entire composite datum curve (make sure the entire curve highlights.) The value entered will be the arc length along the pipe feature's centerline which will determine the length of the coil.
-
9. Create a split curve to keep part of the composite curve on one side of the datum point. Select **Insert, Datum, Curve, Split, Done, Query Sel**, pick the entire composite curve (make sure the entire curve highlights), and then pick the datum point. Keep the portion of the curve to one side of the point.
-
10. Now the helical swept surface using variable section sweep and the trajpar parameter can be created. Select **Feature, Create, Surface, New, Advanced, Done, Variable Section Sweep, NormToOriginTraj**. First select the origin (spine) trajectory by selecting the entire split curve. Select **Select Traj, Curve Chain, Query Sel**, pick the curve (make sure the entire curve highlights), **Select All**, and **Done**.

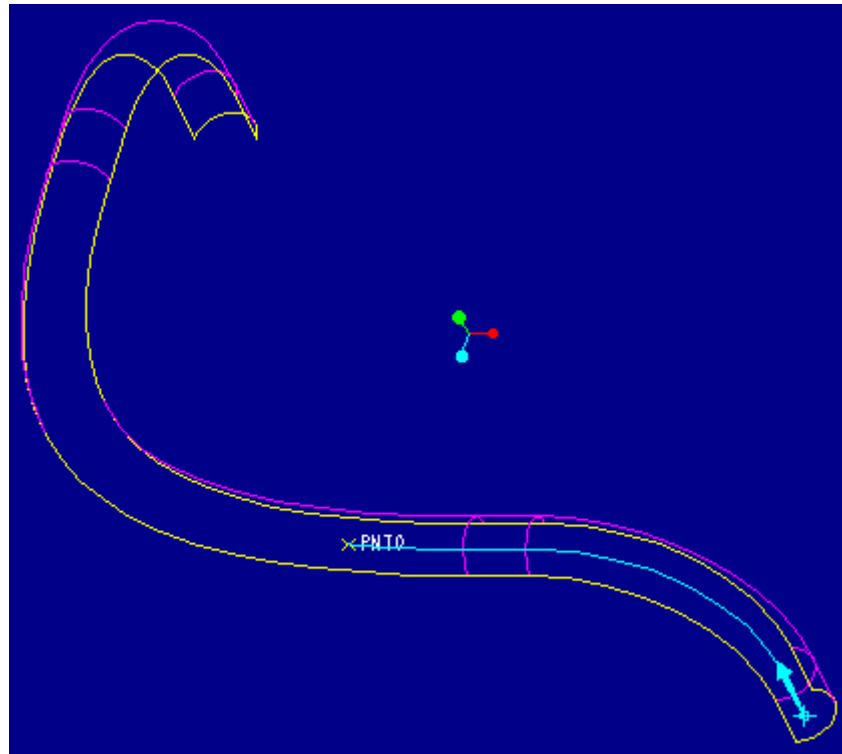


Figure 5

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11. Next when prompted to "Specify the trajectory defining the section horizontal vector." select the same edge of the surface as in [step 3](#). The entire edge can be selected then select **No Join**.
-
12. Select **Done** when prompted "You may specify another trajectory or select "Done"" since no more trajectories are needed, and select **Open Ends, Done, Origin Start**. Then the section for the sweep will spin into view. Sketch a line starting at the origin trajectory (represented by the little blue "x" at the center of the cross hairs) which goes up and to the right at an angle. Create a dimension for the length of the line and for the angle of the line from the horizontal. Change the length of the line to be the radius out to the centerline of the coil (the radius of the pipe plus the radius of the coil) and change the angle to be 30 degrees.

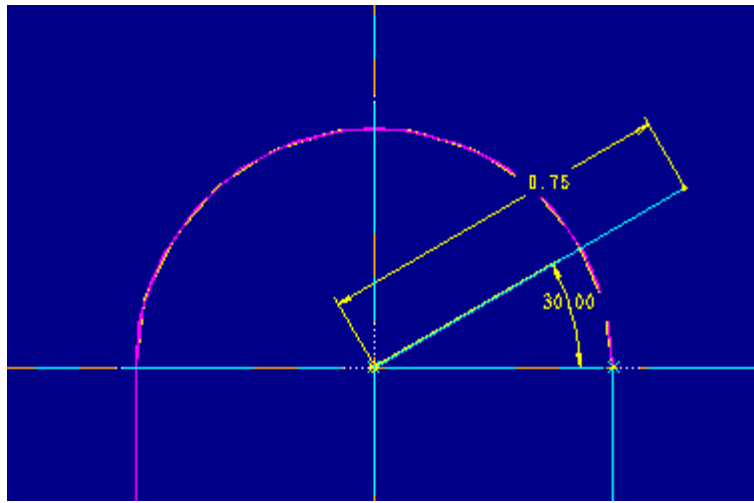


Figure 6

13. Now create a relation using `trajpar` to define the helical sweep. To create a helical sweep there must be rotation and translation occurring at the same time. Sweeping section plane along the trajectory will provide the translation component, so a relation must be created using the `trajpar` parameter to control the rotation component. relation which must be created will define the change in angle of the sketched line. In symbolic form the relation is:

$$\text{angle_sketch_dimension} = \text{trajpar} * 360 * N$$

trajpar is a parameter which changes from 0 to 1 as the sweep progresses along the origin trajectory
N is the number of complete revolutions (360 degrees) which the helix will make

In this example the relation is:

$$\text{sd4} = \text{trajpar} * 360 * 5$$

Notice that when the section regenerates the line rotates to 0 angle because the value of `trajpar` is zero at the start of the trajectory. After the section regenerates successfully, select **Done**, then **OK** from the SURFACE:Variable Section Sweep dialog box. The resulting spiral surface feature is shown in the following image:

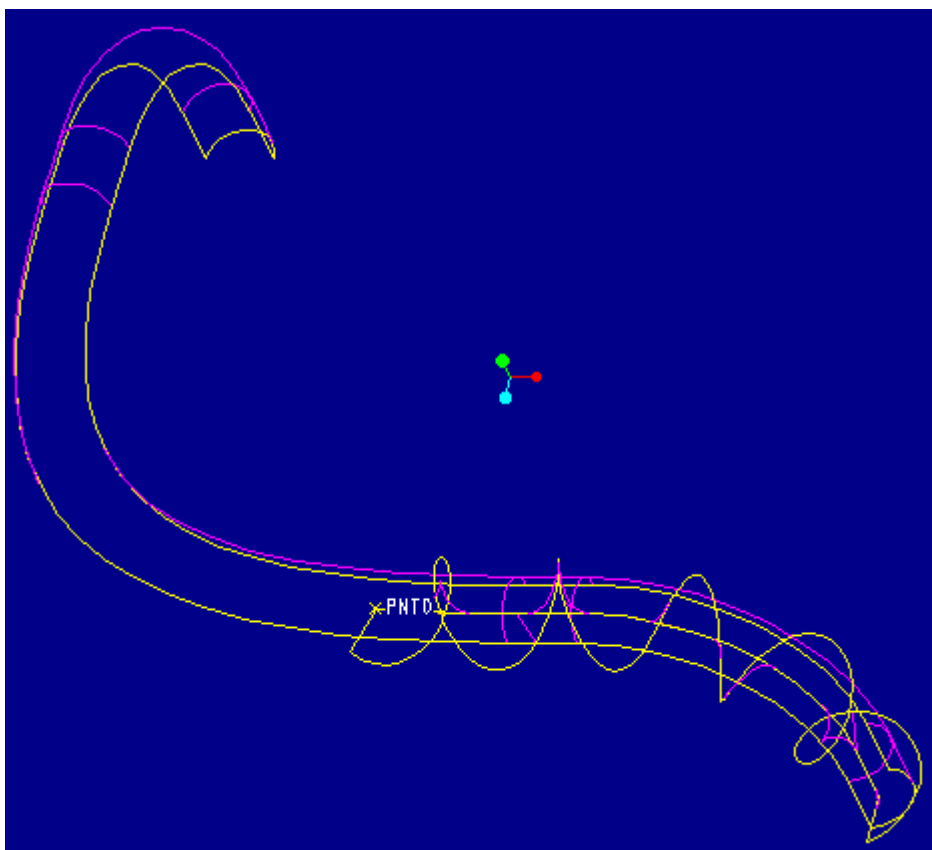


Figure 7

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14. Next create the actual coil protrusion using a regular sweep and selecting the edge of the spiral surface. Select **Feature, Create, Protrusion, Sweep, Thin, Done, Select Traj, Boundary Chain**, and pick the spiral surface. Then select **From To** and select the vertices defining the beginning and end of the edge of the spiral surf. Select **Done** from the CHAIN menu.

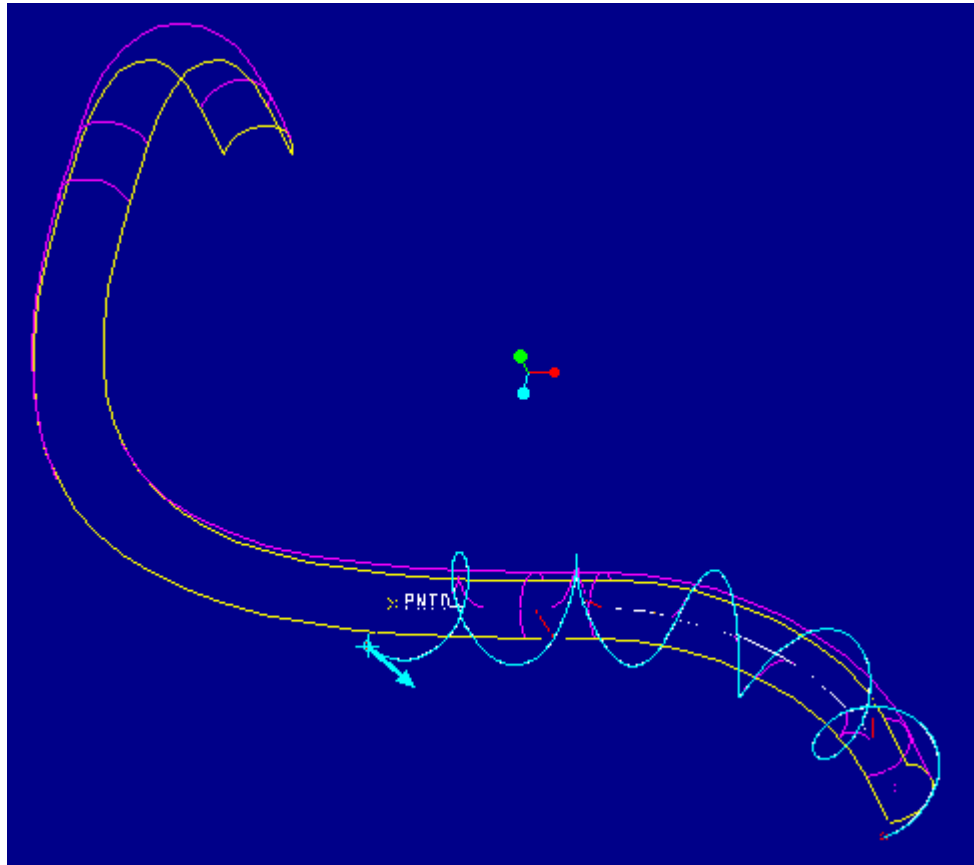


Figure 8

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15. Select **Okay** when prompted "Select upward direction of the horizontal plane for sweep section." The section for the sweep will now spin into view. Sketch a small circle representing the outer diameter of the coil centered on the cross-hairs of the sweep trajectory.

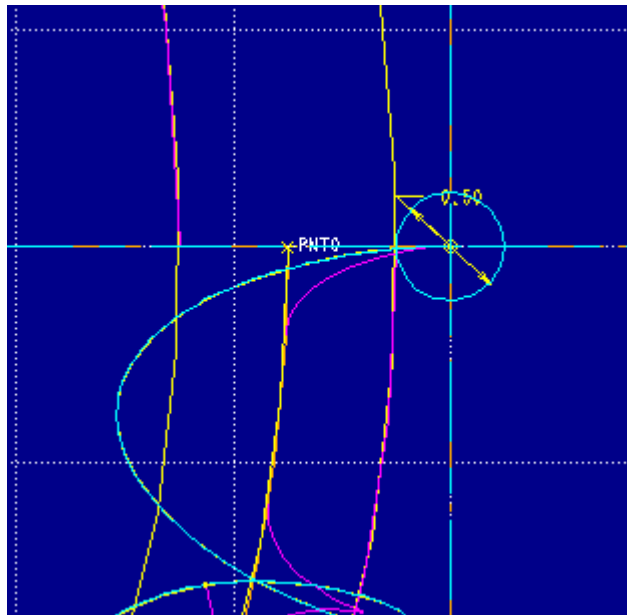


Figure 9

16. Then proceed to add thickness to the inside. After the feature is completed it will look like this:

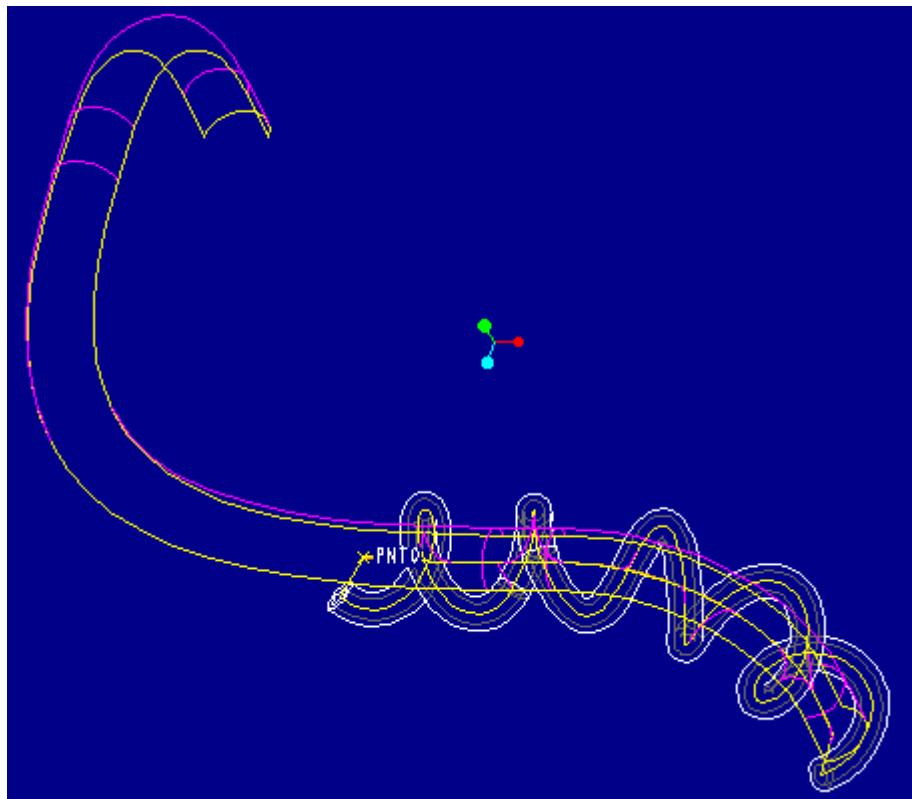


Figure 10

17. Modifications which can then be performed include:
1. the number of rotations the spiral will make by changing the number of rotations parameter in the spiral sweep
 2. the radius of the spiral centerline by modifying the dimension in the spiral sweep
 3. the diameter and thickness of the coil protrusion
 4. redefining the surface features to add or remove surfaces to change the length of the coil

After making some of these modifications and placing the construction surface features on blanked layers, the final assembly can look like this:

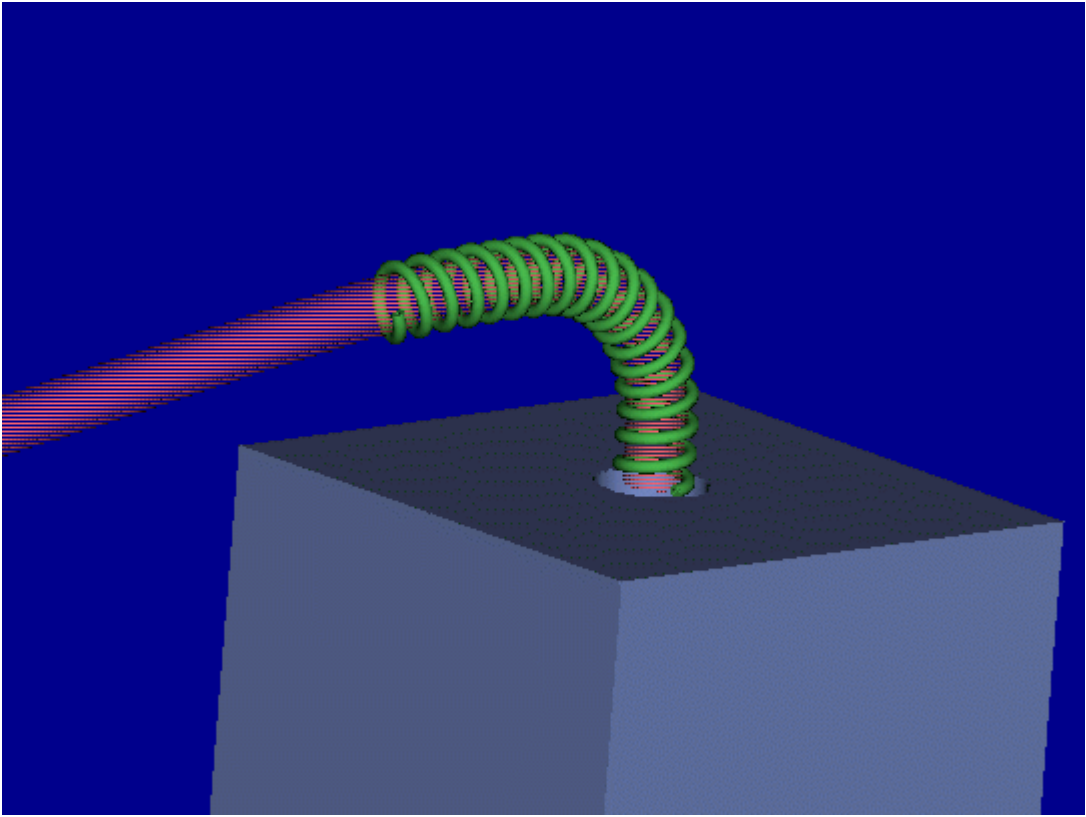


Figure 11

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