

Creating Cables Using Creo 3.0 Style Feature

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1.0 Introduction

The style function in Creo can be used to quickly display cables, or create custom curves to use in swept features. Following are the steps required to create a cable between two connectors. Using assembly references, a curve created using the style feature can also allow you to quickly update the shape and location of a cable, as well as adjust the profile on-the-fly.

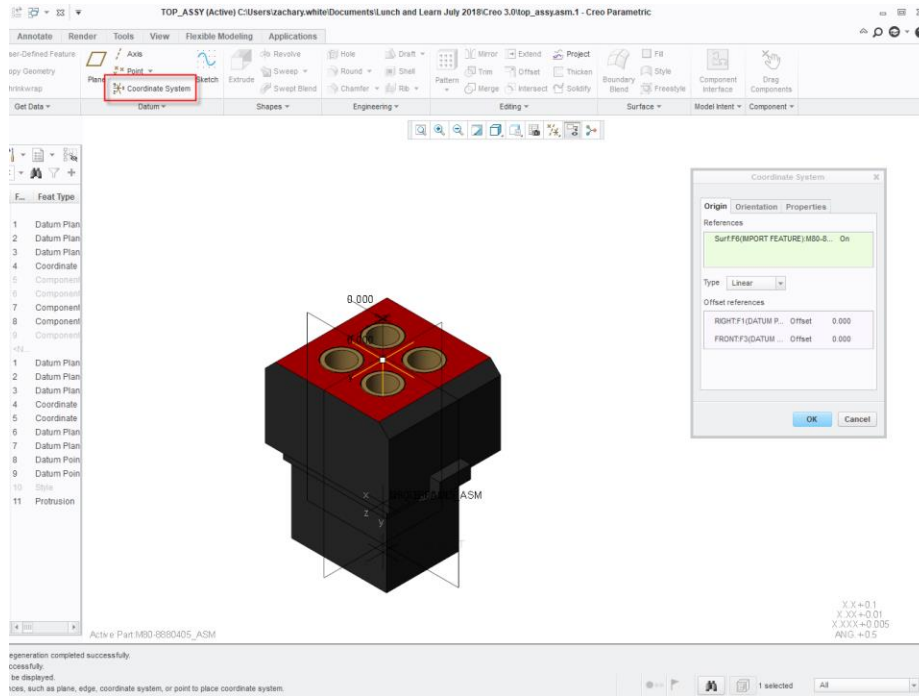
2.0 Cable Creation

In this example, we will route a cable between the two connectors.



2.1 Create Coordinate Systems

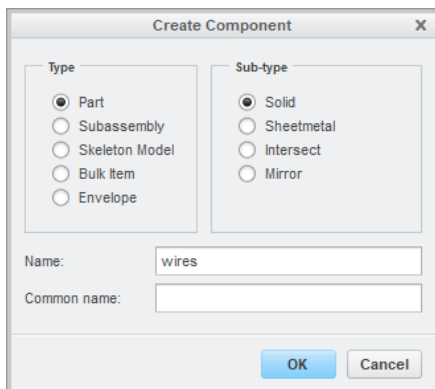
In each connector create a coordinate system to act as a reference start and end point of the curve.



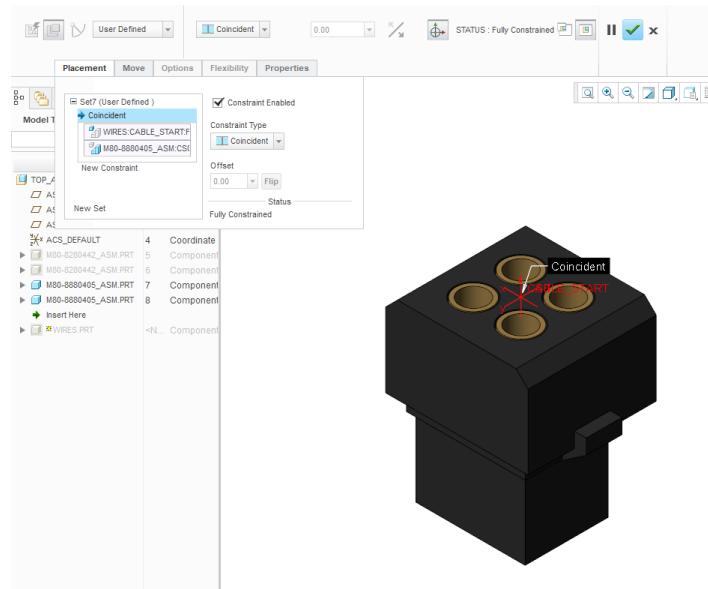
2.2 Create Wire Part

With both connectors shown in the assembly and the assembly active, create a new part which will represent the wires in the connector assembly.

Select: *Model>Create>Part*



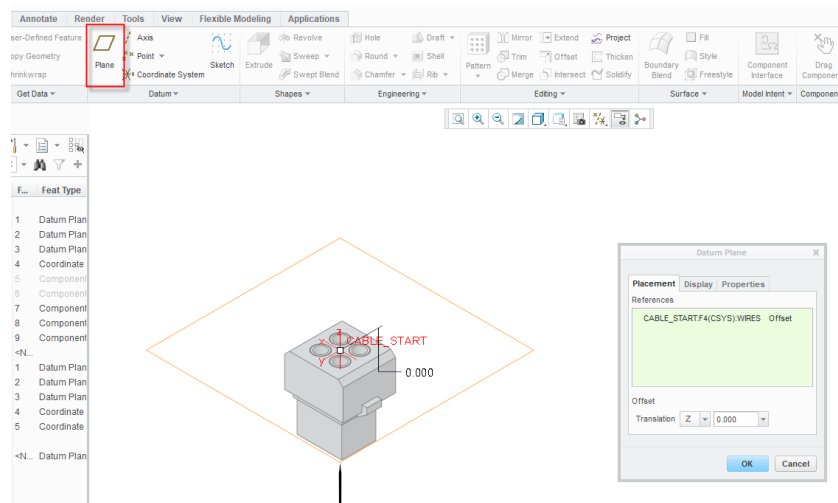
Constrain this new part in the assembly using a coordinate system reference. Select the default coordinate system of the new part and the coordinate system of the connector or assembly.



2.3 Create Reference Features

With the wire part assembled, any further part editing should be done in the assembly with the wire part activated. (*Select: Right Click the part>Activate*)

Next, create reference planes parallel to the back side of the connector, or normal to the direction in which the cable will exit the connector. Do this for both connectors.



The last reference feature needed is a datum point at the start and end of the cable. Creo will not allow the ends of a style feature to reference a coordinate system, but placing a datum point at the “start” and “end” coordinate system will solve this problem.

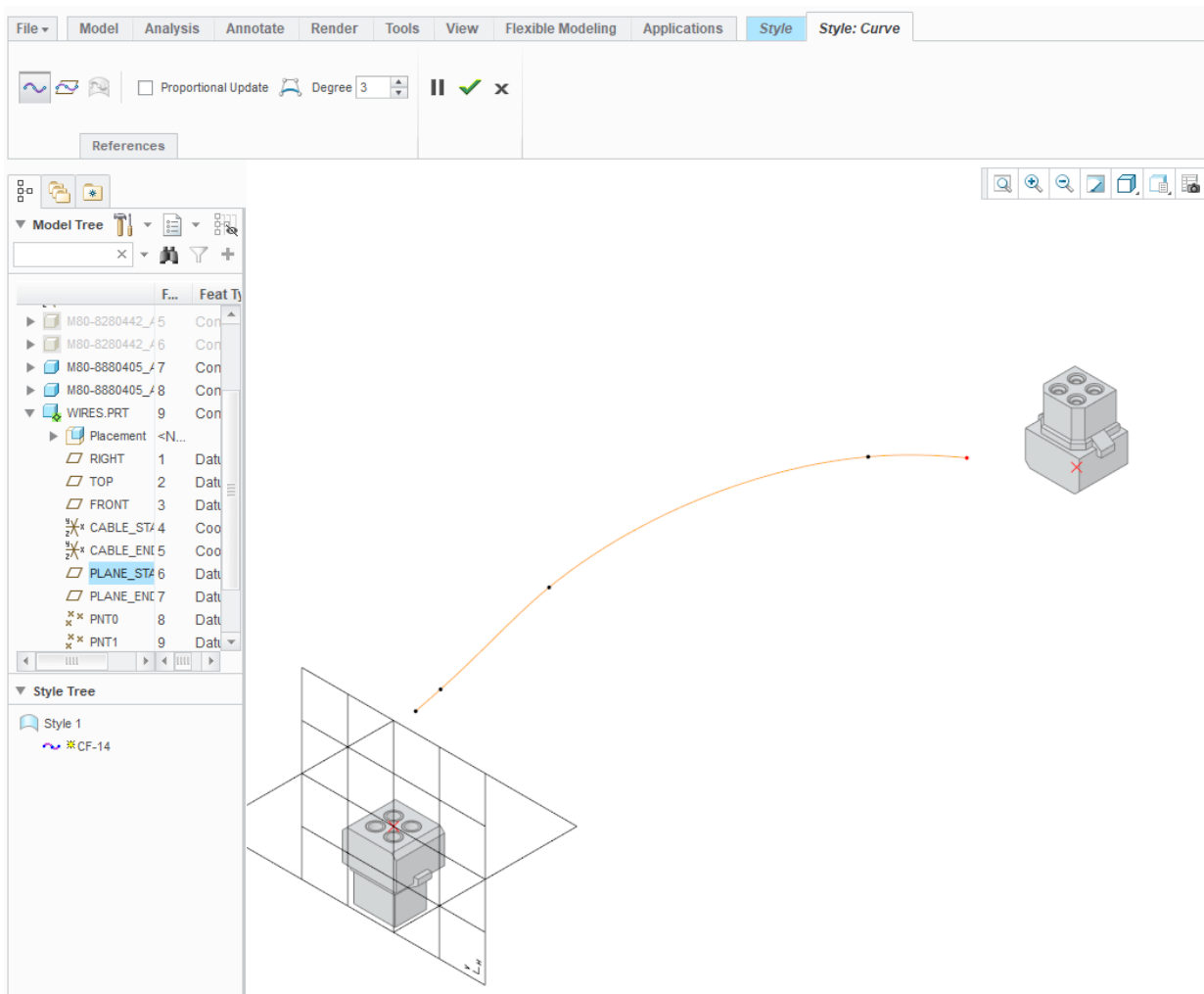
Tip: Re-name your reference features to make future selection in the tree easier (example: CABLE_START, CABLE_END).

2.4 Style Feature

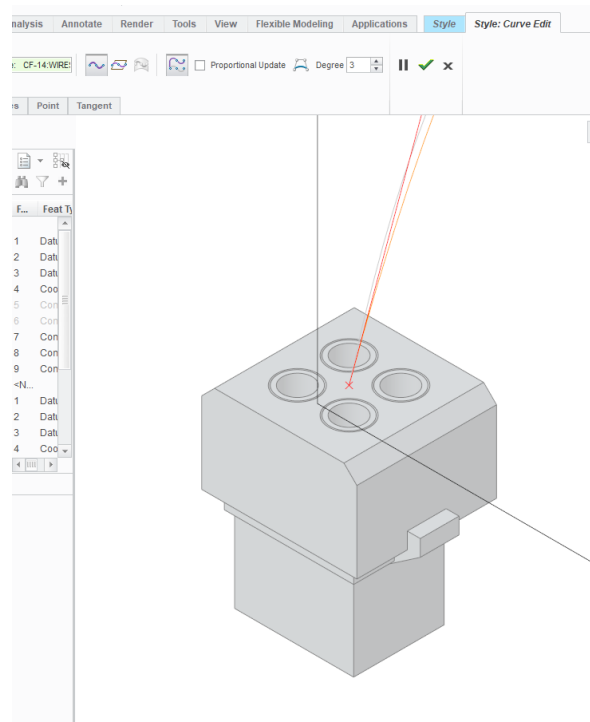
With reference geometry created and the cable part active, now we can create the style feature.

Select: Model>Surface>Style<Curve

Start by clicking a few points between the two connectors, don't worry about exact placement, this will be adjusted.



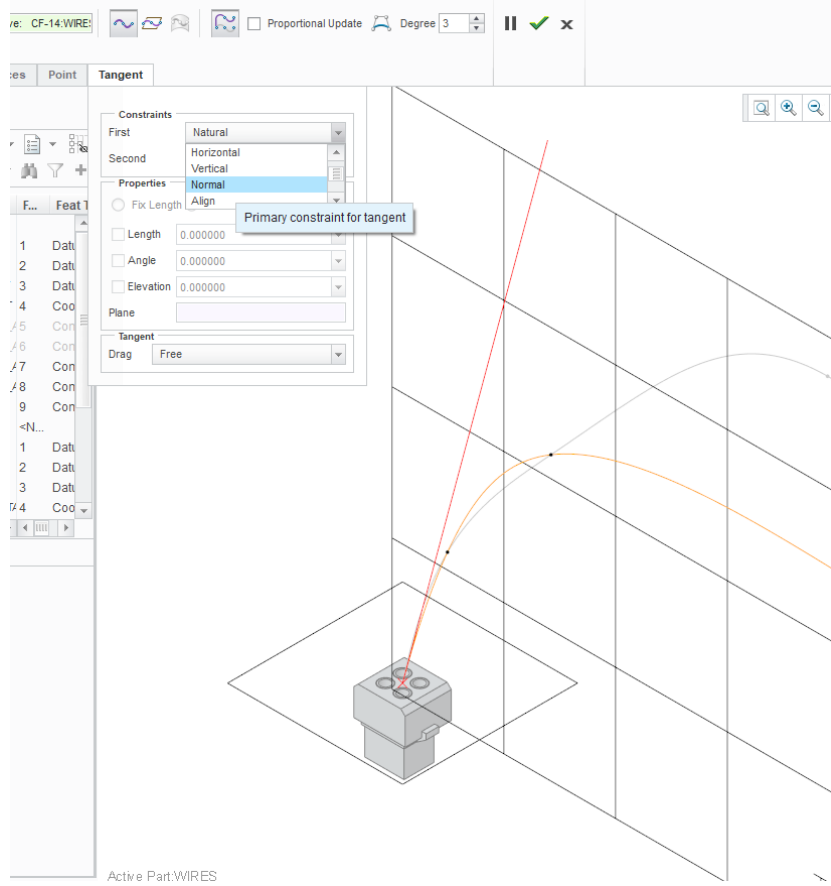
The next step is to constrain the end points. With datum points shown, select the “curve edit” option. While holding the shift key, click and drag the end points to the datum points previously created. They should snap into place. Do this for both connectors.



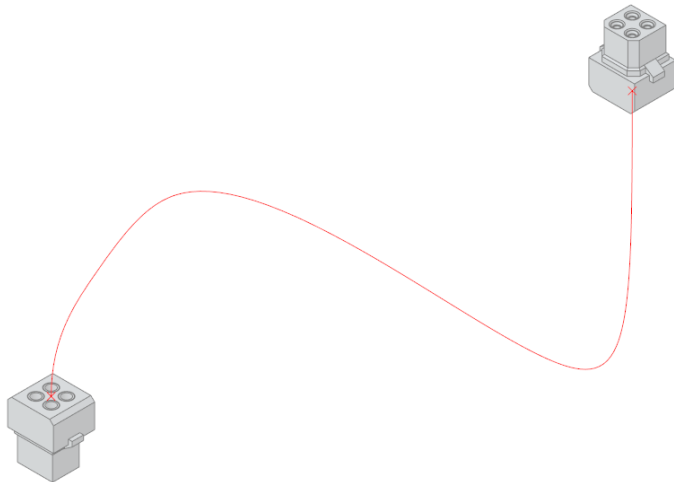
Now that the end points are placed on the connectors, we must now make the curve “normal” to the back surface of the connector.

To do this, click on each end point until a red line appears, this represents the direction in which the curve will begin. Navigate to the tangent tab and select “normal” from the constraints drop down. Creo then prompts you to select a plane to make the curve normal to, select the reference plane previously created.

*Select: Tangent>Constraints>Normal
>Datum Plane*



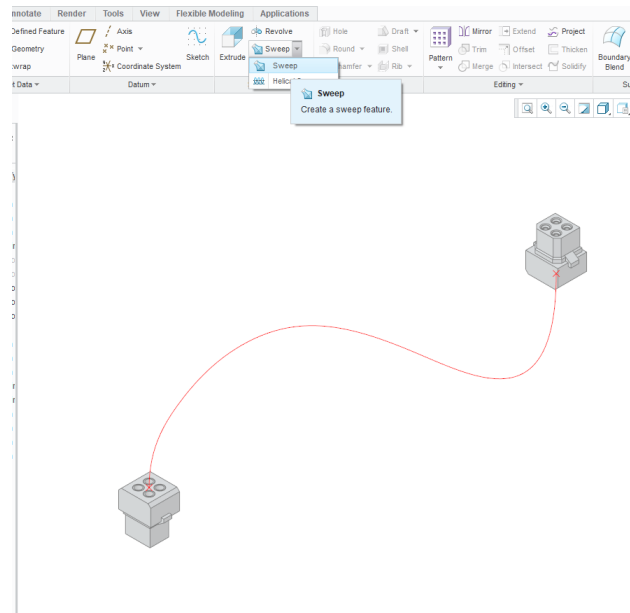
Active Part: WIRES



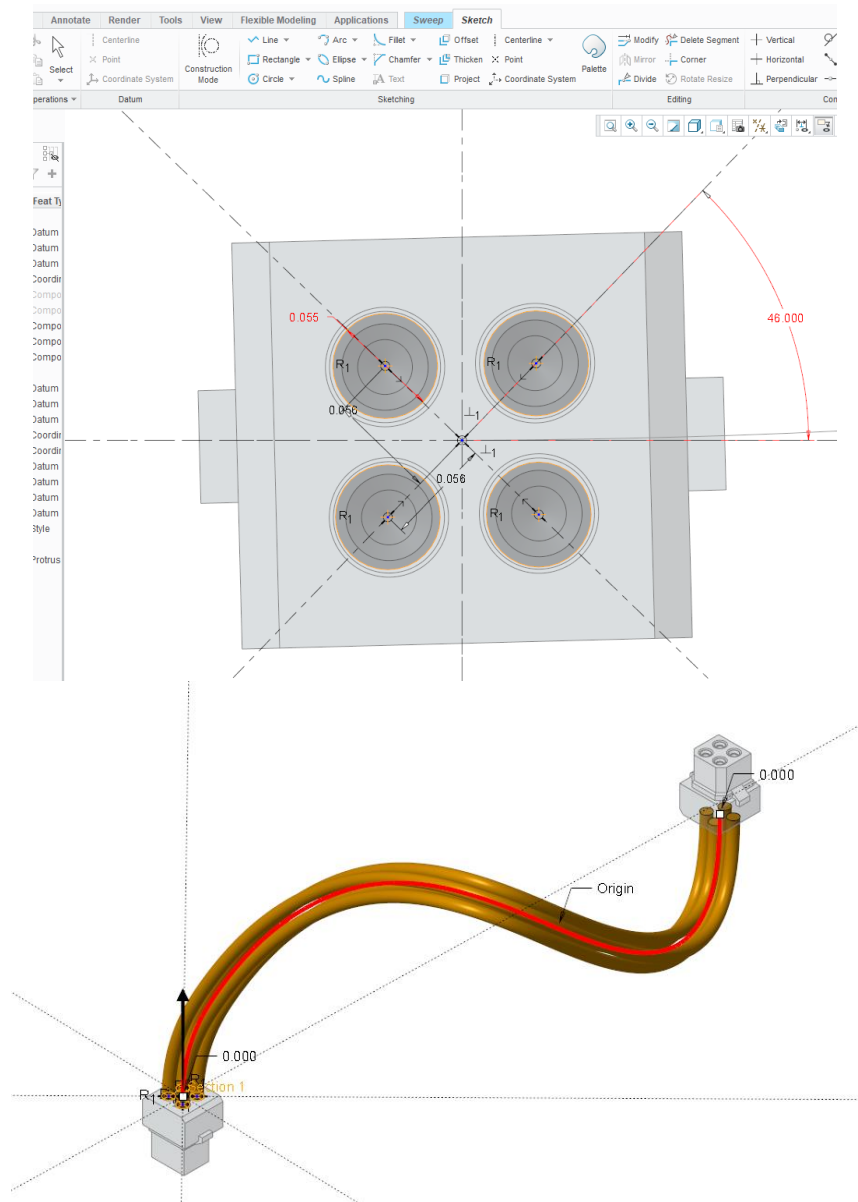
Now that the style curve is created, you can edit the profile of the curve by dynamically dragging the points, or by specifying coordinates for the points for follow.

2.5 Swept Profile

With the profile of the wires now created, we can use the sweep feature to create the wires. To do this highlight the curve and select the sweep feature in the model tab.



Select your sketching tool and make your cross-section.



Click the check mark and accept the shape, either specifying a solid or surface.

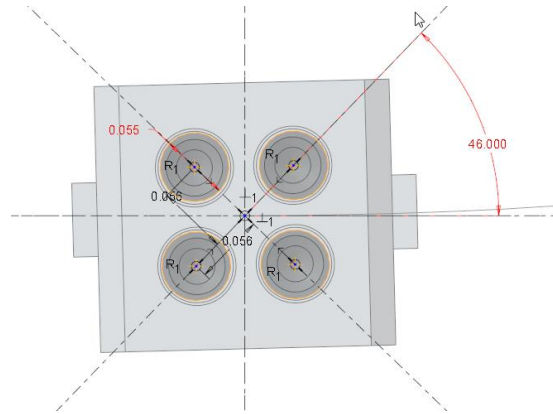
2.6 Twisted Pairs

For twisted pairs a small modification to the cross-section sketch can be made to give the wires a “twist” along the length of the sweep profile.

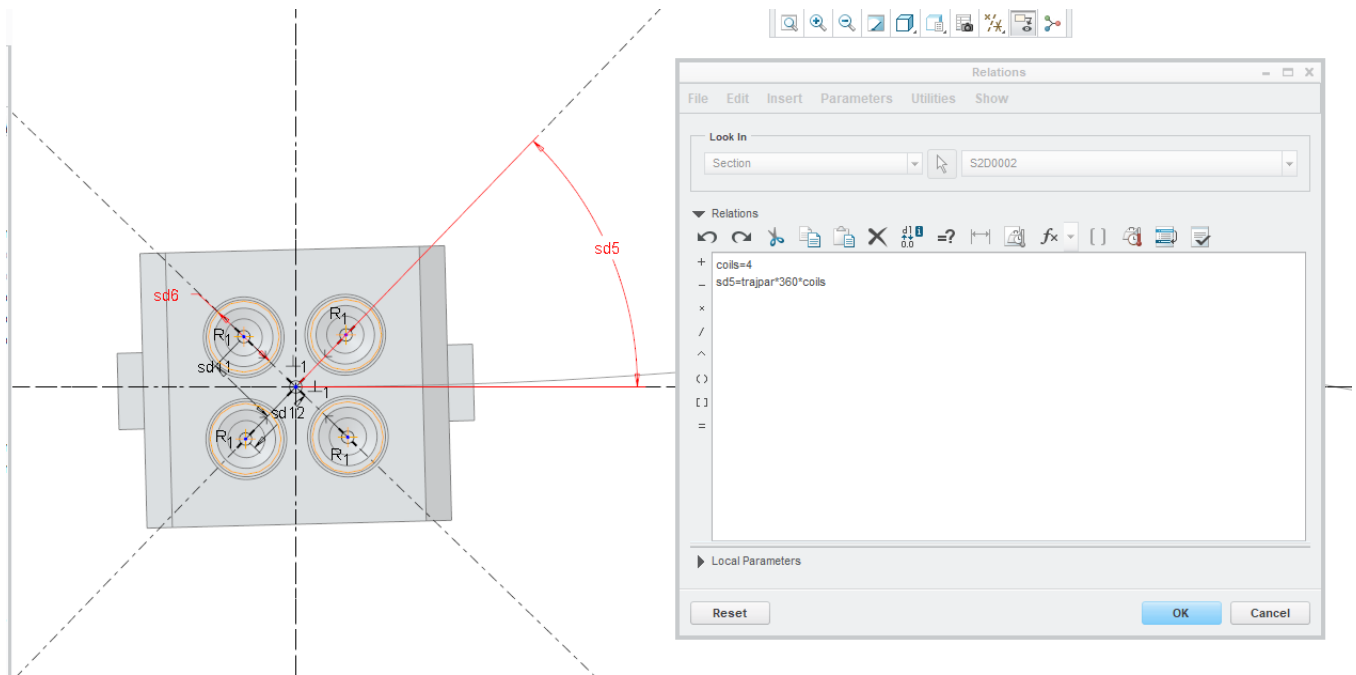
For this, we will use a parameter called “trajpar”. Trajpar is a parameter given to all trajectories that varies continuously from zero (0) to one (1) from start to end. If we create a relation to vary the twist angle, we can multiply it by 360 and create one full

twist along the trajectory. Trajpar x 360 = 0 at the beginning, and 360 at the end. If you desire more twists you can simply multiply by the number of twists. Ex. Trajpar x 360 x 10.

Make your swept section look something like this, giving it an angular dimension like below.



Now we will write a relation to drive the angular dimension, taking note of what the parametric name of the angular dimension is.



Lastly, make sure to select the variable section option in the sweep tool, if this isn't done the twist will not apply to the sweep feature.

