Aalto University
School of Engineering

EXERCISE 4.1 – MODELING A HOOK

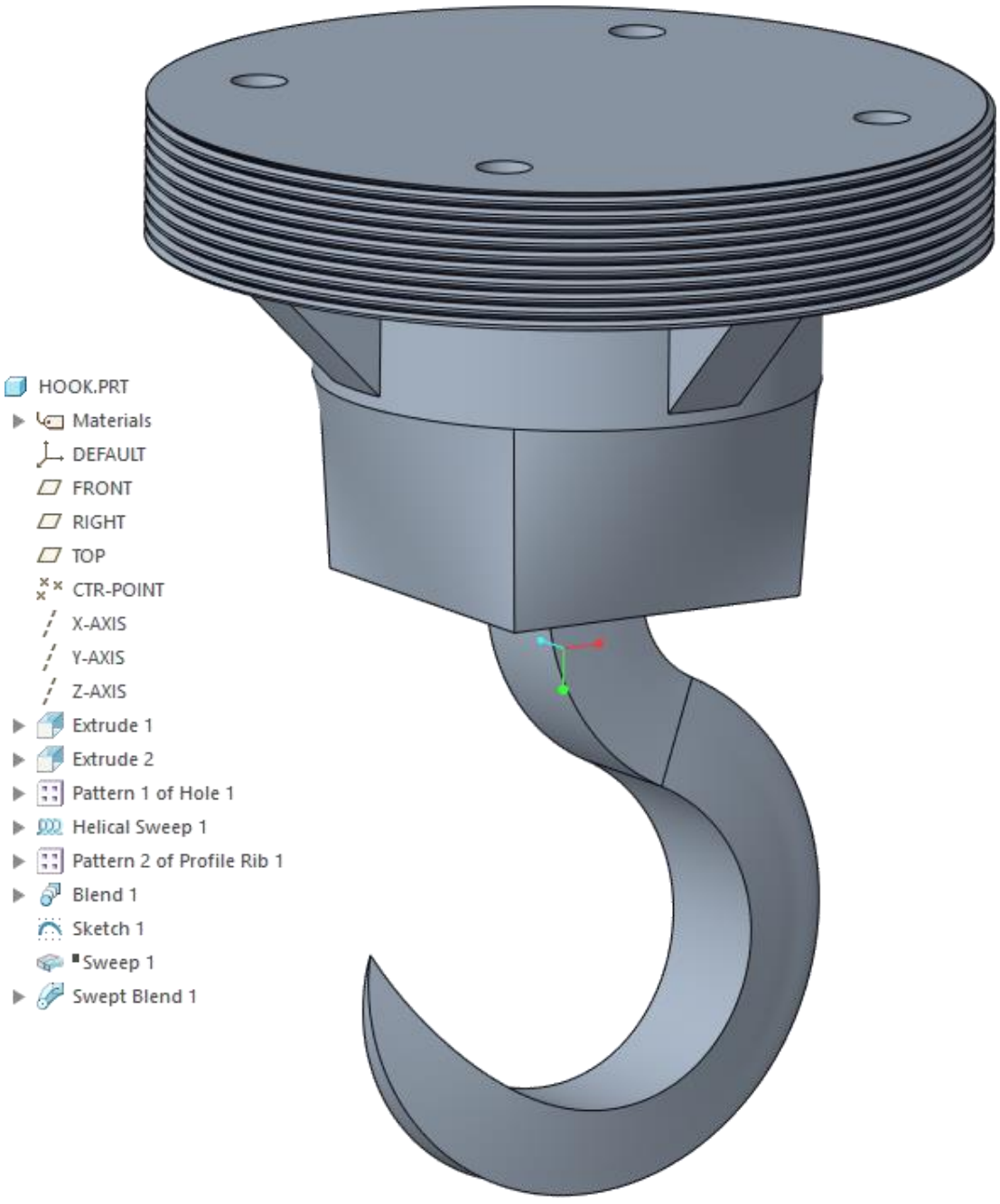

Figure 1: A hook and its model tree.

Learning Targets

In this exercise you will learn:

- ✓ Creating Holes
- ✓ Using Helical Sweep
- ✓ Creating Profile Ribs
- ✓ Using Offset in sketching
- ✓ Using Blend
- ✓ Using Sweep
- ✓ Creating Datum Points
- ✓ Using Swept Blend

Overall, this exercise is about special shape creating tools and techniques. It assumes that using the basic methods to add material (Extrude, Revolve) and basic sketching are known. Program version is 6.0.2.0.

Getting Started

Create a **new** solid part model and name it as HOOK. Then create a cylinder on the TOP datum plain using **Extrude** ( ) as seen in Figure 2.

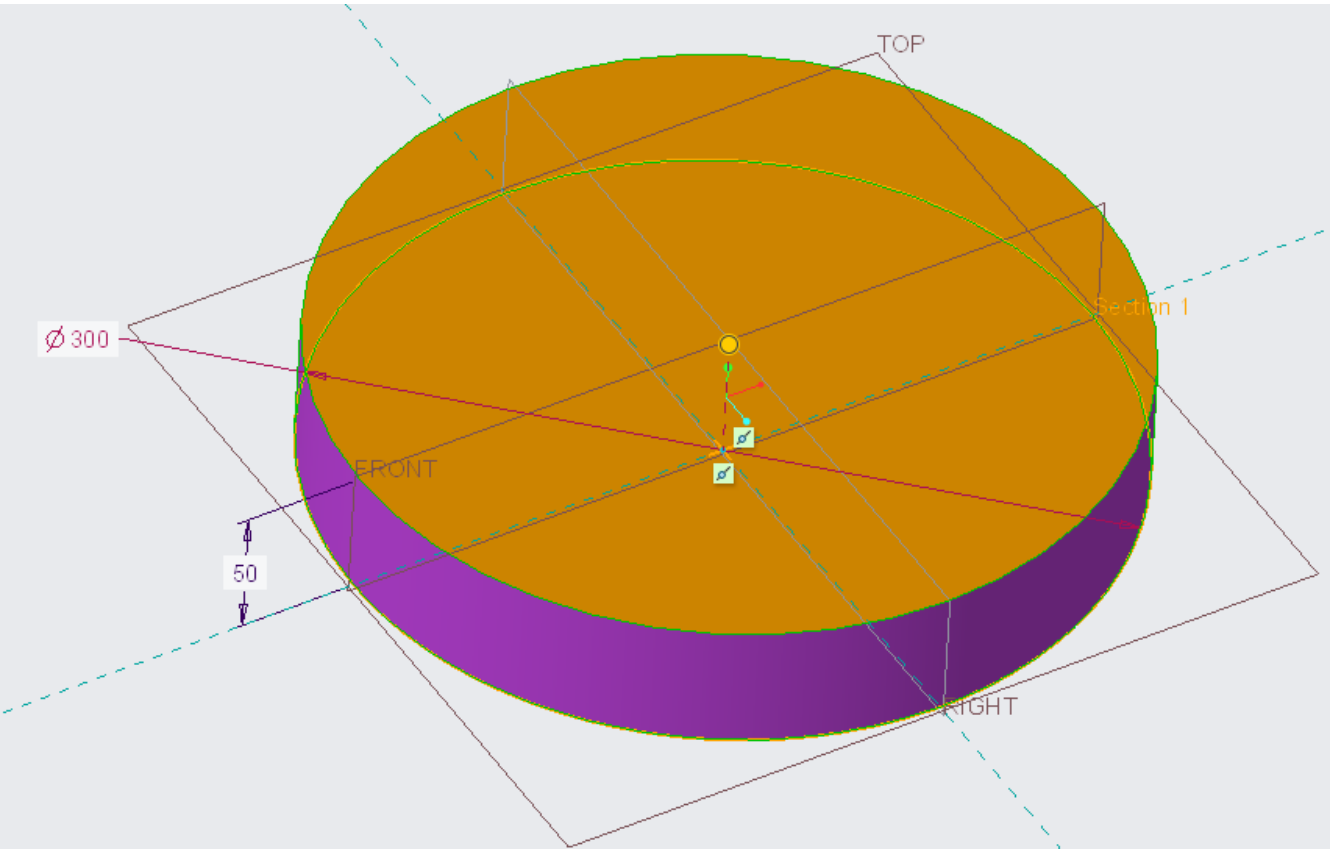

Figure 2: First user-made feature (Extrude).

Using Offset in Sketcher

Next step is to create a middle cylinder. Using **Extrude** ( ), create a sketch on the upper surface of the part. In the sketch, use **Offset** ( ) from *Sketching* group and choose the upper face of the cylinder using **Loop** as a type (Figure 3). When prompted, enter a negative offset value of **- 60** (60 mm inside from the edge). **Close** the *Offset* tool. Now you have a closed sketch, which follows the profile of the previous feature (symbols indicates that, see Figure 4). Accept the sketch, extrude it by **50** (Figure 5) and accept the feature.

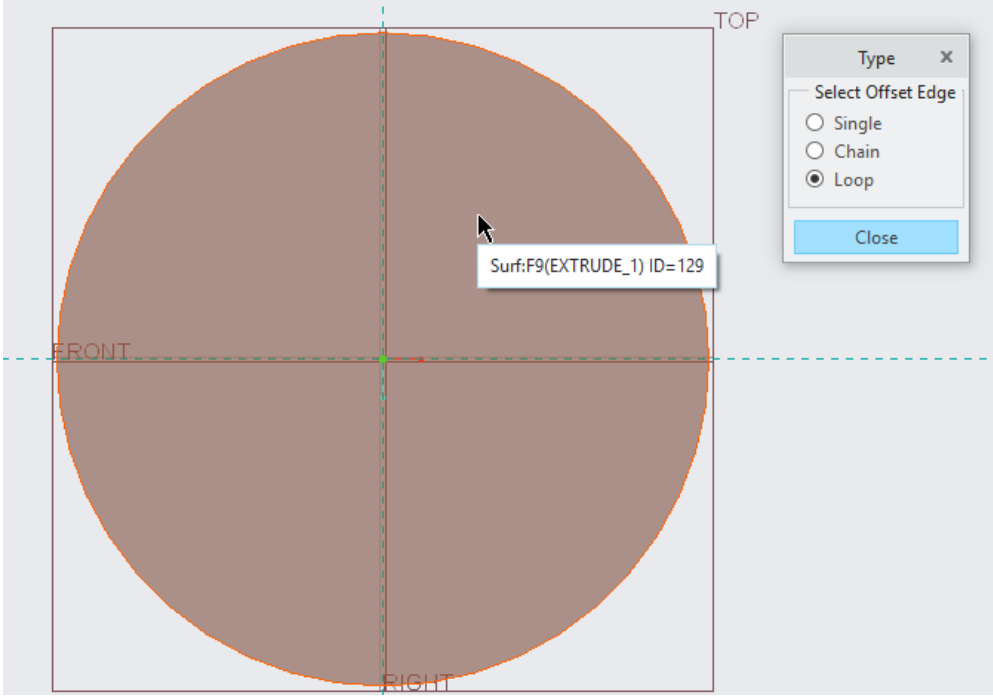

Figure 3: Offset tool is active and selecting part's top surface.

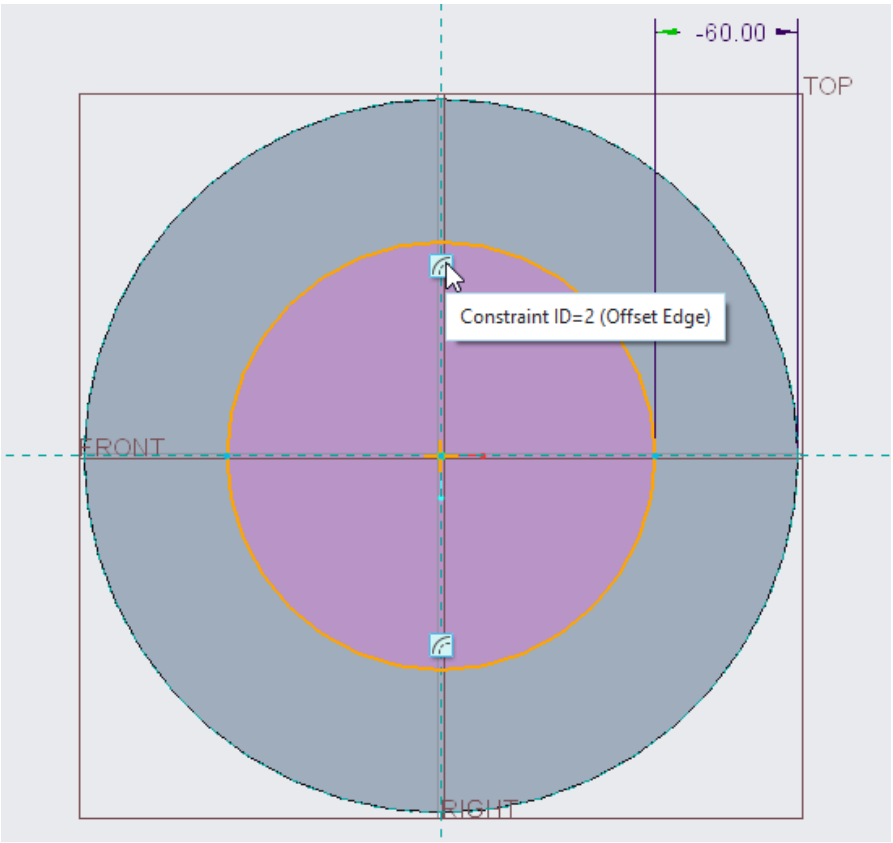

Figure 4: Ready to accept sketch. Notice the -60 dimension and Offset Edge symbols on the arcs.

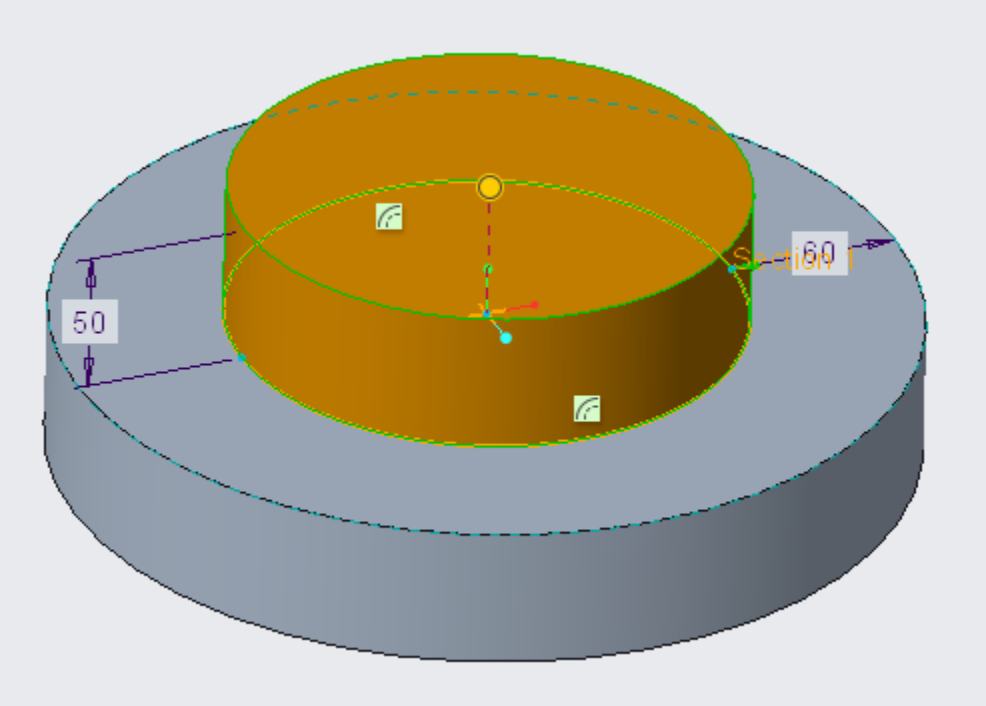

Figure 5: Extruding previous sketch by 50.

With Offset tool (and with Project tool) you can fast copy complex profiles. Even if the parent feature is changed a lot, the sketch follows its profile. This works extremely fine with profiles which includes several lines, e.g. a profile with four lines is changed to have five lines.

Creating Holes

Turn **Axis Display** ( ) on, so you can see axes and use them as references. Next we create a mounting hole using **Hole** (from *Engineering* group). First, the tool wants a surface reference to start making the hole, so choose the upper surface of the first feature (Figure 6).

From the Placement tab, change hole's placement *type* to **Diameter**. Then click *Offset References* field and choose the axis of the cylinder and holding **CTRL** choose FRONT plane as a second reference. Give the hole placement dimensions as seen in Figure 7.

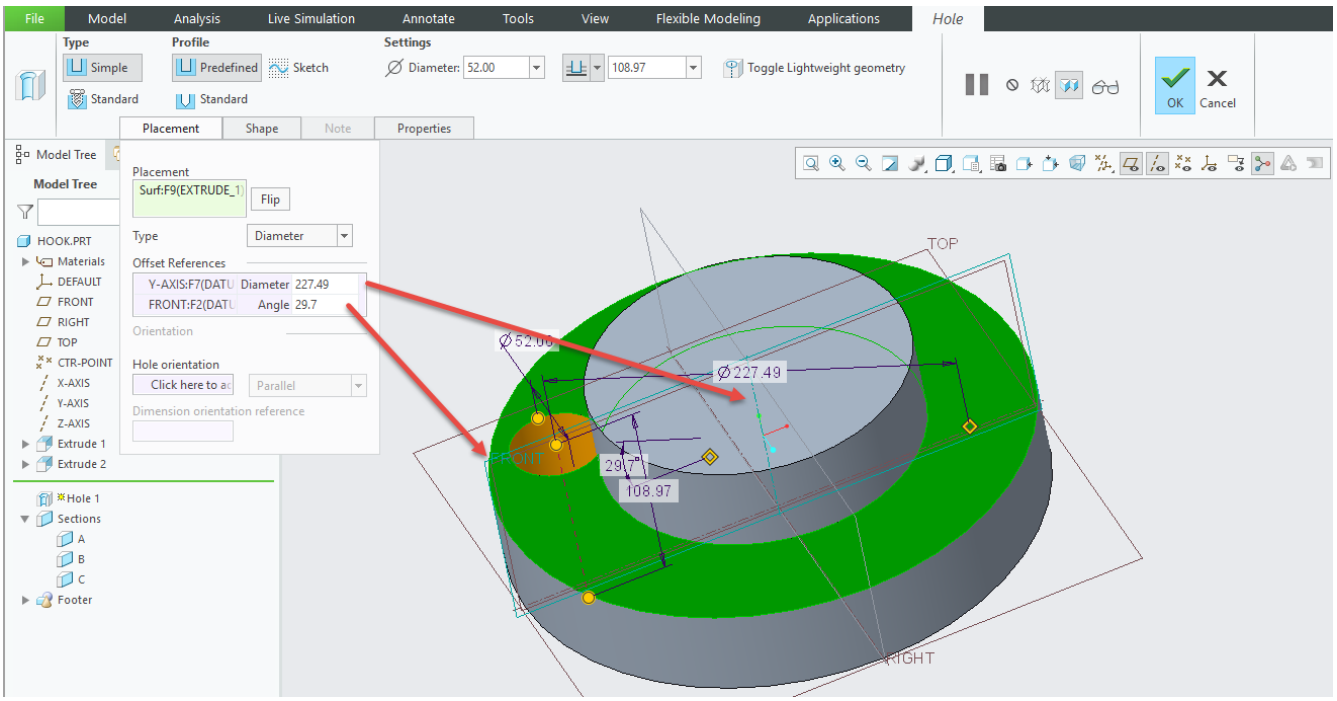

Figure 6: Placement surface (in green) and Offset References selected (axis and FRONT datum).

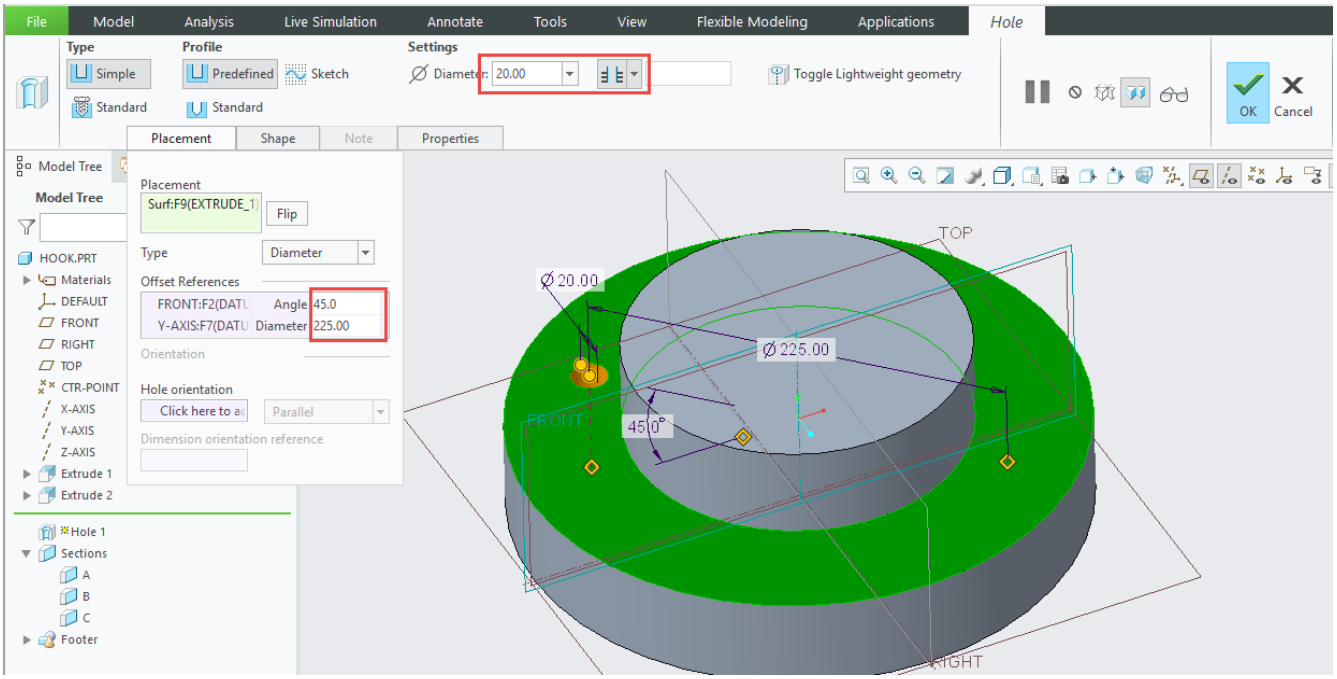

Figure 7: Hole placement values changed.

Next step is to define the profile of the hole. Choose **use standard hole profile** ( ) from the dashboard and then **Adds counterbore** ( ). Choose **Shape** tab and define the profile values of the hole (Figure 8). When ready, accept the feature (**MMB**).

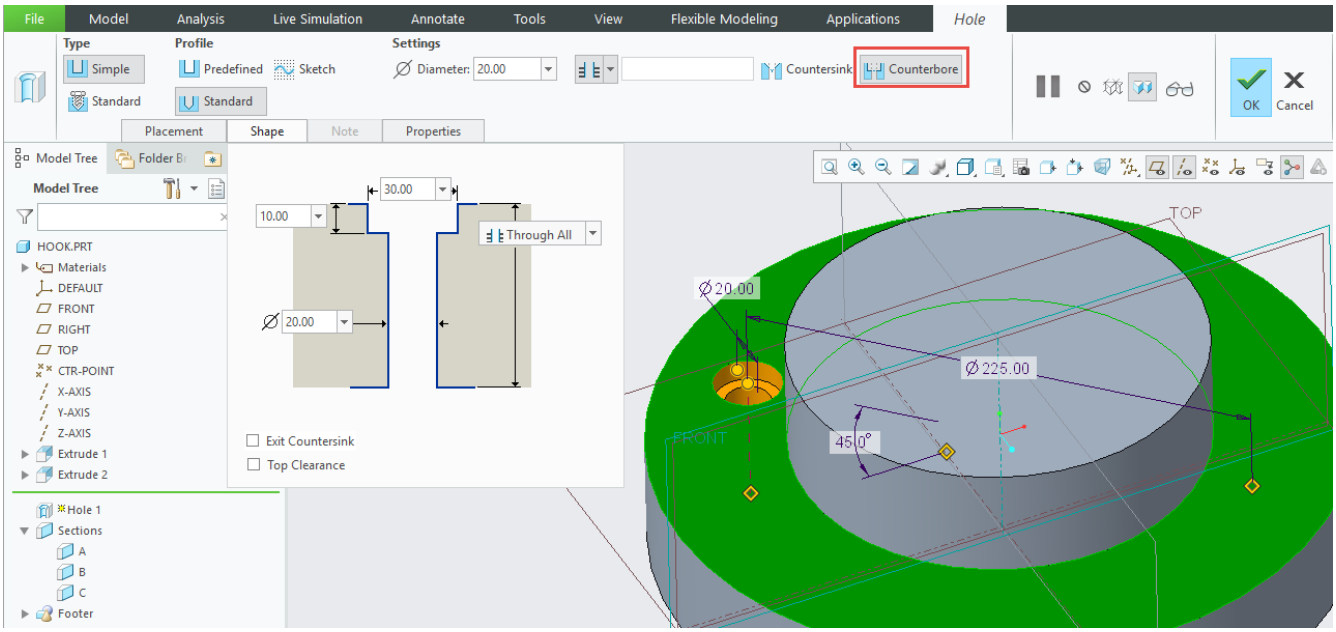

Figure 8: Ready to accept *Hole* feature.

Use Axis **Pattern** ( ) to create four holes (4 per 360°).

With Hole tool you can define profiles of all kind for holes from the simplest cut to the most complicate standard and custom profiles. The program also creates an annotation for holes, which can be used in documentation, and in manufacturing drawings (this is possible with standard holes).

Using Helical Sweep

Next, we create a screw-like side for the bigger cylinder shape. For that we use **Helical Sweep** () which is located in *Shapes* group (Figure 9).

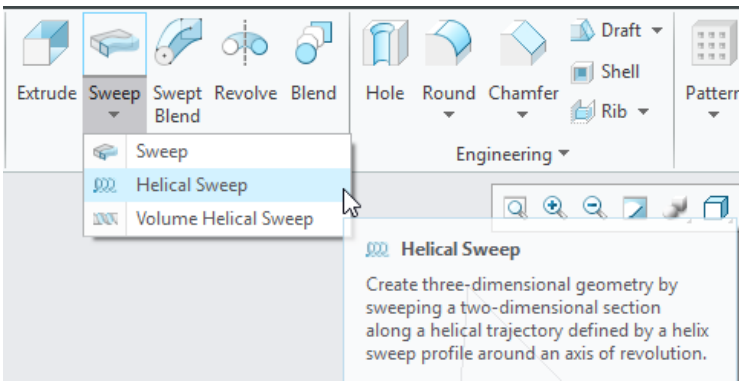

Figure 9: Selecting *Helical Sweep* from *Shapes* group.

First we need to define *Helix Profile*. Select **References** tab and **Define**. A *Sketch* window appears. Select FRONT as a sketching plane and RIGHT (*Orientation Right*) as a *Reference* plane. Click **Sketch**.

Open *References* window (From *Setup* group or hold **RMB** and select **References**). Remove all references that the program has preselected and choose the ones showed in Figure 10. To make things easier, set **Plane Display** () off.

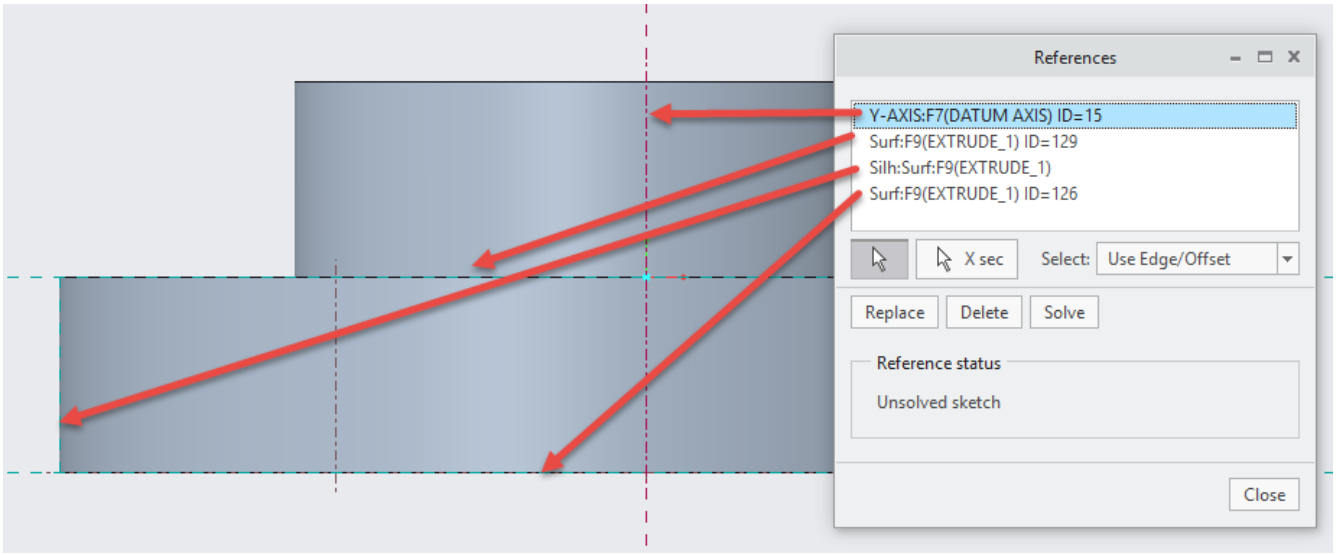

Figure 10: References for the sketch. Notice the silhouette surface (Silh:Surf) reference.

Next step is to draw a helix profile. Using **Line** (), create a straight line as shown in Figure 11. Start from the bottom to have the arrow pointing upwards.

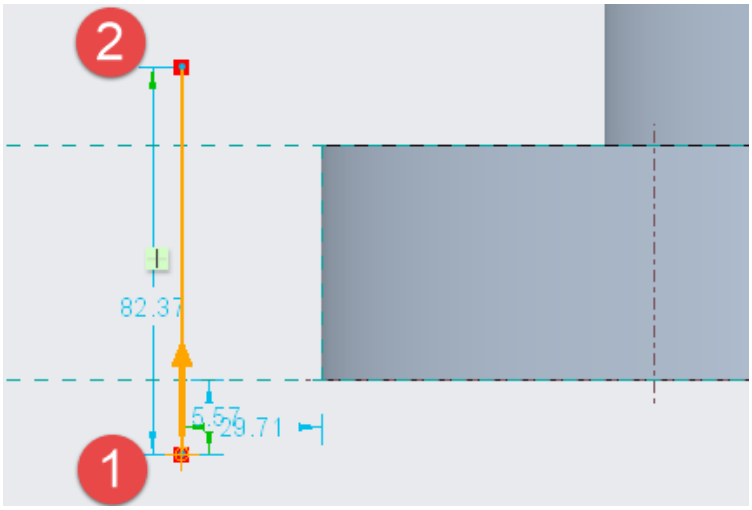

Figure 11: Sketched line, started from 1 and ended in 2. Dimension values doesn't matter.

Then, select that just made line, hold **CTRL**, select the vertical reference line (silhouette surface) and select **Coincident** from the menu (Figure 12).

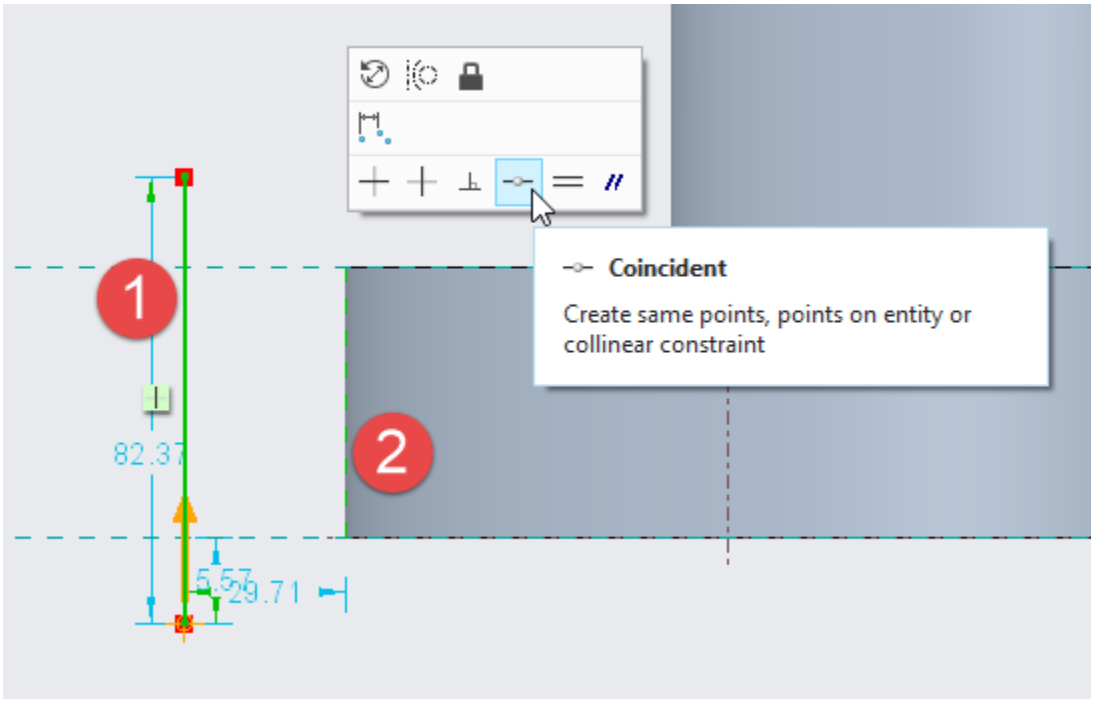

Figure 12: Line and reference selected, selecting *Coincident* from the menu.

Now the coincident constraint appears between those two selected entities. Using **Dimension** (), create dimensions showed in Figure 13. Then select the lower dimension, hold **CTRL**, select the upper dimension and select **Equal** from the menu (Figure 13). This makes dimension 2 to be depending on dimension 1. To symbolize that, the E1 dimensions have appeared.

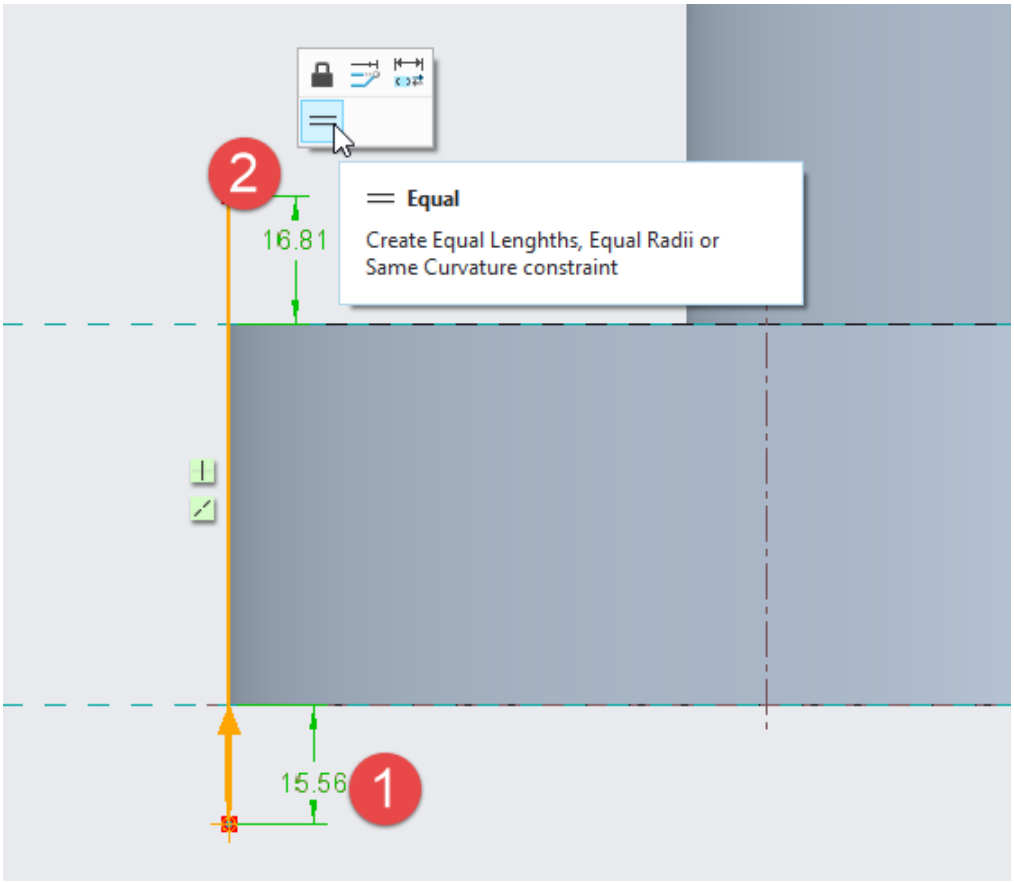

Figure 13: Two created dimensions. Dimension 1, then dimension 2 selected and selecting Equal from the menu.

Change the only dimension to **5** (number 1 in Figure 13). Create also a **Centerline** (from *Datum* group) on top of the vertical axis (Figure 14). When ready, accept the sketch.

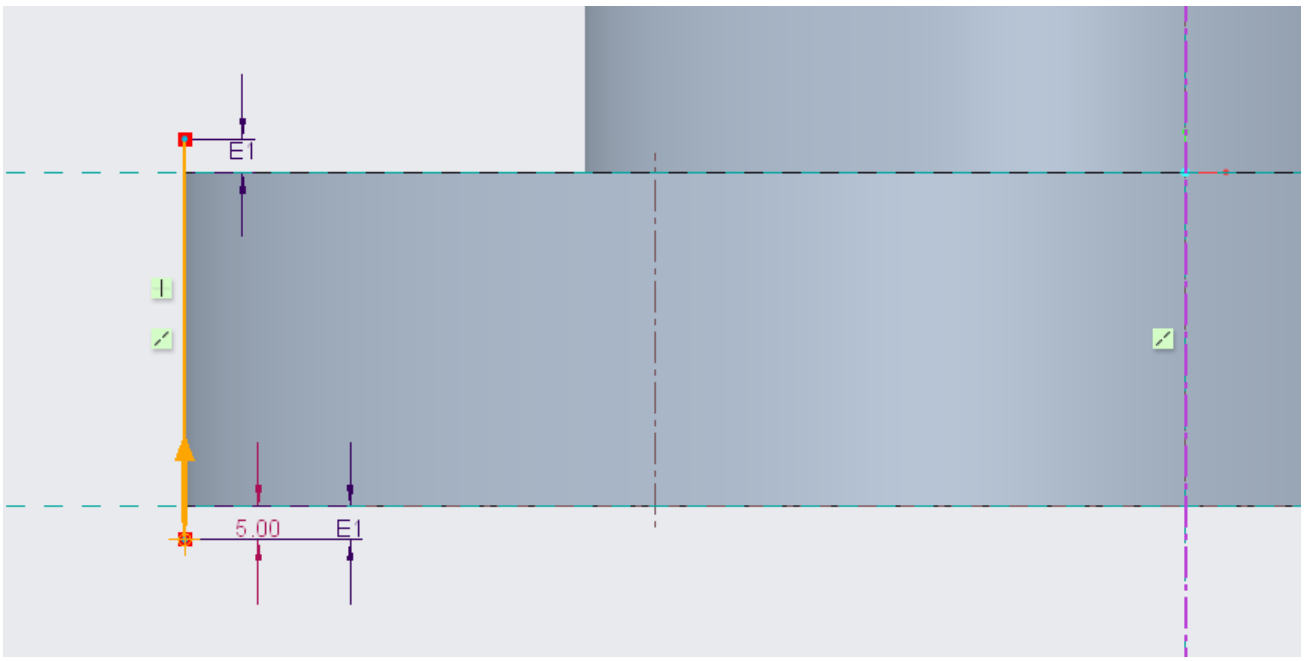

Figure 14: Ready to accept *Helix Profile* sketch. Notice the purple Axis of Revolution.

We are back in the dashboard. Select  from the dashboard to create a sweep profile. We are back in the sketching mode. Sketch a line from bottom of the geometry to the start point of the sketch. Select that line and select **Construction** from the menu (Figure 15).

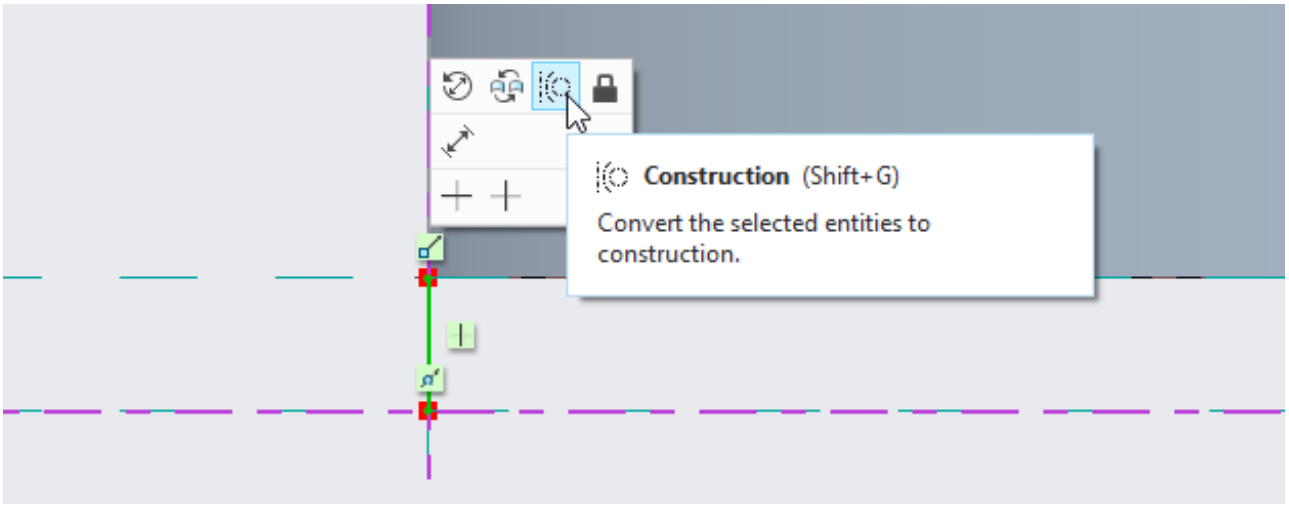

Figure 15: Sketched line, selecting *Construction*.

Using previously sketched construction line and selected references, create a shape as seen in Figure 16. Notice that there is a mid-point, horizontal, vertical and two tangency constraints! When ready, accept the sketch.

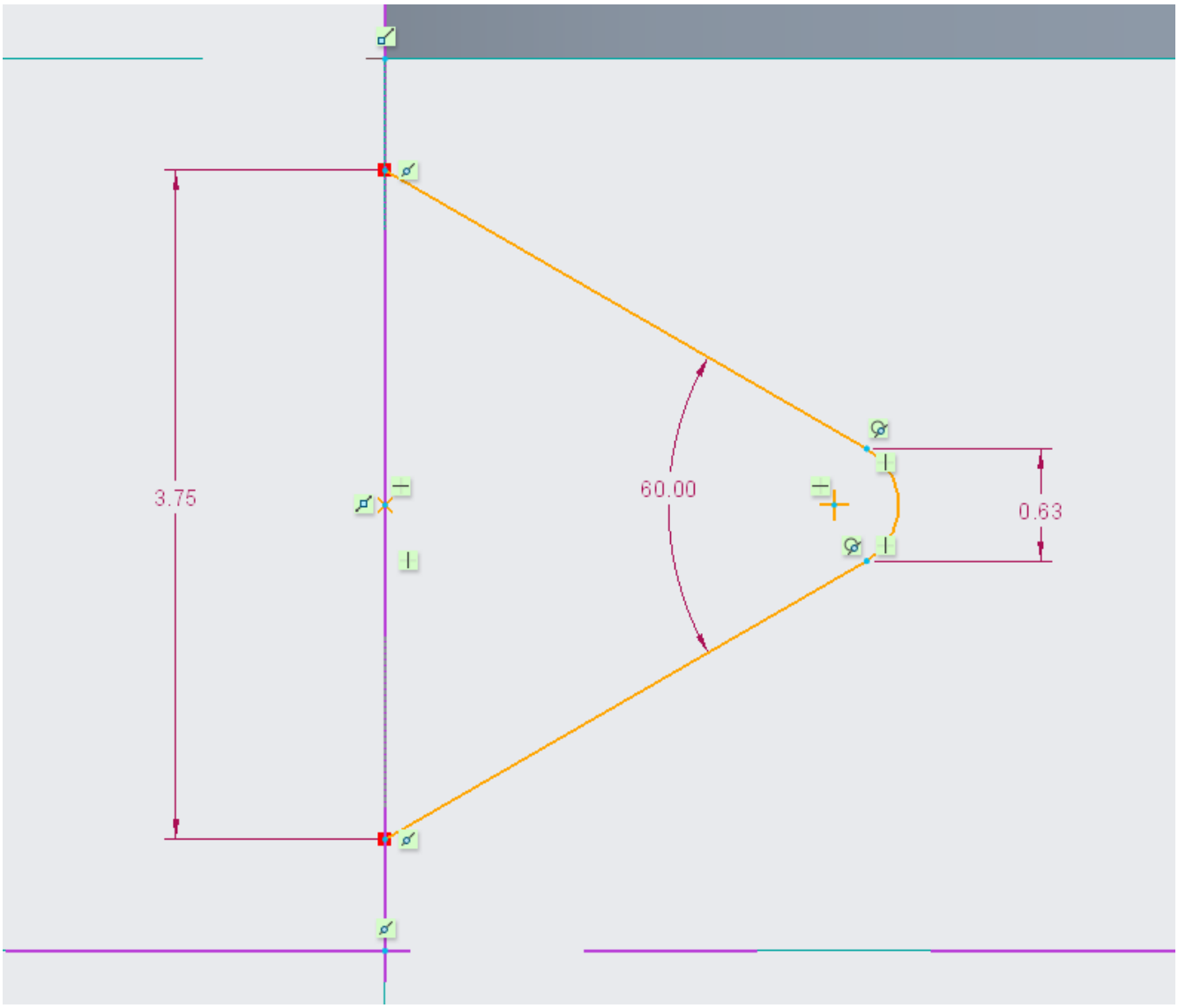

Figure 16: A sketched shape containing three elements (and one constructional line).

The preview may be active, just ignore it. Change **Pitch Value** ( ) to **5** and set **Remove Material** ( ) on (we are removing material, not adding) from the dashboard. The preview should look like in Figure 17. If needed, flip the cutting side ( ). When ready, accept the feature.

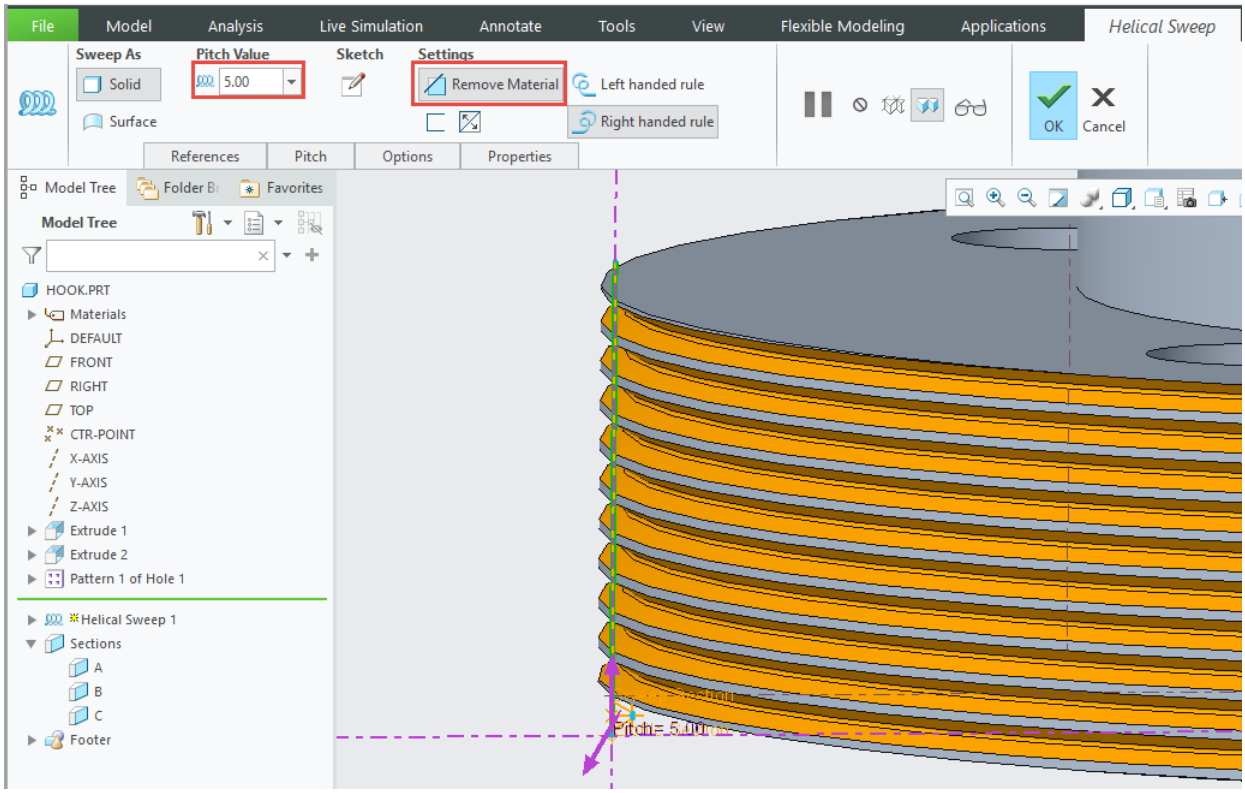

Figure 17: Ready to accept Helical Sweep.

With Helical Sweep, you can create features like screws and springs. The idea is that you rotate the profile over centerline along the sketched helix profile with defined pitch.

Time to save your model?

Using Ribs

Next we create a supporting rib for the part. Select **Profile Rib** () from *Engineering* group (Figure 18).

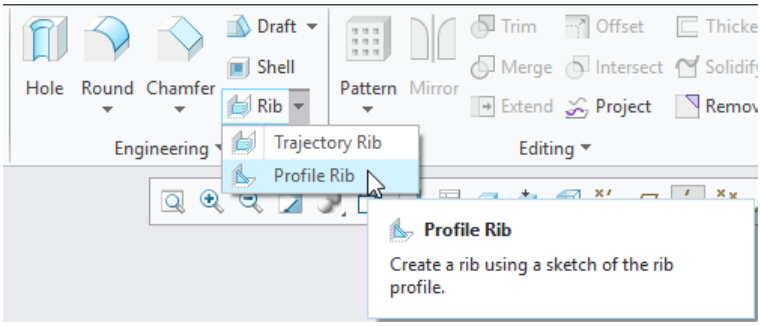

Figure 18: Selecting Profile Rib from *Engineering* group.

Hold **RMB** and select **Define Internal Sketch** from the menu. Select FRONT plane as a sketching plane and RIGHT (*Orientation Right*) as a *Reference* plane (if not auto-selected). Go sketching. Select three surfaces as references and create an open sketch as shown in Figure 19.

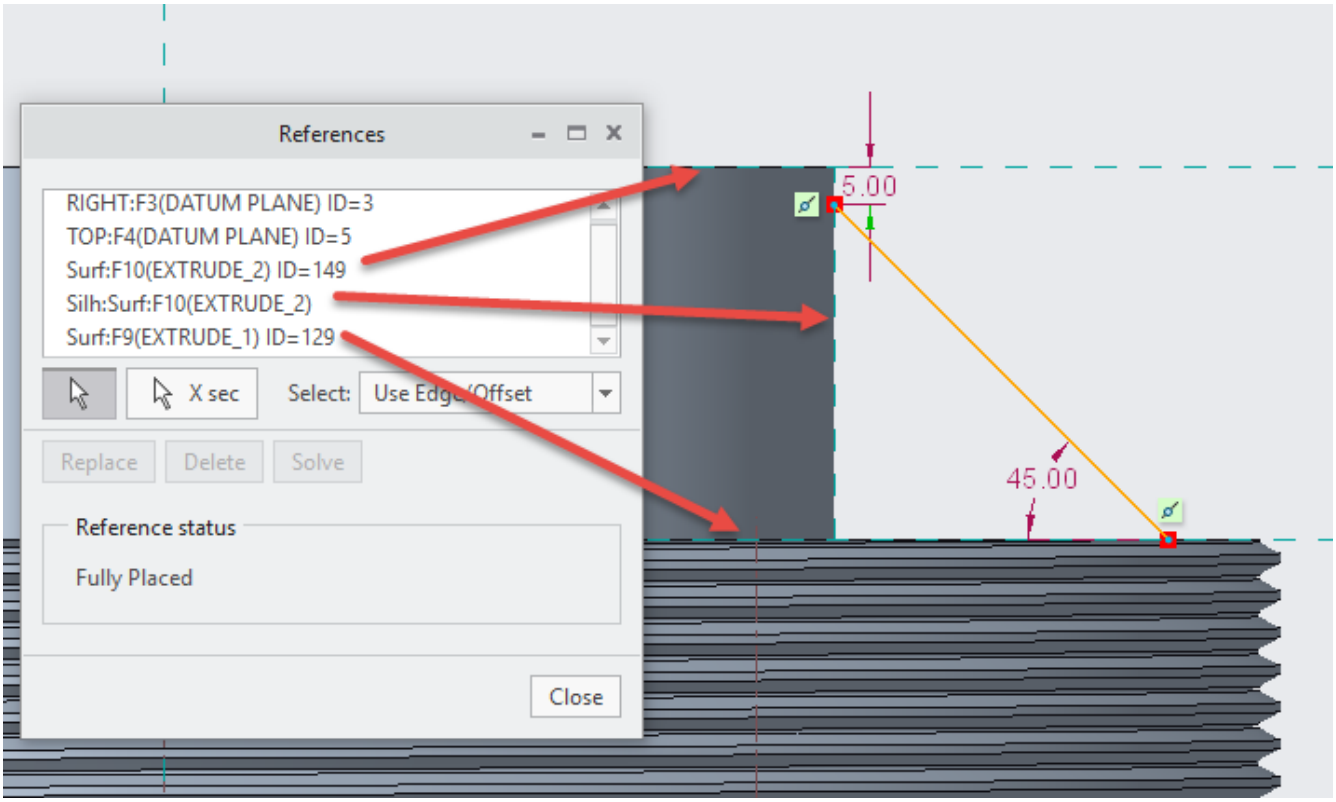

Figure 19: Sketched open profile (only one line) and references used for the sketch.

When ready, accept the sketch. Next we need to define to which side of the sketched line material is added. There is an arrow pointing upwards, click on it to make it point downwards and give a value of **30** to the thickness of the rib (Figure 20).

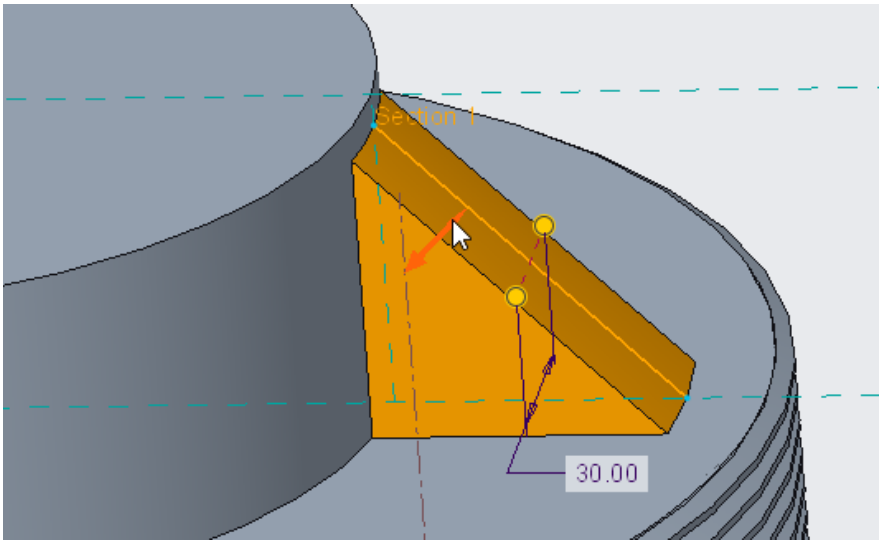

Figure 20: Arrow to change the side of the material highlighted.

Accept the feature. As you can notice, the rib profile follows the shape of the upper cylinder. If you make the same sketch and use *Extrude* to add material, it won't (Figure 21). You don't have to do this.

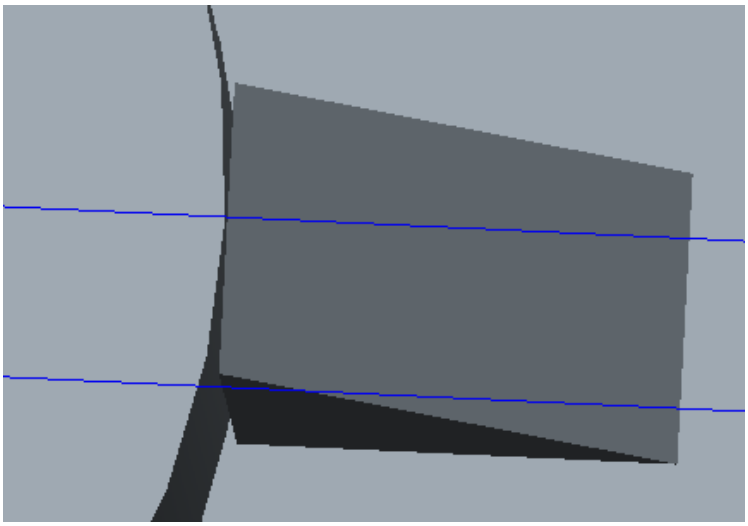

Figure 21: Same profile (closed sketch) using Extrude. Notice, that the shape is connected to the upper cylinder only on the sketching plane.

Make a **Pattern** ( ) of ribs (*Axis* pattern, 4 per 360).

With Rib you can create supporting elements which are very important when designing parts for e.g. plastic molding.

Using Blend

The next task is to create a feature that starts with a circular cross-section and ends with a rectangular cross-section. For that we can use **Blend** ( ) from *Shapes* group (Figure 22).

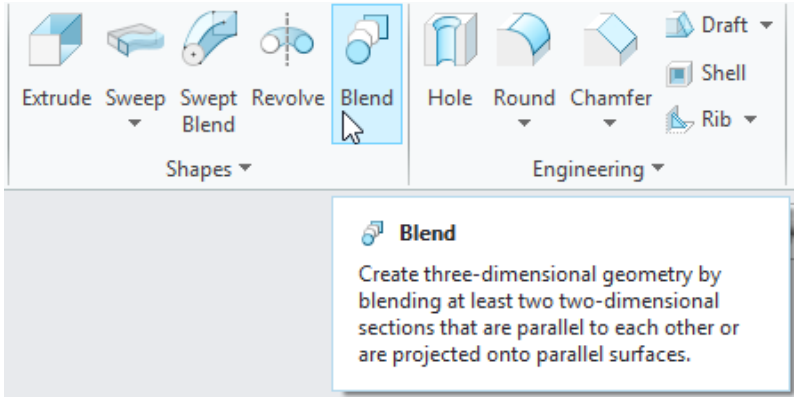

Figure 22: Selecting Blend from Shapes group.

Select the top surface of the part as a sketching plane (hold **R**MB, select **Define Internal Sketch**). Select **References** (RMB menu), select the side surface of the upper cylinder as a reference (Figure 23) and sketch a circle using the previously created reference to lock diameter.

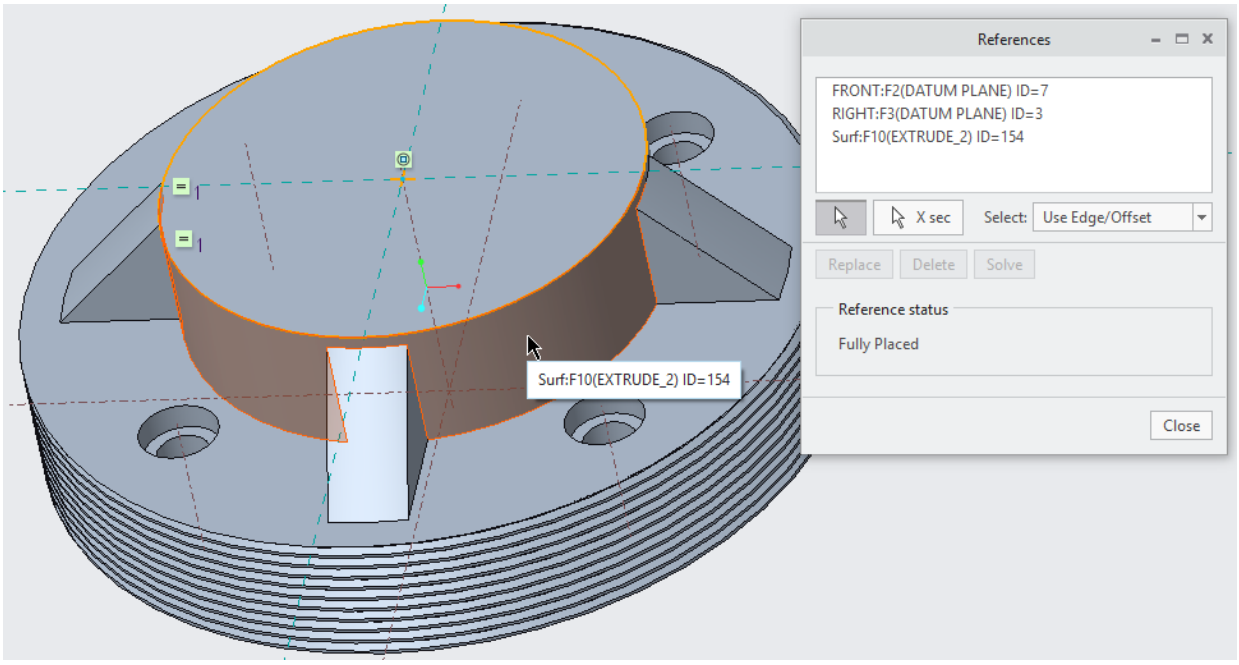

Figure 23: Selected side surface as a reference for the sketch.

Now we have a sketched circle. In Creo, all circles consist of two arcs. That means that all circles have two points. Because the plan is to create a shape that's cross-section starts as a circle and ends as a square, we have a problem. The problem is that square has four points (four corners). When *Blend* creates a shape, it connects all points in different cross-sections together. The *Blend* tool needs, that all cross sections have equal amount of points. We can't remove points from square, so we need to add points to circle.

To make dividing the circle easier, create two construction centerlines (from *Sketching* group) as showed in Figure 24.

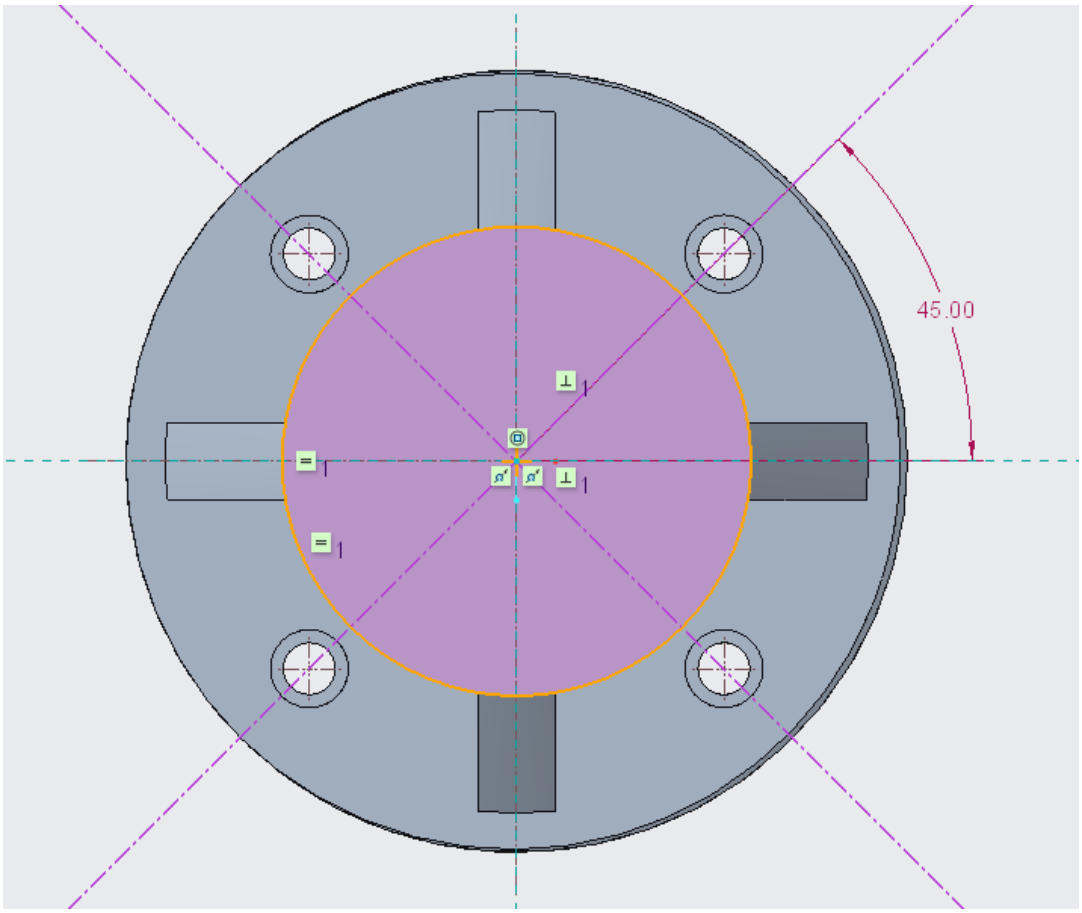

Figure 24: Two construction centerlines created. Notice the perpendicular constraint.

Then we use **Divide** () from *Editing* group. Select that tool and then select all four places, where the circle and centerlines cross each other (Figure 25). This divides the circle into four arcs and therefore sketch has four points. The yellow arrow has appeared. This marks the starting point to connect sketches together (every sketch in blend has the same arrow). The

default rotation is to positive side (counter-clock wise) and for this sketch it is fine. If you need to change the starting point, select the point from the sketch, hold **RMB** and select **Start Point** from the menu (Figure 26). If you need to change the rotation direction, do the same thing (if the point is selected again, it rotates).

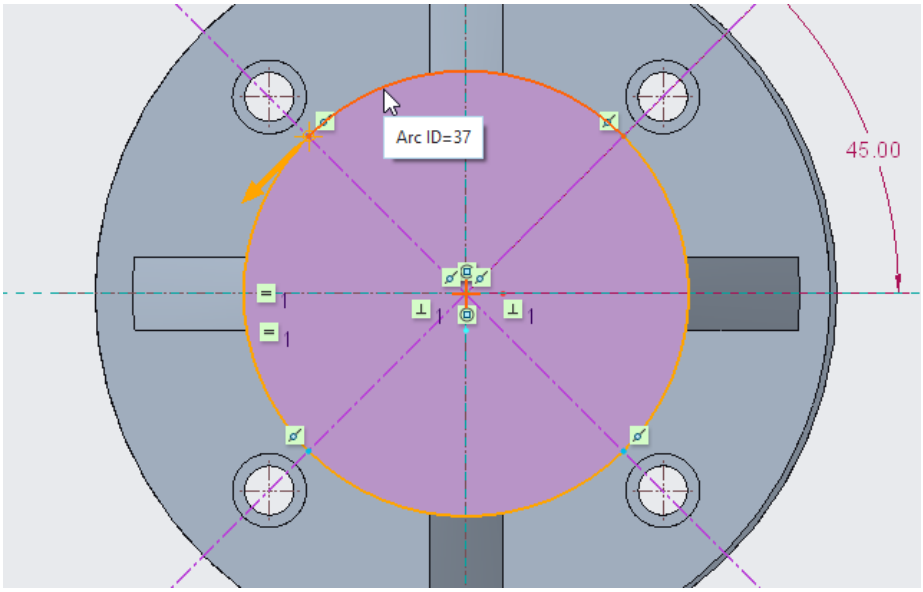

Figure 25: Three arcs created by dividing a circle, selecting the last one.

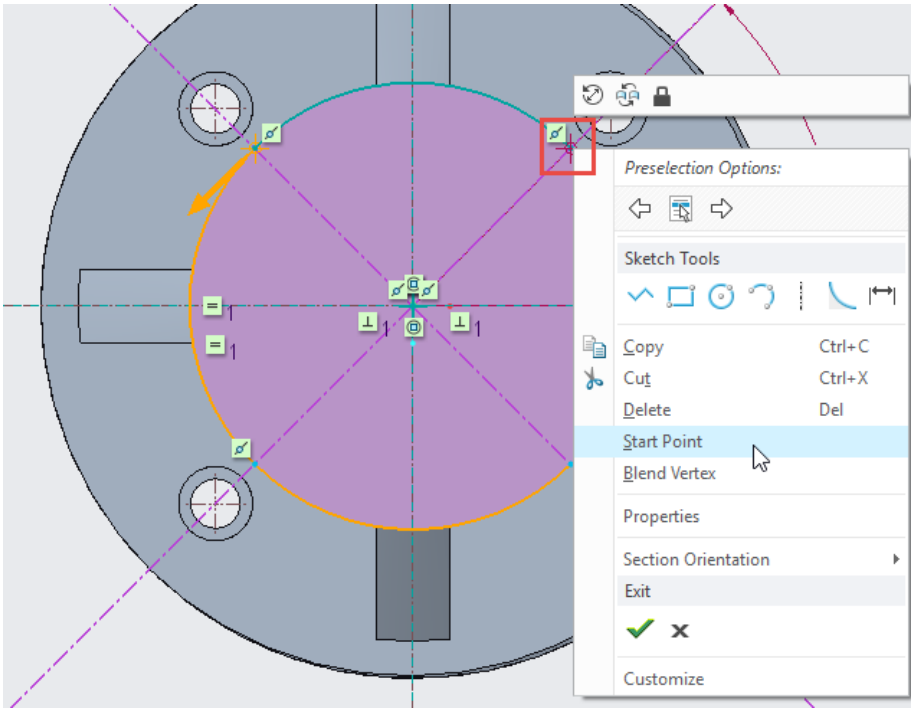

Figure 26: Point selected (dark red) and choosing *Start Point* from RMB menu.

The first sketch is ready; it should have four points. Accept the sketch to get back to *Blend* dashboard. Select **Sections** tab, change the distance (*Offset* from Sketch 1) to **75** and select **Sketch** to create a new cross-section (Figure 27).

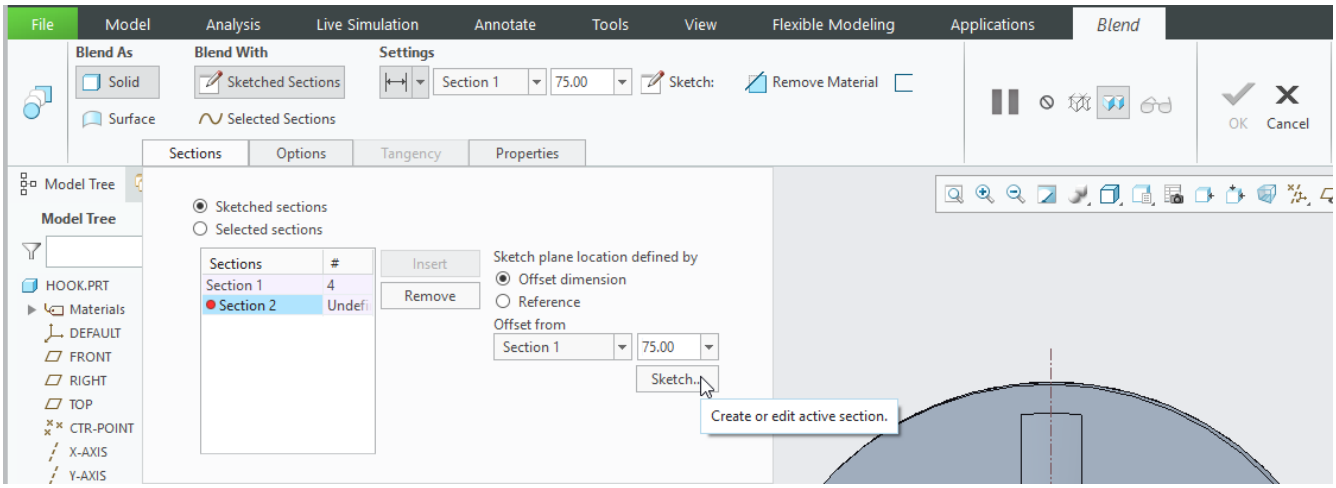

Figure 27: Distance changed to 75 and creating a new sketch.

We are back in the sketching mode. Create a square as shown in Figure 28. Be sure, that the yellow arrow showing the start point for the sketch is in the right place (i.e. at same place as in the circular sketch). When ready, accept the sketch. If the preview looks like in Figure 29, accept the feature. If not, from *Sections* tab you can modify existing sketches.

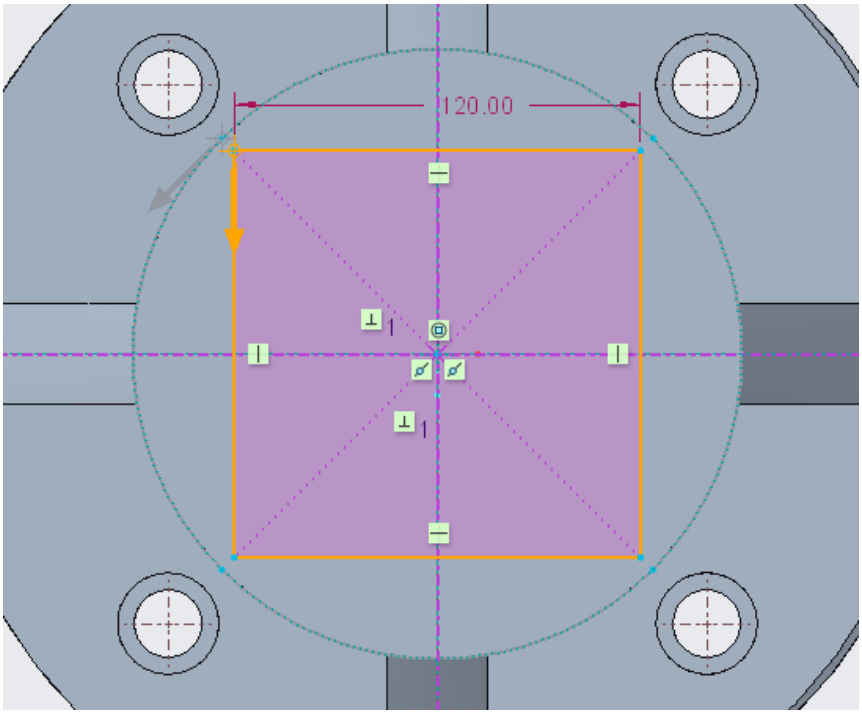

Figure 28: Ready to accept sketch.

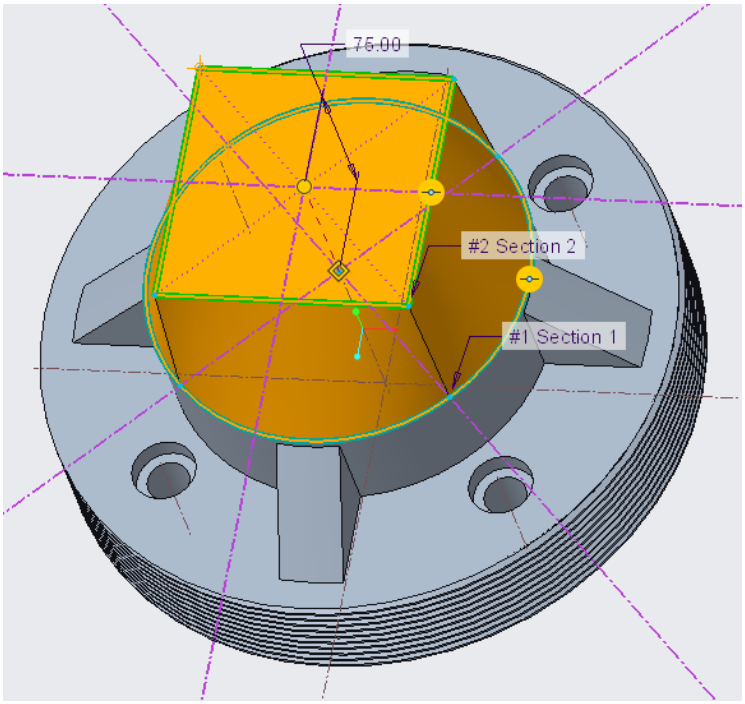

Figure 29: Ready to accept feature. Notice how the geometry is created.

With blend you can create features with two or more different cross-sections. In this exercise, we used parallel blend (sketches are parallel) but there is also a rotational blend.

Using Sweep

When using *Extrude*, the sketched shape is always extruded normal to the sketching plane. With *Sweep*, you can define the trajectory that the sketch follows.

Creating external sketch

The *Sweep* tool needs a sketch to be its trajectory. First task is to create an external sketch using **Sketch** () from *Datum* group. Select FRONT plane as a sketching plane and RIGHT (*Orientation Right*) as a reference plane. Create a sketch as seen in Figure 30. To make life easier, use two *Tree Point / Tangent End Arc*, one *Horizontal* and one *Perpendicular* constraint. When ready, accept the sketch. Because we used an external sketch, we have now Sketch 1 feature in the model tree.

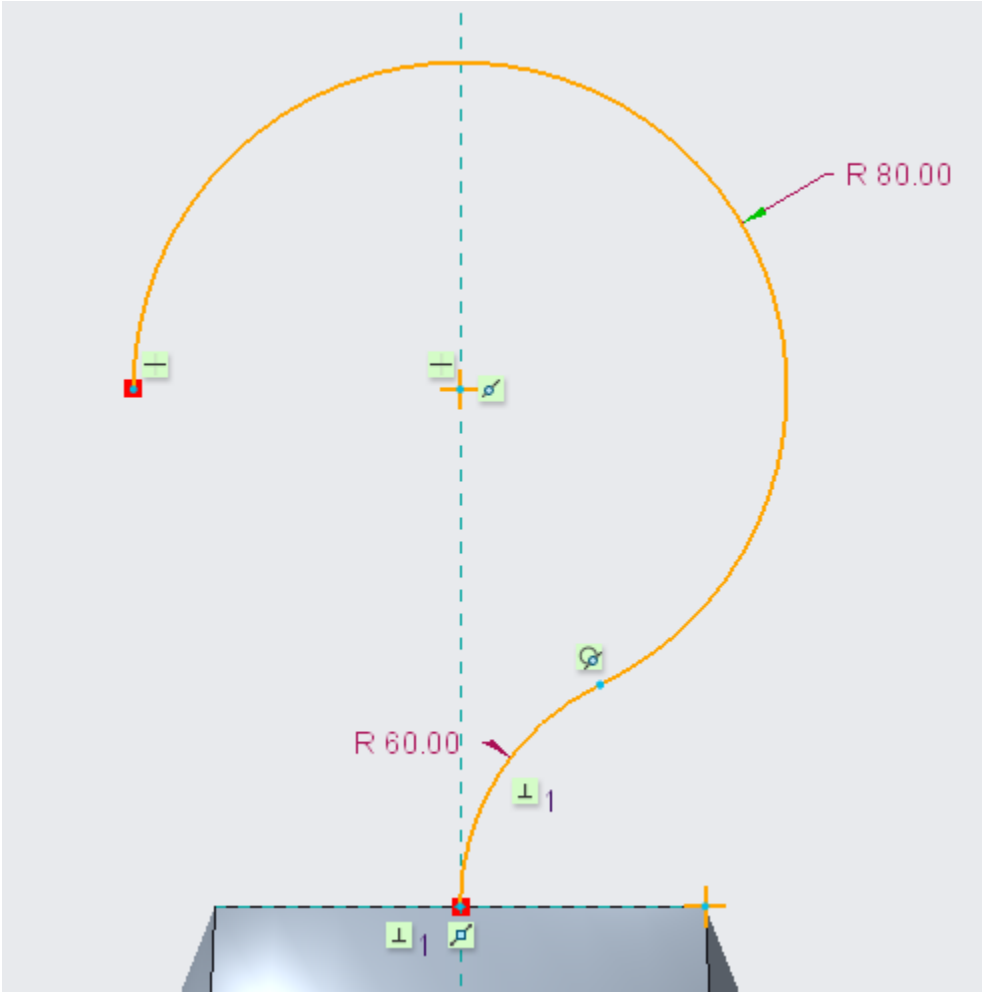

Figure 30: Ready to accept sketch. Notice the constraints (green).

Creating Sweep profile

Select **Sketch 1** from the model tree and select **Sweep** ( ) from *Shapes group*. The *Sweep* dashboard appears and the previously made sketch is selected as a trajectory (i.e. sketched cross-section is always normal to it). Check that a purple arrow is located at the bottom point of the sketch (if not, click the arrow to change its location). Select **Create or edit sweep section** ( ) from dashboard to sketch a cross-section. The sketching mode opens. By using **Center and Axis Ellipse** ( ) from *Sketching* group sketch a shape as seen in Figure 31. When ready, accept the sketch.

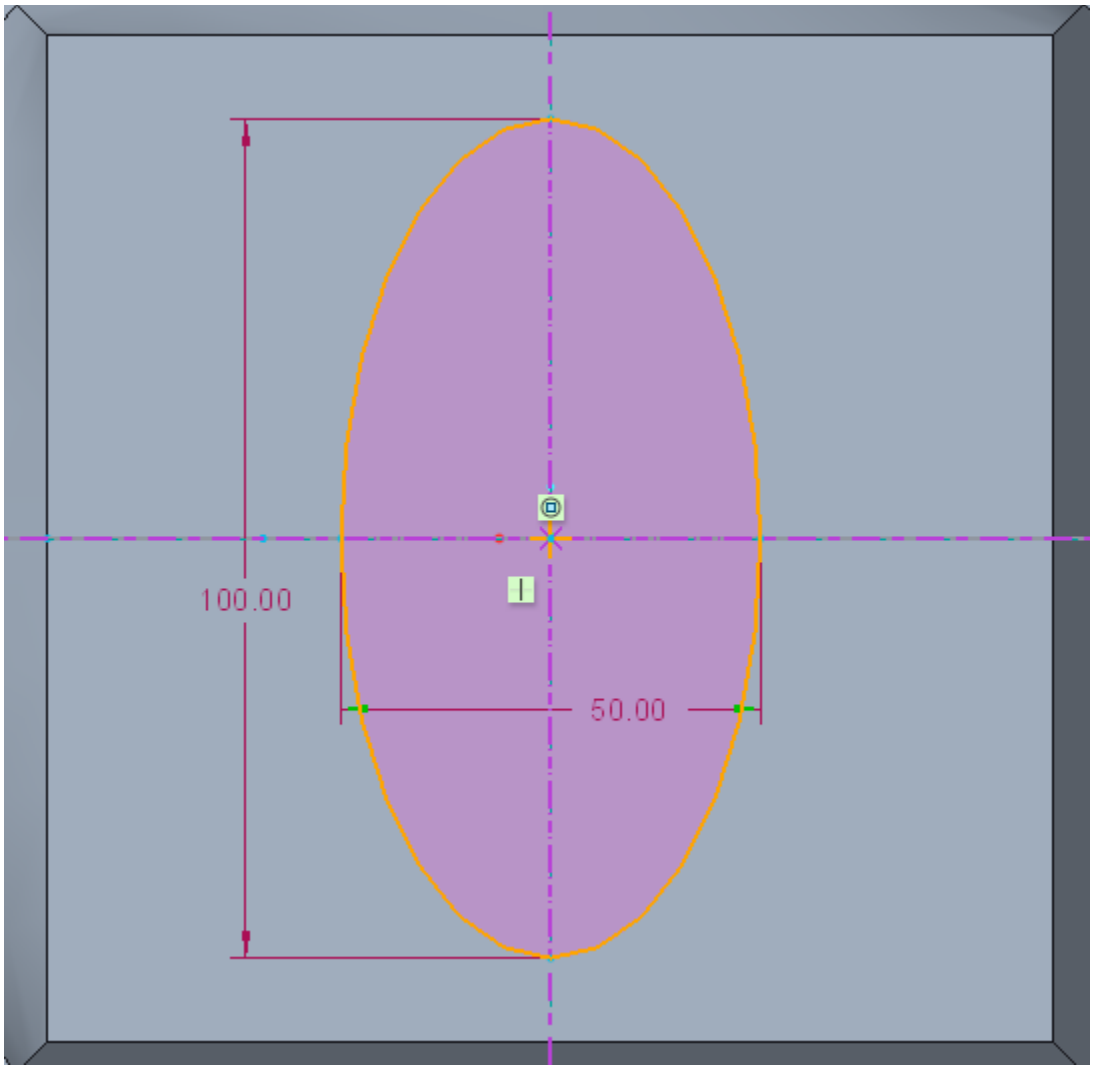

Figure 31: Sketched ellipse.

The sweep preview should look like in Figure 32. Accept the feature.

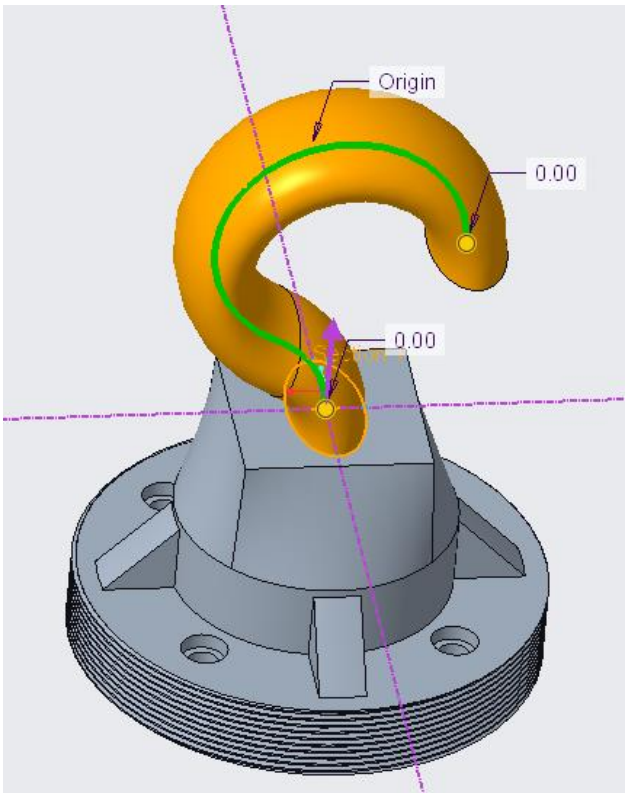

Figure 32: A preview of the Sweep feature.

Suppressing Features

Let's be honest: our hook looks terrible! The real hook profile is more complex than what we just created. Suppress the sweep by selecting Sweep 1 from the model tree, click **RMB** and select **Suppress** ( ) from the menu. The suppressing is like deleting, but with an option to undo it (there is a command unsuppress). Be sure, that the Sketch 1 is not suppressed; we use it to create a better hook.

Using Swept Blend

Swept Blend is a tool that combines the trajectory of the *Sweep* to the cross-section control of the *Blend*. Select **Swept Blend** ( ) from the *Shapes* group. Select the **Sketch 1** (if needed) to be as a trajectory. Cross-sections can be sketched in the places where line (or arc) ends. The plan is to create a shape where we have five different cross sections. To make places

where to put cross-sections, *Point* tool can be used. Select **Point** ( ) from the right side of the dashboard (Figure 33).

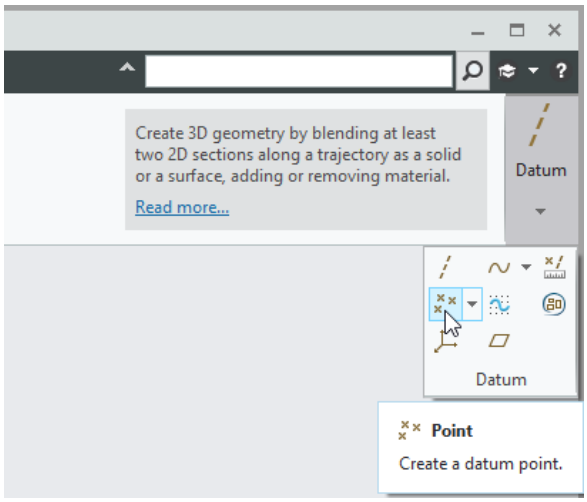

Figure 33: Selecting Point tool from Datum from the dashboard.

A *Datum Point* window appears. Select Sketch 1's biggest circle and create two points there (Figure 34 and Figure 35).

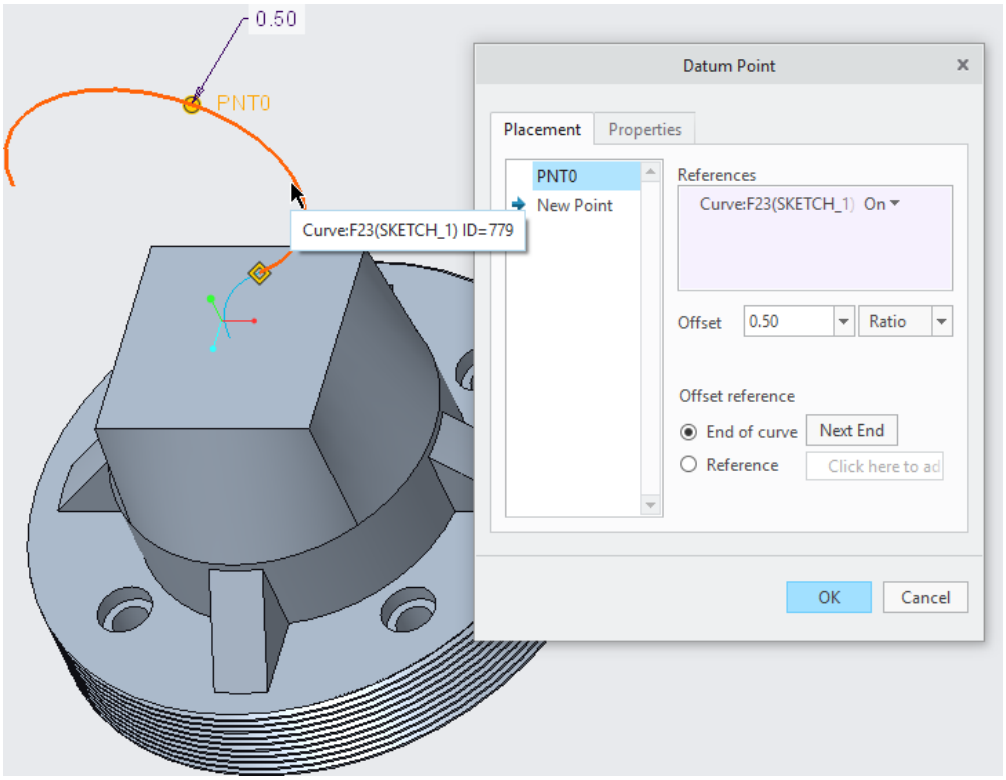

Figure 34: First point created and selected arc highlighted.

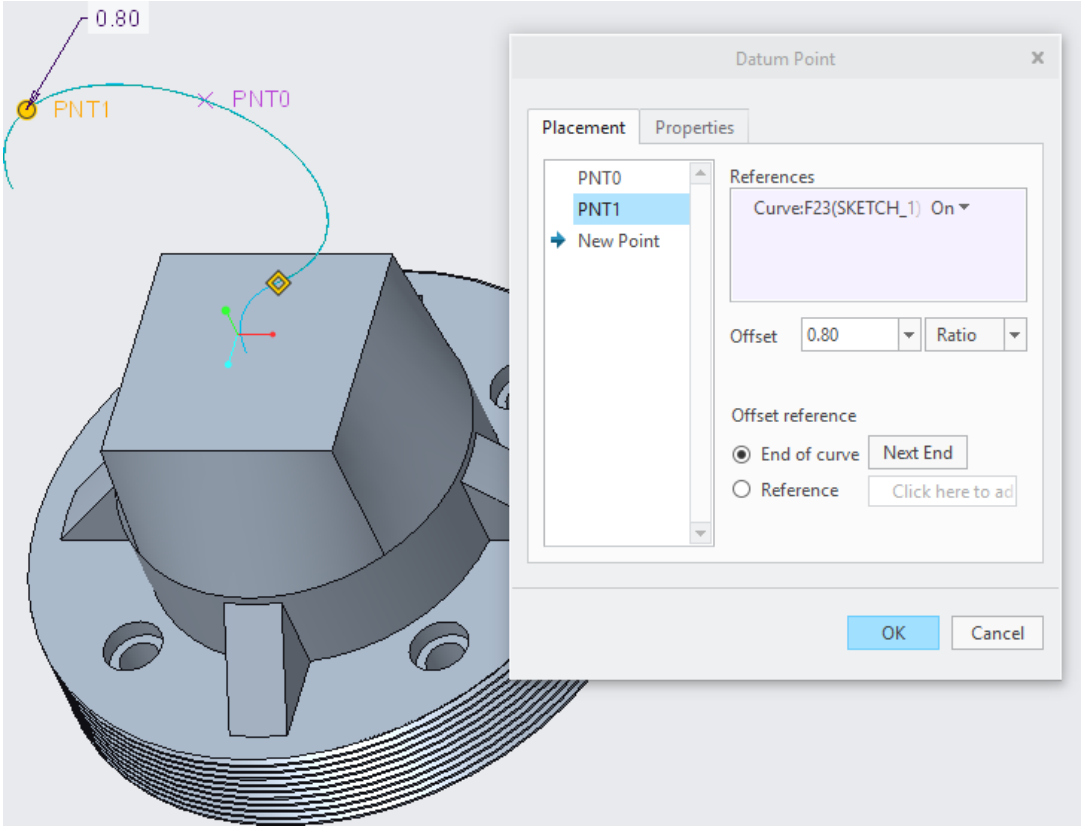

Figure 35: Second point created. The box with black circle symbolizes the start point of the arc. It can be changed by selecting *Next End*.

When creating datum items using the dashboard of some tool, the created items are internal items to that tool. That means, that the values can be changed by defining Swept Blend (i.e. they are grouped together) and that these points are only visible to the parent feature.

When ready, accept the point by selecting **OK**.

Creating cross-sections

To see the previously created points, set **Point Display** ( ) from *Graphics* toolbar on. To continue with *Swept Blend*, select **Resume** ( ) from the dashboard (Figure 36).

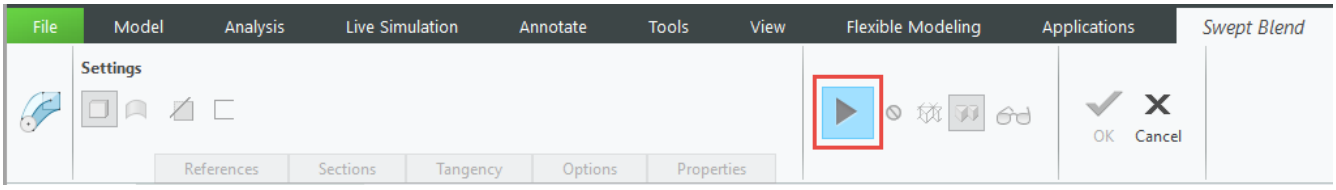

Figure 36: Resume button in the Swept Blend dashboard.

Open the **Sections** tab from the dashboard. Then select the start point of the Sketch 1 (bottom point, should be highlighted in green) where the arrow is (if needed to change, select some other point). Click **Sketch** to make a sketch to that point (Figure 37). When the sketch is ready, accept it.

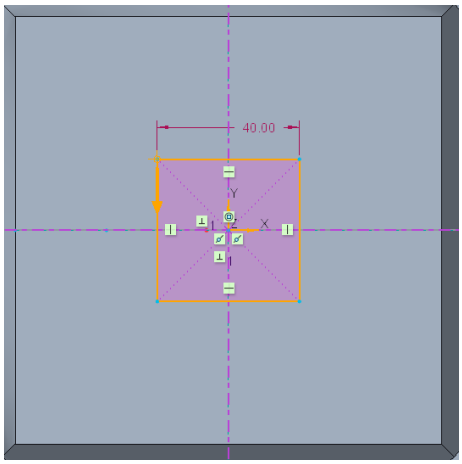

Figure 37: Section 1 for Swept Blend.

In the **Sections** tab, click **Insert** to create a new sketch. Select the end point of the smaller arc in the Sketch 1 as a sketch location (Figure 38). Create a sketch as seen in Figure 39. When ready, accept that sketch.

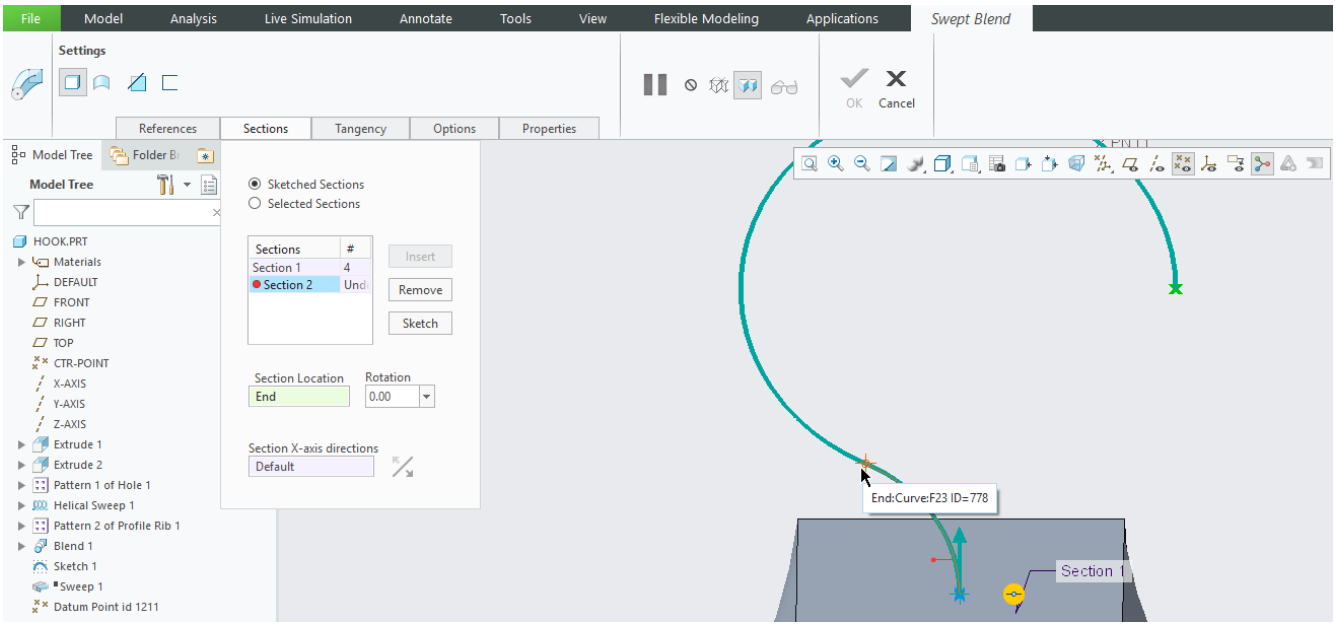

Figure 38: Selecting a point for Section 2.

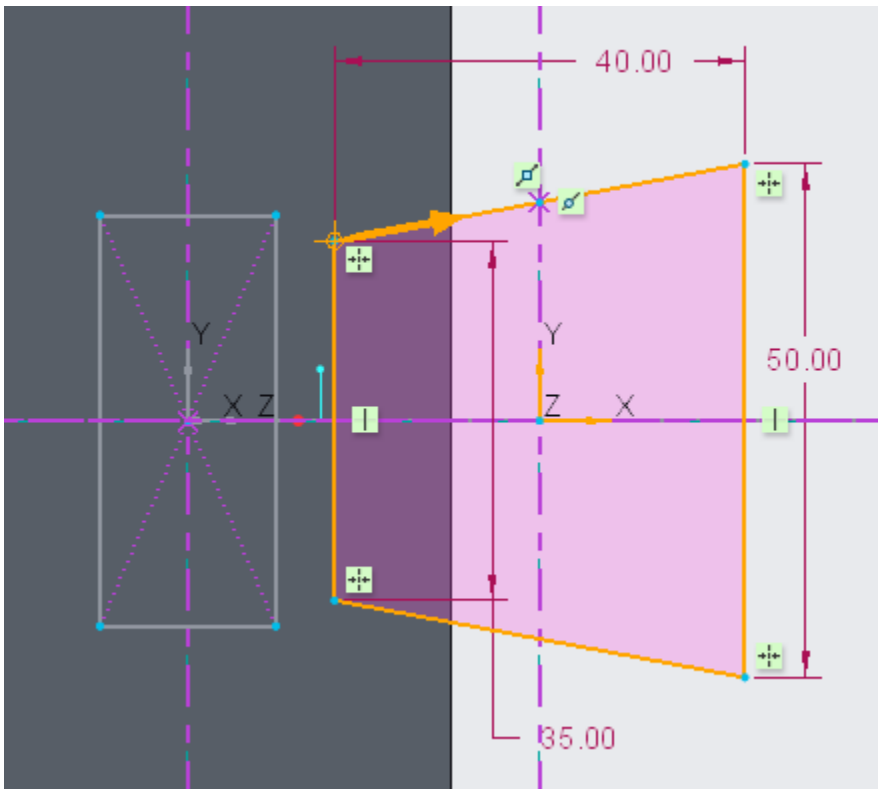

Figure 39: Sketch for Section 2. Notice the symmetry (centerline!) and mid-point constraints.

Now we have two cross-sections defined, three to go. Create Section 3 to **PNT0** (Figure 40) and Section 4 to **PNT1** (Figure 41).

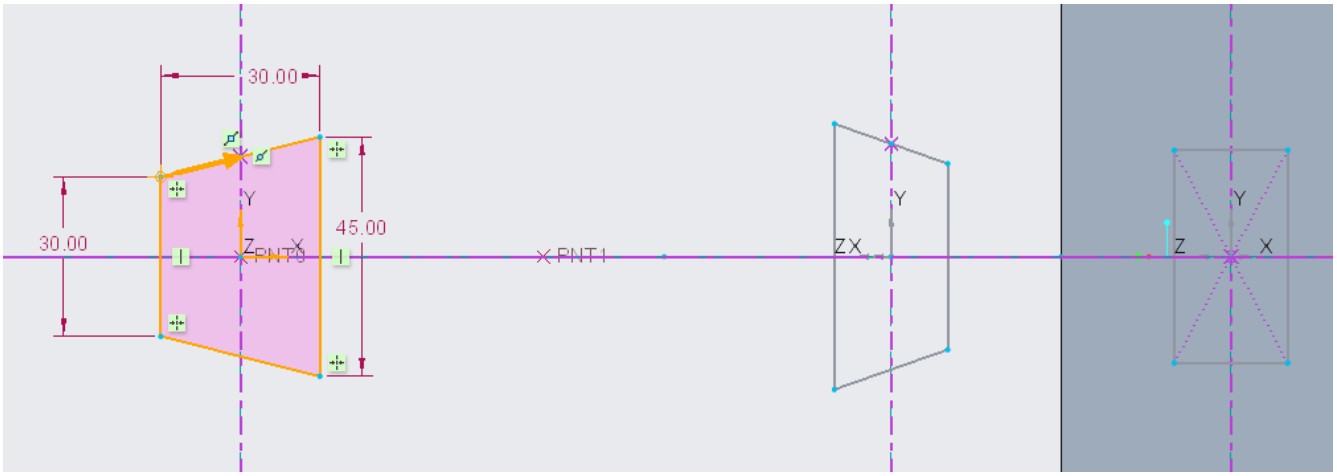

Figure 40: Sketch for Section 3.

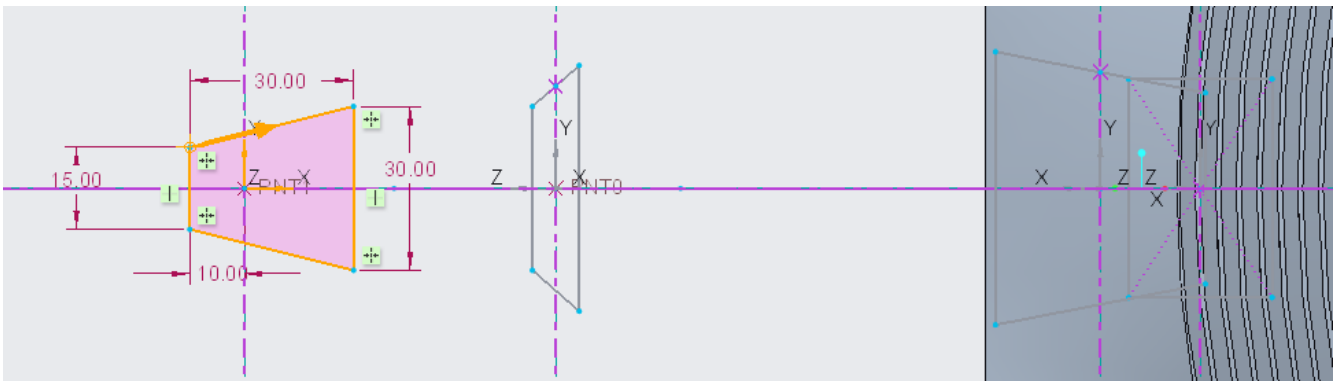

Figure 41: Sketch for Section 4.

One last section needs to be defined. As a last section (Section 5) we create just a point there! This works, because the shape is defined by lines that go through defined points in the sketches (point order in the sketches is defined with an arrow), so all lines will end in one place. So, as Section 5, sketch a **Point** (⁘)!

When the point is created, the preview should look like in Figure 42. Notice that the longer side in the sketch points to inside the circle (trajectory). If needed, through *Sections* tab all sketches can be redefined. If preview looks fine, accept the feature.

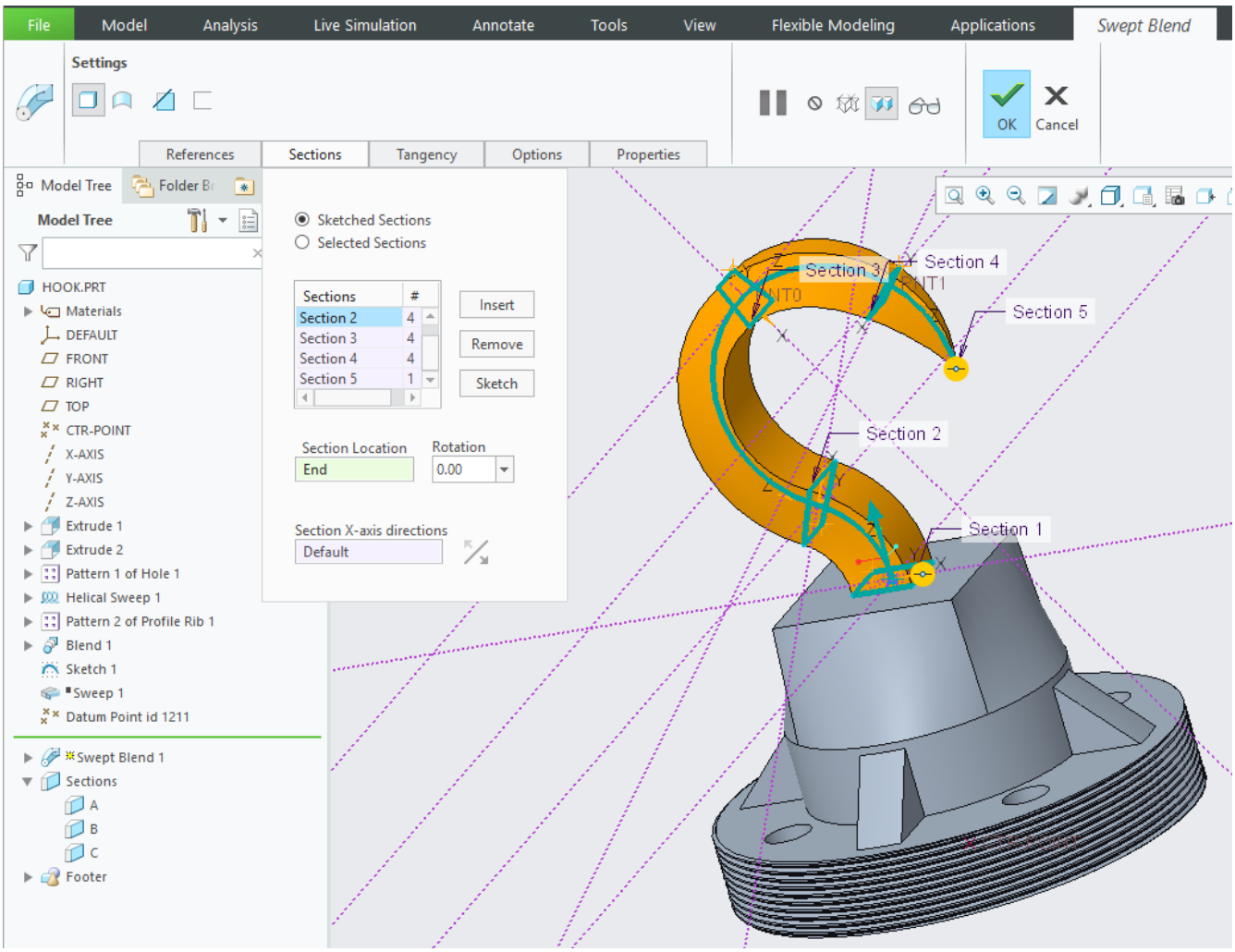

Figure 42: Ready to accept Swept Blend.

Your model should look like the one in the first page (Figure 1). Time to **save** your model.

With Swept Blend you can create very complex features; you need one trajectory and at least two sketched sections.

Parameters and Relations

Create the following parameters:

Parameter Name	Type	Value
OFFSET_VALUE	Real Number	60
BLEND_HEIGHT	Real Number	75
HOOK_RADIUS	Real Number	80

Create relations as following (Figure 43).

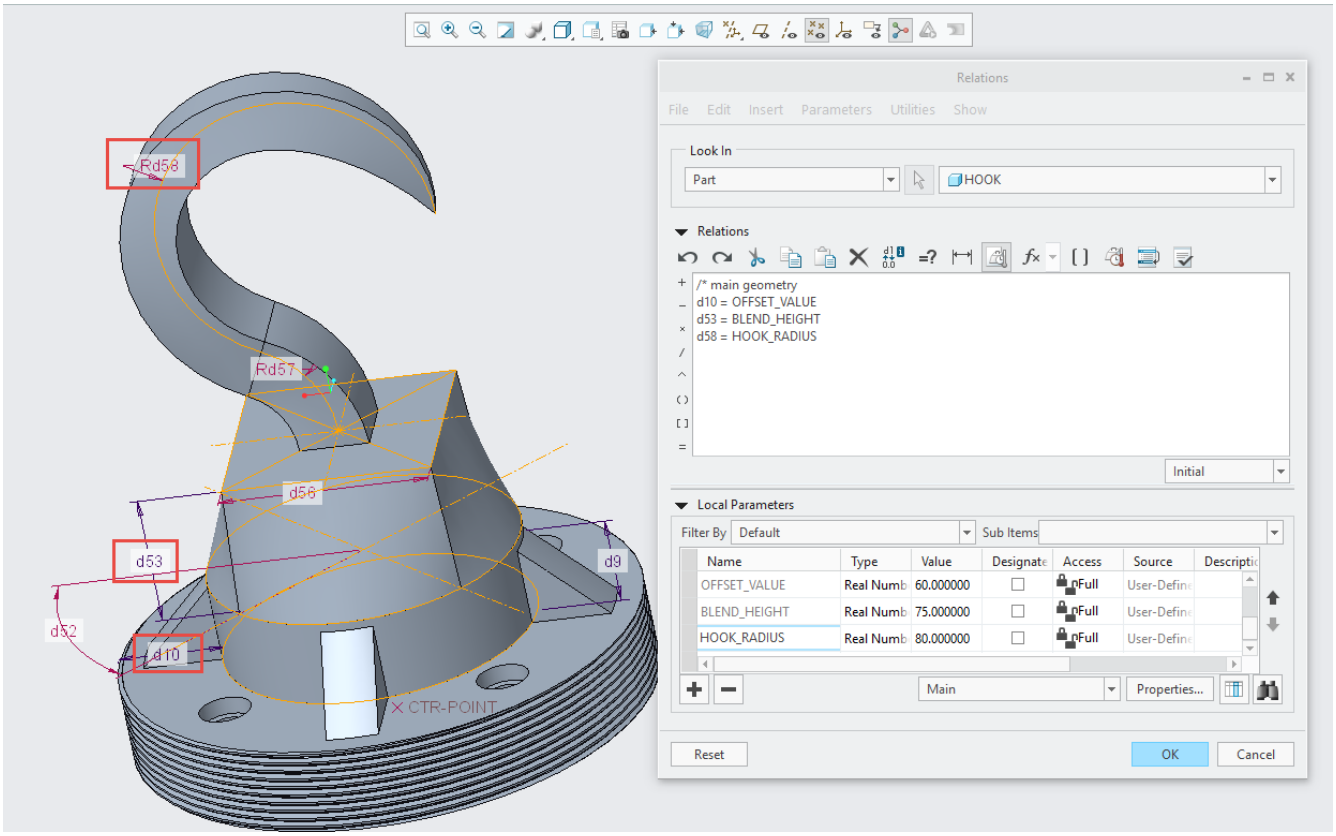

Figure 43: Created parameters and relations. You may have different dimension names!

This concludes this exercise. Time to save your model.