Aalto University
School of Engineering

NX.1 – MODELING A BELT WHEEL

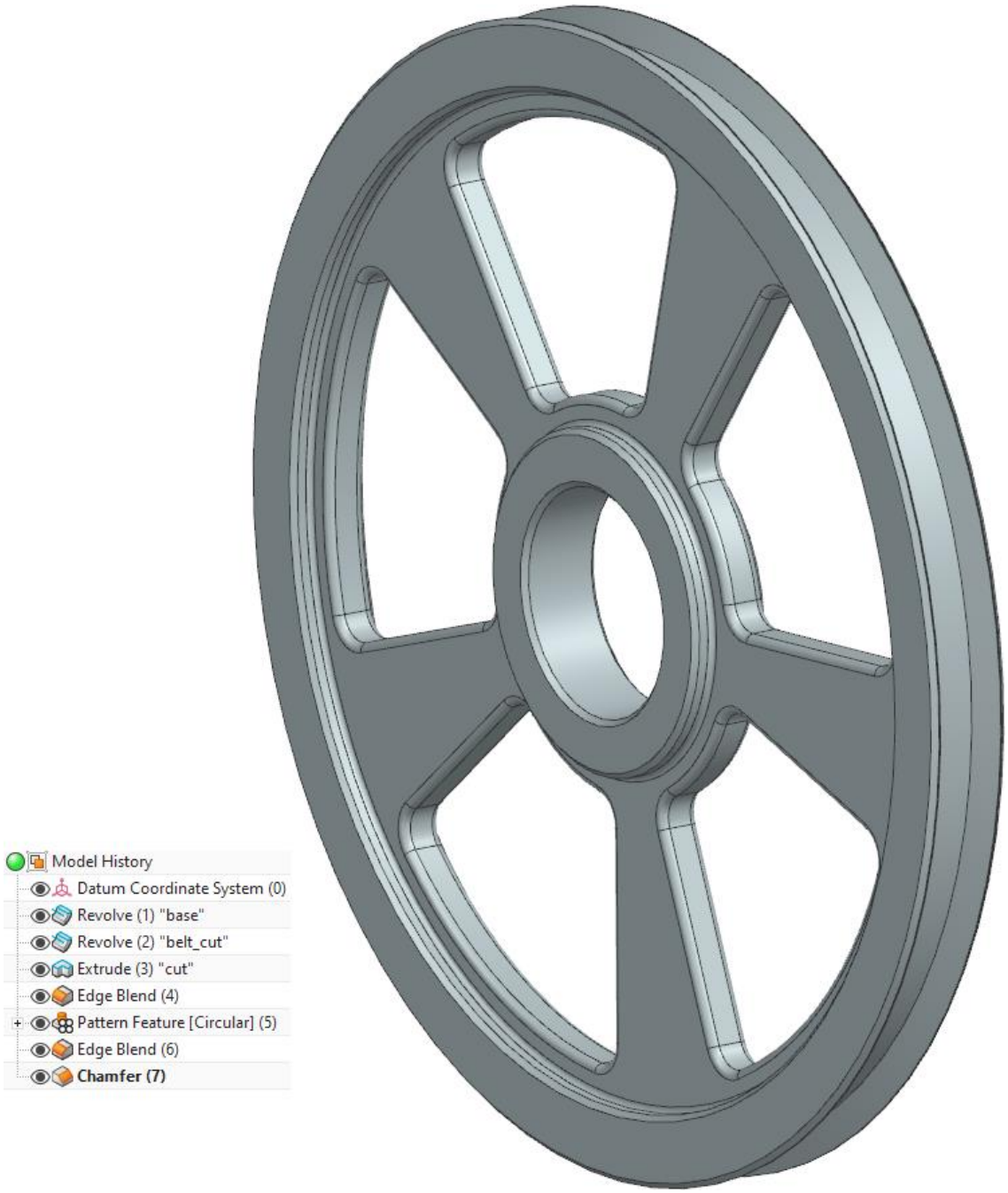

Figure 1: The belt wheel and its Model History.

Learning Targets

In this exercise you will learn:

- ✓ Create a part
- ✓ Using revolve and extrude to add and remove material
- ✓ Using references
- ✓ Using edge blend and chamfer
- ✓ Using pattern
- ✓ Get used with NX

Utilized program version in this exercise is Siemens NX 2027.

Some terms utilized:

- LMB: Left Mouse Button
- MMB: Middle Mouse Button (press the wheel)
- RMB: Right Mouse Button.

Getting Started

Start NX from **Start menu** (, use *Start menu* search).

When the program starts, it opens a *Discovery Center* screen. Select **New** ( ) from top-left or *New* from File-menu. Select **Model** from **Model** tab and give a name to your model (in this case belt_wheel) and select a working folder (Z:\Documents\ for example) (Figure 2). Click **OK**.

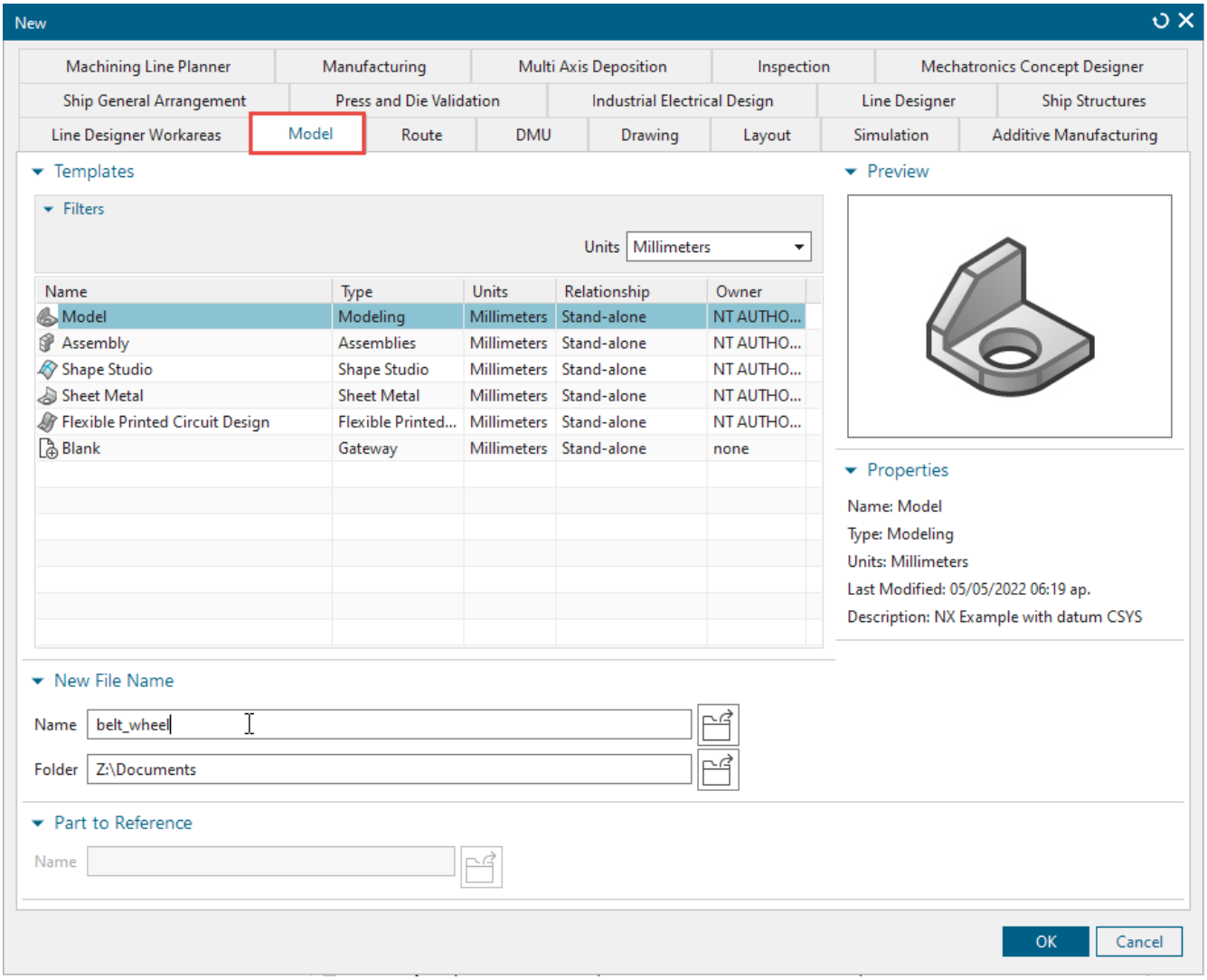

Figure 2: Creating a new model.

Chacing the User Role

NX has several user roles for different modeling purpouses. In this exercise, we will utilize **Advanced** role. From the left side, select **Roles → Content → Advanced** (Figure 3).

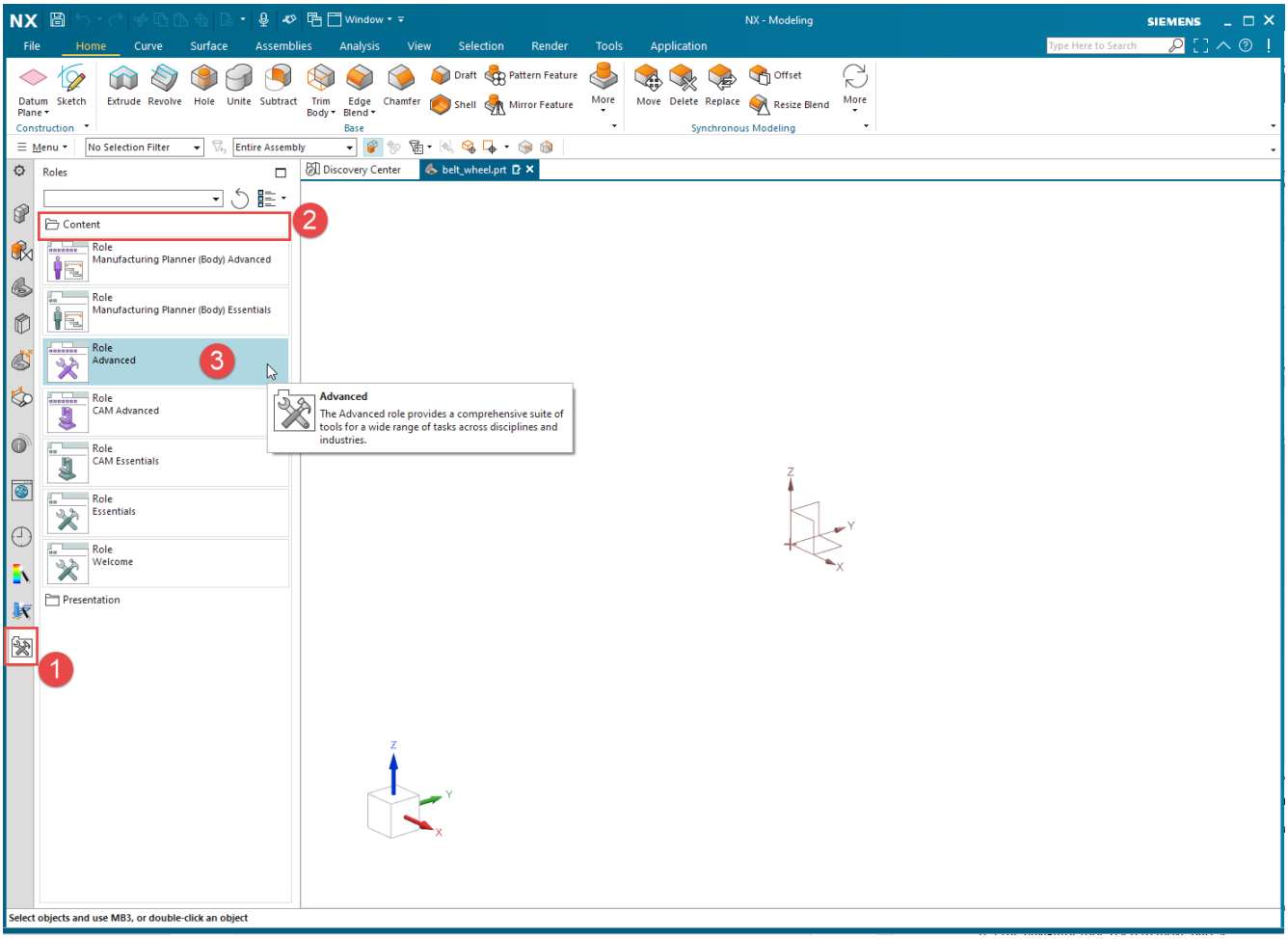

Figure 3: Selecting a new user role.

Go back to **Parn Navigator** () from the left side.

The User Interface

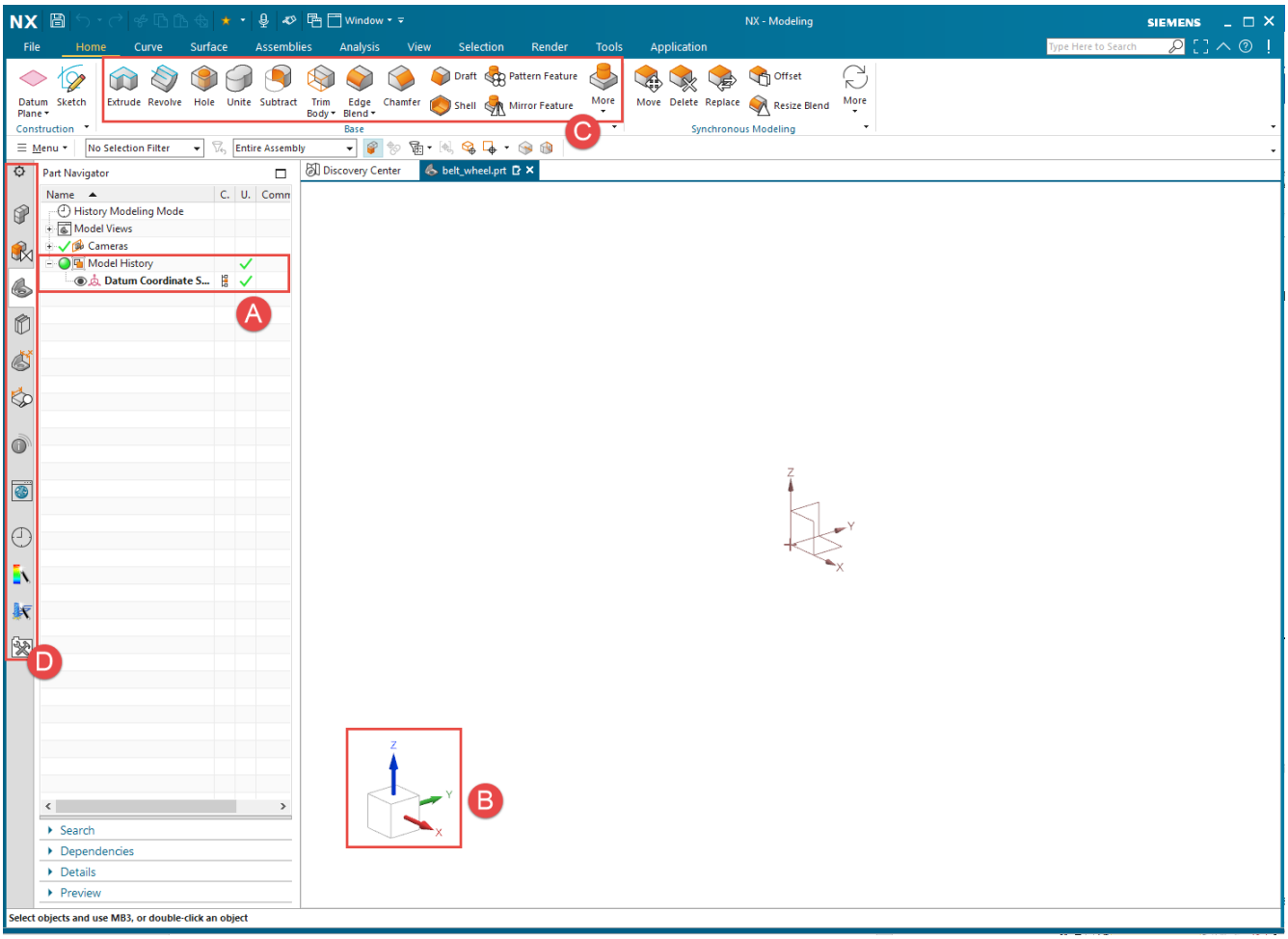

Figure 4: The user Interface of the Model mode.

The basic user interface of the modeling mode is presented in Figure 4.

Areas needed in this exercise are highlighted in red:

A. The Part Navigator (model tree) with basic planes (Datum Coordinate).
B. The navigator tool. Used to move part.
C. Material creating and editing tools. The tools that can create or remove material.
D. Resource bar. To change visible models.

Moving a View

The view orientation can be controlled by the mouse and keyboard. Please notice, that the datum planes in NX are always the same sized.

Panning

Hold **Shift** and **MMB** to pan the view.

Zooming

Use mouse scroll to zoom gradually.

An alternative option is to hold **Ctrl** and **MMB**.

Rotating

Hold **MMB** to rotate view.

Other

Fit ( , in *View* tab) is a very useful tool, can be activated with **CTRL+F**. **Trimetric** () view can be activated by pressing **Home**.

Using Revolve

Predefinitions

The first task is to create a revolve that represents the main geometry. Select **Revolve** () from *Base* group (Figure 5). The *Revolve* window appears. Create a sketch by selecting  symbol from *Section* field (Figure 6).

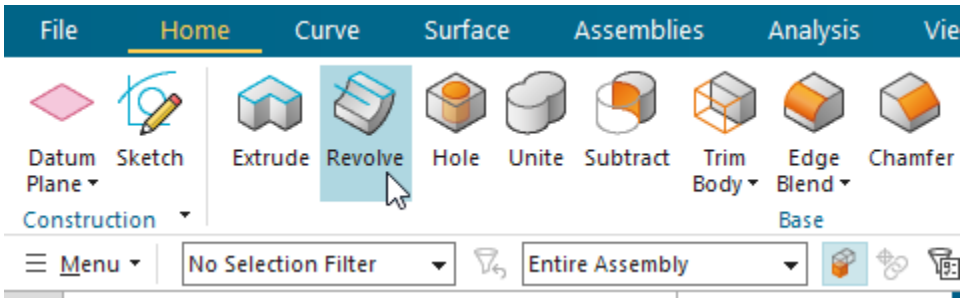

Figure 5: Selecting Revolve from Base group.

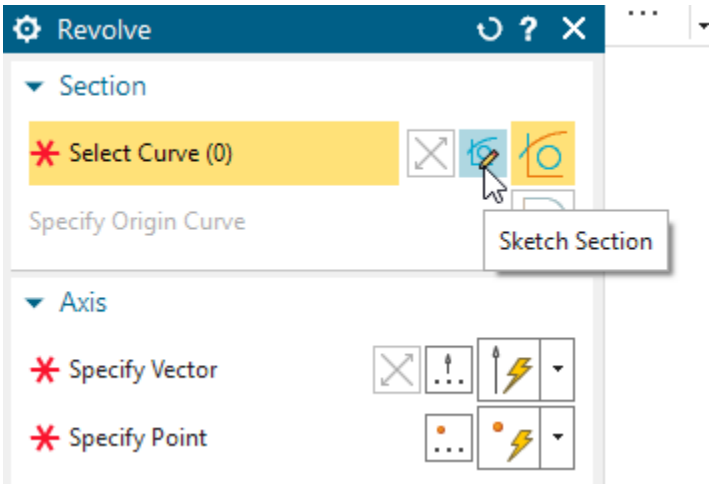

Figure 6: Revolve window and *Sketch Section* button highlighted.

Sketching

Next select **YZ Plane** (Right) as a sketching plane from the graphical area (Figure 7). Press **OK** when selected to start the sketching mode.

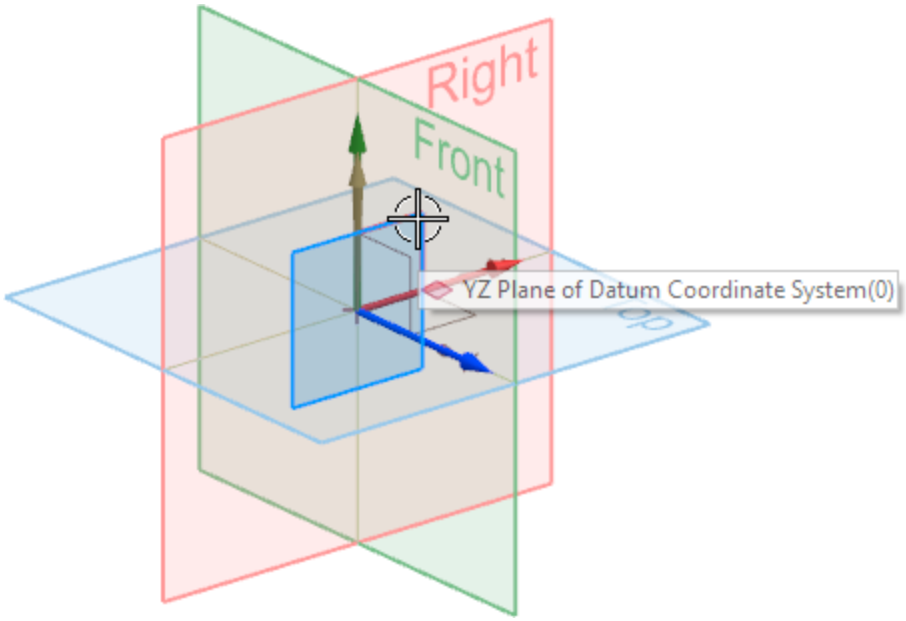

Figure 7: Selecting YZ Plane (Right) as a sketching plane.

Shape

View will be reoriented into 2D, and Sketch mode will be activated. Select Profile tool from Curve group (Figure 8).

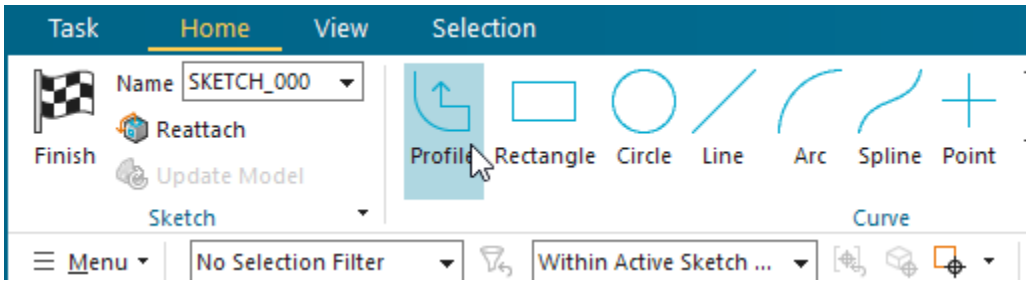

Figure 8: Selecting Profile from Curve group.

Sketch an open profile as seen in Figure 9. You can close the loop by pressing **MMB** and the *Profile* tool by pressing **MMB** second time.

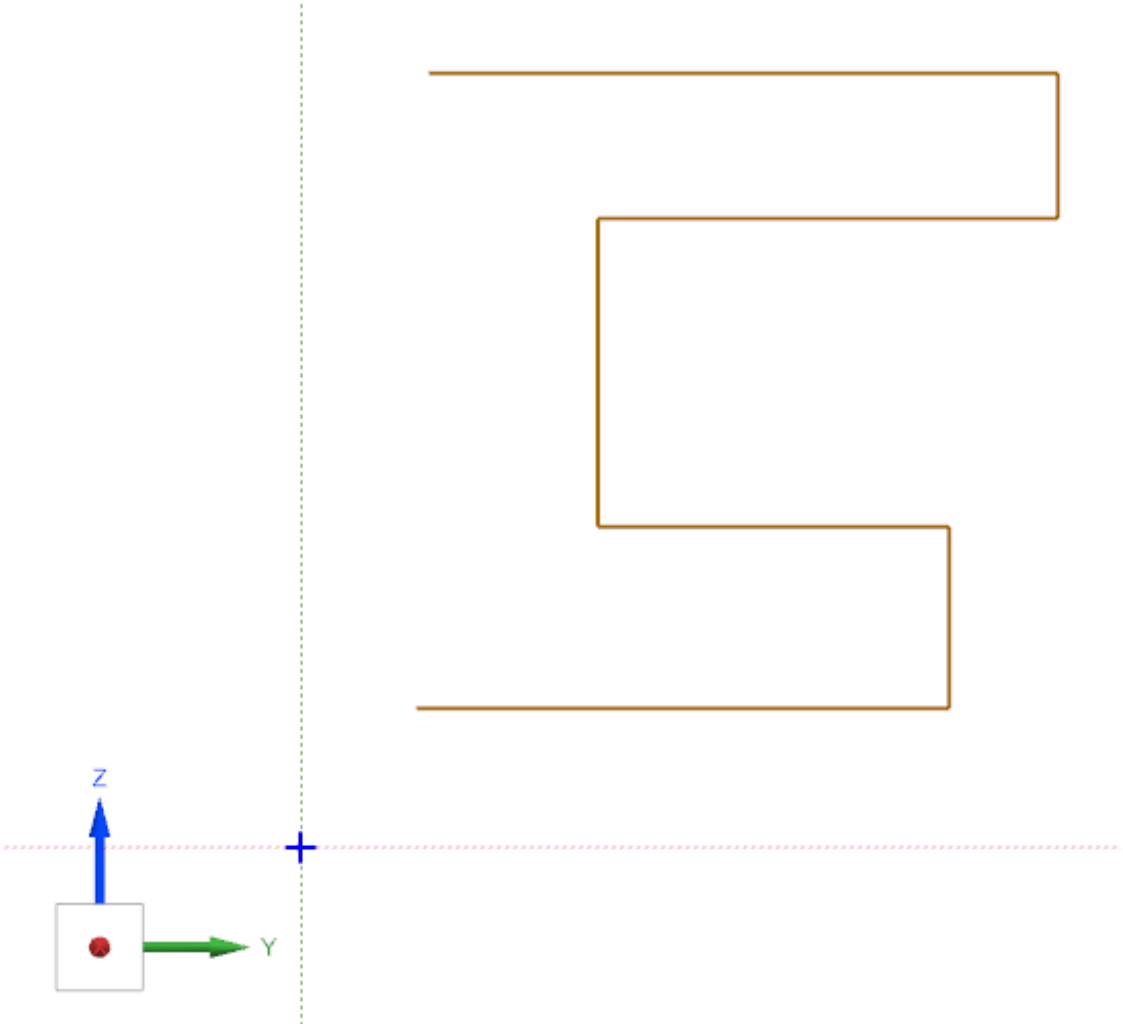

Figure 9: Sketched open profile.

Constrains

On the bottom of the graphical area a notification about movable curves (in this case 7) is given (Figure 10). This means that sketch is not fully defined and has several degrees of freedom. To fix this, several geometric constraints and dimensions are needed.

Sketch is partially defined with 7 movable curve(s)

Figure 10: Text on the bottom of graphical area.

First we attach two horizontal open-ended lines (top and bottom line in Figure 9) to start on a vertical Z-axis (blue axis in the orientation "box"). Constraints (inlcudind dimensions) are created first by selecting entities (points, lines, etc.) and then selecting suitable constraint from the list.

First, select the open-ened point from the topmost horizontal line (Figure 11).

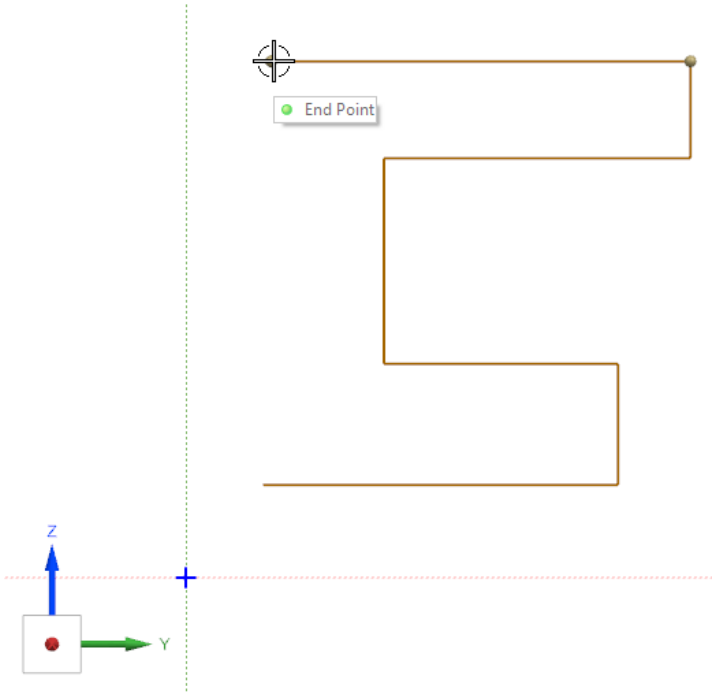

Figure 11: Selecting the End Point from topmost line (in green).

Then, select the vertical reference line and change recommented constraint type (dimension) into **Coincident** () from sketching region top bar (Figure 12).

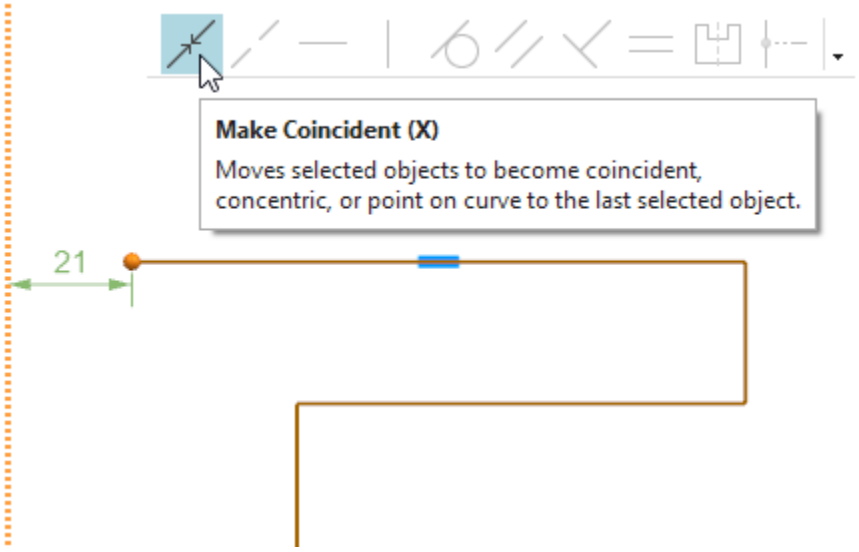

Figure 12:Selecting *Make Coincident* to replace automatically offered dimension.

Bottom horizontal line will be attached into vertical reference line with different workflow. This will create the same result. When nothing is selected, select **Make Coincident** ( ) from the top sketching bar (Figure 13).

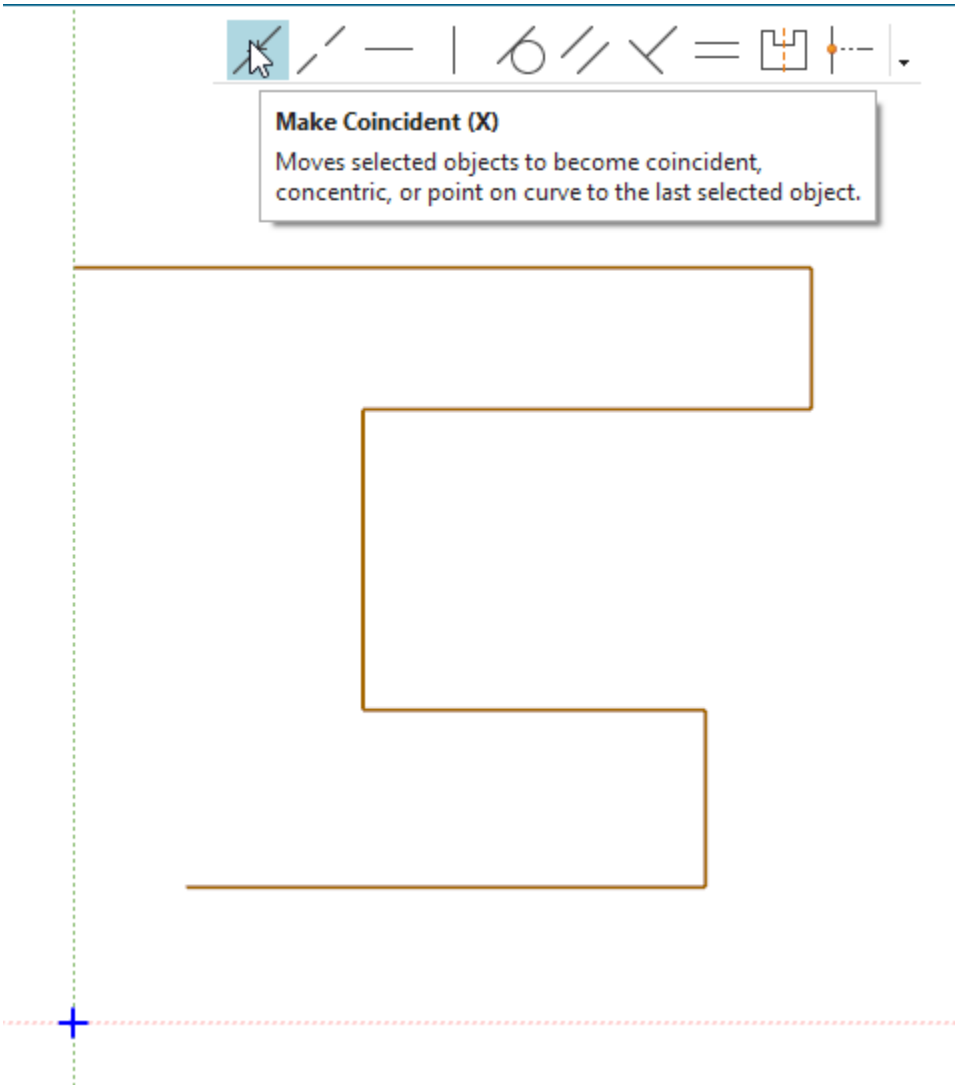

Figure 13: Selecting Make Coincident from the toolbar.

As *Motion Curve or Point*, select the end point of bottom horizontal line, and as *Sationary Curve or Point*, select the vertical horizontal line (Figure 14). **OK** to accept constraint.

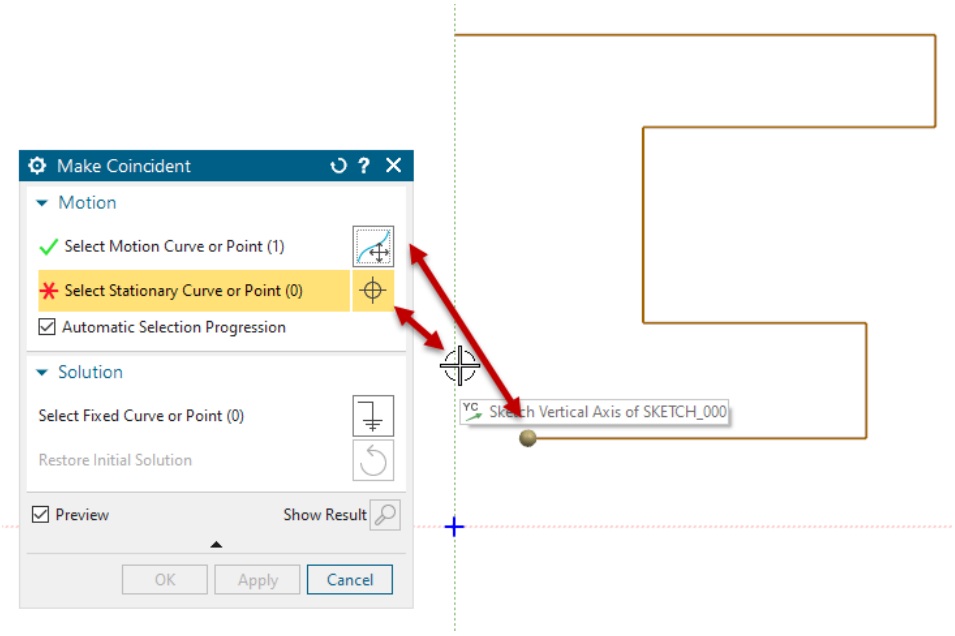

Figure 14: Make Coincident window, point selected and selecting vertical reference line.

Next an *Equal* (=) contrain is added. Select the two lines highlighted in the picture and select **Make Equal** (Figure 15).

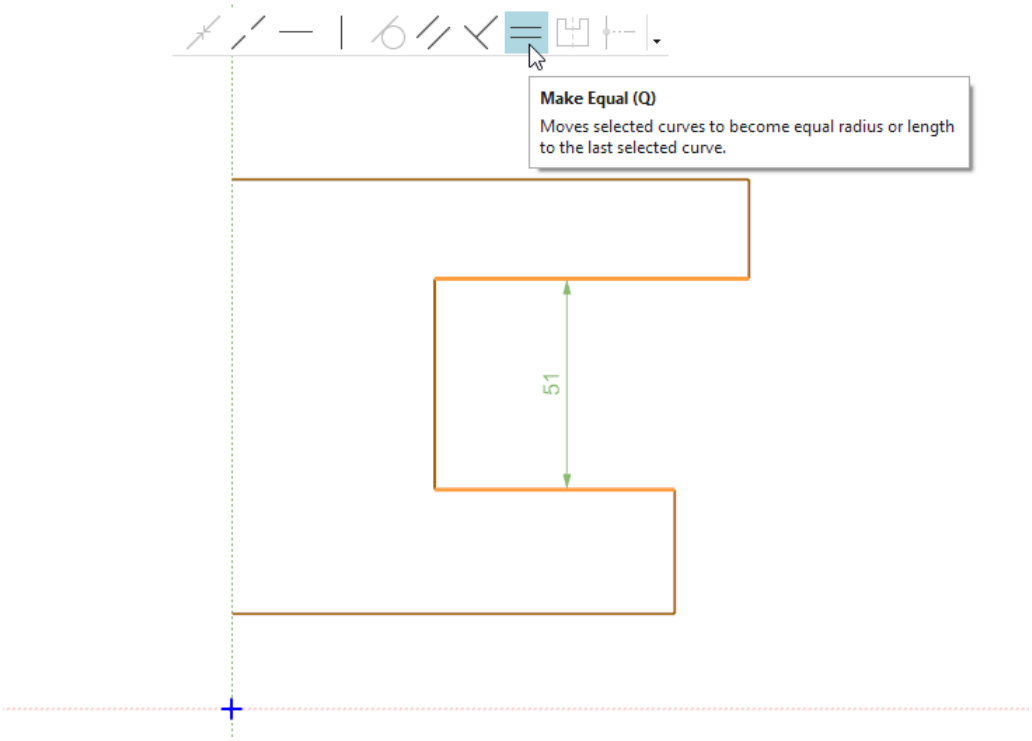

Figure 15: Two lines selected (highlighted in orange) and selectin Make Equal.

Mirroring

Theaim is to create an I-profile and only half of it is sketched. Next the existing curves are mirrored along the Y-axis. Select all the sketched lines and select **Mirror** ( ) from *Curve* group.

Select **Select Centerline** and select *Sketch Vertical Axis* as a mirroring centerline as seen in Figure 16. If the preview looks valid, press **OK**. Now you should have an I-shaped sketch. Notice the shaded closed surface in light blue.

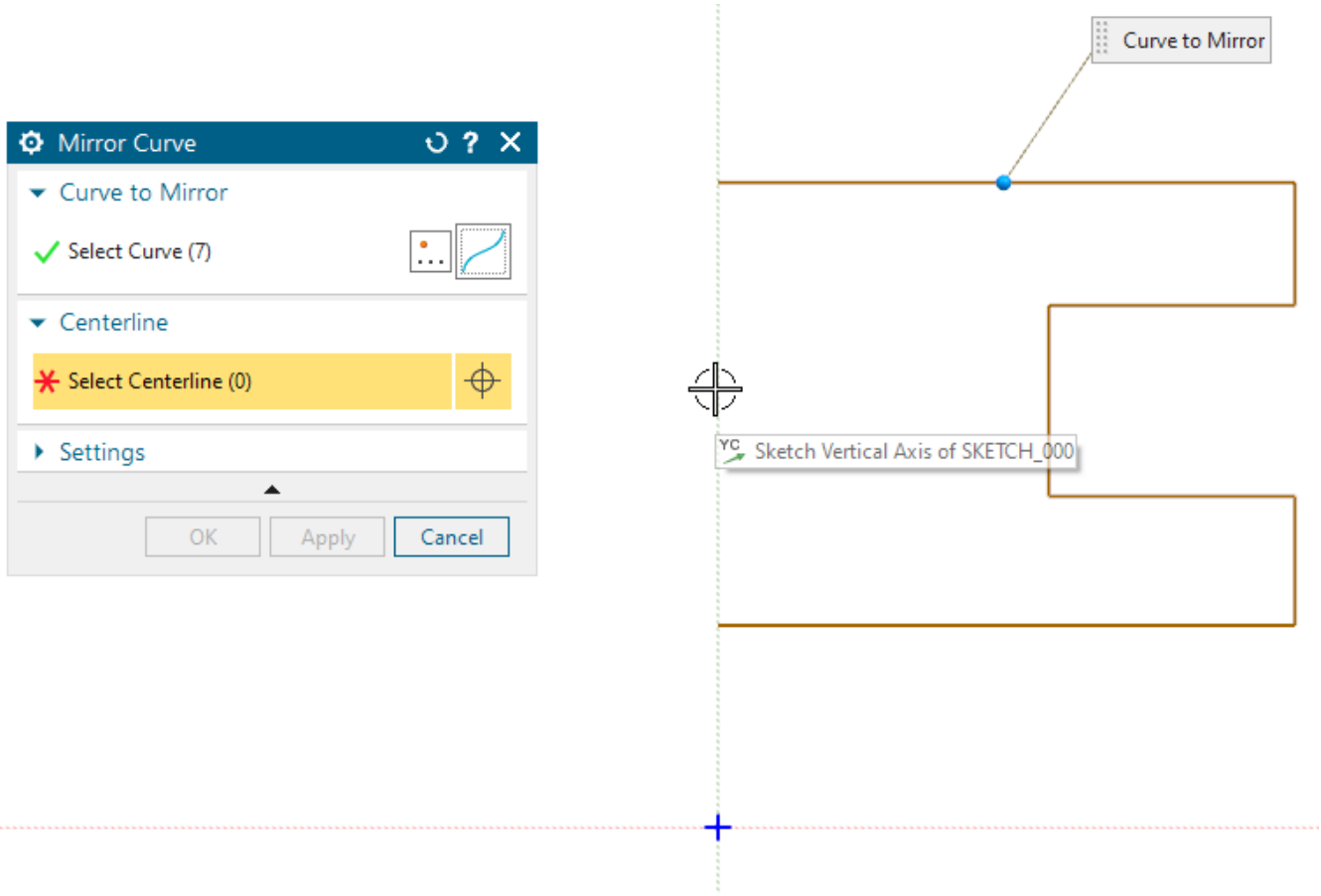

Figure 16: Selecting vertical axis as Centerline.

Dimensions

The main shape is now defined, so the next task is to define dimensions. Workflow of creating dimensions is like creating constraints: select two lines/points and select the dimensions you want to create.

Select the bottom horizontal line and then horizontal symmetry line (left in Figure 17). Then select the offered dimension to lock it and thus create it (right in Figure 17)

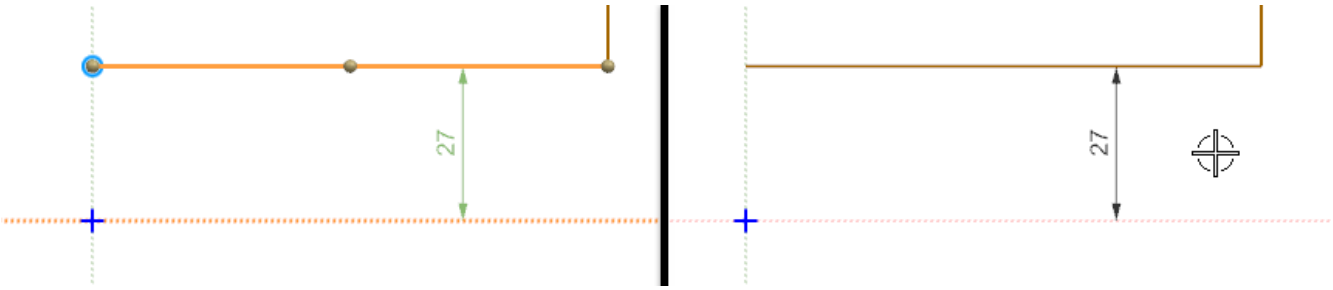

Figure 17: On left: Line and reference selected, program offers dimension between (green). On right, dimension selected and accepted (black).

The geometry will be created with revolute tool, so it means that this created dimension can be seen as radius dimension. It can be leaved as it is, but for design intent and further reuse purpose a diameter dimension is preferred. To create a diameter dimension, select the previously created dimension, **RMB** and select **Convert to Half Diameter** (Figure 18).

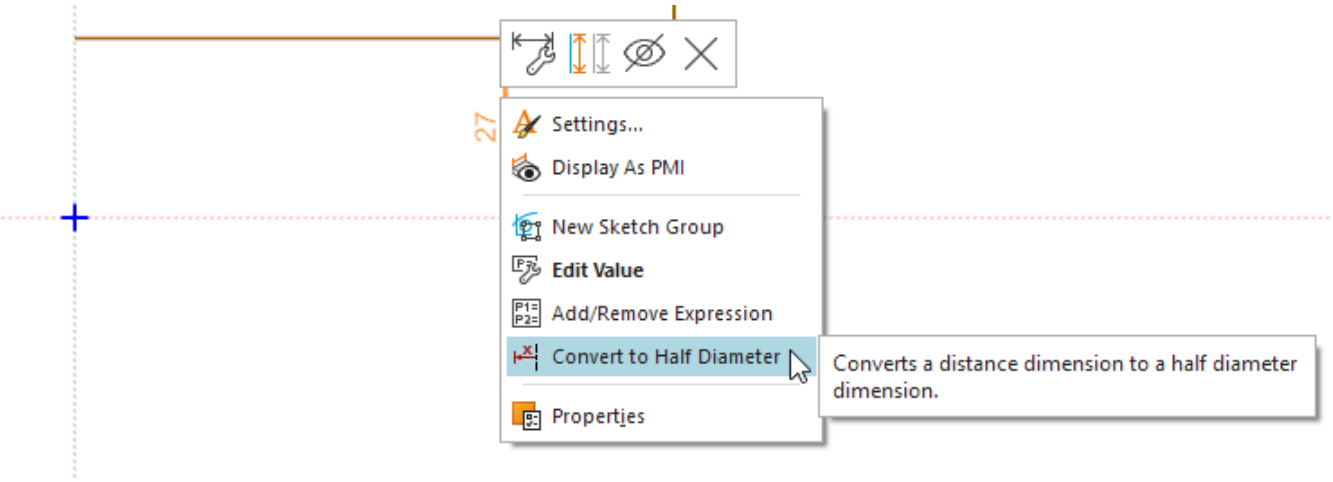

Figure 18: Changing dimension to Half Diameter.

Using the dimension creating workflow, create the rest needed dimensions as seen in Figure 19. When ready, select **Finish** (🏁) from *Sketch* group.

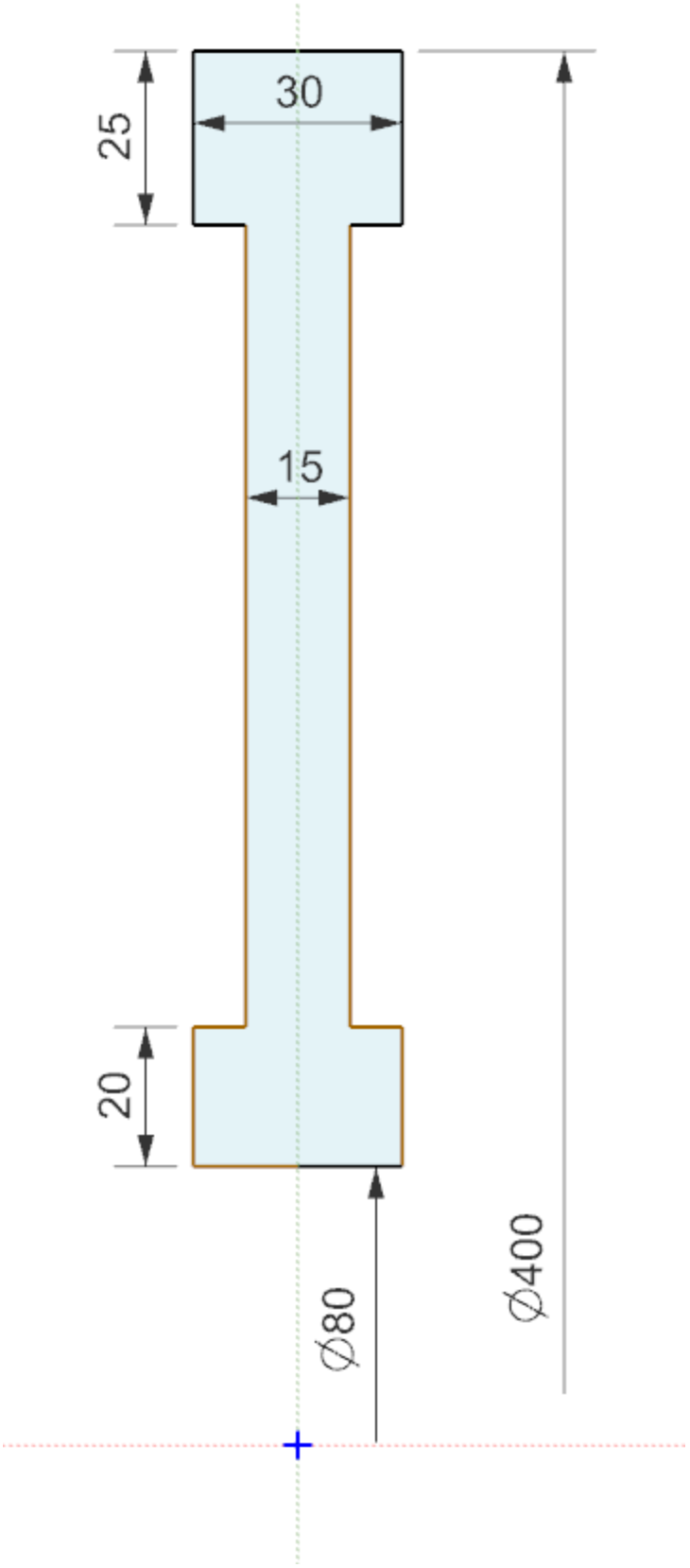

Figure 19: Finalized sketch.

Finalizing Revolve feature

The sketch mode is closed and Modelin mode is back on. Next Revolute feature's Axis needs to be defined. Click **Specify Vector** from *Axis* field and select *Y Axis* (Figure 20).

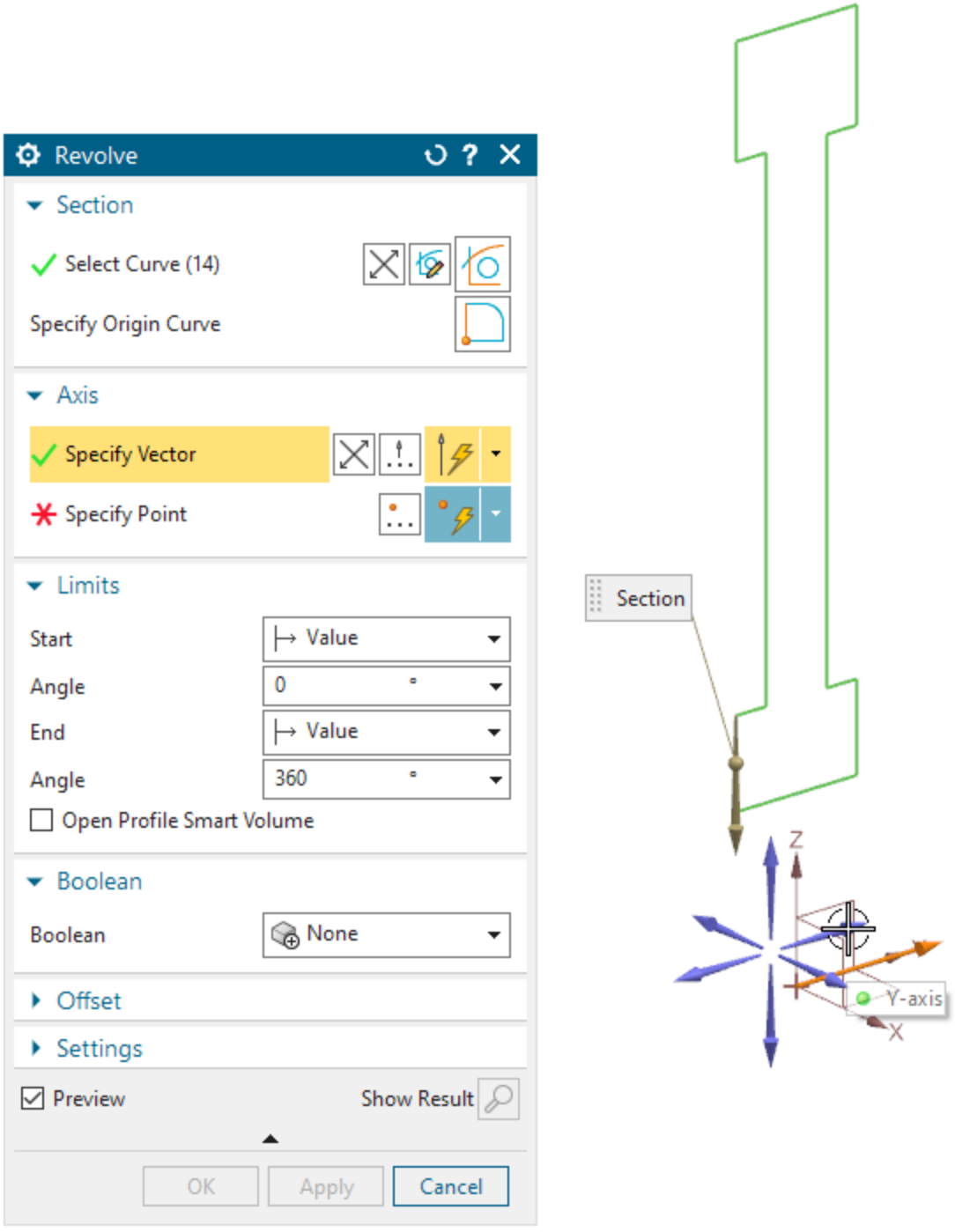

Figure 20: Selecting Axis for revolve feature.

As **Specify Point**, select the coordination point (Origin of Coordination System) as seen in Figure 21.

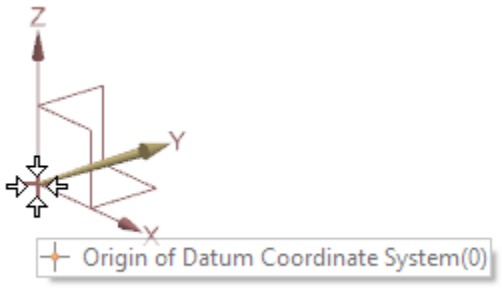

Figure 21: Selecting the point for revolve feature.

When the point is selected, the geometry preview will be activated. All values should be fine (360 deg rotation), just click **OK** to accept the revolve feature. Next, select **Revolve (1)** from *Model History*, **RMB** and select **Rename**. Give name *base* and hit **Enter**. Save your model (**CTRE + S** or).

Using Revolve to Cut Material

Next a cut for the belt is created. Select **Revolve** (, *Base* group). A *Revolve* window opens. Select **Sketch Section** ( ) from the *Section* field and select *YZ Plane* from the list (press … to open *QuickPick* menu). Click **OK** if needed.

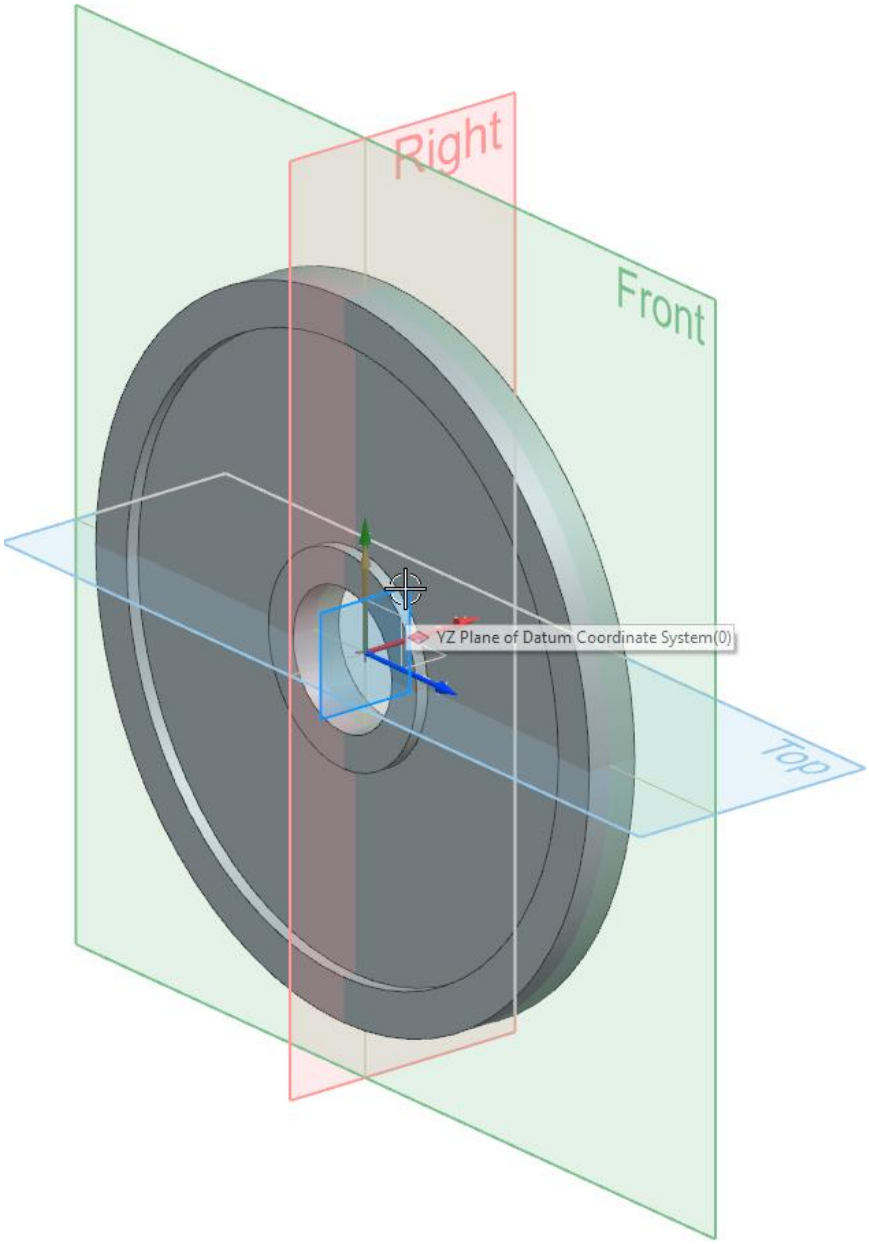

Figure 22: Selecting YZ Plane.

Using sketch references

We want to attach revolve to the top of the *base* geometry allowing fast changes of the main dimensions. Select **Project Curve** () from *Include* group (Figure 23).

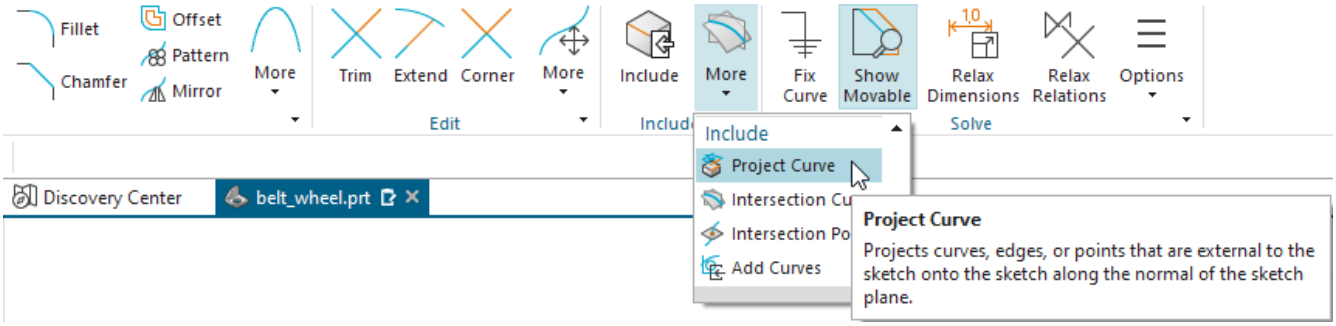

Figure 23: Selecting Project Curve.

Select two vertical edges as seen in Figure 24. Click **OK** to project those edges.

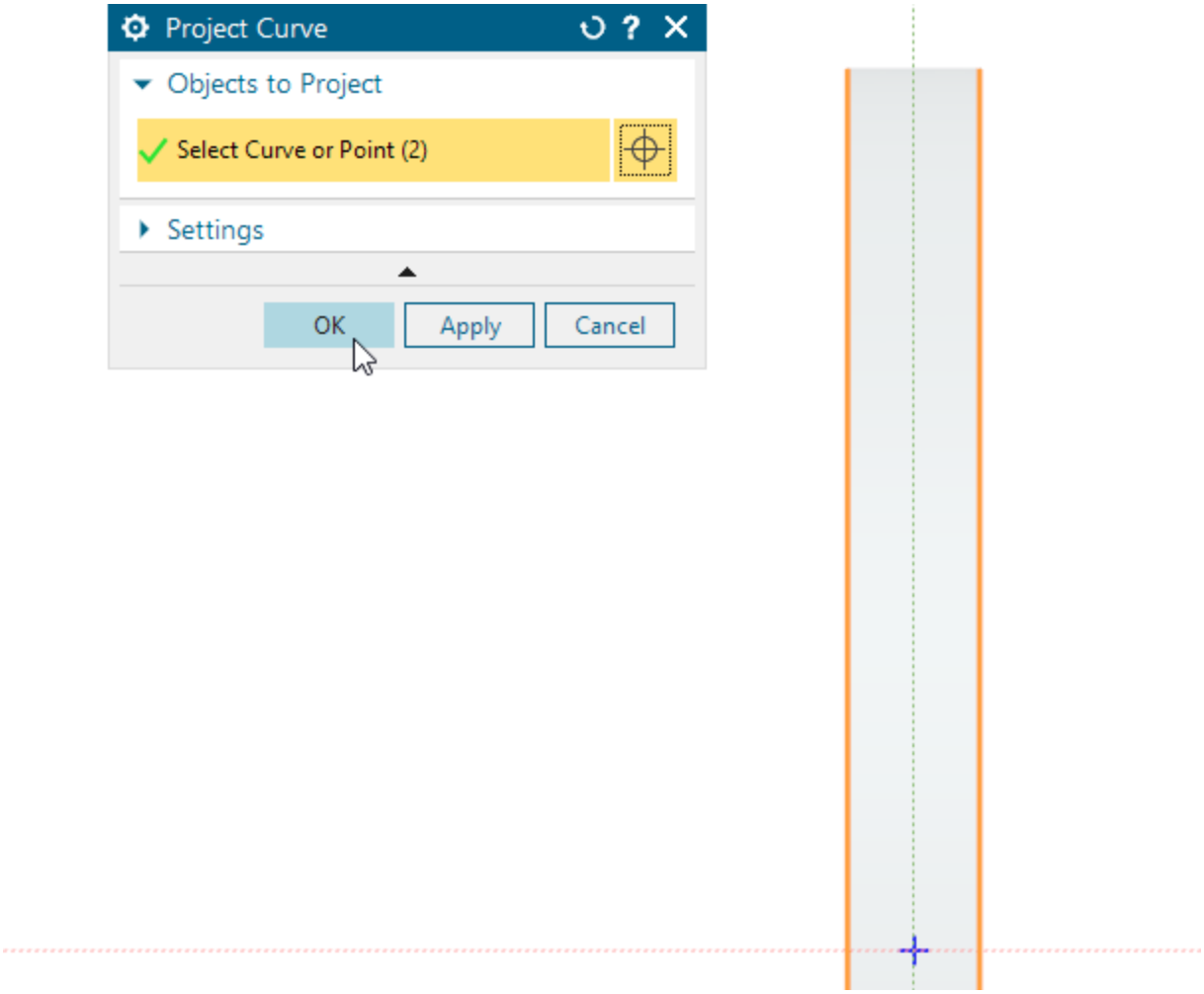

Figure 24: Projected edges (highlighted in orange).

Now there is two lines in the sketch. Select those lines one-by-one (so they are highlighted in orange), hold **RMB** over one of those lines and select **Convert to References** (, Figure 25).

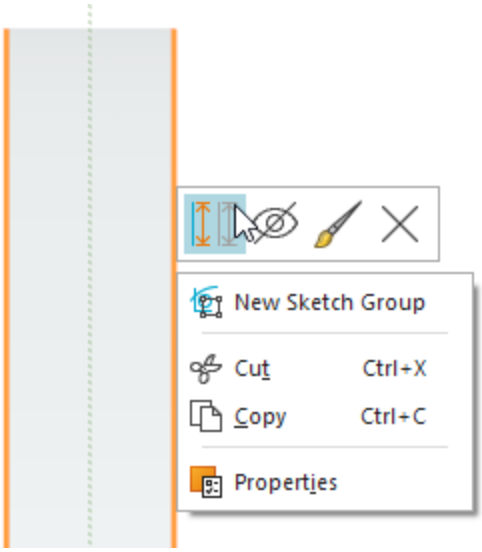

Figure 25: Hovering cursor over Convert to References.

Now there is two reference lines in the sketch. Next, using **Line** (, *Curve* group), sketch a horizontal line on the end of the reference lines (Figure 26) and convert that line to **Reference** ( ).

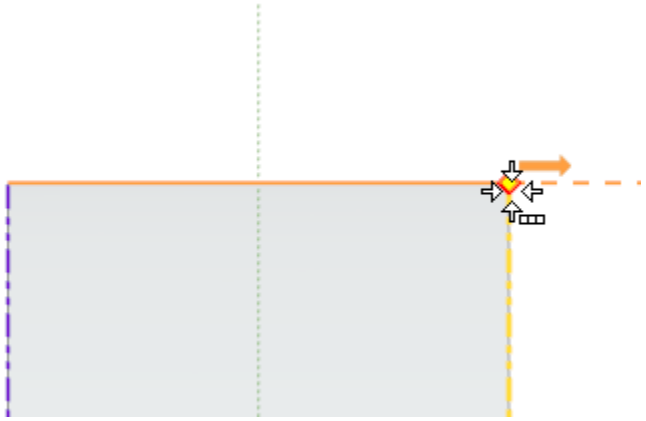

Figure 26: Sketching a line. First point selected, selecting the second one.

Using **Profile** (, *Curve* group) and the help of the horizontal reference line, sketch a closed shape as seen in Figure 27.

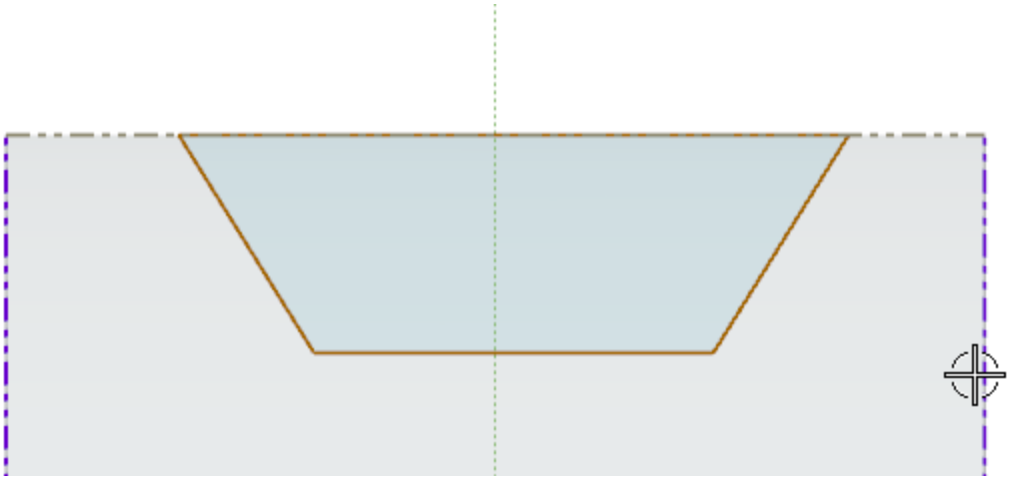

Figure 27: Sketched closed shape with four lines. Notice the closed shape shading.

Select both non-horizontal lines and apply **Make Equal** (=). Using **Make Symmetric** ( ) tool, select both non-horizontal lines and the vertical reference line (Figure 28).

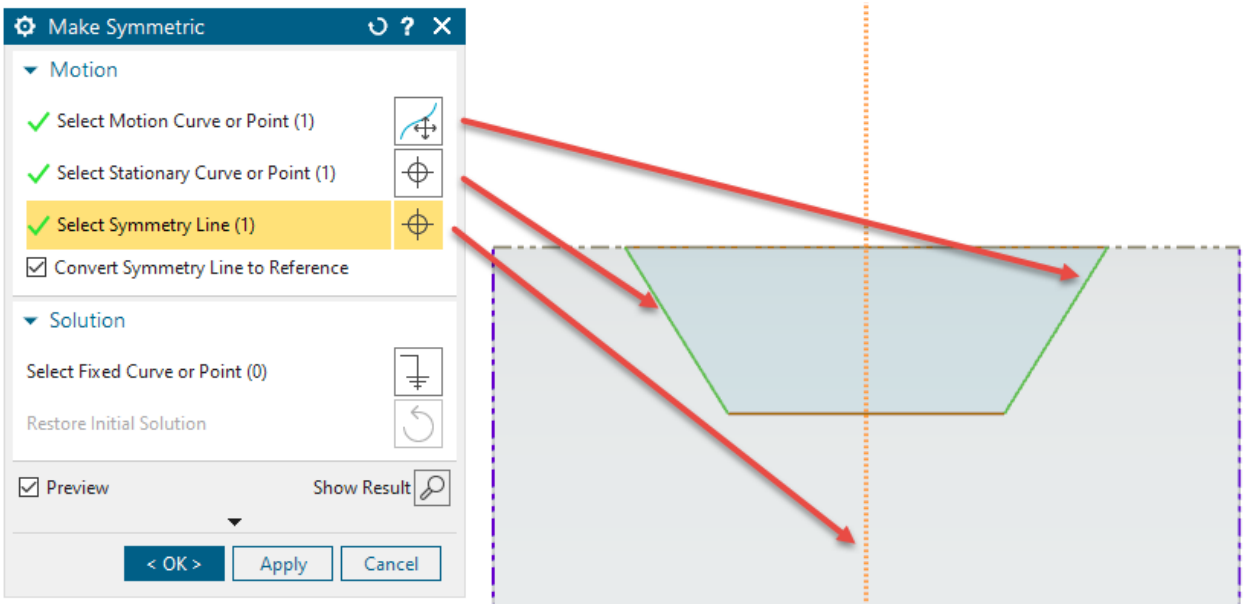

Figure 28: Make symmetric and selected lines.

The shape is ready. Create dimensions as seen in Figure 29.

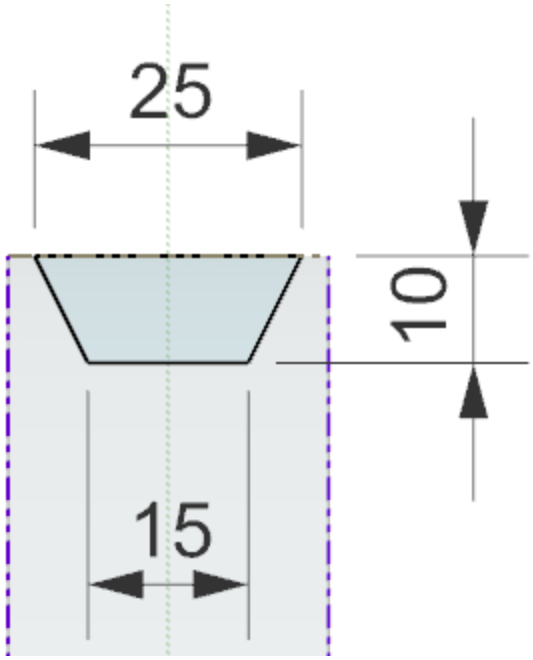

Figure 29: Current state of the sketch.

When ready, **Finish** (or Ctrl + Q) sketch.

We are back in the 3D mode. Select **Y Axis** as *Vector* and **Coordination Point** as *Point* (same way as with the first revolve feature). To remove material, select **Subtract** ( ) from *Boolean* drop-down menu (Figure 30).

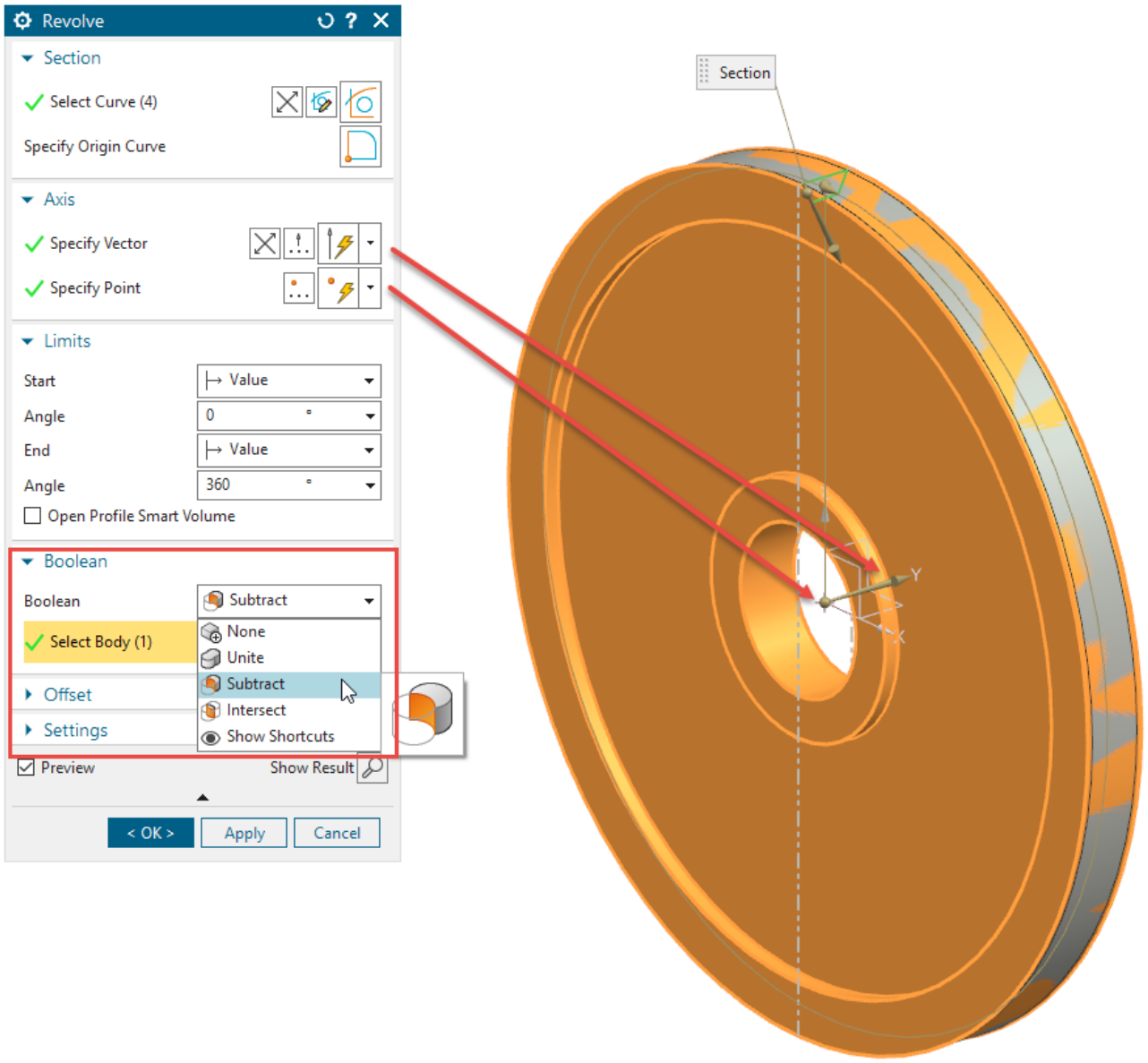

Figure 30: Selecting *Subtract* as *Boolean* operation.

Click **OK** (or **MMB**) to accept the revolve feature. Now there should be a cut to a belt. Rename *Revolve (2)* to *belt_slot*. Save your model (**Ctrl + S**).

Redefining an Existing Feature

We need to change the dimensioning schema of the *belt_slot* feature. Now there is a parameter/dimension for the thickness of the belt, but we want to have the functional diameter (for ex. calculating the gear ratio).

To redefine an existing feature, just **double-click** *blet_slot* in *Model History*. Then click **Sketch Section** () to redefine the sketch. Remove the belt hight dimension and replace it with half-diameter as seen in Figure 31.

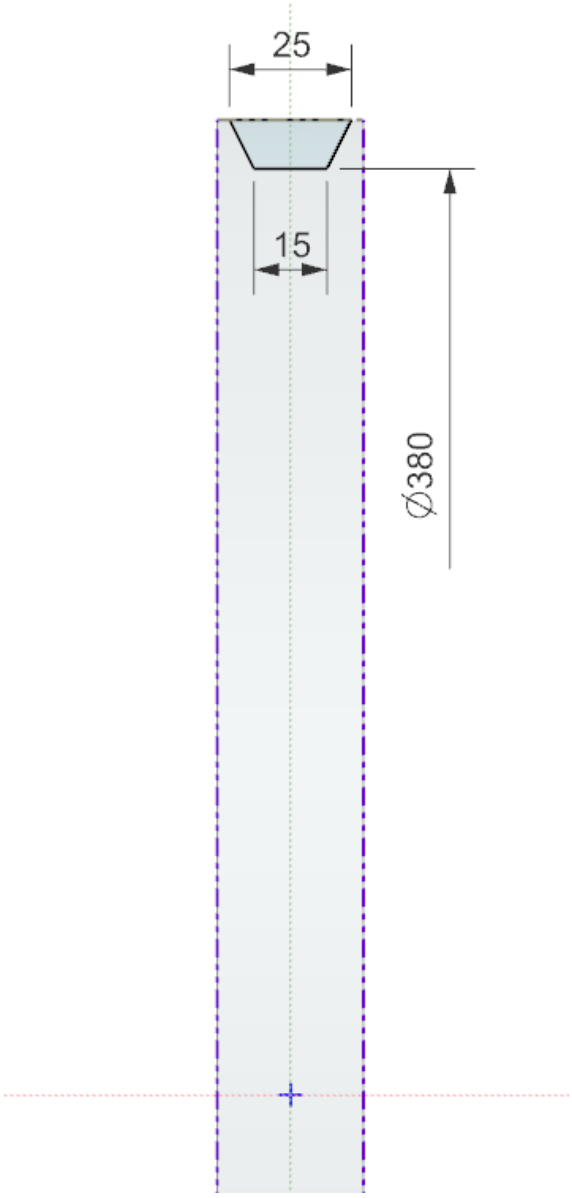

Figure 31: Updated sketch.

When ready, **Finish** () the sketch and accept the revolve feature (**OK**).

Remember to save your model (**Ctrl + S**).

Using Extrude

The next task is to create lighteners to the wheel. Select **Extrude** (, *Base* group. From *Extrude* window, select **Sketch Section** ( ) and select **XZ Plane** as a sketching plane (Figure 32). **OK**.

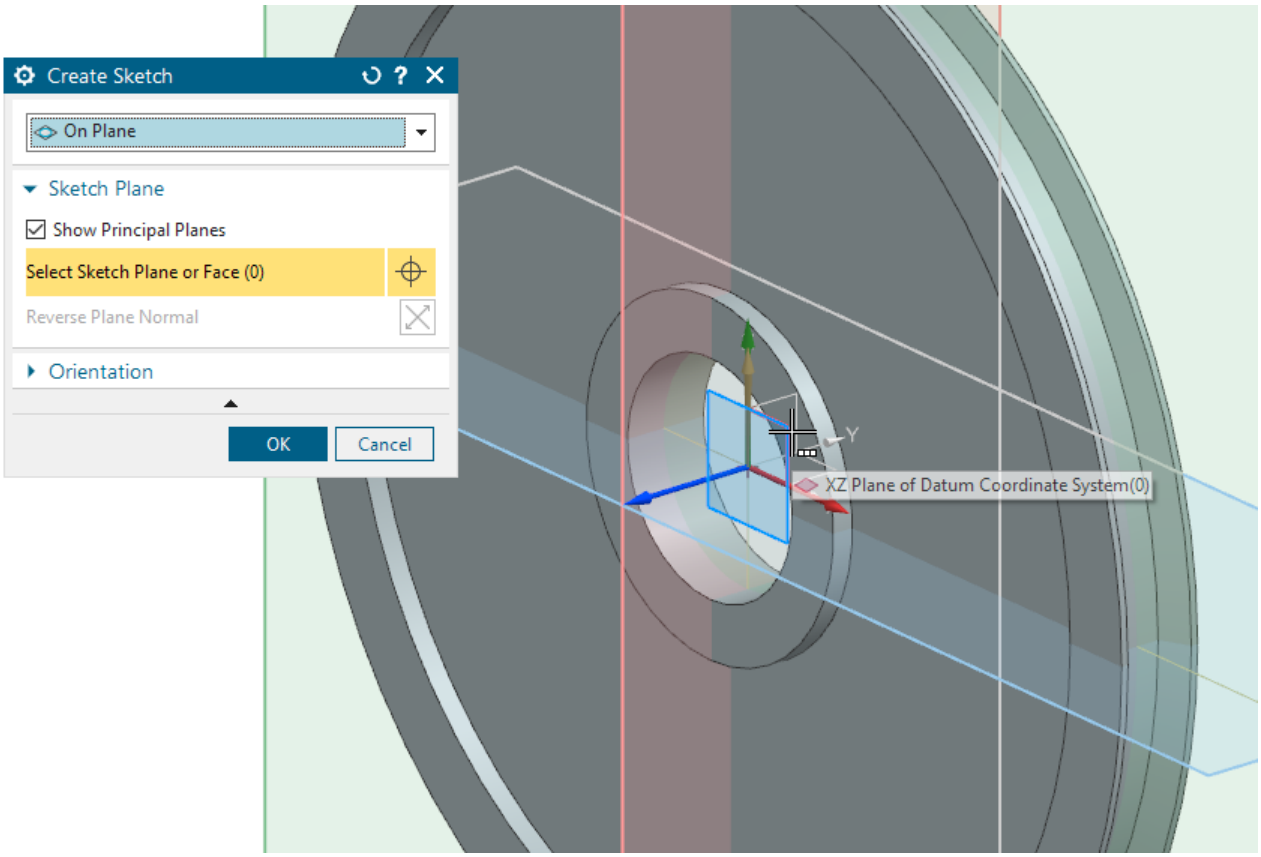

Figure 32: Sketching plane for the lightener.

The sketching mode opens. Select **Arc** (, *Curve* group). An *Arc* window appears. Select **Arc by Center and Endpoints** as *Arc Method* (Figure 33).

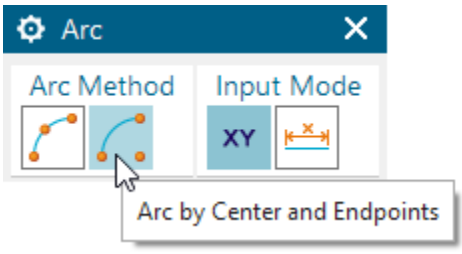

Figure 33: Selecting *Arc by Center and Endpoints* as *Arc Method*.

Sketch two arcs using coordination center point as the arc center point. Then using **Line** (, *Curve* group), sketch two lines that connect previously sketched arcs (Figure 34).

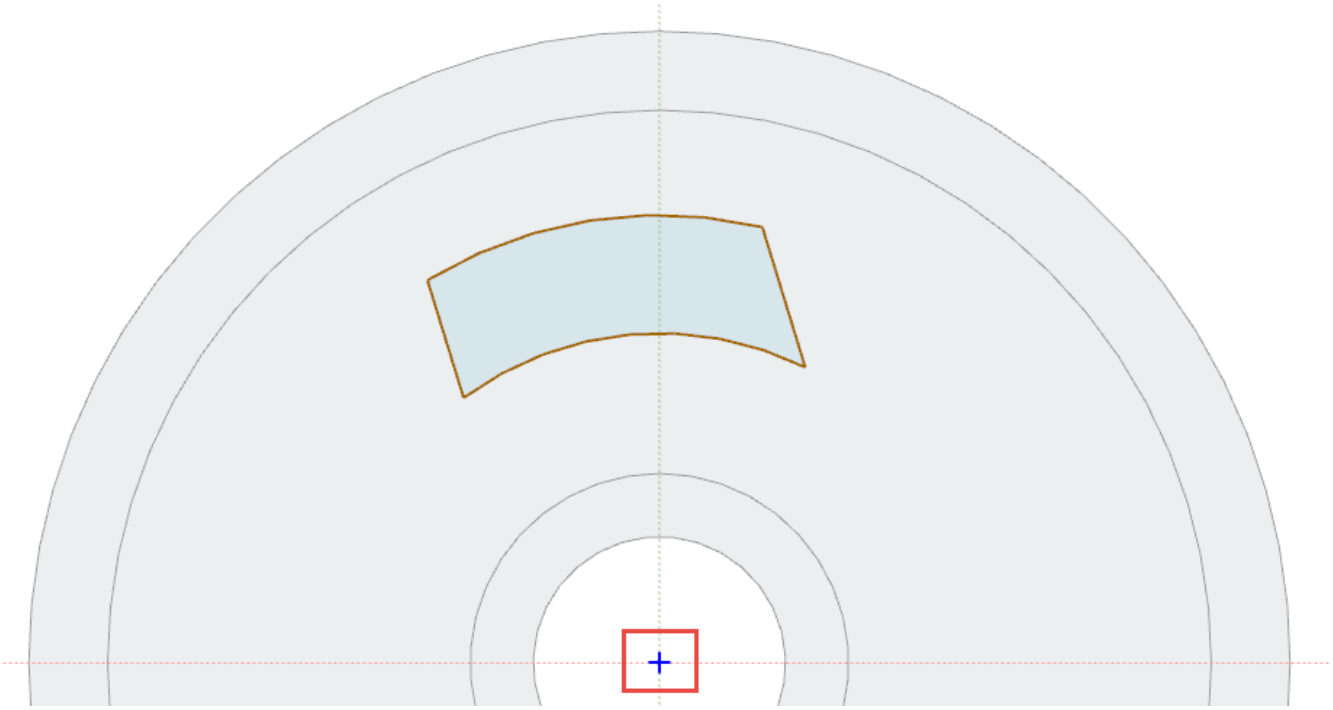

Figure 34: Two arcs and two lines sketched. Ignore the dimension values!

Create two **Perpendicular** ( ) constrains by first selecting line and then selecting the bottom arc (Figure 35). Do the same to other line and bottom arc. (solver may alert about curve calculation – this should be fine).

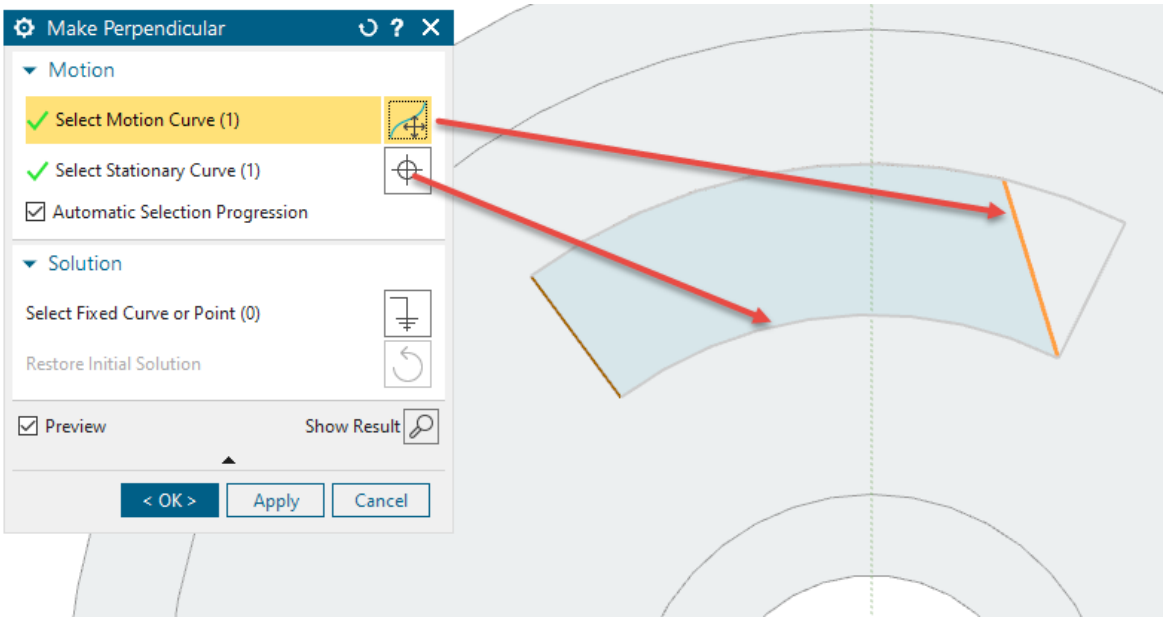

Figure 35: Creating Perpendicular constraint. Lines selected, selecting arc. On left side the comstrain is already defined.

Select **Make Symmetric** () and place arc to be symmetric along **vertical** reference axis (Figure 36).

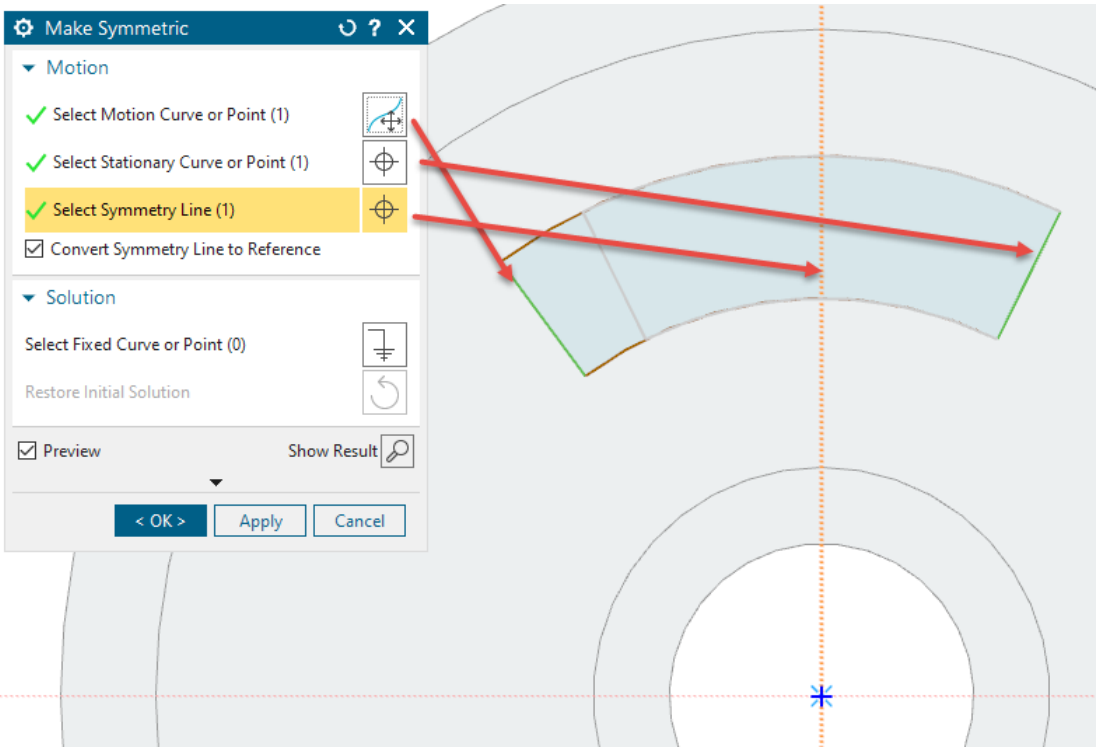

Figure 36: Selected objects for Make Symmetric.

Using **Project Curve** ( ) tool, select the edge from the base feature and **Convert to Reference** ( ). Your sketch should look like in Figure 37.

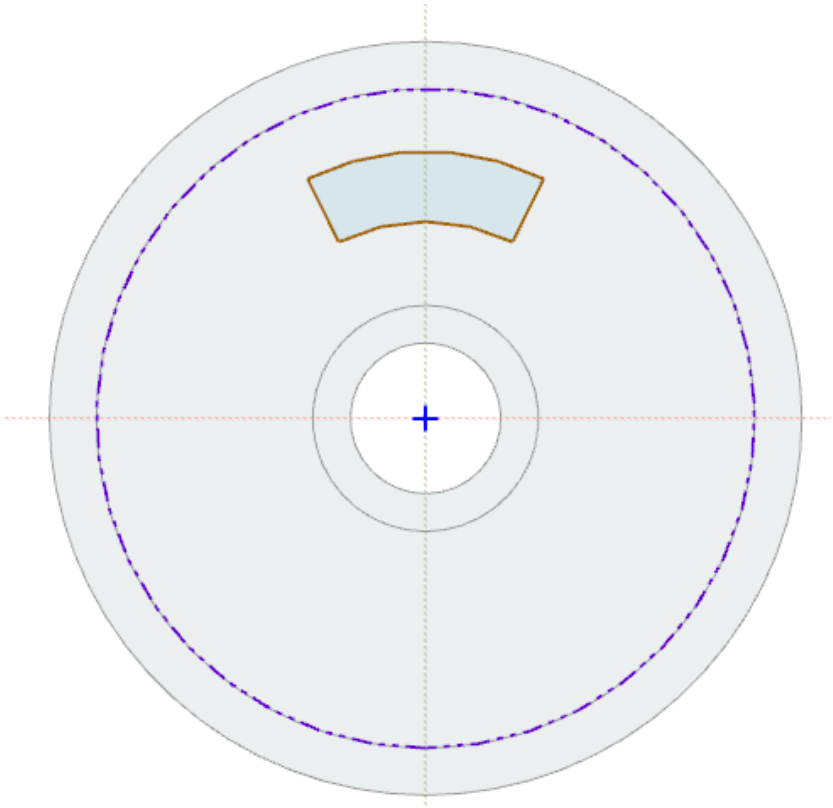

Figure 37: Current sketch status. Notice the reference circle.

Now the shape is ready, time to add some dimensions. This time **Rapid Dimension** ( ) tool is utilized. The tools is not available in the sketching mode UI, so select searc and type "**rapid**" to find the tool (Figure 38). (this tool can also be activated by pressing key **d**)

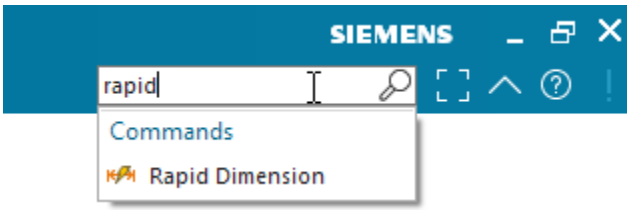

Figure 38: Search box and rapid as input.

With *Rapid Dimension* tool, create a dimension between sketched form's top arc and reference circle so, that arc's middle point is utilized and using *Tangent Point* from the other arc (Figure 39).

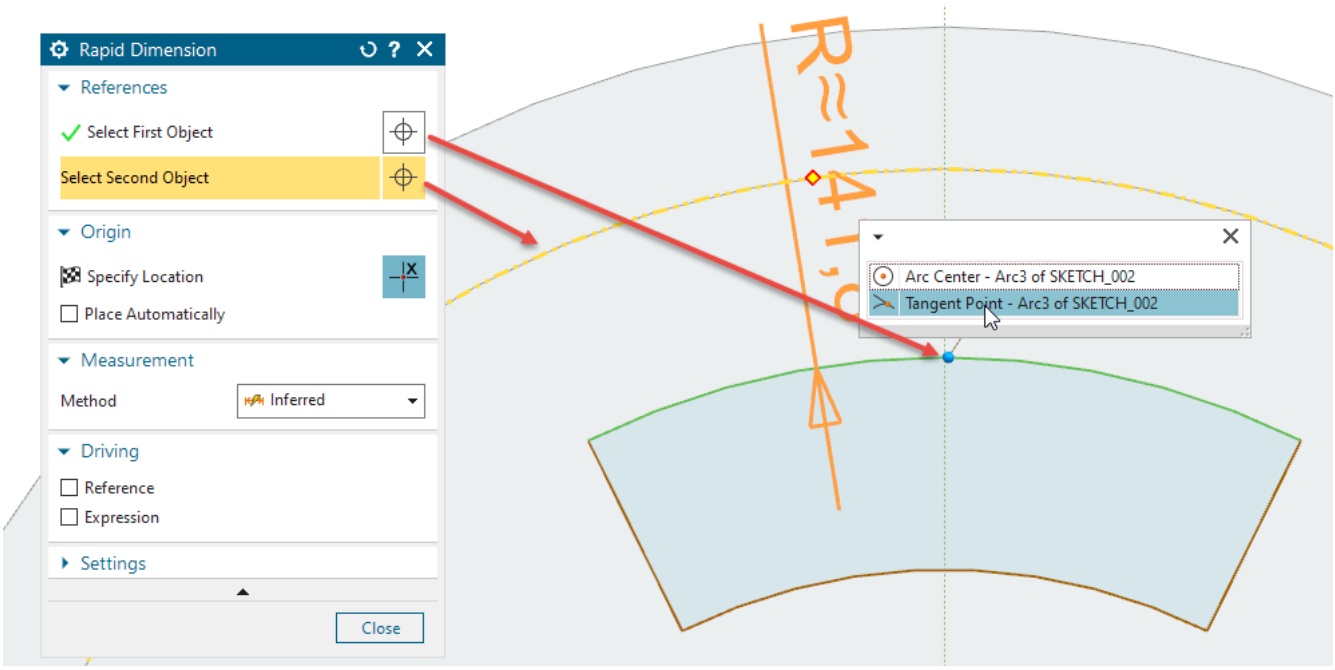

Figure 39: Sketched arc selected, selecting Tangent Point from the reference arc (circle).

Project also the second **curve** from the base feature (see picture). To finalize the sketch, create missing dimensions and give values as seen in Figure 40.

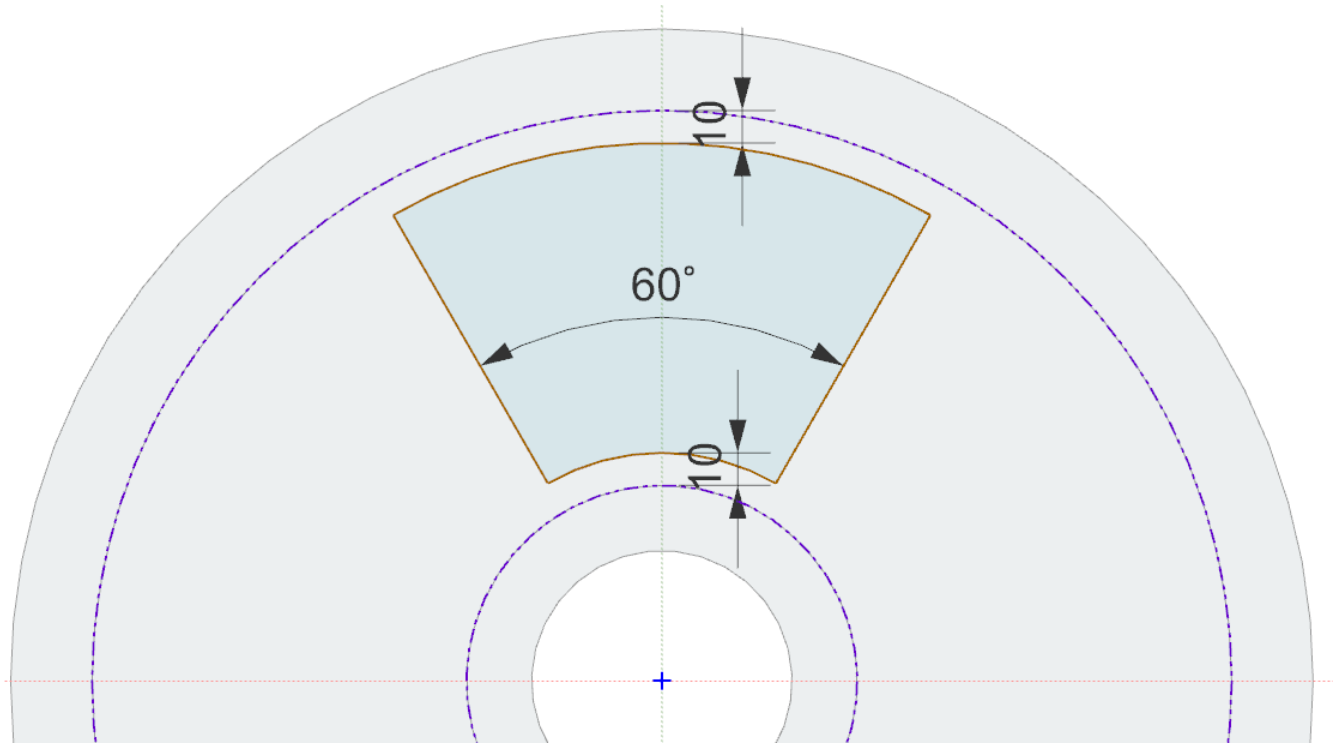

Figure 40: Ready-to-accept sketch.

The sketch is ready, accept it (**Ctrl + Q** or). In *Extrude* window, in *Limits* field, change *Start* to be **Through All** and *End* to be also **Through All** (Figure 41). Check that *Boolean* is **Subtract** ( ) and accept the feature (**OK**). Rename the feature as *cut*.

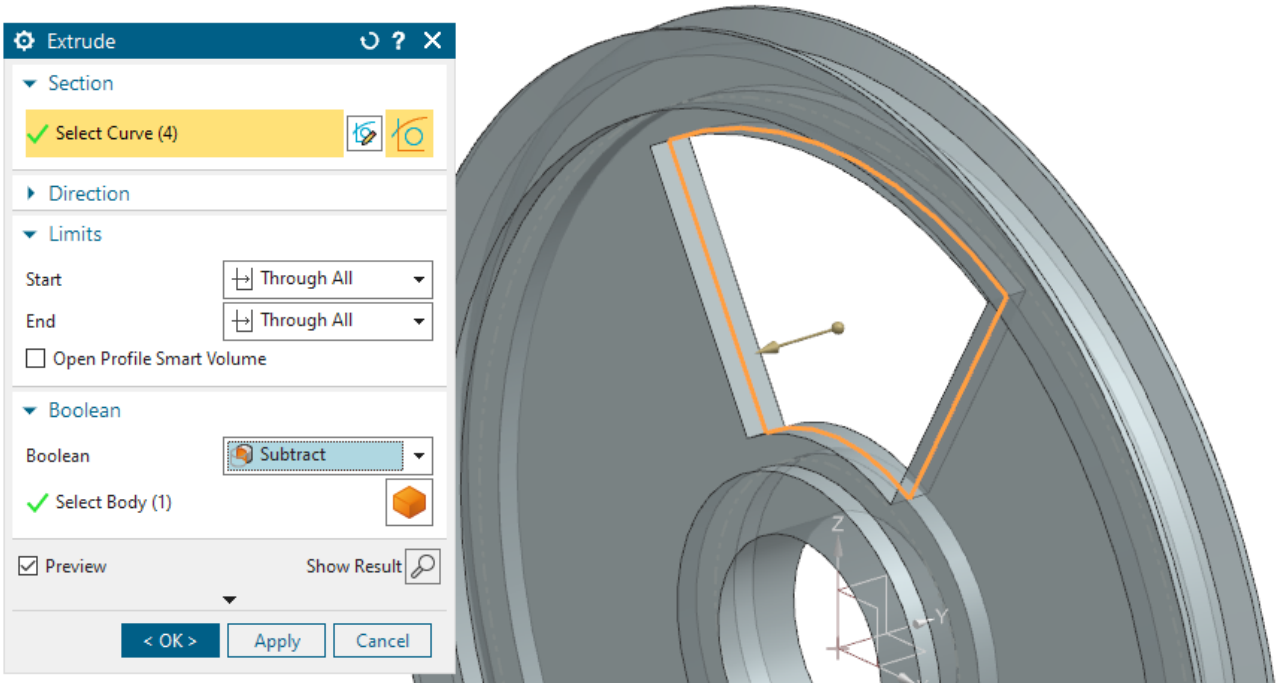

Figure 41: Ready-to-accept Extrude feature.

Using Edge Blend

The edges of the previously created pocked are sharps, so rounding is neede. Select **Edge Blend** ( ) from the *Base* group. Using radius of **10** mm, select the edges shown in Figure 42.

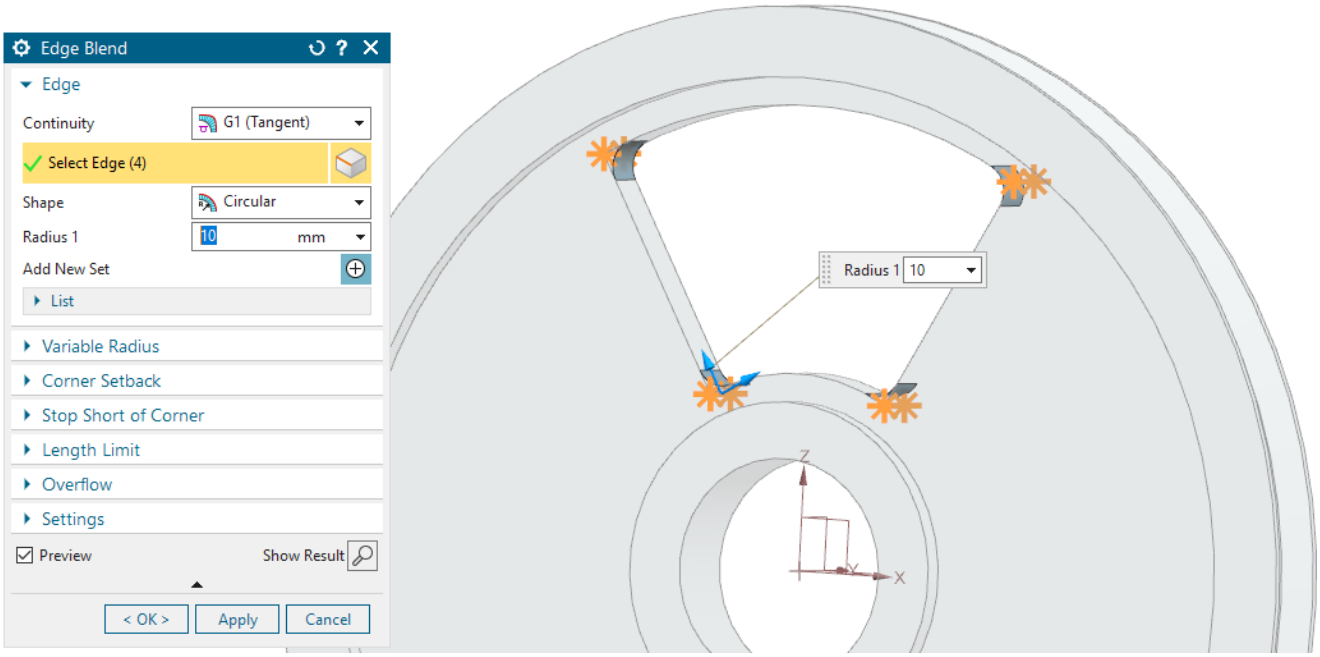

Figure 42: Selected edges for the edge blend of 10.

Then, select **Add New Set** ( ) and using the value of **3 mm**, select remaining edges of the *cut* feature (Figure 43). When ready, accept the feature (**OK**).

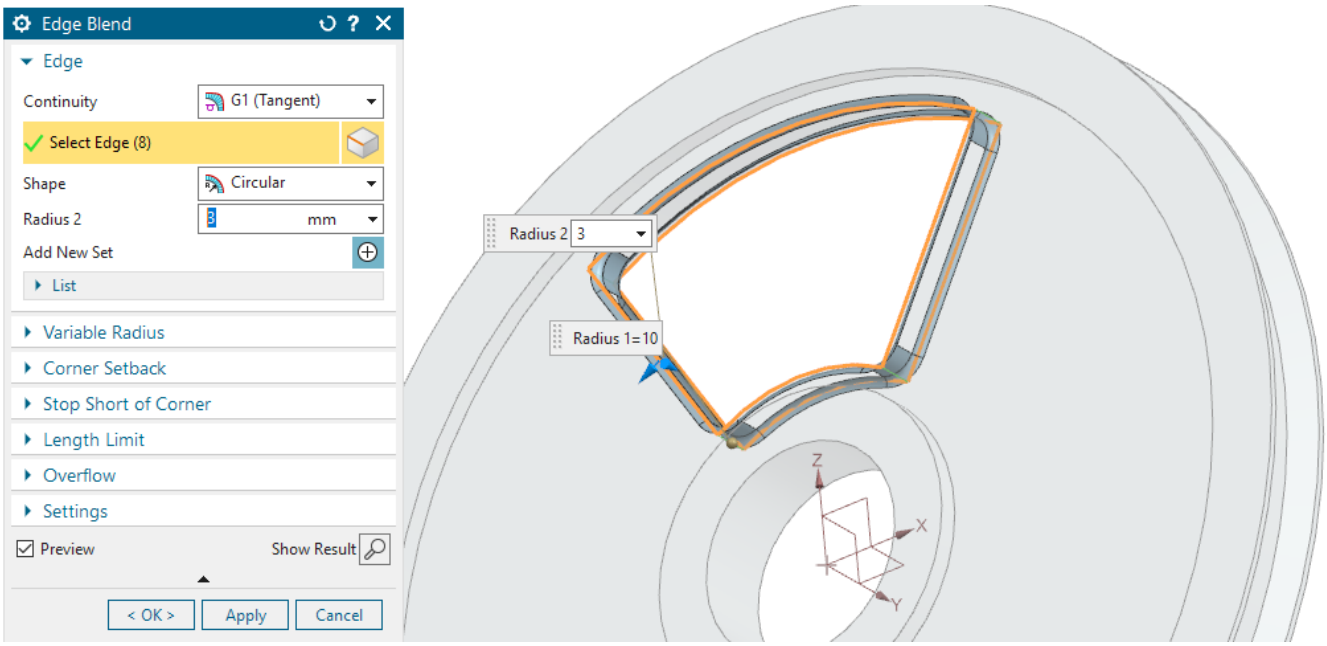

Figure 43: Edges for the second set selected (highlighted in orange).

Using Pattern

To pattern features, select **Pattern Feature** (, *Base* group). The *Pattern Feature* window opens. Select *cut* feature from the *Model History*, hold Ctrl and select *Edge Blend* feature. Select **Circular** ( ) as *Layout*. Select **Y Axis** as *Rotation Axis* and **Coordination Point** as *Point*. To *Angular Direction*, select **Count and Span** and give values as seen in Figure 44 (5 and 360 deg).

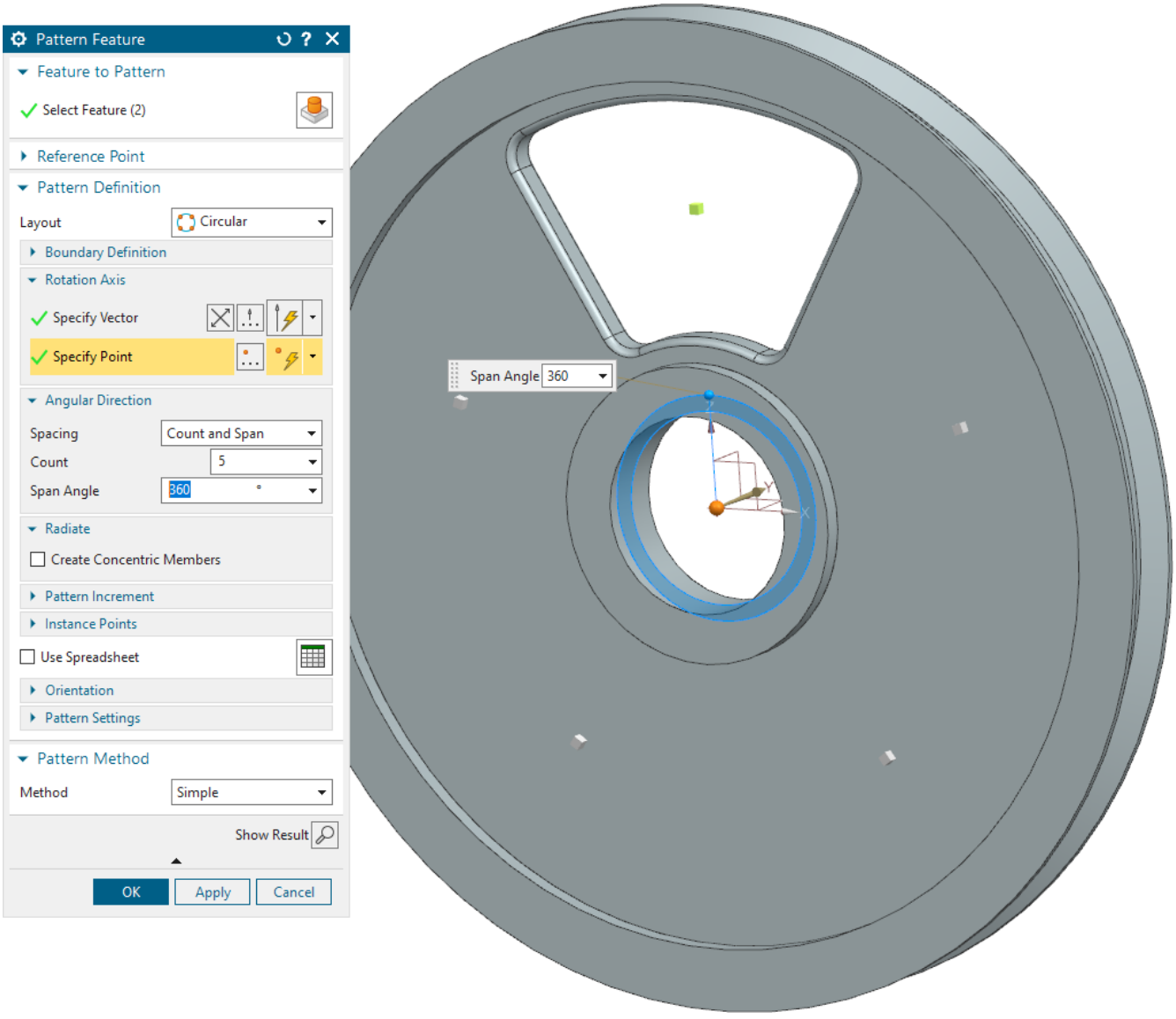

Figure 44: Ready-to-accept pattern feature.

When ready, accept the feature (**OK**).

Redefining the Patterned Feature

Select *cut* feature from the *Model History* by double-clicking it, select **Sketch Section** ( ) and change the 60 degree dimension into **50** (select the 60 value and give a new one). Click **OK**. The patterned features should also update (Figure 45).

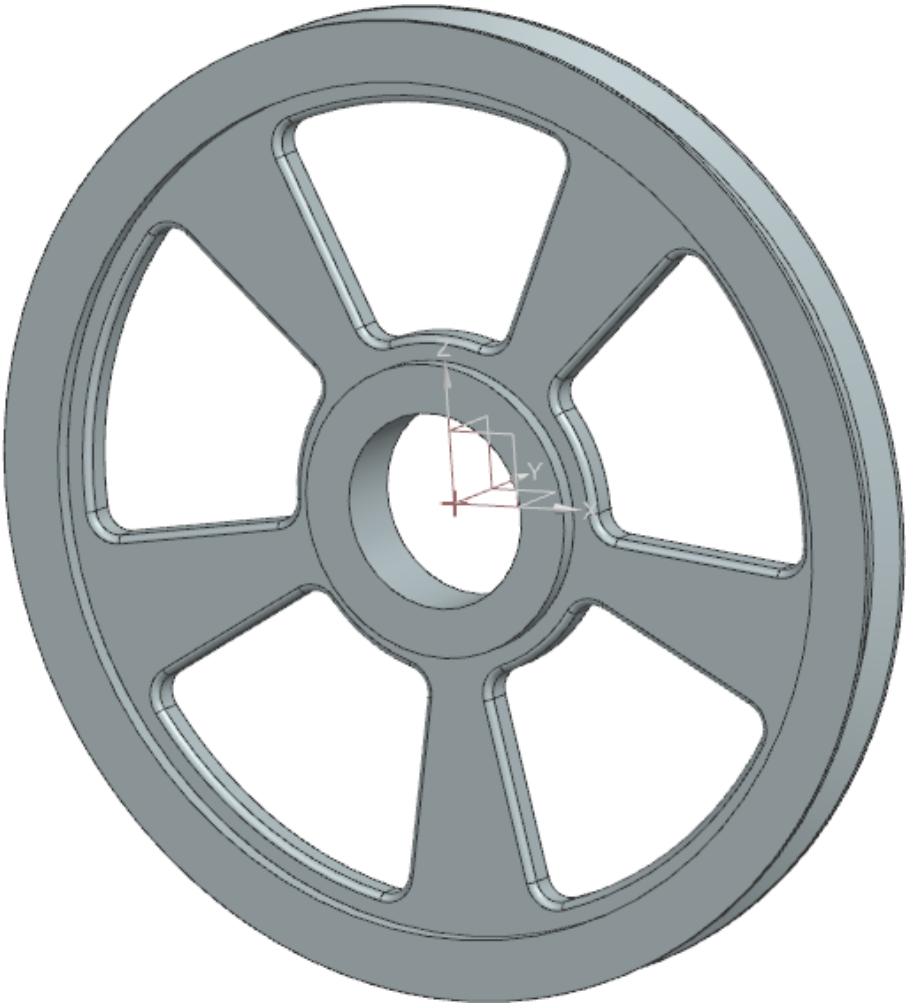

Figure 45: Current state of the model.

Using Edge Blend

Crete **Edge Blend** (, *Base* group) for four edges as seen in Figure 46. Use the value of **2**.

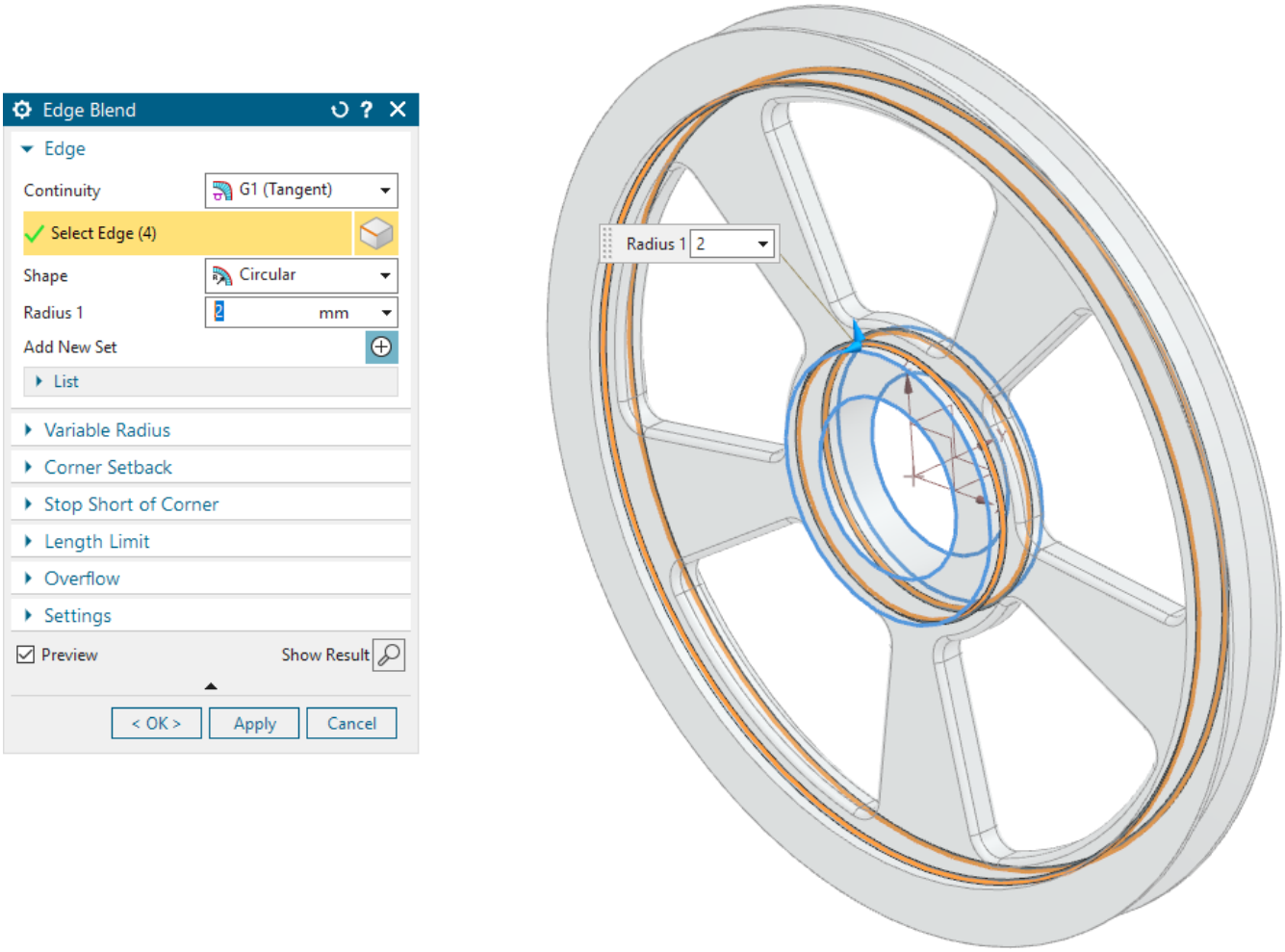

Figure 46: Selected four edges highlighted in orange.

Using Chamfer

Select **Chamfer** ( , *Base* group). Using the value of **2**, select the remaining sharp edges to be chamfered (Figure 47). Accept the feature (**OK**).

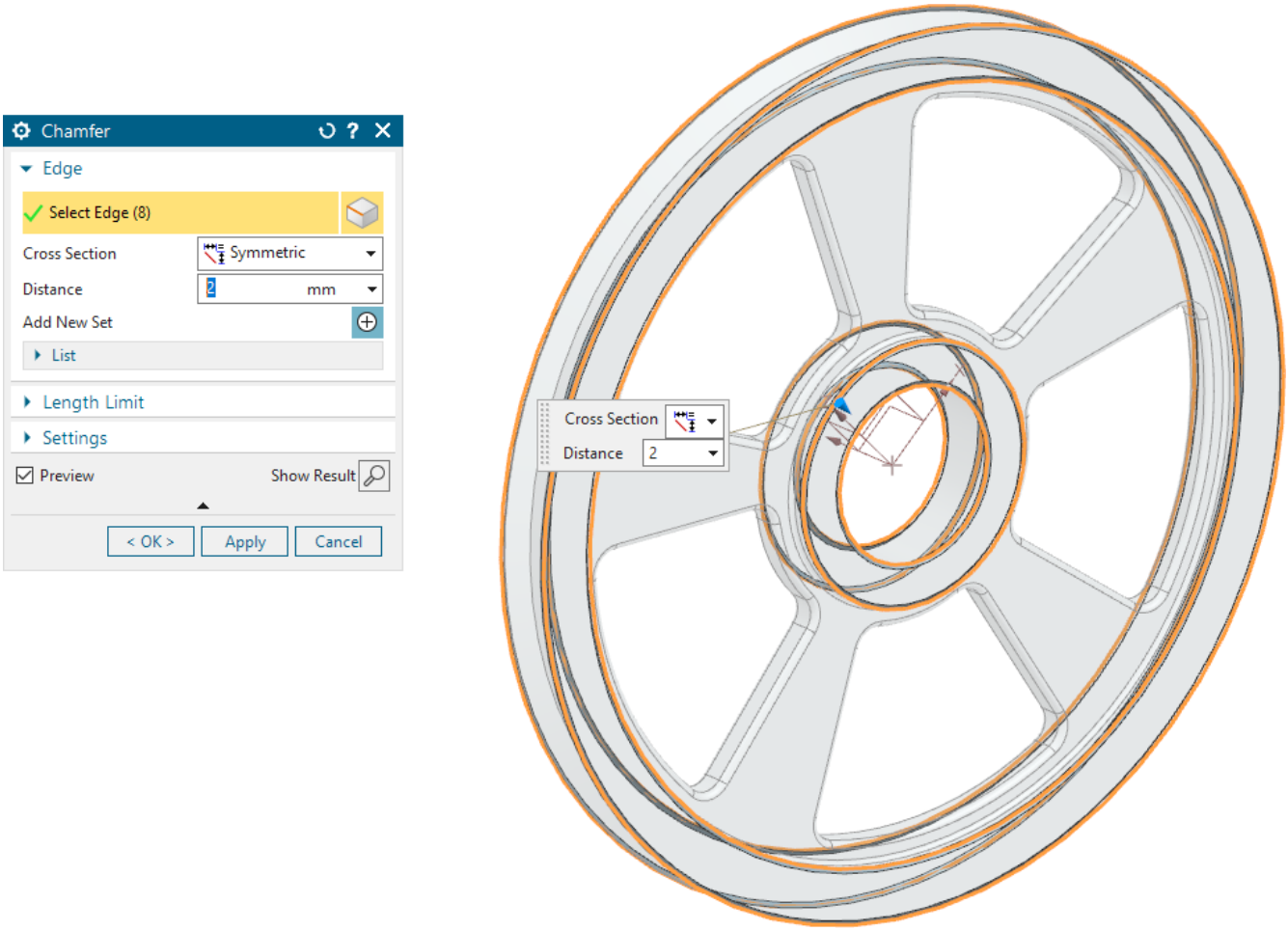

Figure 47: Eight edges selected for chamfering (highlighted in orange).

Save your model. This concludes this exercise. Your model should look like in Figure 1.