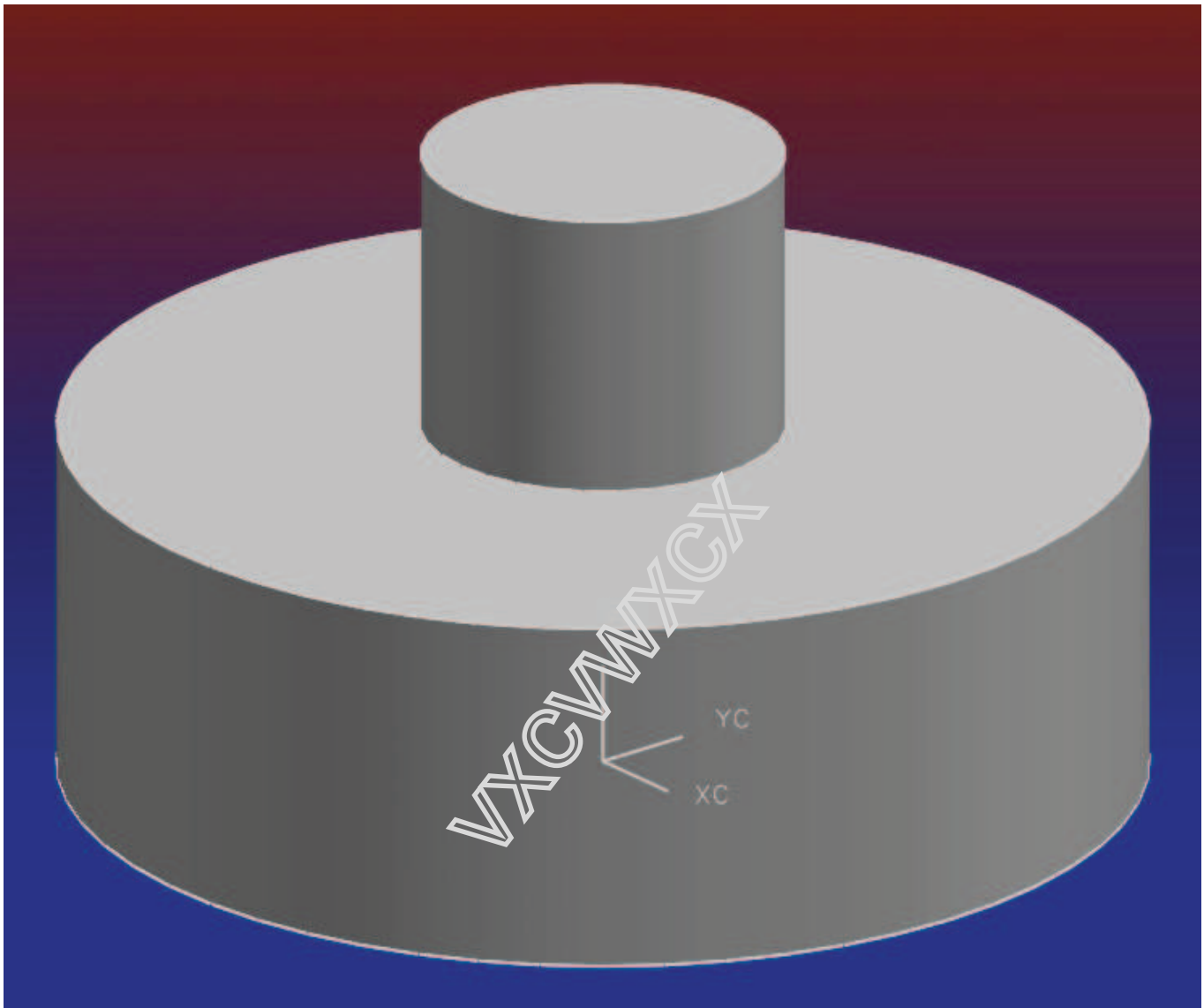


Using a Sketch Feature for Creating a Machining Relief - Cylindrical

This article will focus on using a sketch feature to create a machining relief in a model.



Start with a sketch that will define the planar profile orientation of the expected relief cross section.

In this case a simple relief sketch is created defining the material being removed and some control curves that help define the profile for parametric alteration.

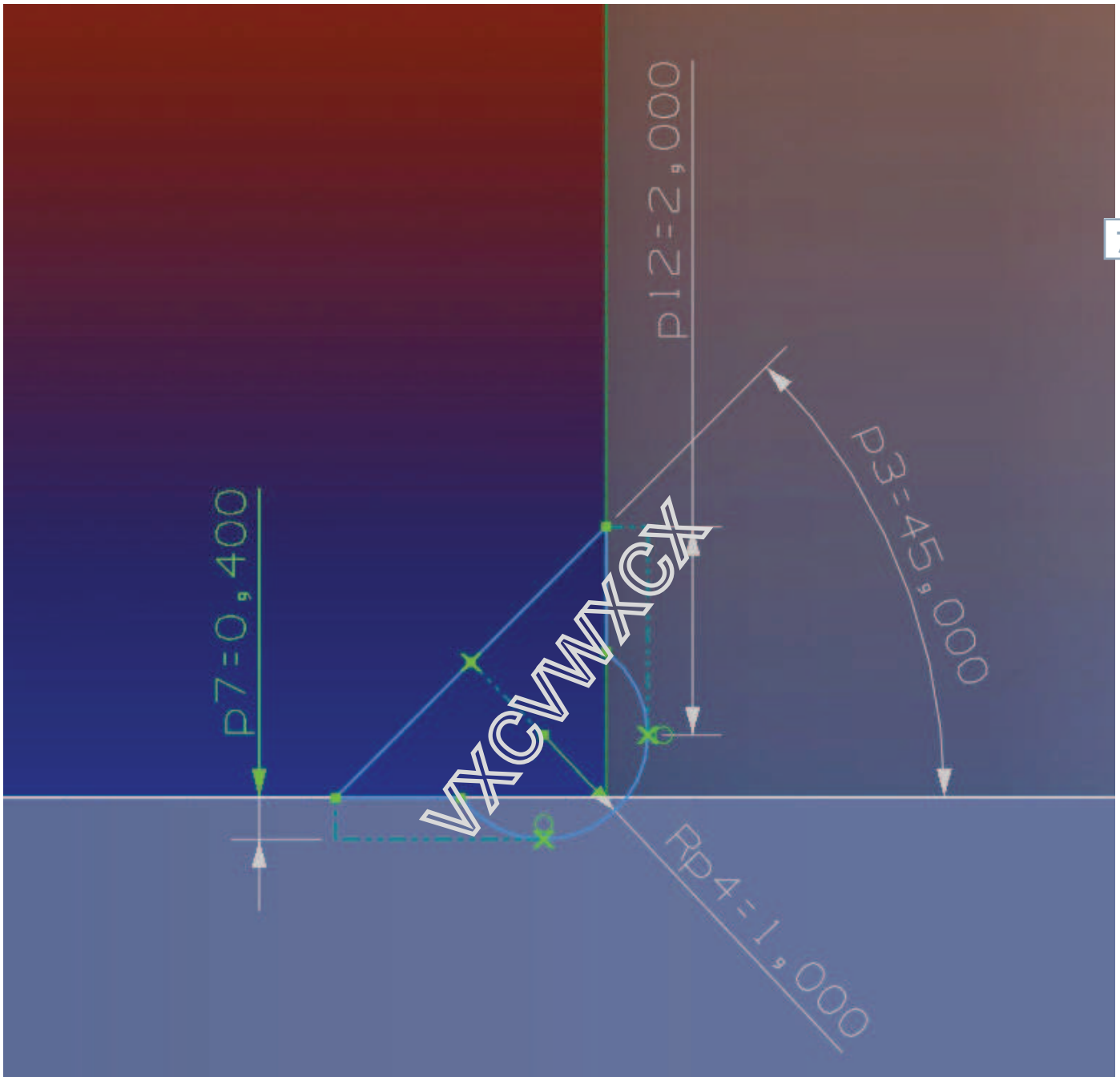
The relief, as seen in the image, protrudes below the surface enough to remove material from the intersection of the 2 surfaces.

Construction intersection feature curves are used to geometrically constrain the sketch as collinear vertical and horizontal to the model surfaces. (The GREEN lines)

Reference Sketch curves are used to provide additional parametric definition for both geometric and dimensional constraints.

Continued from previous

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The sketch is finished.

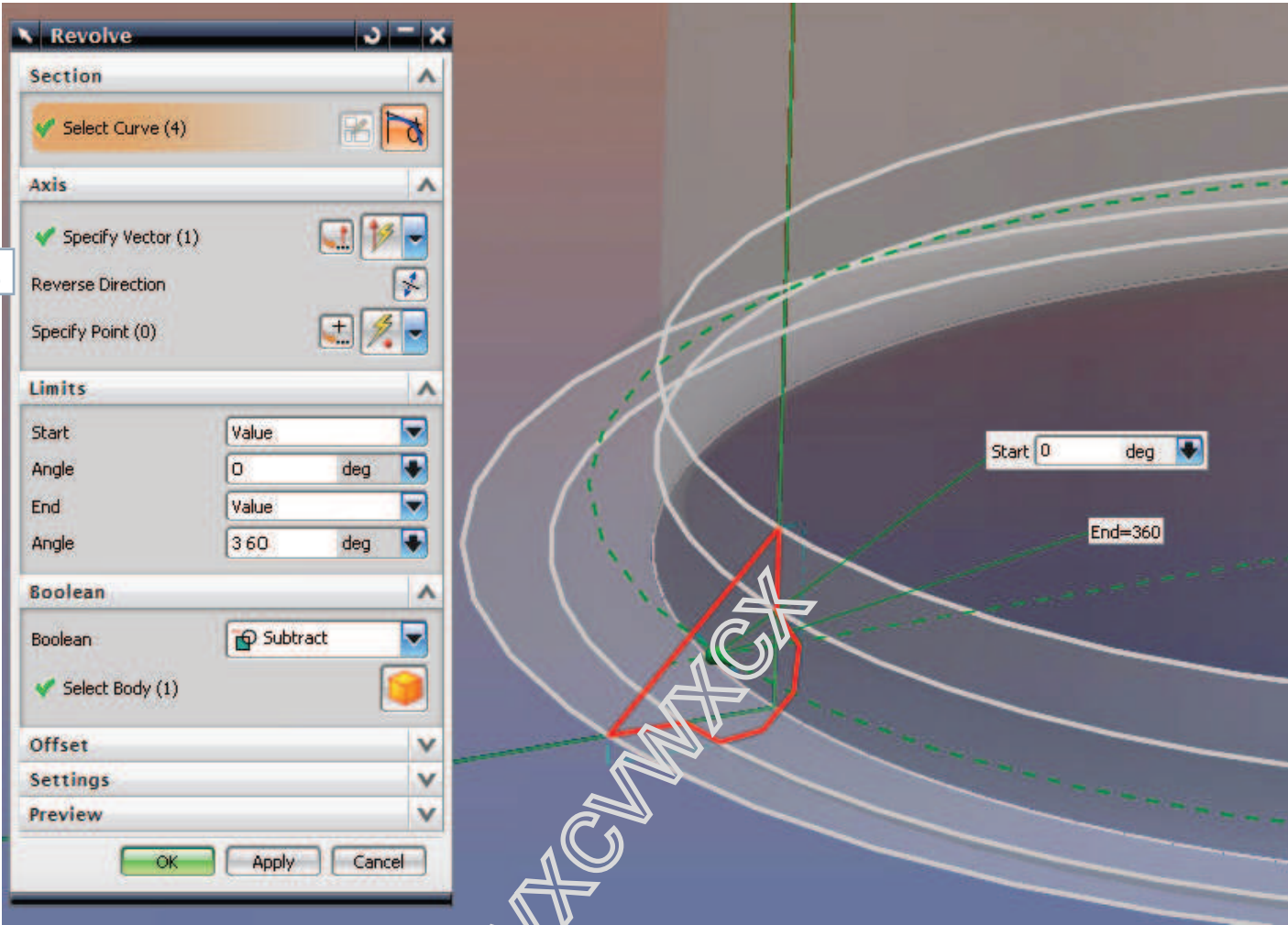
Revolve of the sketch just created: **Insert => Design Feature => Revolve.**

The vector direction is defined by the cylindrical center axis and a Boolean Subtract is also applied.

>> NX

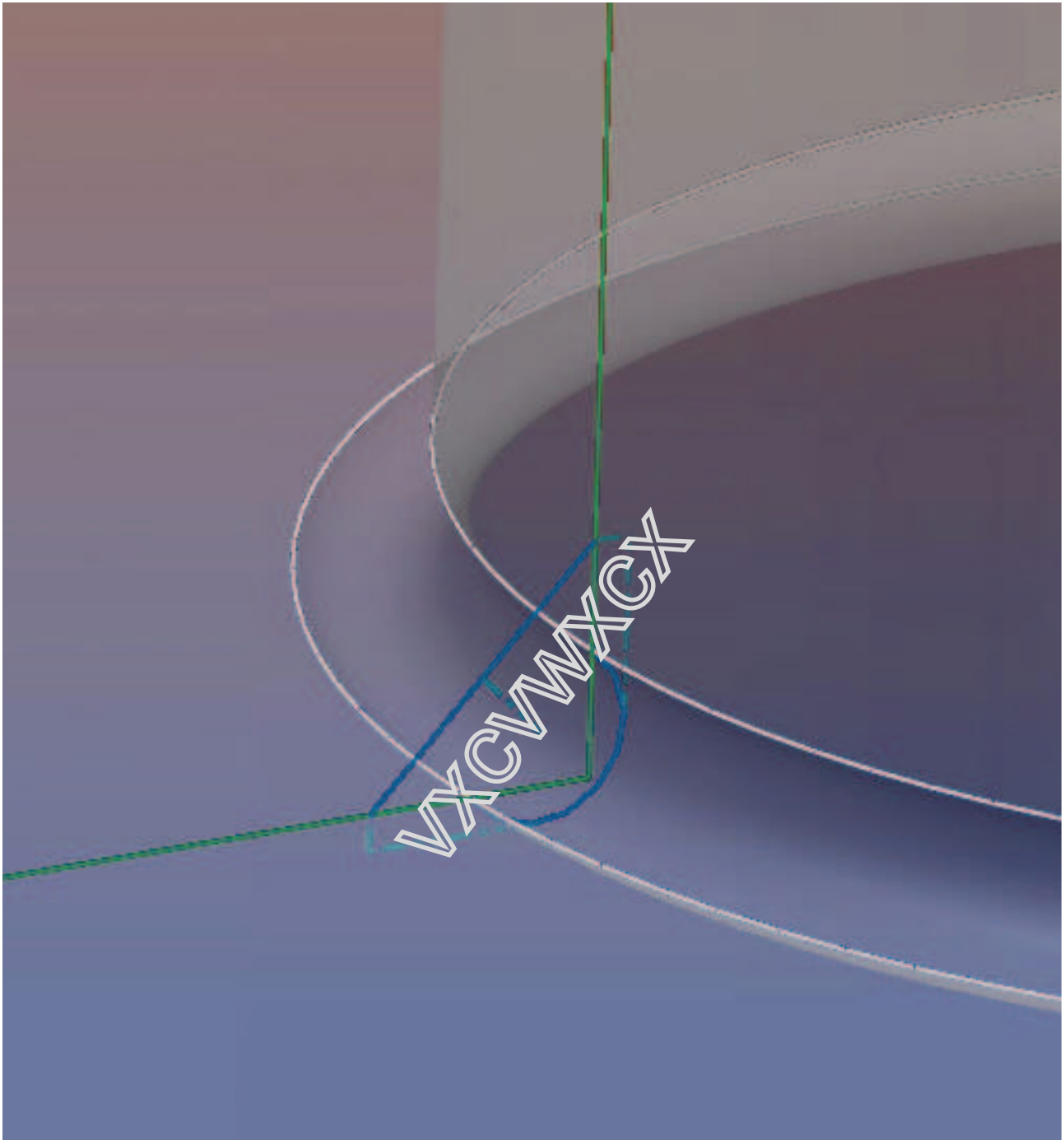
Continued from previous

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The Revolve is completed and subtracted.

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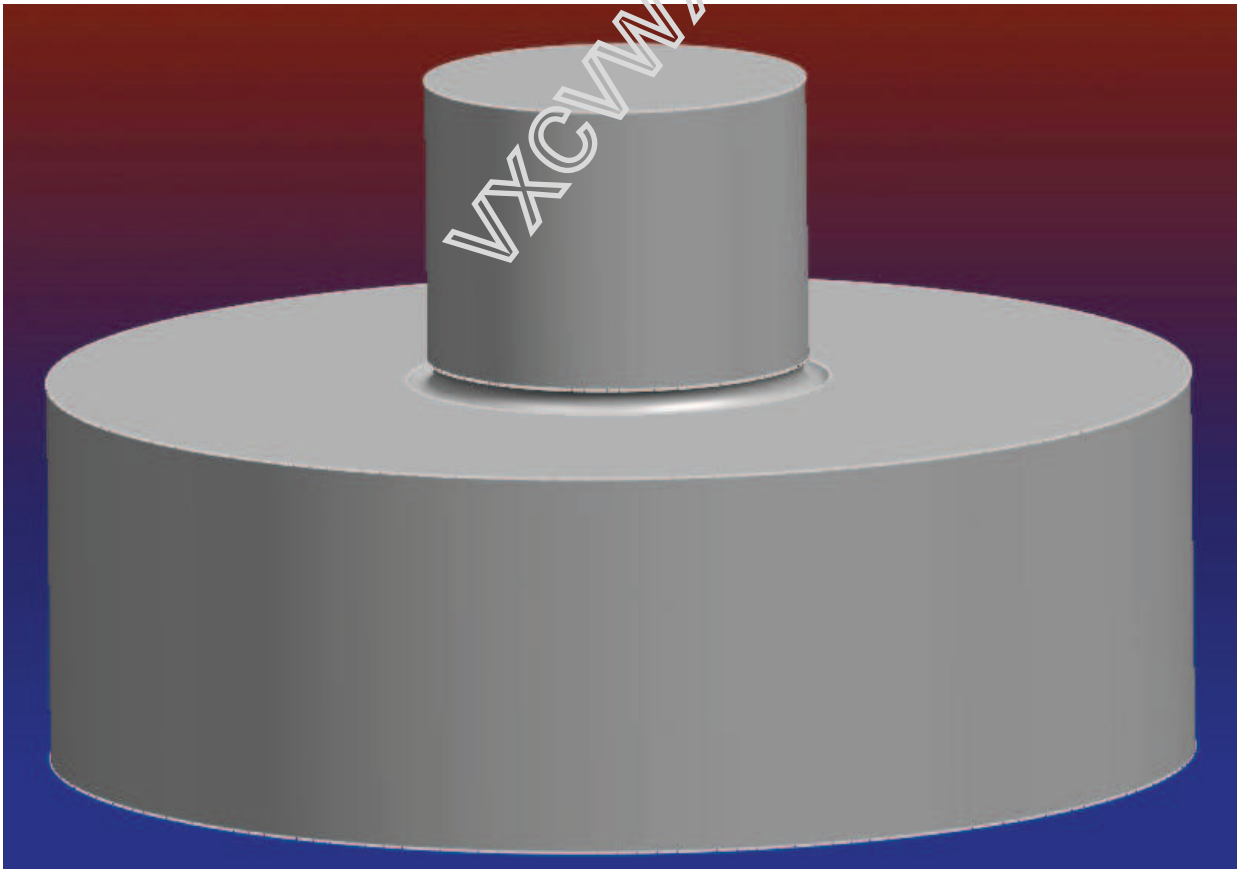
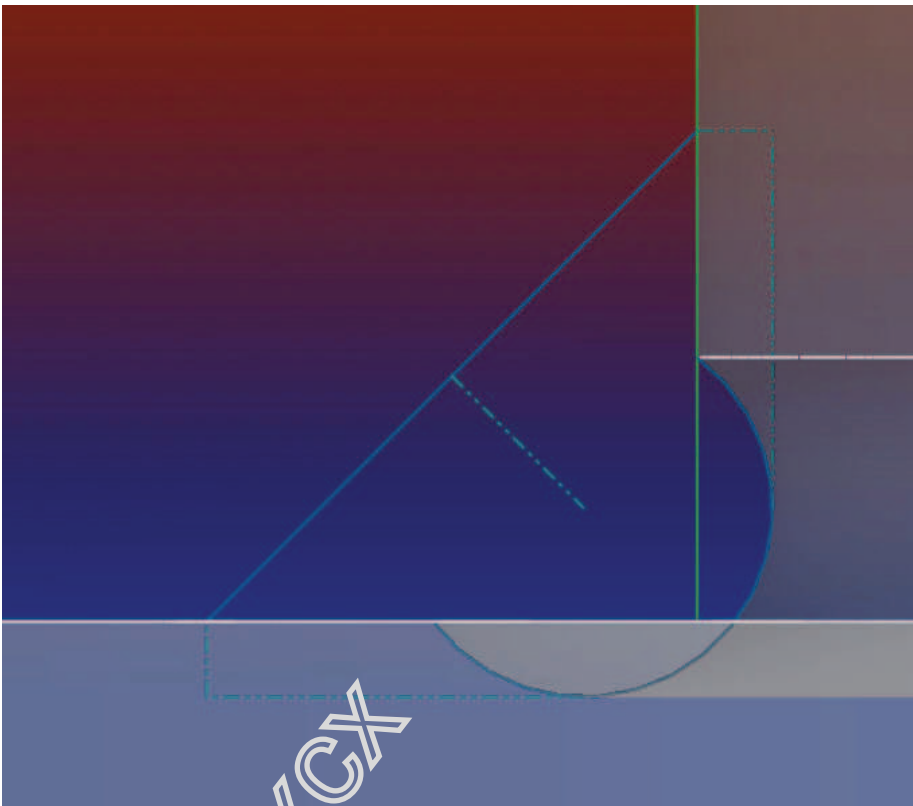
Here is another view of the sketch defined and the revolve created.



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The finished model with the sketch/revolve feature producing a relief feature in the part model.

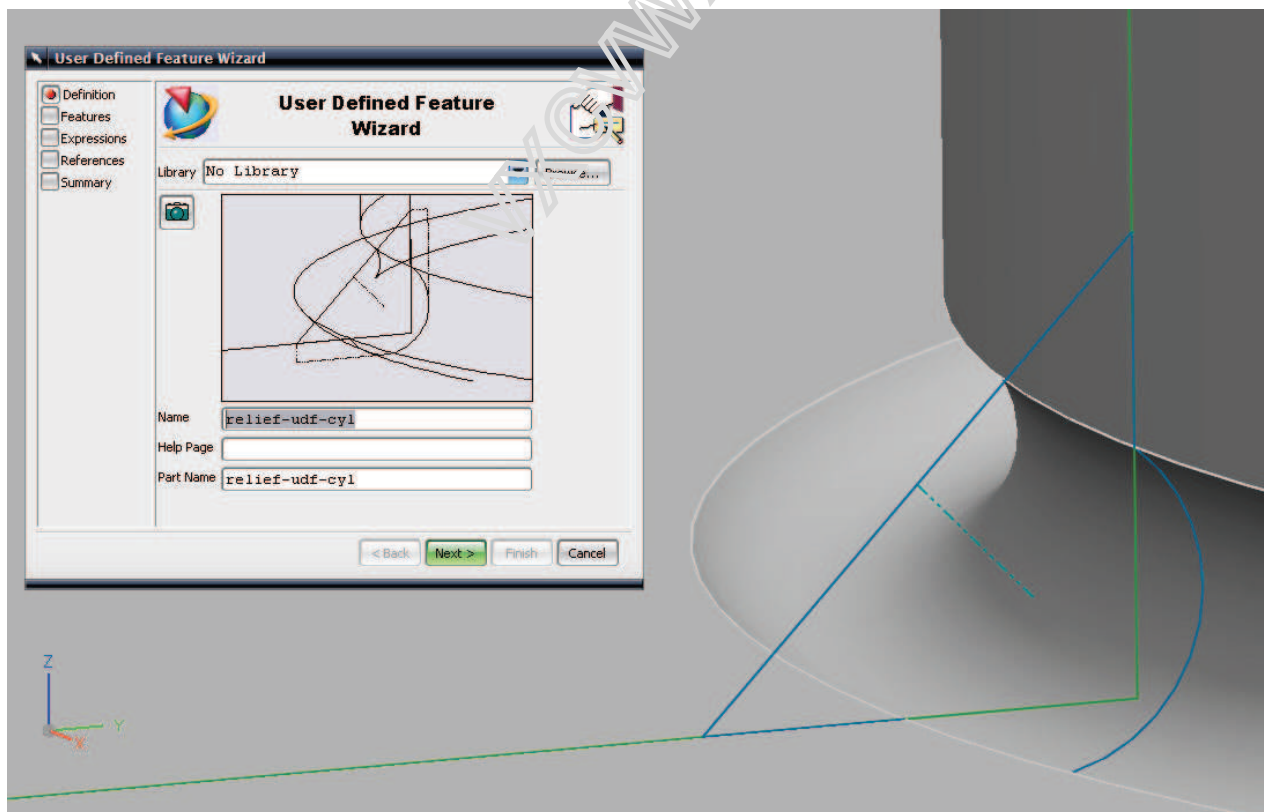
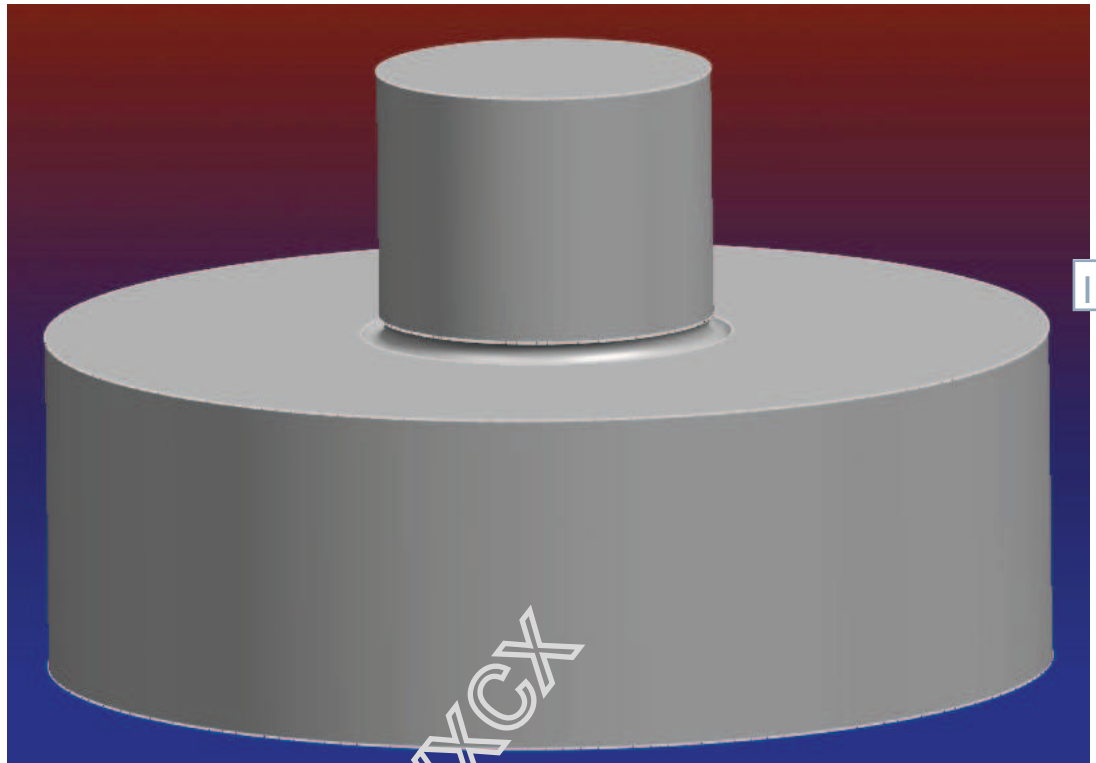


Stephen Chlopecki

Creating a UDF for an Interactive Parametric Machining Relief - Cylindrical

Using the part and sketch from the article, [*Using a Sketch Feature for Creating a Machining Relief - Cylindrical*](#), the following operations will create a UDF from the sketch and import the sketch to a new model and apply the necessary parametric values and object associations.

1. Select **File => Export => User Defined Feature**.
2. Create a snapshot for the expected UDF
3. Insert a name for the **Name** and **Part** name fields.

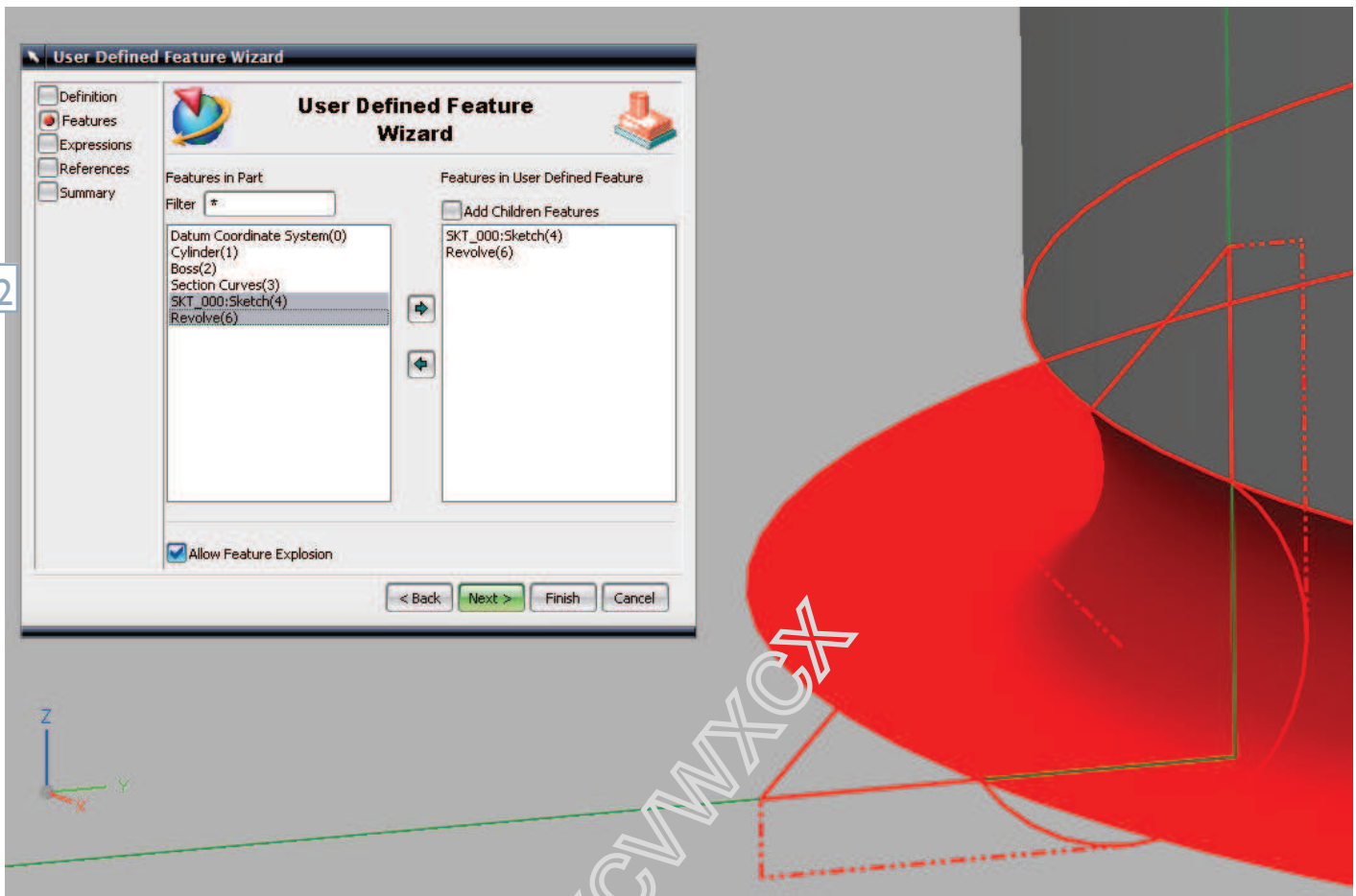


4. Select the necessary features to export for this UDF:

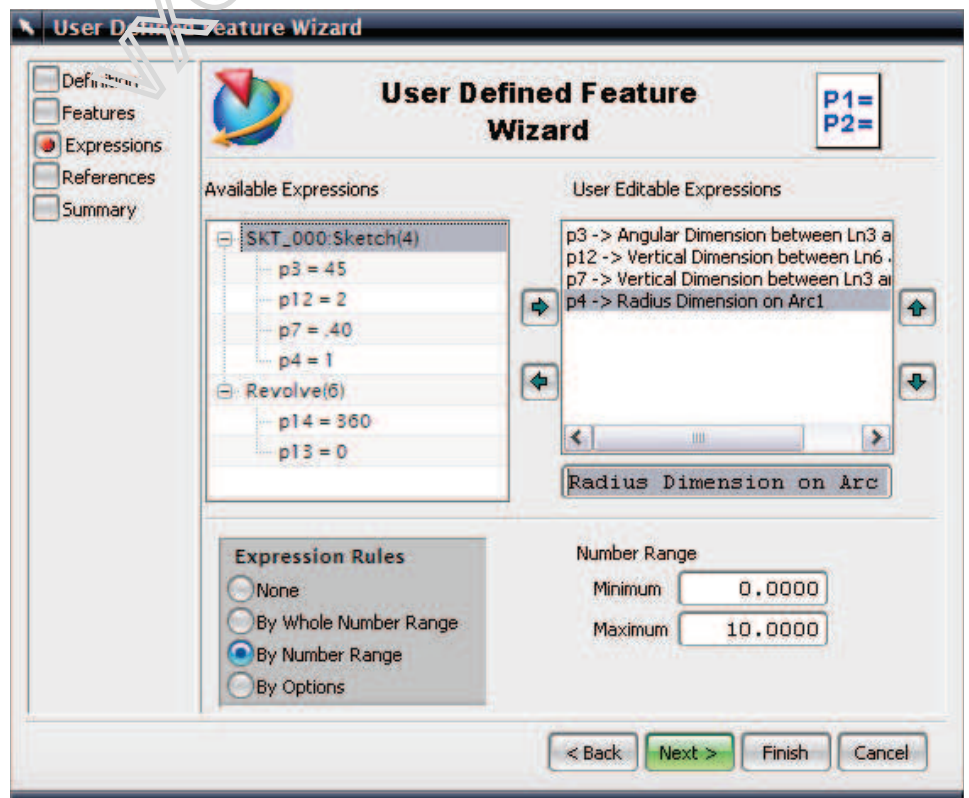
- **Sketch(4)**
- **Revolve(6)**

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5. Add the sketch dimensional parameter values.
6. Define the number ranges for each:
 - Angle set as 0 to 180
 - Others set as 0 to 10



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Notice the listed external object constraints that will be required to place the UDF.

Take note of these because setting up the part that this UDF is to be placed in will greatly increase the smoothness of the operation.

7. Finish the UDF creation.



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>> **NX**

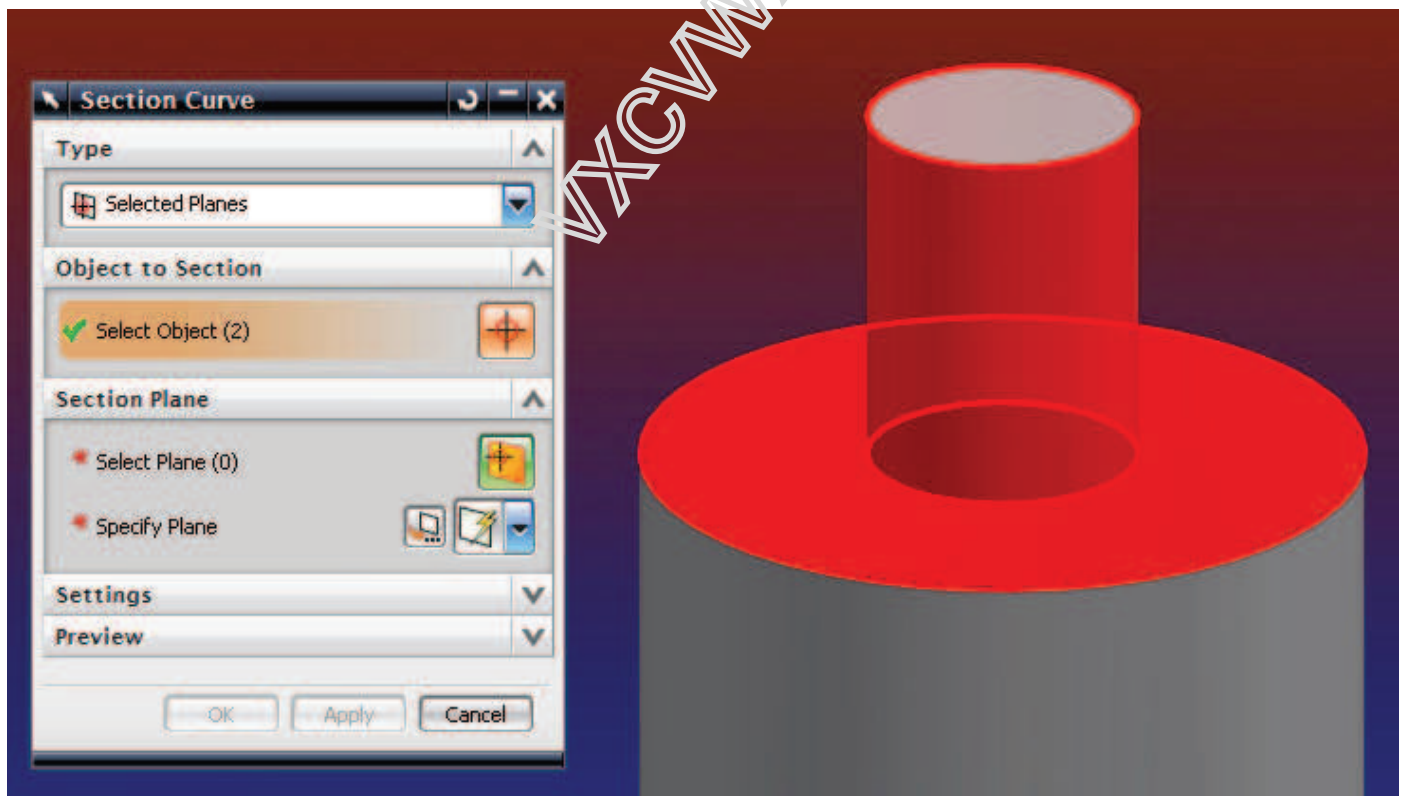
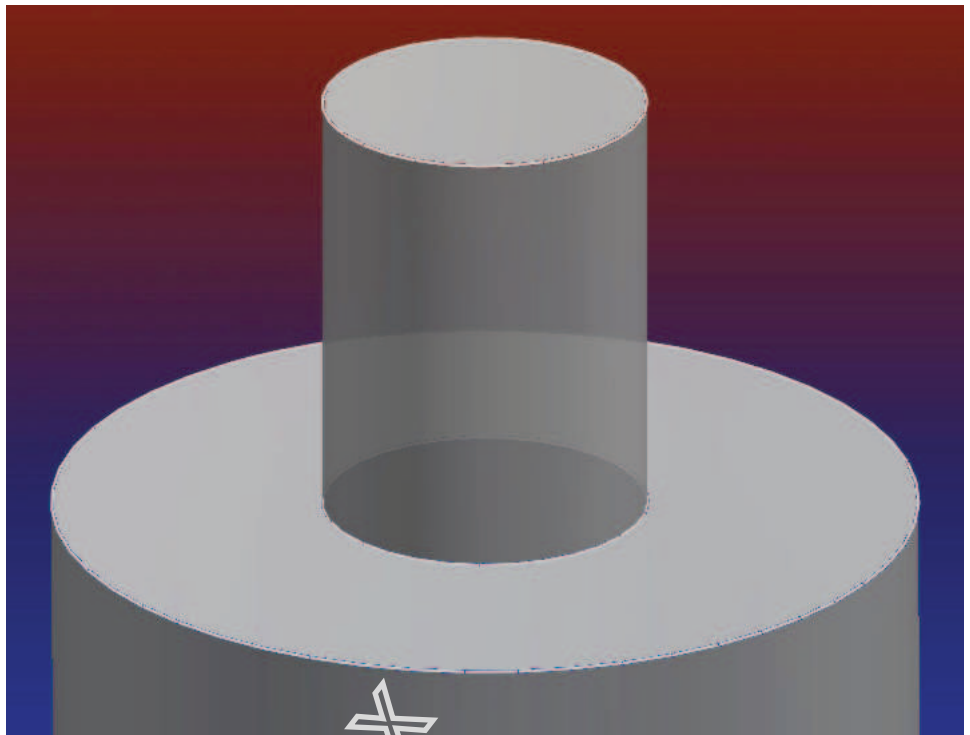
Continued from previous

8. Open the next part that this relief UDF will be created in.

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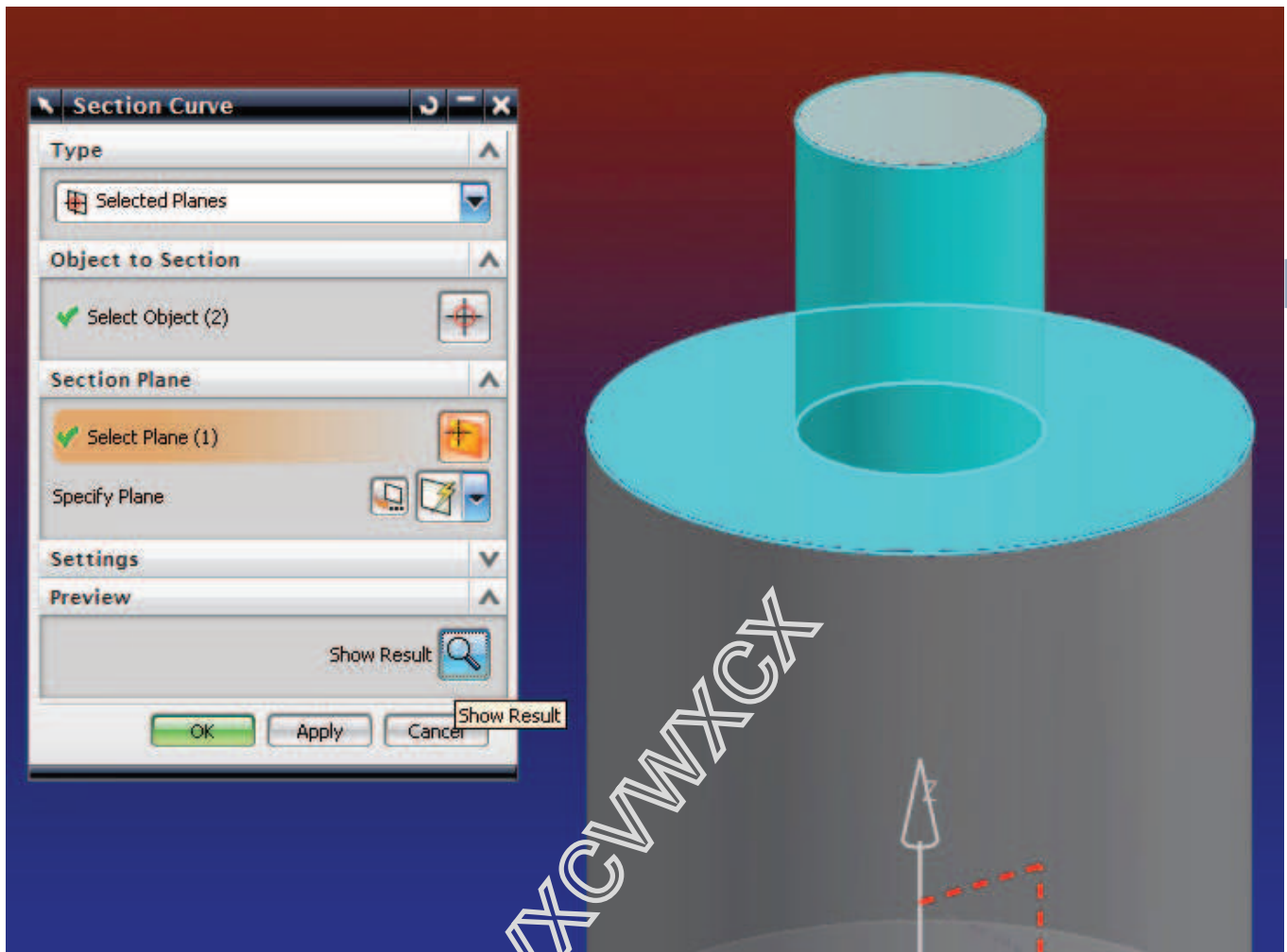
9. It is necessary to create some of the external referenced objects prior to inserting the UDF.

- Select **Insert => Curves from Bodies => Section**.
- Select the 2 faces for the horizontal and vertical sketch geometric constraint definitions



- Select the ZC-YC plane as the section curve.

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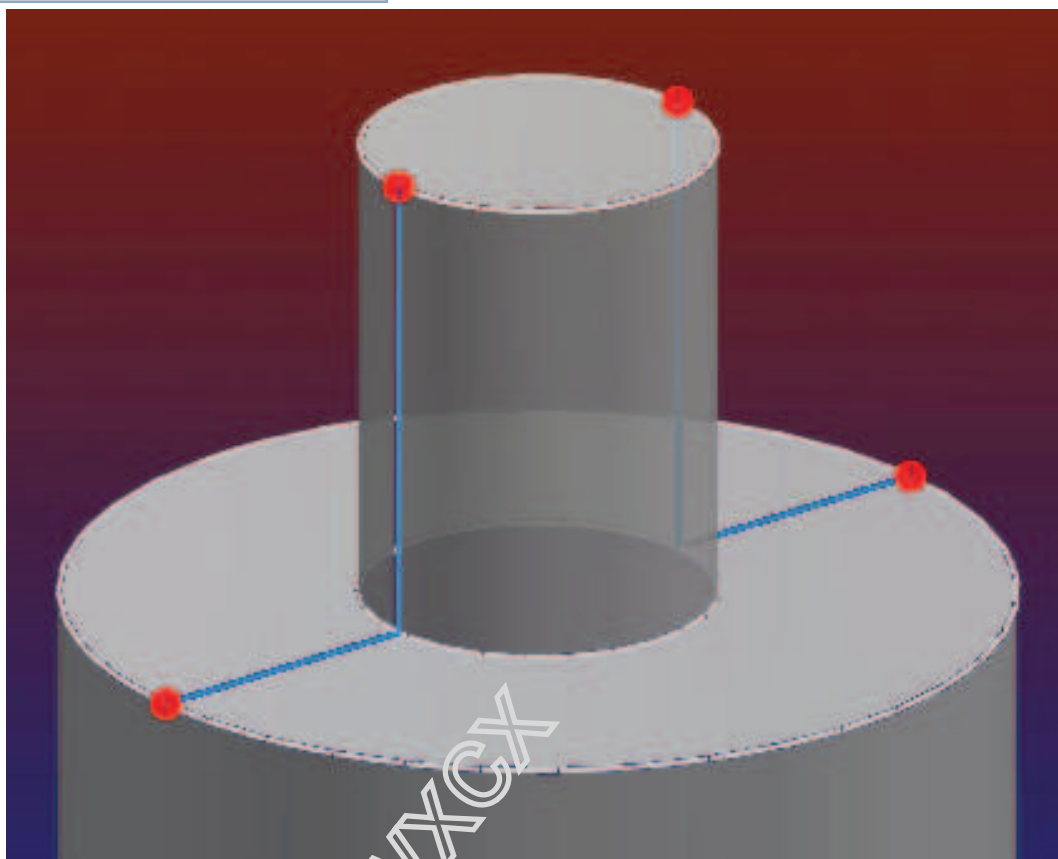
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- Preview of curves
- OK



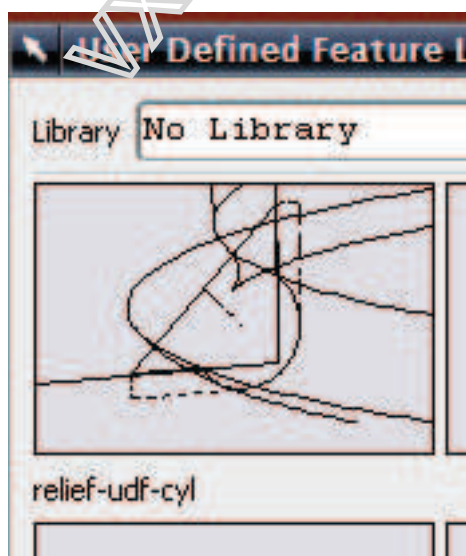
Continued from previous

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10. Next, insert the UDF.

- Select **Insert => Design features => User Defined Feature**
- Select the UDF created; **relief-udf-cyl**.



11. Notice the UDF dialog will appear and display all available parameters that can be changed for the UDF.

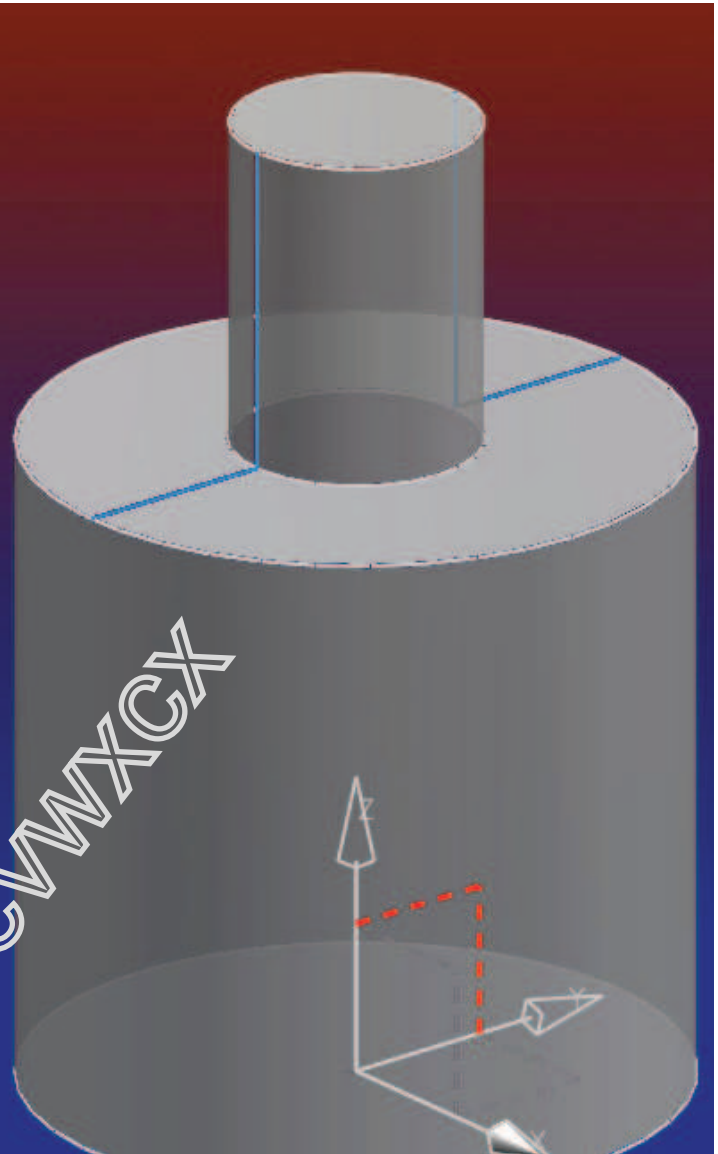
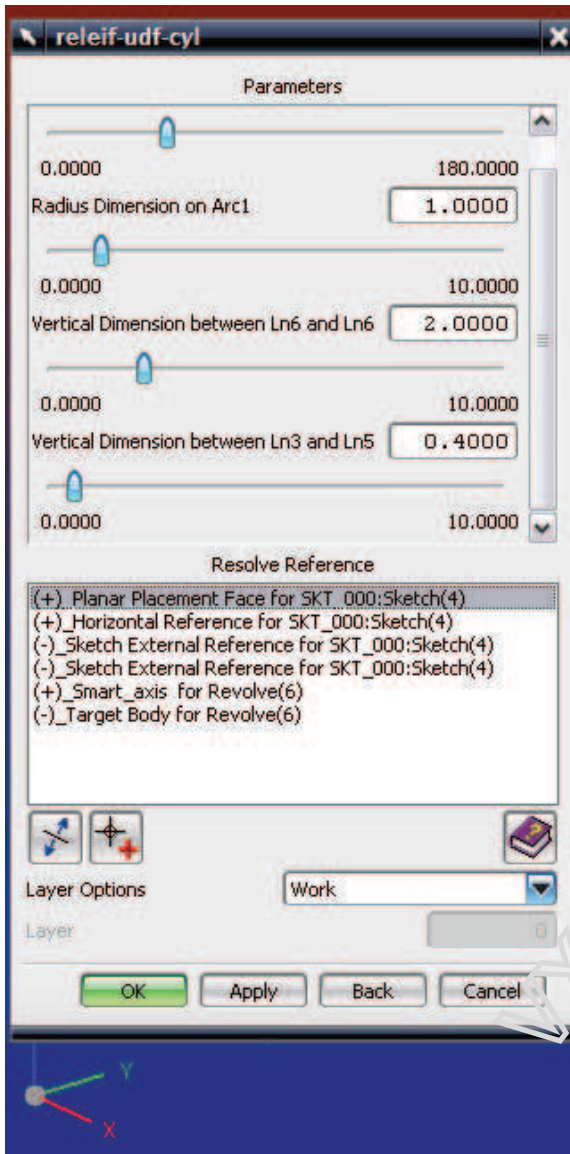
- Select the Sketch Planar Placement face.

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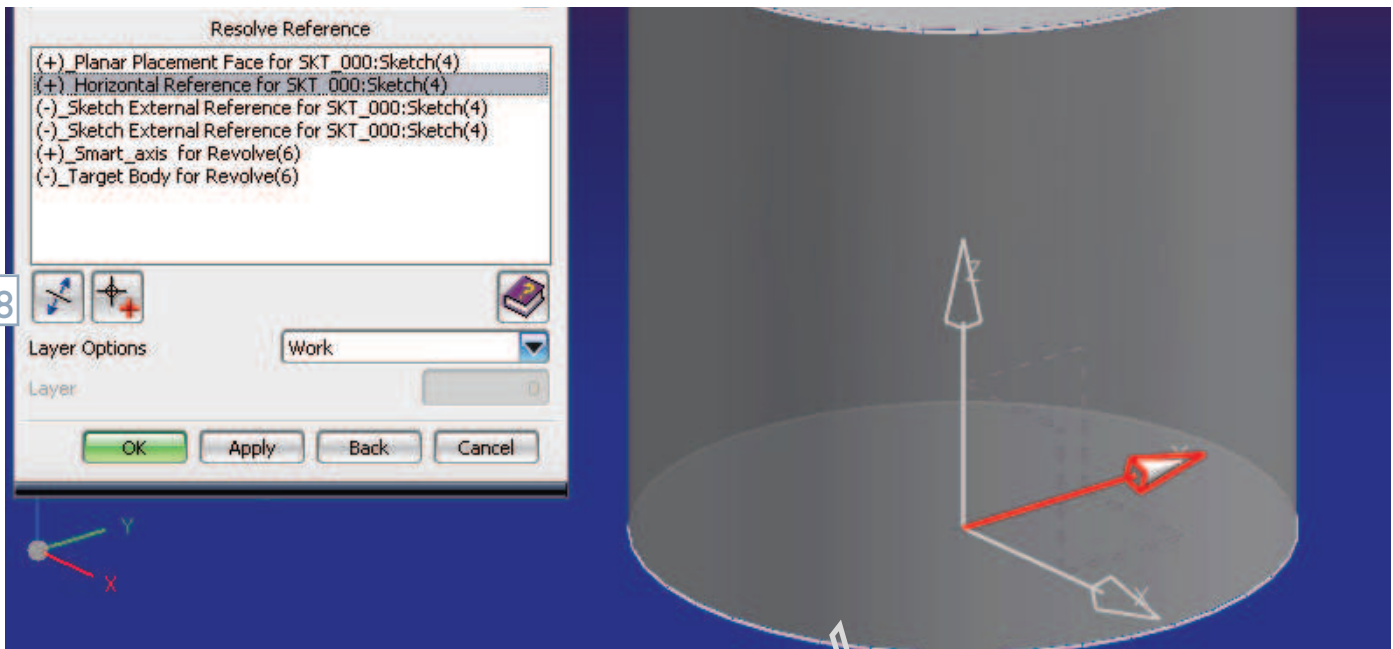
- Select the Horizontal Reference direction (if necessary).



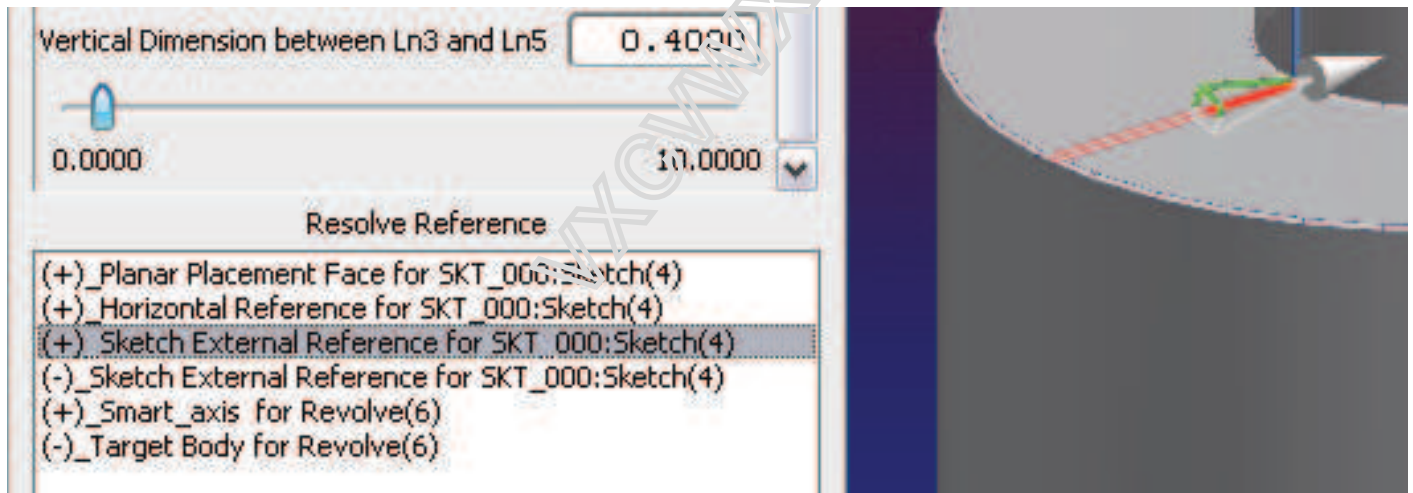
>> NX

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- Select the first External reference for the sketch geometric constraint.

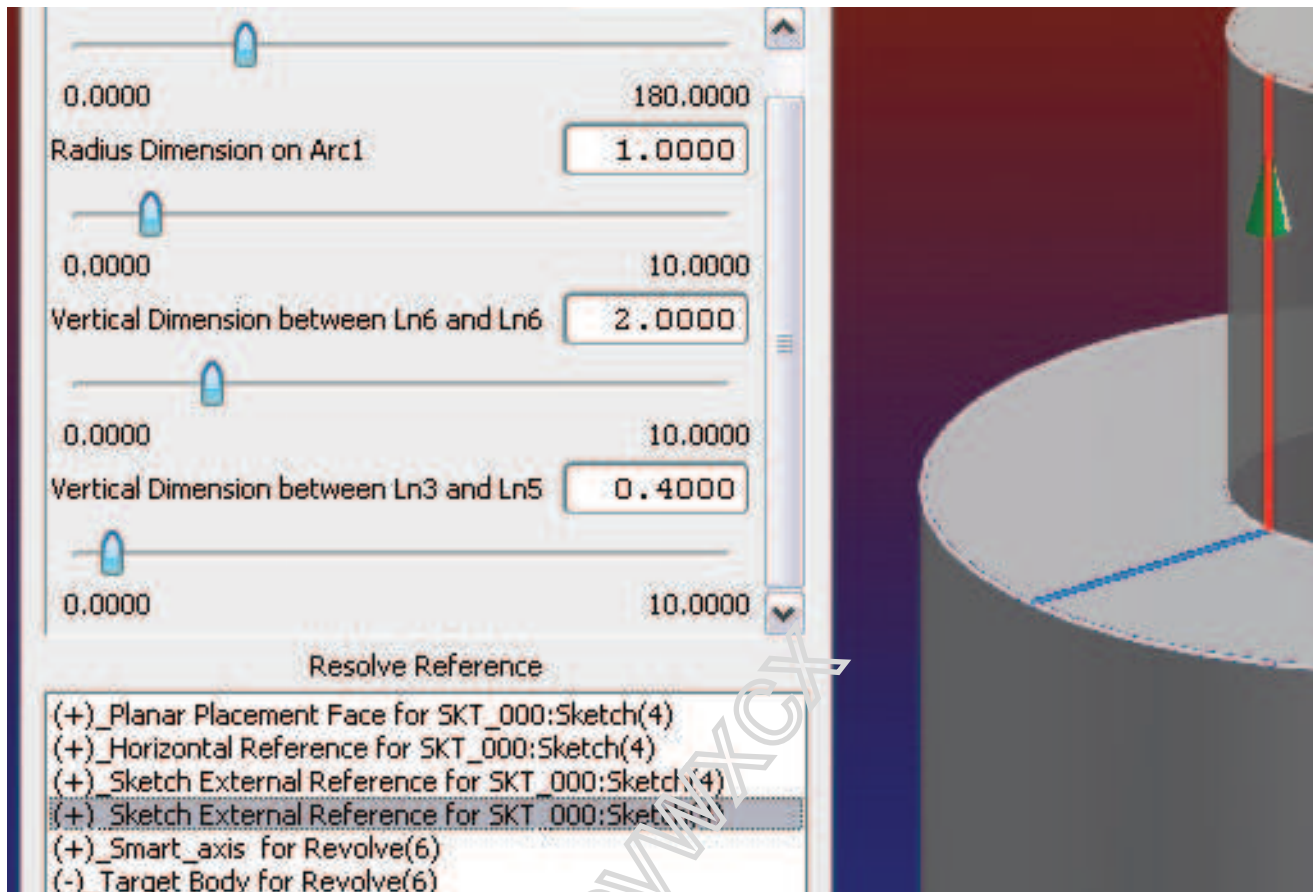


- Select the second External reference for the sketch geometric constraint.

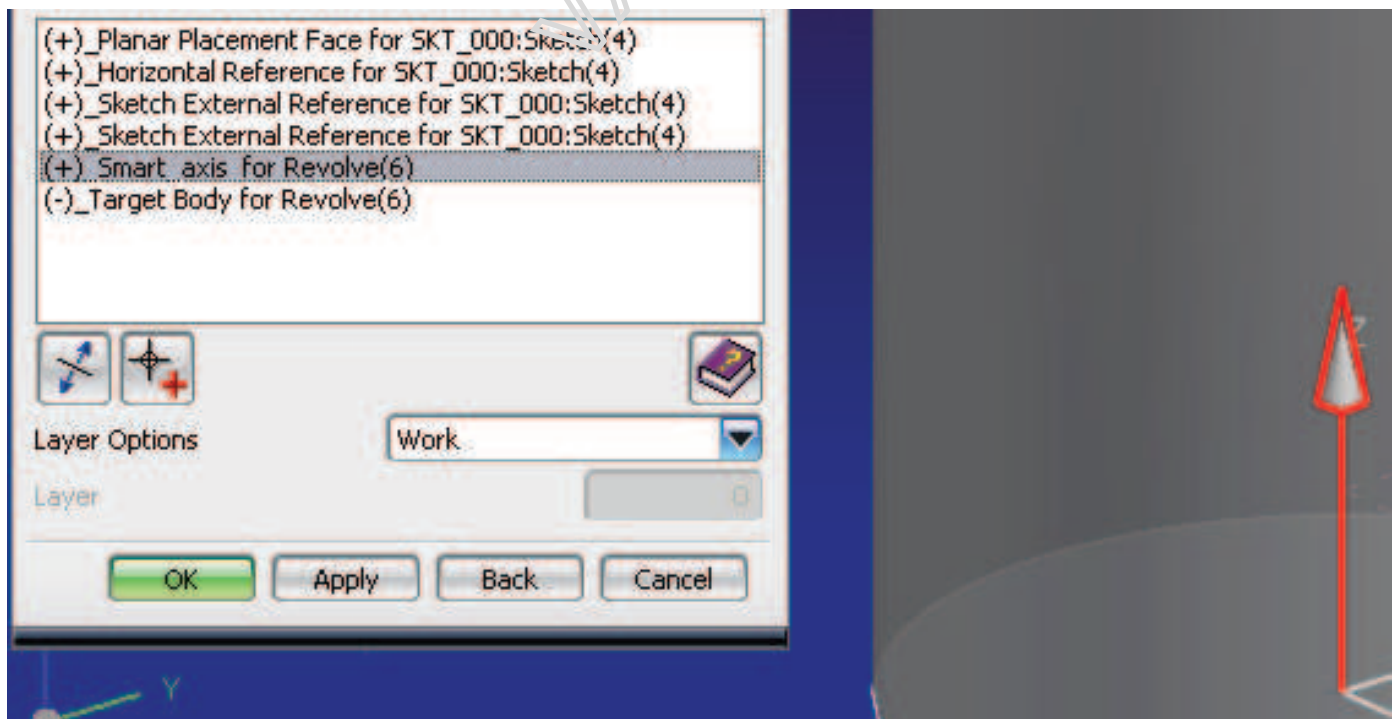


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- Define the Revolves axis of revolution.



- Select the object to perform the subtract Boolean.

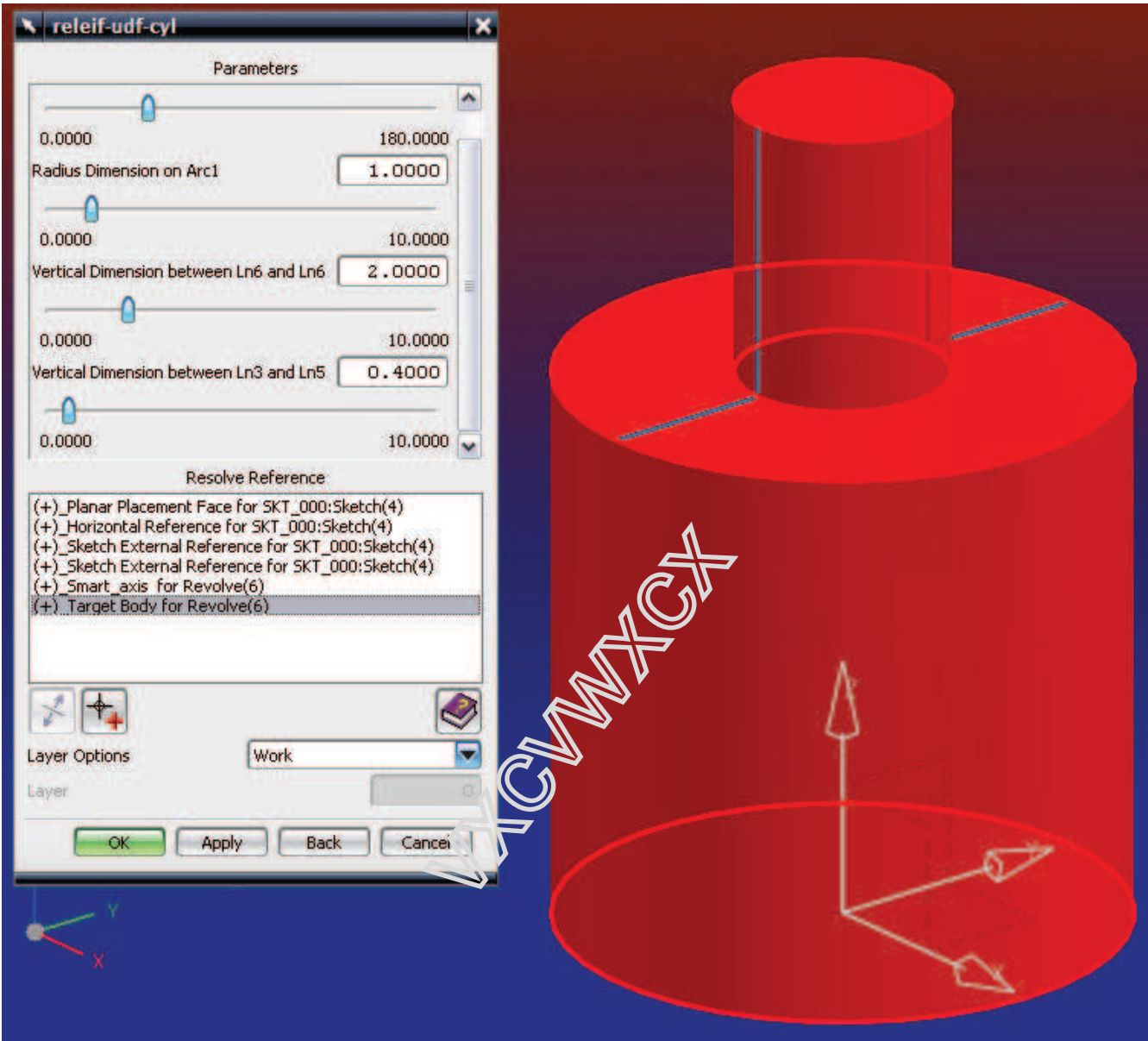
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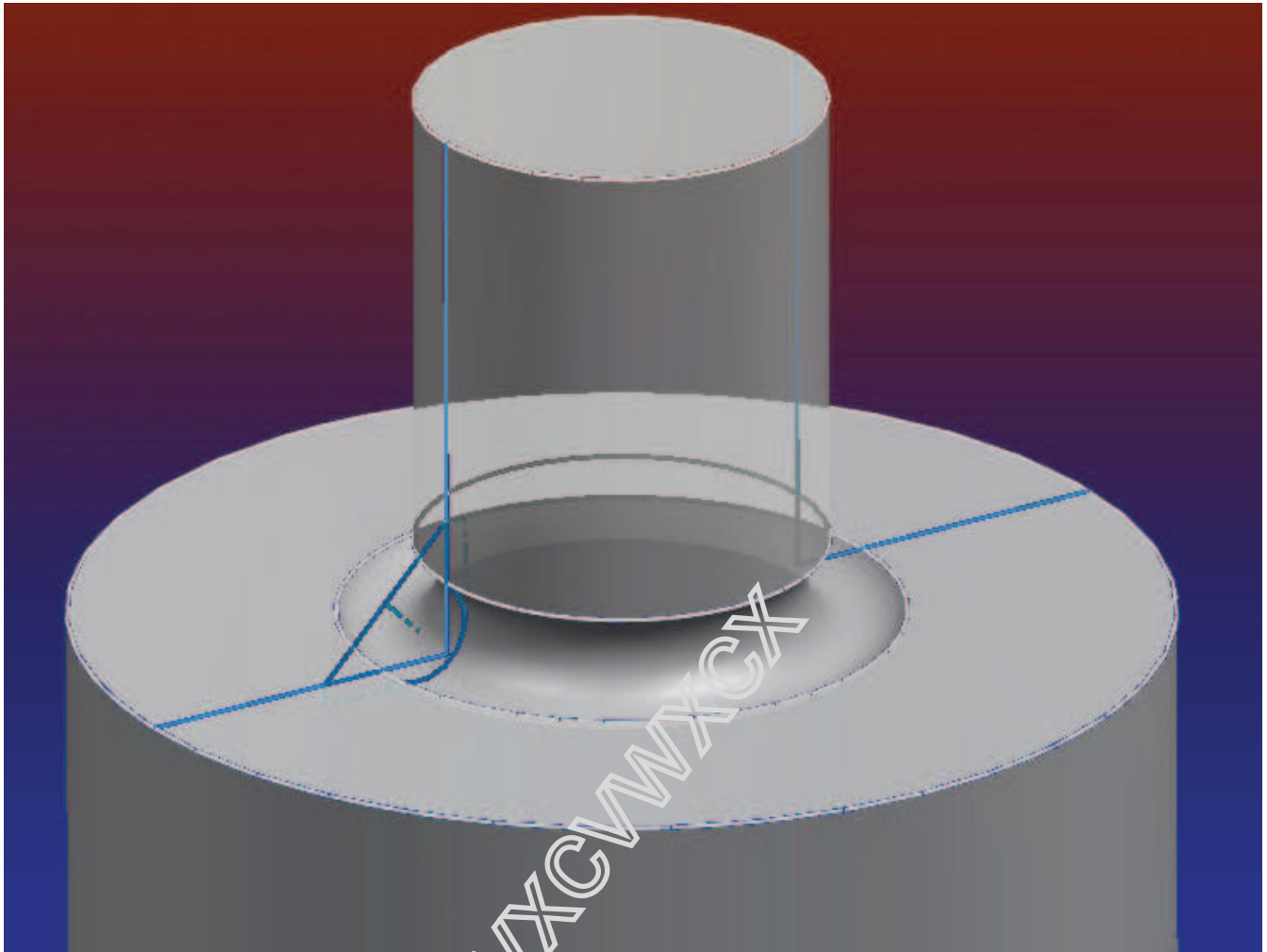


- OK.



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Now the relief UDF feature is added with the parameters specified during creation.

