## 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.









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.





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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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## <u>Stephen Chlopecki</u>



# 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.

Definition

Features

References

Summary



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



- 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

Type  Selected Planes  Object to Section  Section Plane		
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Object to Section		
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• Select the ZC-YC plane as the section curve.





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Section Curve				
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- Preview of curves .
- OK •



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#### 10. Next, insert the UDF.

- Select Insert => Design features => User Defined Feature
- Select the UDF created; relie tudiccyl.

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- 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).



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



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• 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.



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