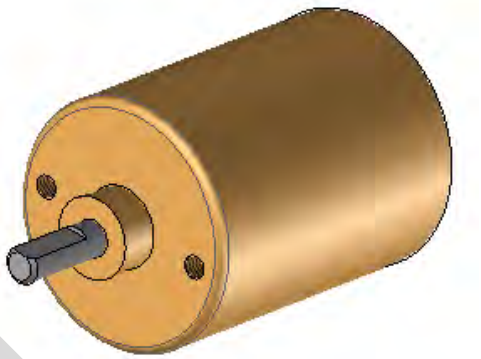
Exercise 1:
Model a Part using Revolved Protrusion

In this exercise you will learn to model a part in 3D using the revolved shape function in TopSolid Design.



Overview:

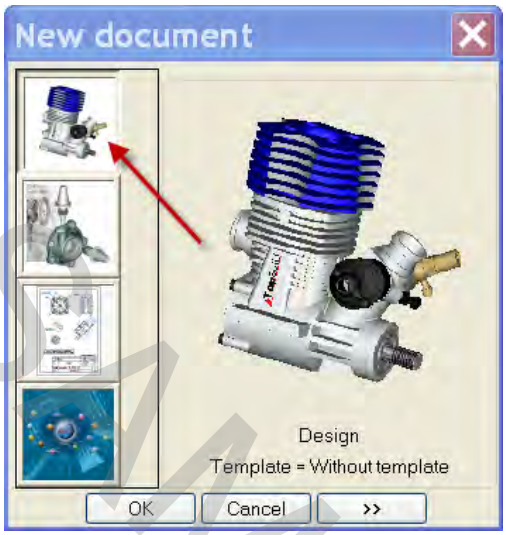
The modeled part is a DC reduction motor from Micromotors - reference L149.6.21

The AGV has two DC gear motors that drive the wheels.

The motors can be purchased from:
Micro Motors s.r.l
Viale Piave, 80/82 – 23879 VERDERIO INF. (LC) ITALY
Tel: 039510611-499 – Fax : 039513617
www.micromotorssrl.com
email: micromo@tin.it

1. **Create a new TopSolid'Design New Document** 

- Start TopSolid and choose **New Document** - **DESIGN** mode
- Select **No Template** from among the standard templates

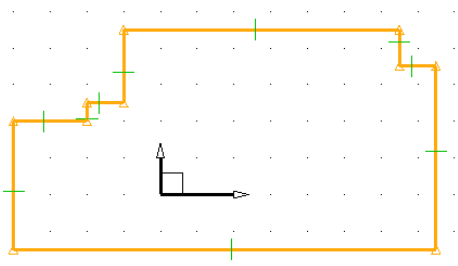


- **Save** the document as : *DC_motor.top* 

Note:
When creating a solid volume or protrusion it is first necessary to create a sketch which will form the base of the protrusion

2. **Create a sketch**

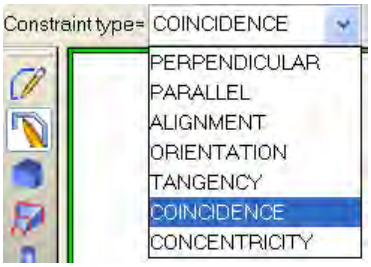
- From the context menu select **Sketch** 
- Choose the function **Create Contour** 
- Sketch, using point to point a sketch which approximates to the following:



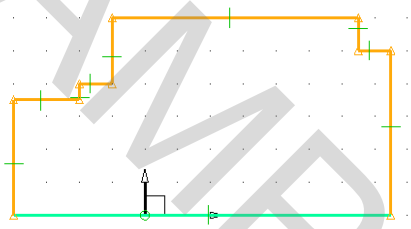
Note:
By default, unconstrained line segments are in orange.

3. Constrain the sketch

- Select the command **Constraint** 
- **Select** the type COINCIDENCE

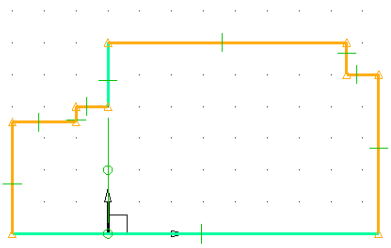


- Make the bottom horizontal line coincident with the origin.

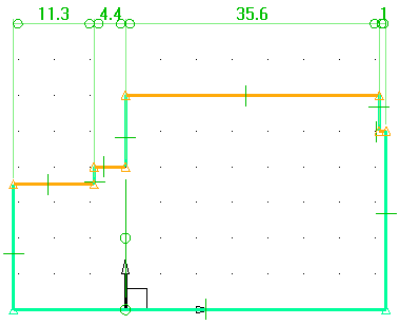


Note:
The segment turns green. This indicates that it is constrained in position.

- Repeat the procedure for the vertical segment in the sketch as follows:



- Select the Dimension  function and dimension the horizontal line segments as follows: **11.3, 4.4, 35.6, 1**

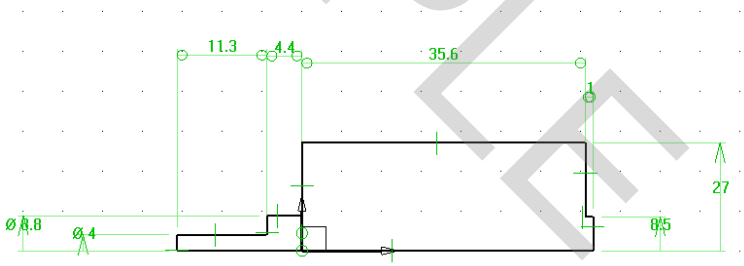


- Click on the segment, click to position the dimension, type in value, and hit the return key
- Select the function **Half Part Dimension**
- Select the x axis as the reference axis and dimension all the diameters

- Use **Modify Element** to change the diameter dimensions to: **4.0, 8.8, 27.0, 8.5.**
- Validate each change by hitting the Return key

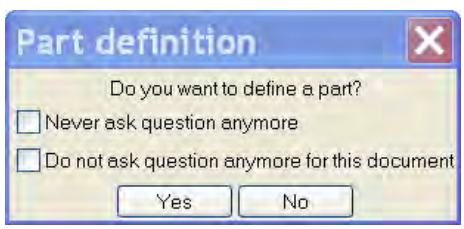
4. Create the Revolved Protrusion

- From the Context menu select **Shapes**
- Select the function Create **Revolved Shape**
- Highlight the Profile
- Select the **X+** as the axis of Rotation

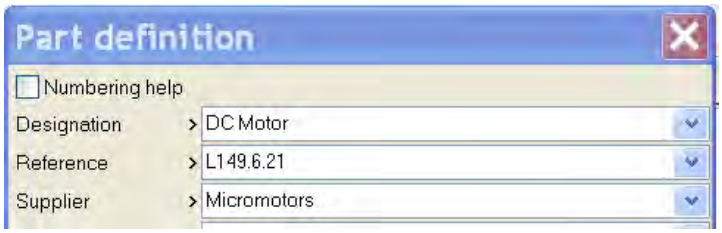


- Choose an angle of **360**° and validate
- Save the Document

Note.
A Dialogue box appears asking if you would like to define the part. This allows us to identify the part and its characteristics which we can exploit later on in the Bills of materials.



- Click **Yes** and highlight the part, then click STOP
- Define the part as follows:

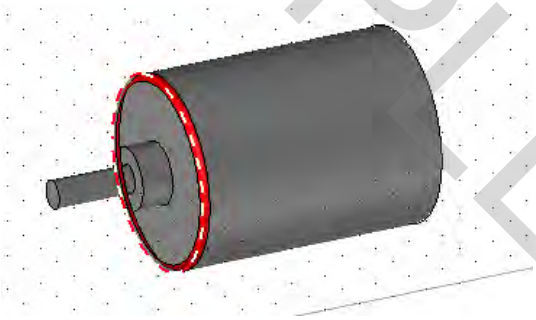


- Click **OK** to terminate

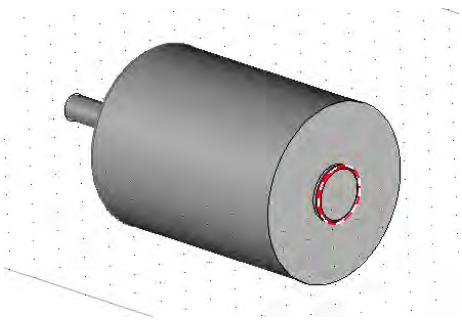
Note.
Don't forget to regularly save your work. A good practice is to click on the save button each time an operation has been successfully completed.

5. Creating Fillets

- Select the function **Fillet**
- Specify a radius of **1mm** and accept using the **Return** button
- Highlight the front edge and click on COMPUTE FILLETS

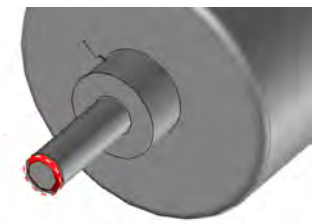


- Repeat the procedure for the **0.5mm** fillet at the back



6. Creating a Chamfer

- Select the function **Chamfer**
- Specify a value for the chamfer: **0.5**
- Highlight the edge and click on **COMPUTE CHAMFERS**



Note:
We are now going to create a cut-out on the rotor arm

7. Create an appropriate coordinate system for the slot

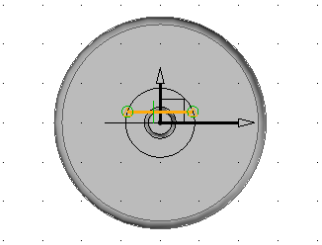
- Select the command **Coordinate System** 
- Select **Coordinate System on Point** 
- Select **Point** 
- Choose **Centre Key Point**  and select the outer edge of the rotor arm
- Click on **Current Coordinate System**  and select the last coordinate system
- Select **Modify Element**  and highlight the coordinate system
- A dialogue box appears which allows you to change the characteristics on the coordinate system.

- Click on the **X Button** to turn the coordinate system **90º** about the **X** axis.
- Click three times on the **Y Button** to turn **270º** about the **Y axis**

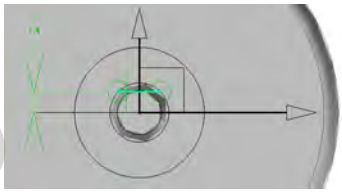
8. Create a contour for the cut-out

- Select **Sketch** from the context menu 
- Click on the **Line**  command and draw a line segment (Approximately as shown)

- Place **COINCIDENCE** constraints  between the ends of the line segment and the edge of the chamfer on the rotor.

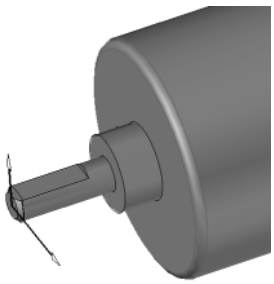


- Select the command **Dimension** and dimension the line segment **1.4mm** from the origin



9. Create a contour for the cut-out

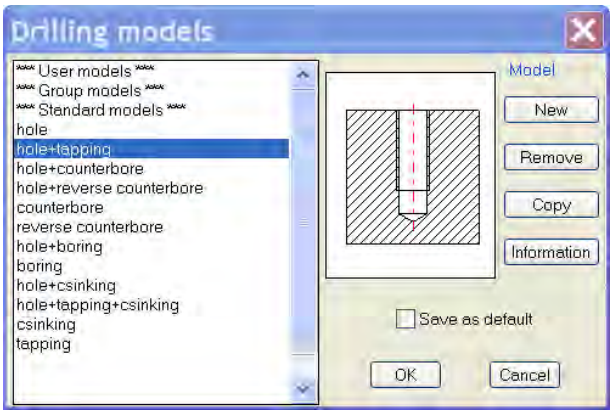
- From the context menu select **Shapes**
- Activate the function **Trim**
- Choose the option **By Sweeping Curve**
- Highlight the motor as the Shape to trim, pick the line as the trimming curve then click the button **>>**
- Enter the length of the protrusion as **8.5** and hit **Return**
- Click on the arrow to change the direction if it is not pointing inwards. Then click **OK** to accept the protrusion direction.
- Click once more on OK to accept the direction of the cut.



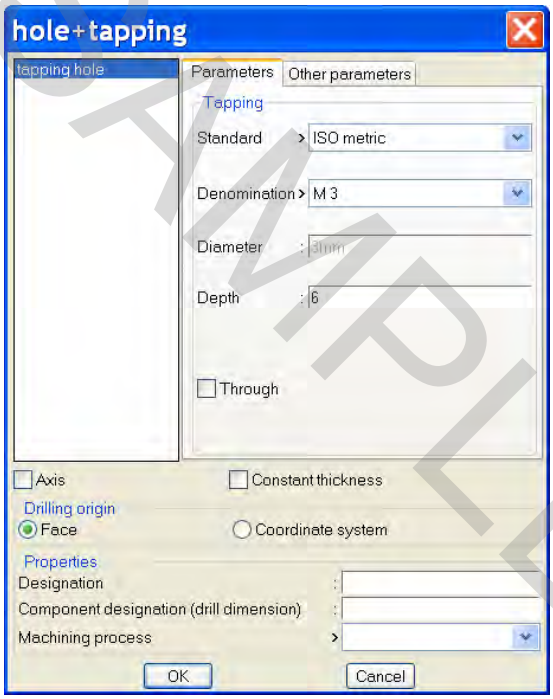
10. Create the Tapped holes

- Select the function **Drilling**
- Choose **Coordinate System = Polar**

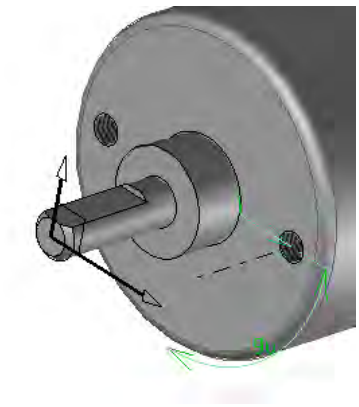
- Highlight the face to drill



- Set the drilling model to hole & tapping and click **OK**
- Set the parameters: **Denomination M3, Depth 6**



- Accept with **OK**
- Select **PROPAGATE**
- Choose **CIRCULAR** , **Z+, 360º**, Total number : **2**
- Select **Modify Parameter** , highlight the position dimension of the first hole.
- Change the angle to **90º** and the radius to **10**



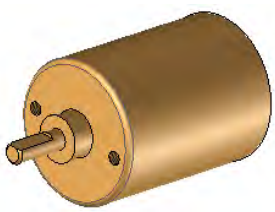
11. Change the graphic Layout

- Click on **Current coordinate System** and, 
- select **Absolute Coordinate System**
- From the context menu select **Attributes**  followed by the function **Elements Visibility** 
- Select the coordinate system and the dimensions to make them invisible.

12. Change the Color of the DC motor

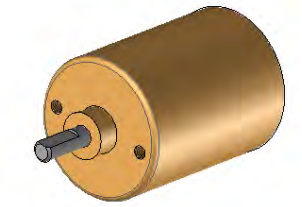
- Click on the color pull down menu
- With the right button clock select the Yellow or Light Gold

- Left click on the part to change its color.



13. Change the shaft color

- From the **Shape Menu** select **Mechanical / Other Operation – Color** 
- Choose a color, then highlight the rotor arm to change its color
- Select Apply Color



- Don't forget to save your work 

Summary:
In this exercise you learnt how to generate a revolved protrusion. You also used other useful tools and functions, and looked at making entities visible or invisible.

This motor was created in isolation as a standalone part which is appropriate for once of standard parts.