Manufacturing Process - Extrusion

Parts have a uniform (constant) cross section
Extrusion

Material (billet), hot or cold, is either pushed or pulled through a die. The long piece can be cut to a desired thickness.
Extrude Command in SolidWorks

Adds material

Removes material
Create a 2D sketch

Select Extruded Boss/Base

Select the sketch

Preview of the solid, there is a problem with the sketch if the preview does not appear, most likely the sketch is not continuous

Choose Blind and enter the thickness and accept
Extrusion

Click to change the direction of the extrusion

Click on the arrow and drag to change the thickness using the scale on the screen

On screen scale
Extrusion

Two directions could be used for extrusion: direction 1 15 mm, direction 2 5 mm
Extrusion

The part can be hollowed out by selecting the **Thin Feature** option and assigning a value for the wall thickness.

- **5 mm wall thickness**
- **Copper material rendered**
Extrusion

To taper the extrusion: select the taper option and assign the desired taper angle in degrees, 15 degrees.
Revolve Command

Select the revolve command, choose the axis of rotation and the amount of rotation in degrees. The axis of revolution could be any line in the sketch.

360 deg. revolution

180 deg. revolution
Revolve
The part can be hollowed out by selecting the *Thin Feature* option and assigning a value for the wall thickness.

![Rendered model](image)