

Lecture #6

SOLID MODELING



This week

You will learn 3D (or solid) modeling.

The steps to follow are:

- Geometric modeling
- 3D modeling (coordinates)
- Line or wireframe modeling
- Surface modeling
- Solid modeling
- Creating 3D parts
- **Assignment # 5**

Geometric modeling

Geometrical modelling is a general term applied to 3-D techniques.

There are three main types of geometrical modelling used.

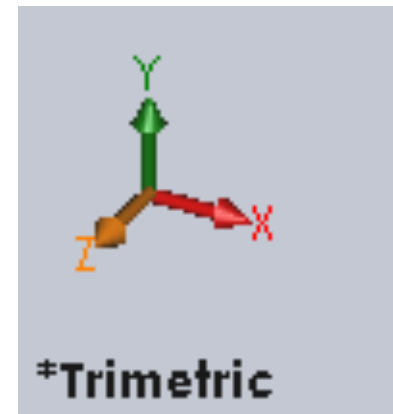
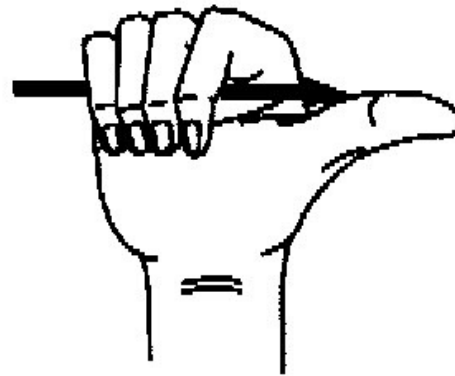
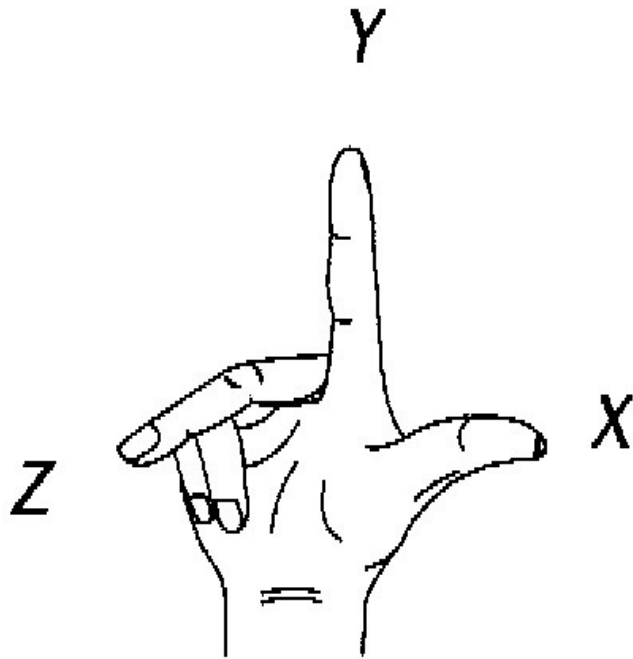
Wireframe (line) modeling

Surface modeling

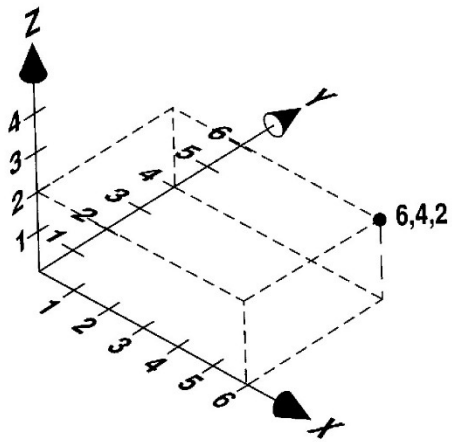
Solid modeling

Geometric modeling

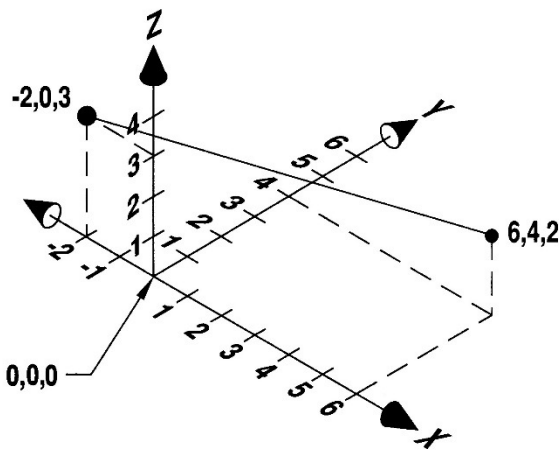
The *right-hand rule* provides an easy-to-understand reference to determine the positive and negative directions of the X, Y, and Z axes.



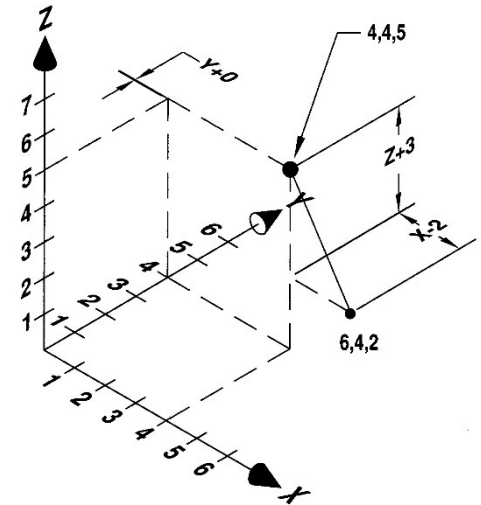
3-D Coordinates



Point coordinate

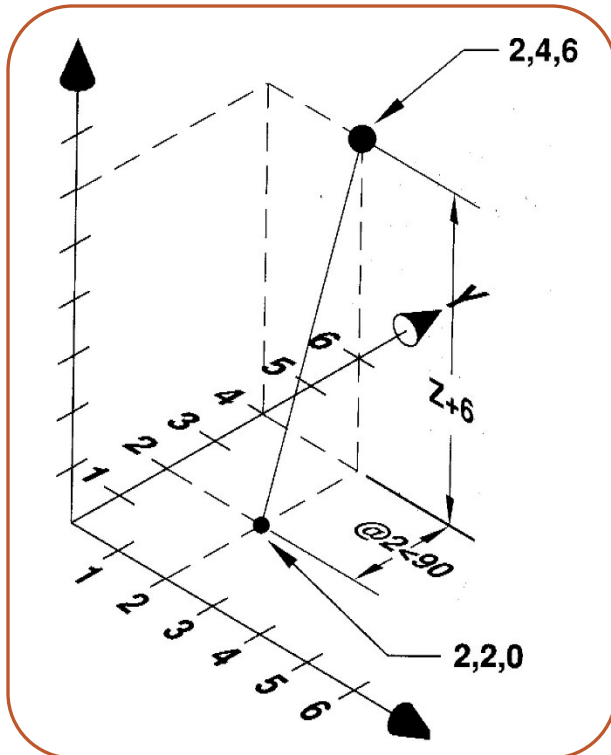


Absolute coordinate

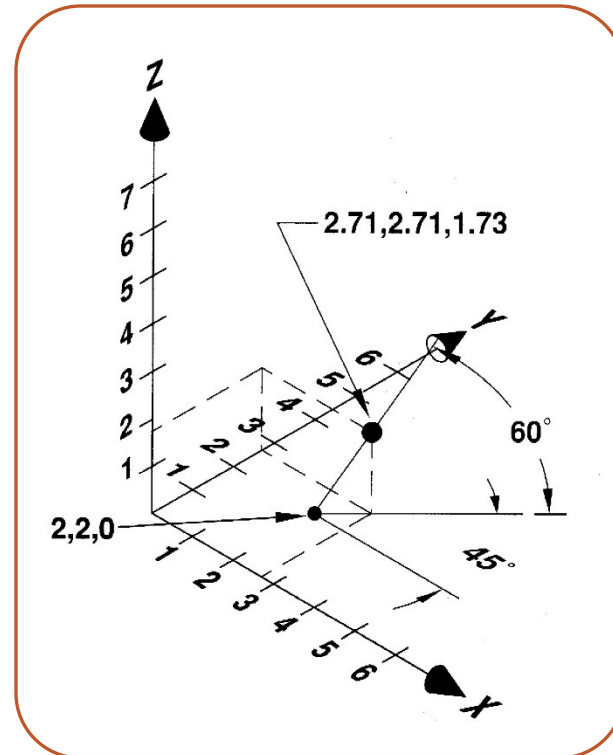


Relative coordinate

3-D Coordinates



Cylindrical
coordinate

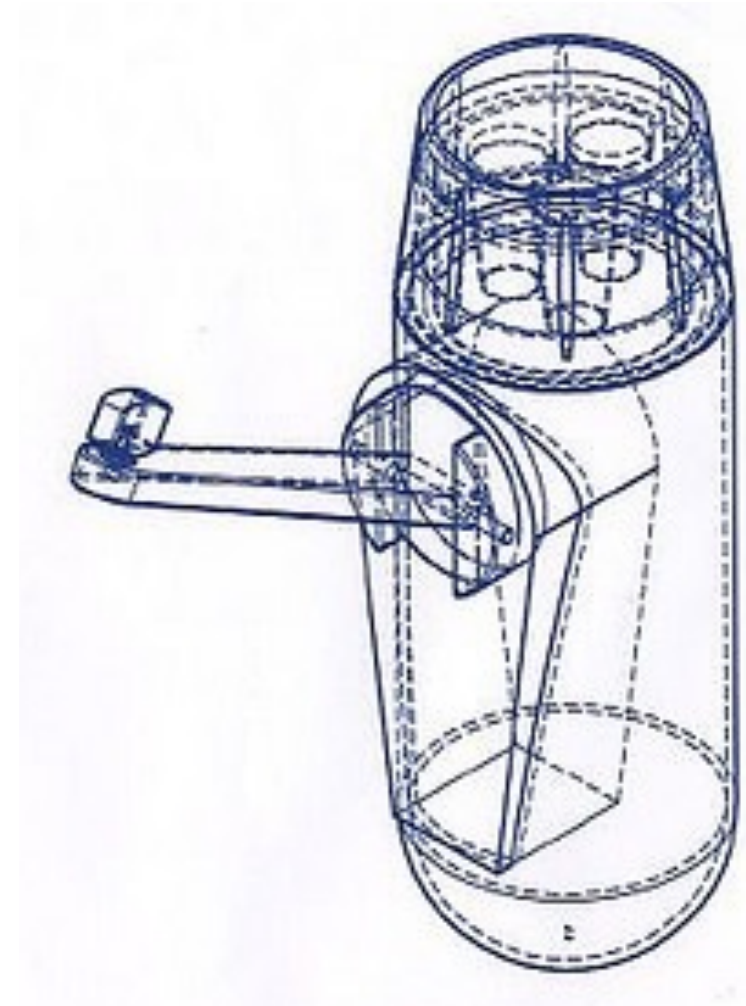


Spherical coordinate

Line or wireframe model

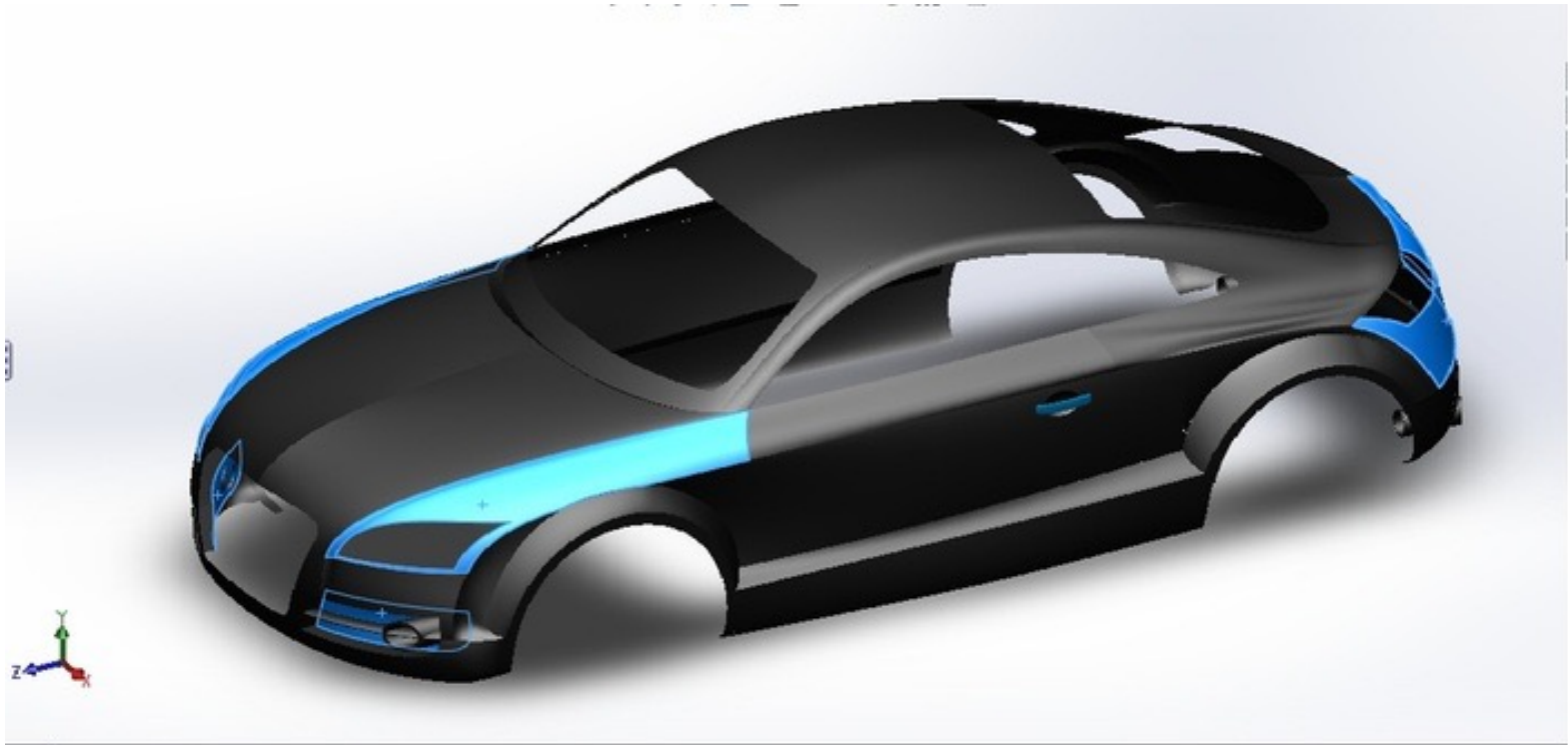
Line modelling or, as it is often called, wireframe modelling is the simplest form of geometrical representation of an object.

The object has the appearance of a frame constructed from wire. It can be quickly displayed and manipulated.



Surface model

Surface models define the surface features, as well as the edges, of objects.



Solid model

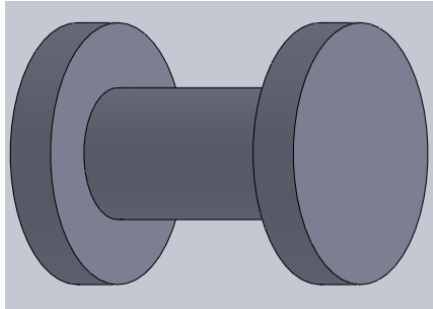
Solid model allows the automatic production of realistic images of a shape and automation of the process of interference checking.

Solid modelling takes the designer into areas of still greater realism than that of surface or line modelling.

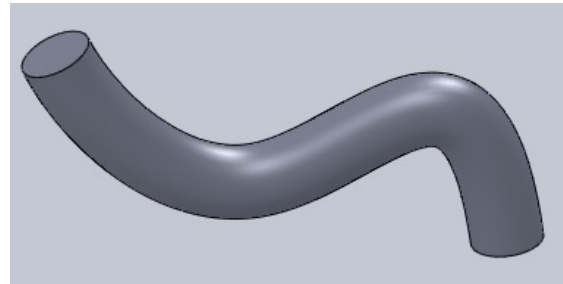


Features

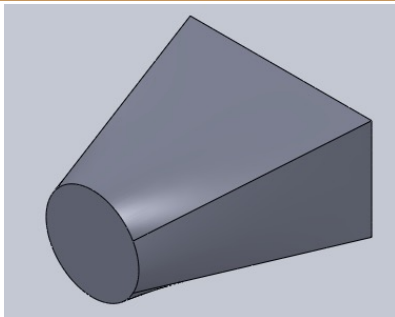
Revolved Boss/Base



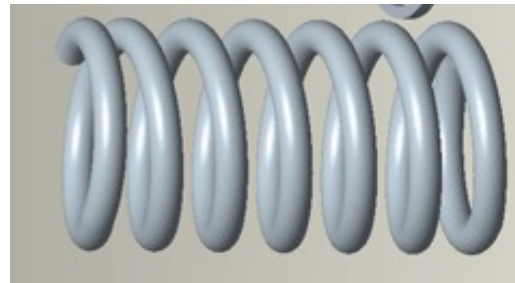
Swept Boss/Base



Lofted Boss/Base



Helix and Spiral



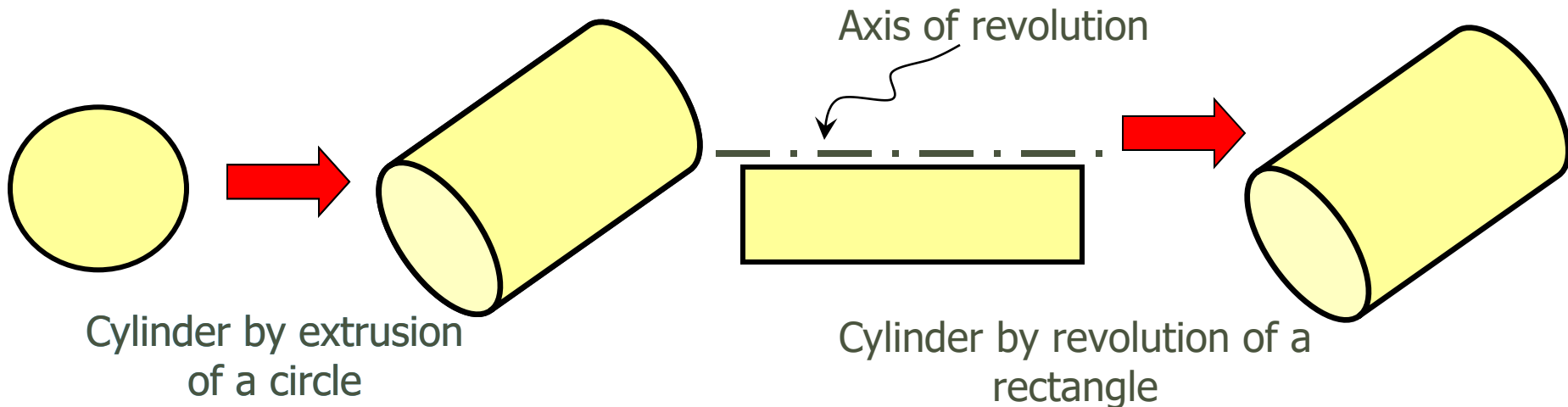
Creating 3D parts

SOLIDWORKS 2014

Modeling strategy

The methods that you choose to create 3-D solids will depend on the shape of the geometry you are trying to model.

Many times, you can use more than one method to achieve the desired result. It is up to you to choose the method with which you are the most comfortable.



Parent & Child relationship

Parametric models are composed of features that have established relationships. Features are built upon other features in a way that resembles a family tree, hence the phrase *parent-child relationship*.

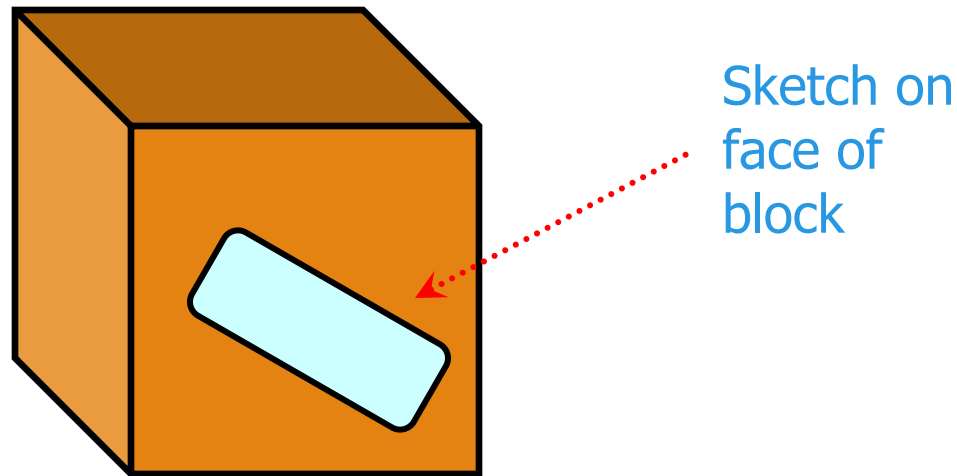
Implicit relationships can be established through the adding of a numeric equation using the relations option.

Explicit relationships are created when one feature is used to construct another.

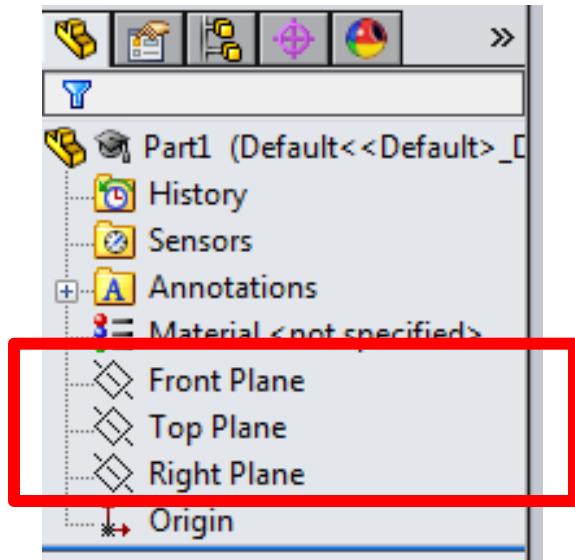
Working (sketch) planes

Note that, from this view, it is really impossible to tell exactly where the shape is located in 3-D space.

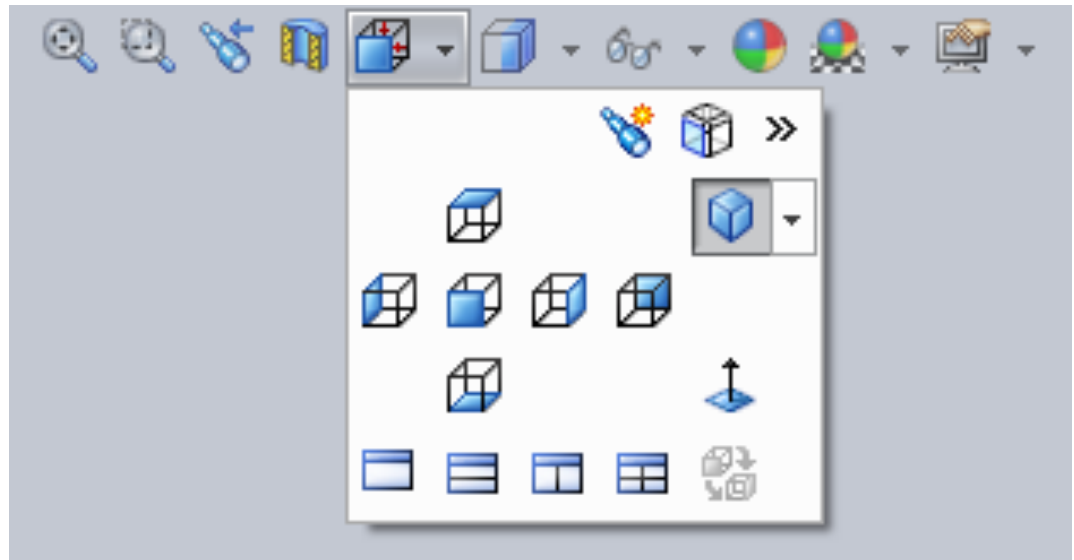
Since you know that it was drawn on the indicated surface, its location is fixed in space.



Working planes

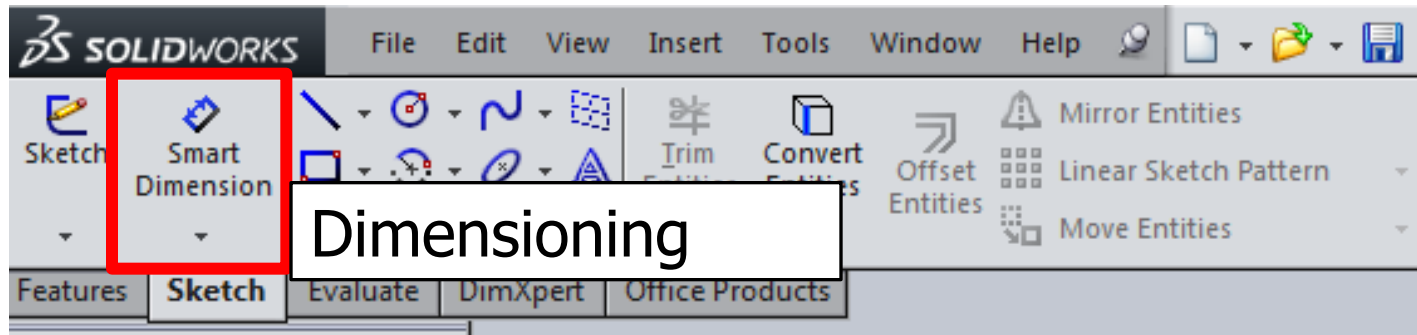
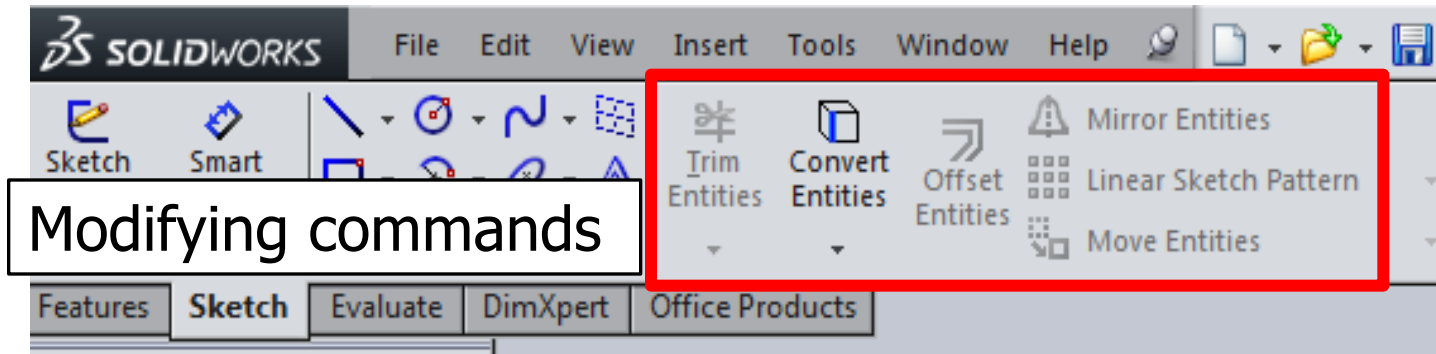
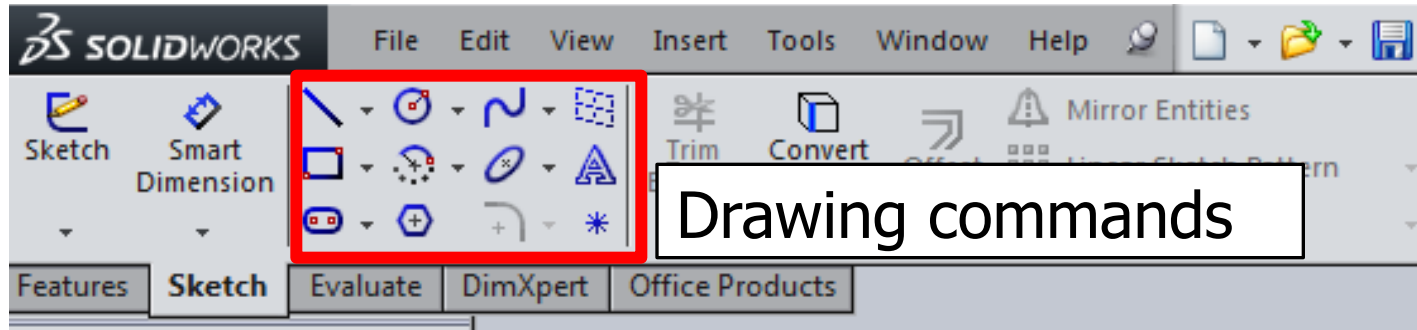


Basic views

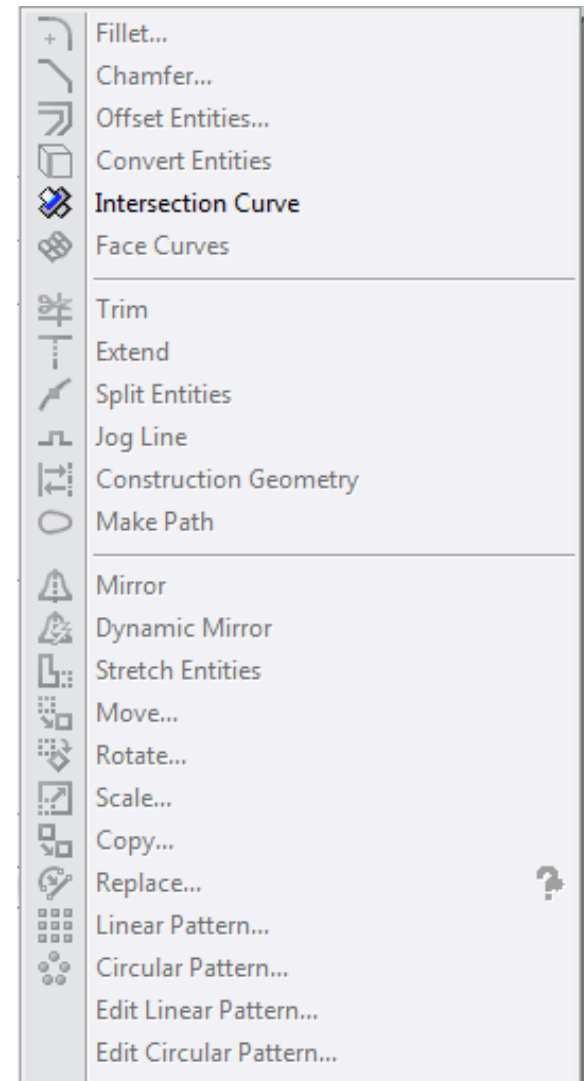
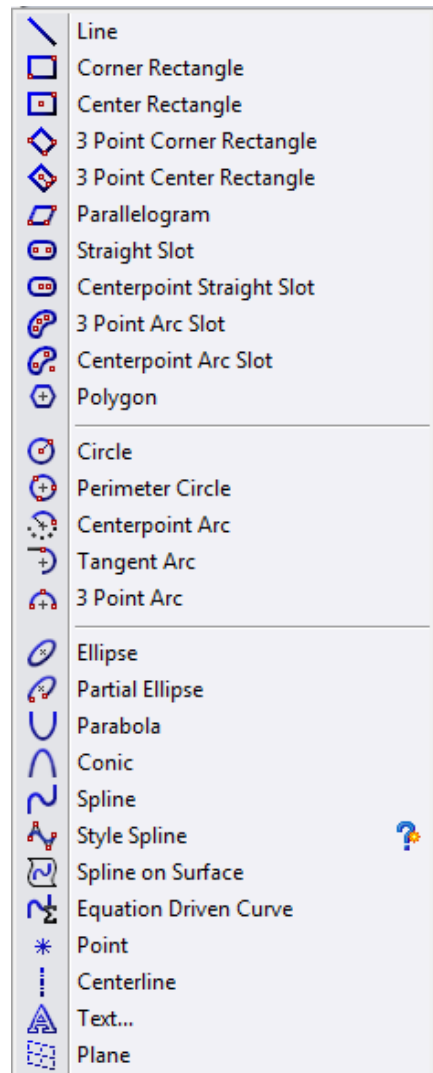
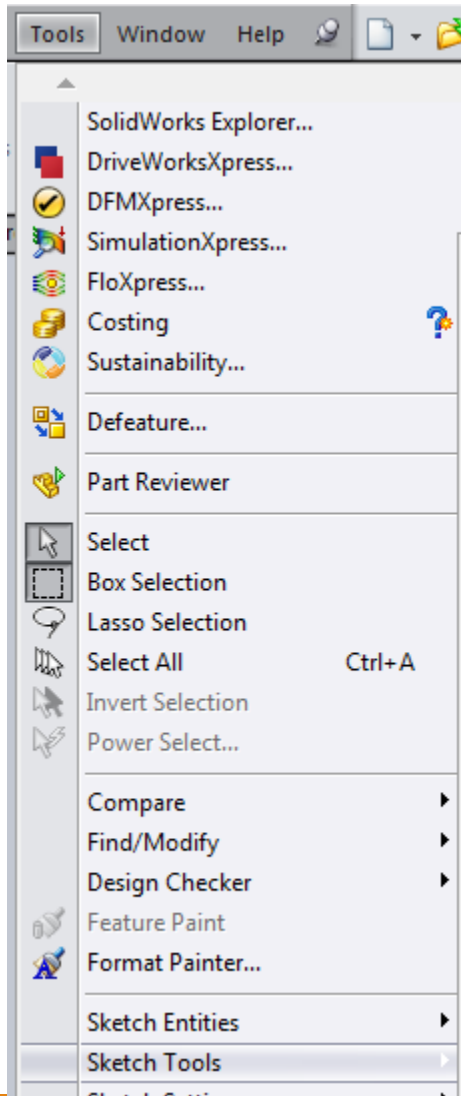


Principle views

Sketching tools



Sketching tools



View tools



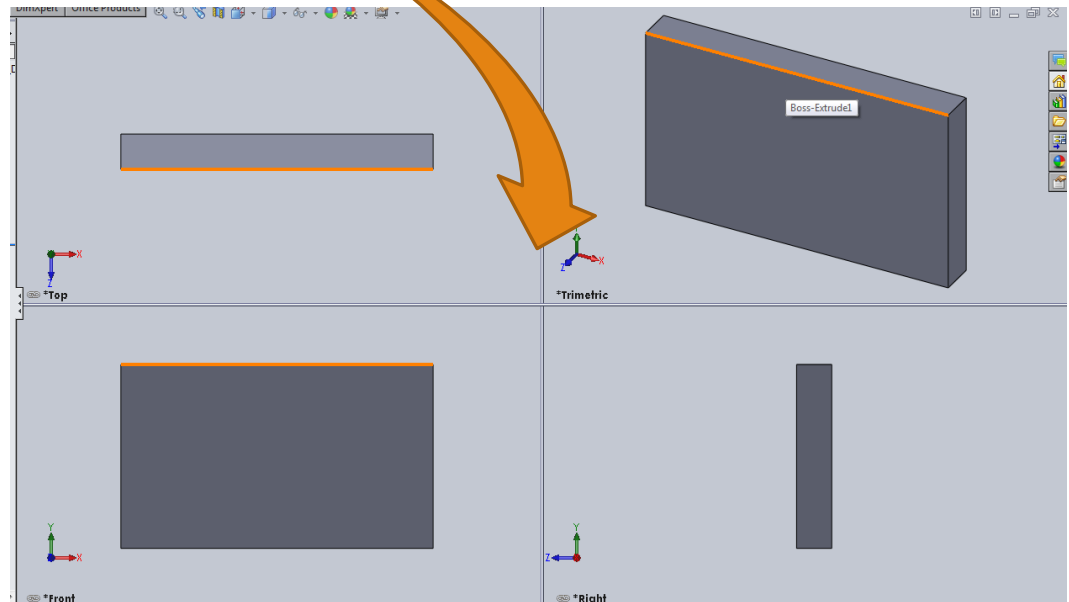
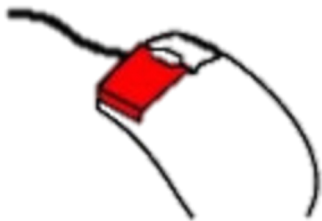
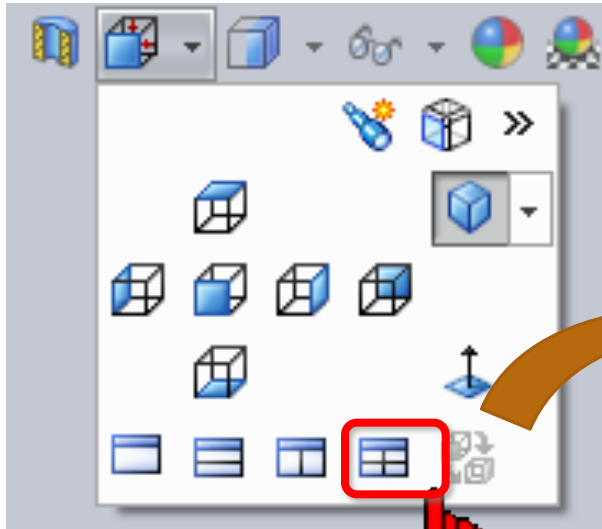
Zoom to Fit (F)
Zooms the model to fit the window.

A callout box with an orange border and a white background. It points to the first icon in the toolbar. The callout box contains the text: **Zoom to Fit (F)** and Zooms the model to fit the window.

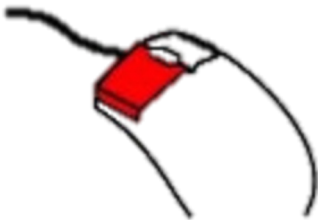
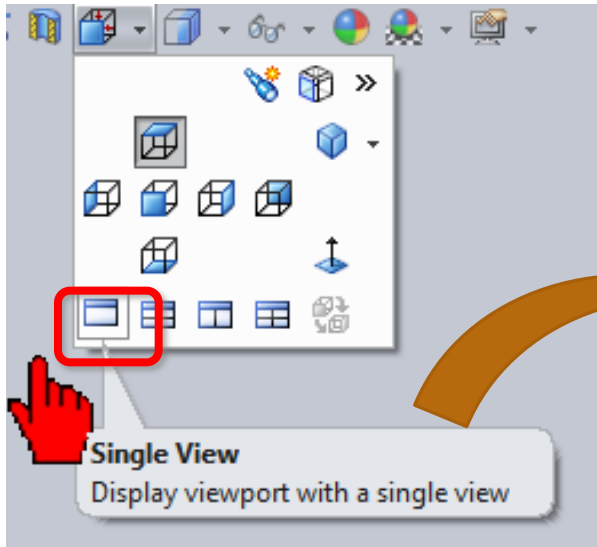
Zoom to Area
Zooms to the area you select with a bounding box.

A callout box with an orange border and a white background. It points to the second icon in the toolbar. The callout box contains the text: **Zoom to Area** and Zooms to the area you select with a bounding box.

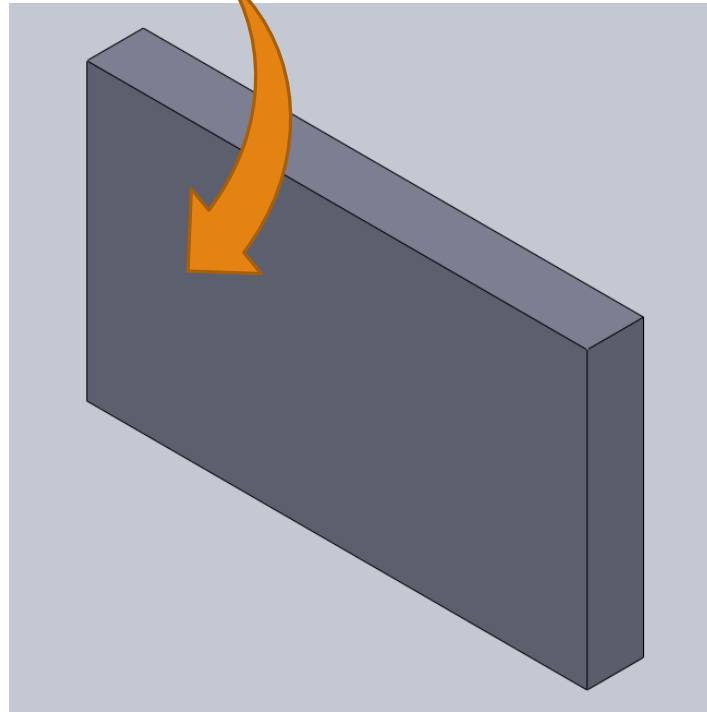
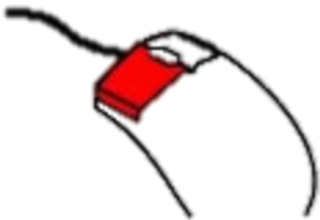
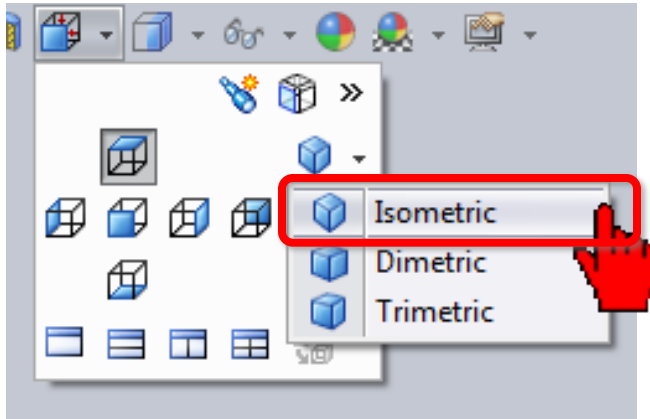
View tools



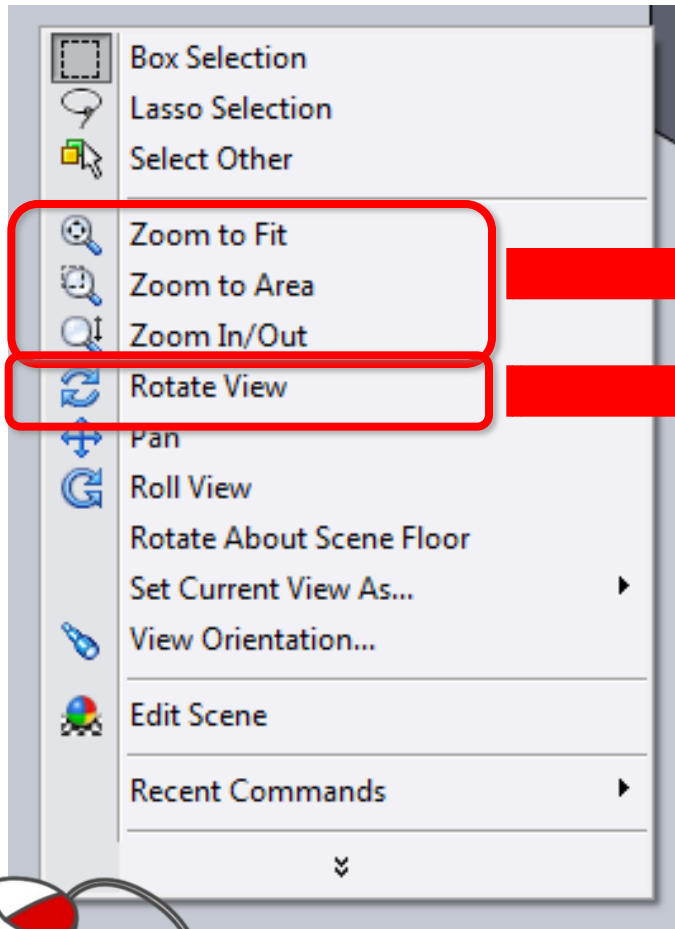
View tools



View tools



View tools



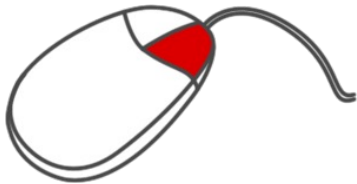
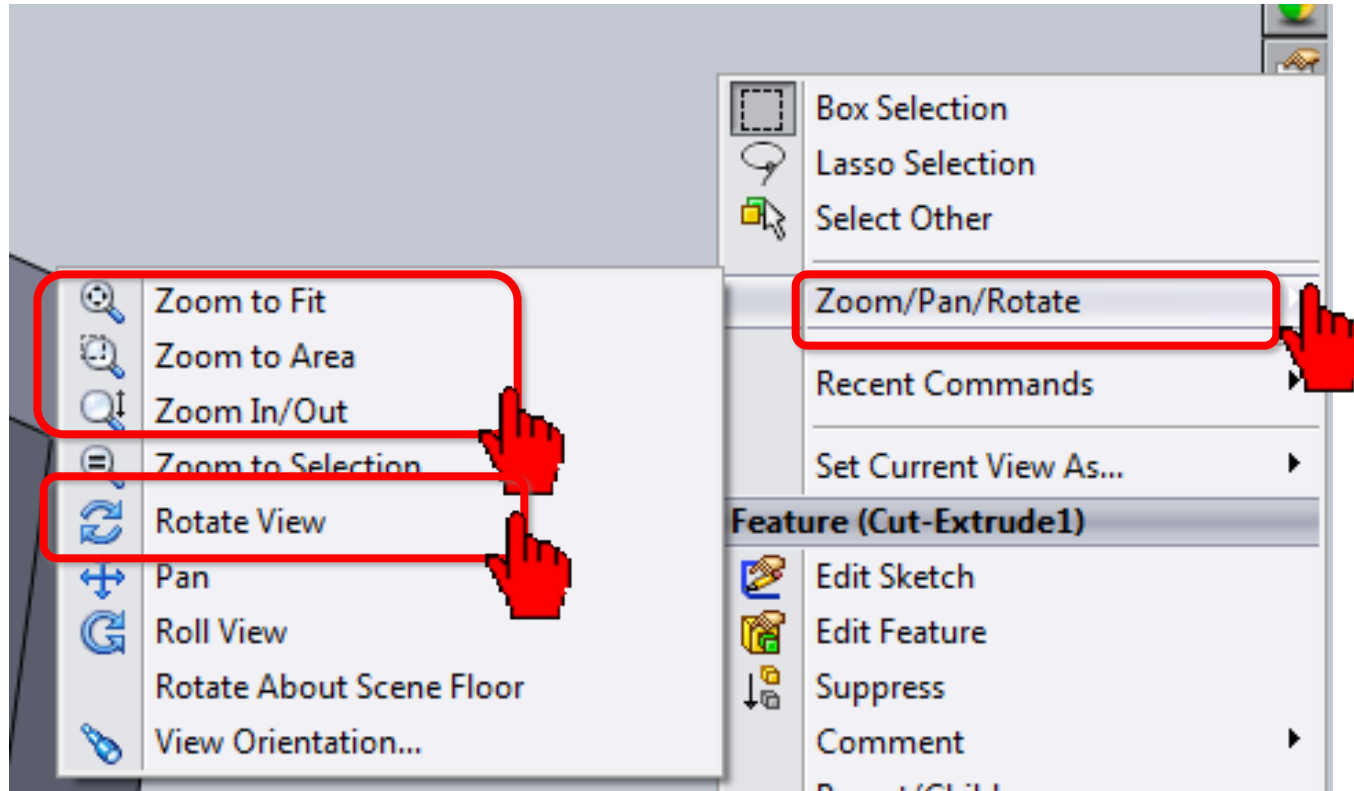
Zoom commands

Rotate object



Hold left button
and rotate

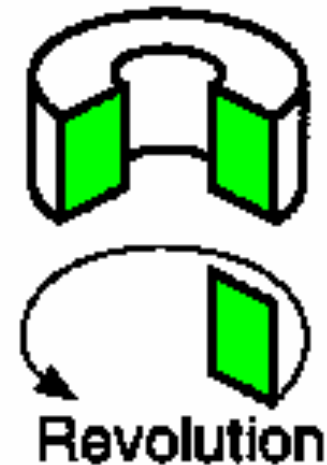
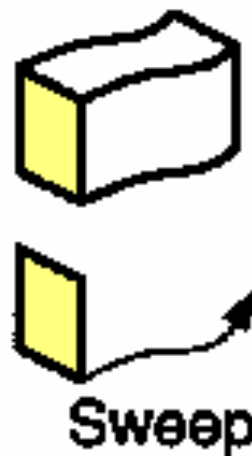
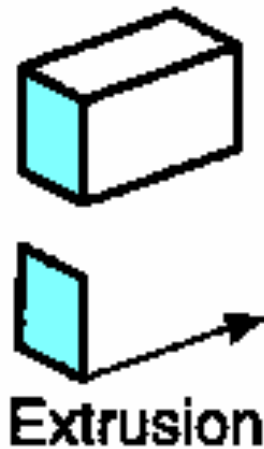
View tools



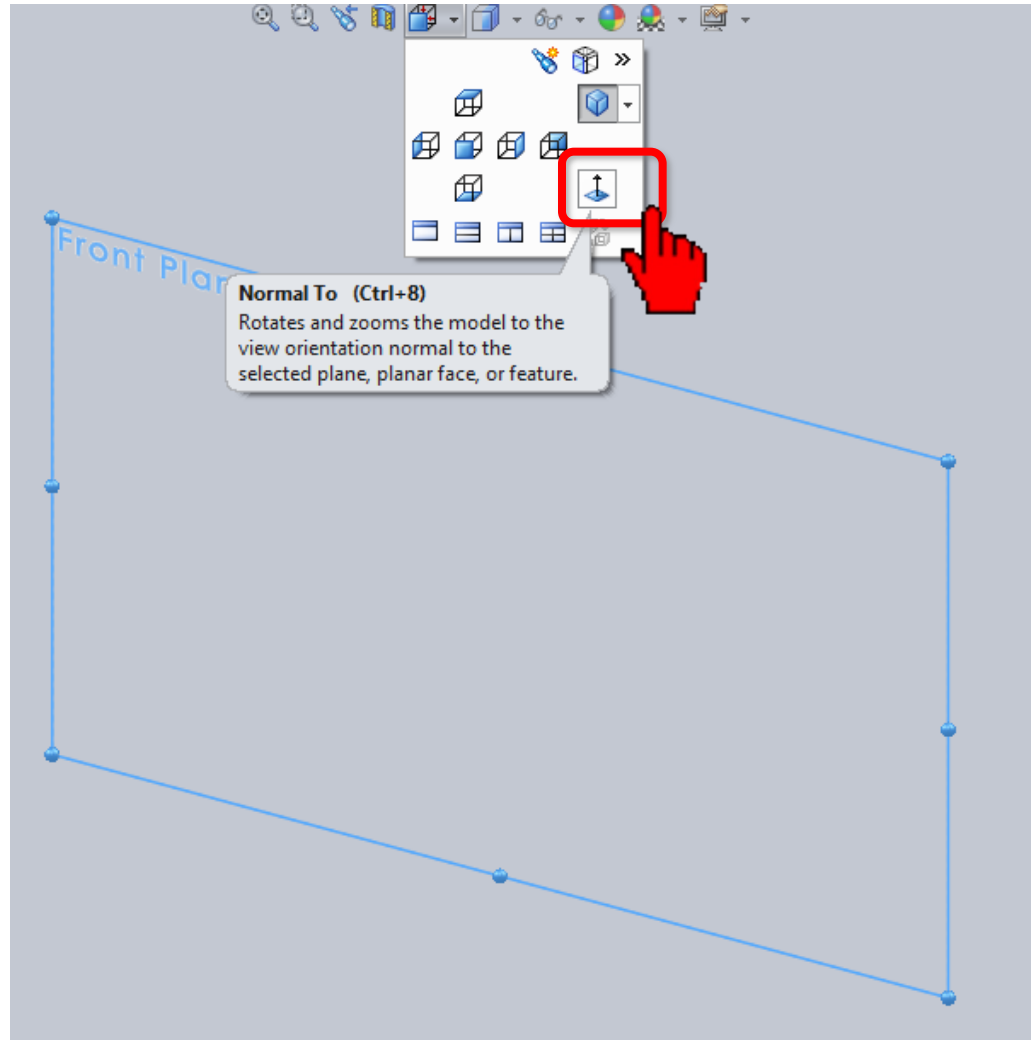
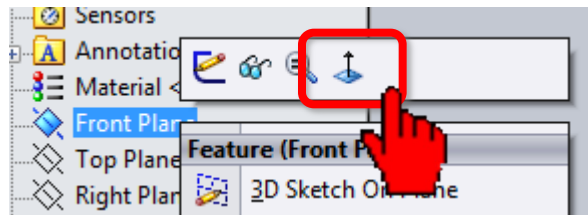
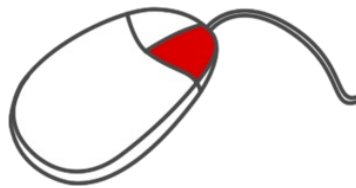
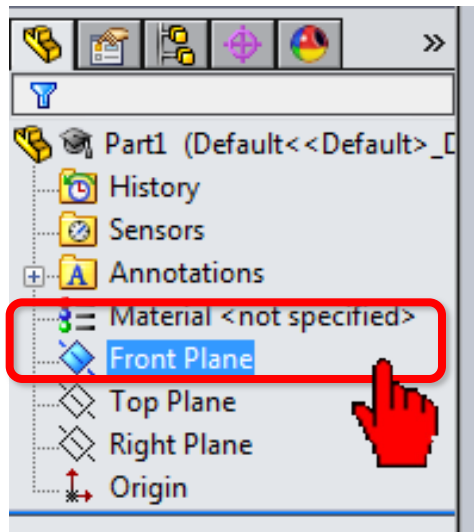
3D parts from 2D geometry

Features can be defined in different ways, such as using wireframe geometry or by getting standard parts out of a catalog.

Some of the basic feature types includes extrude, revolve, fillet, chamfer, and shell.

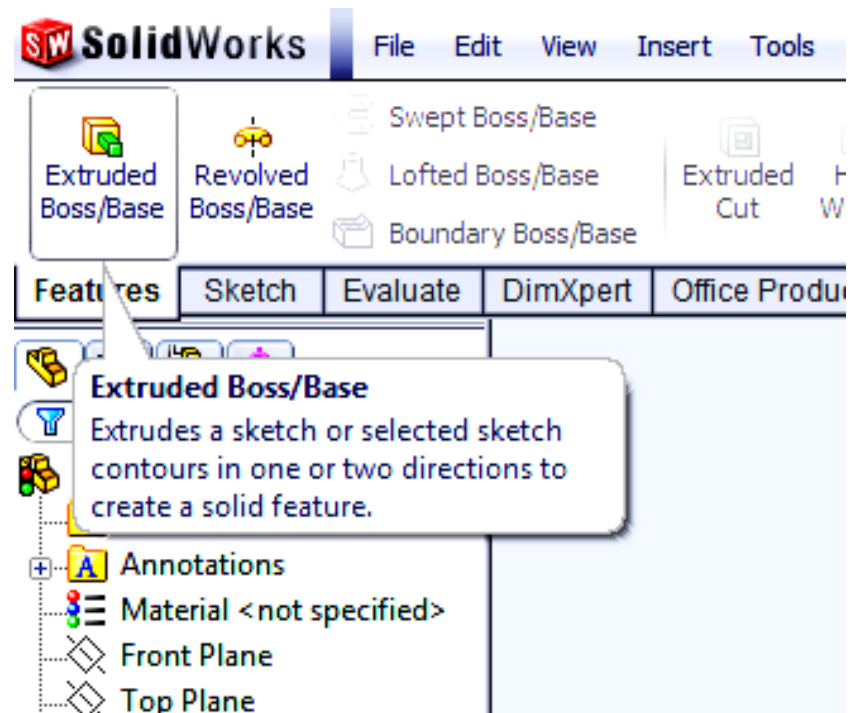
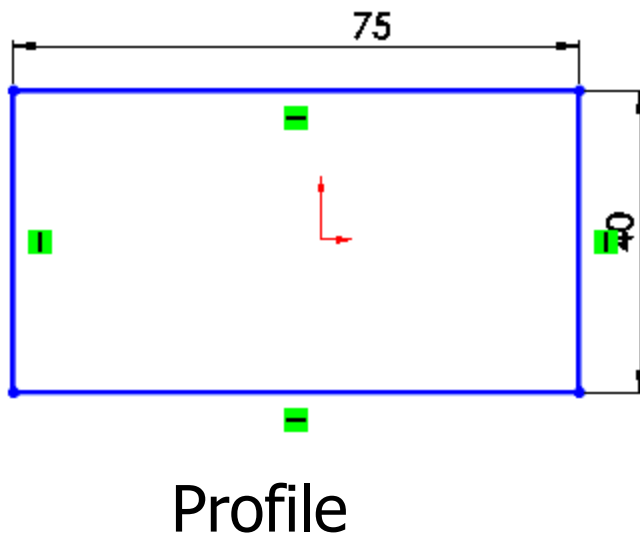


Profile sketch

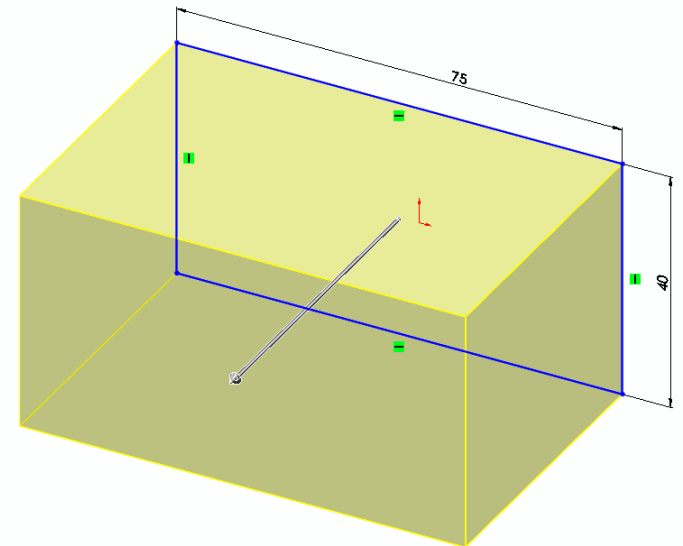
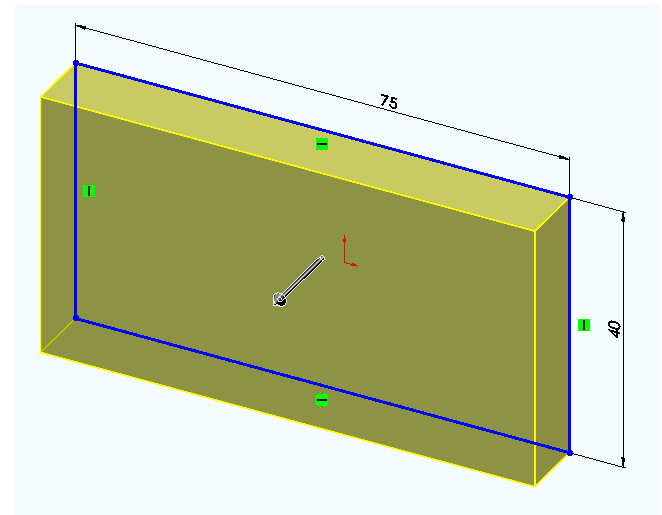
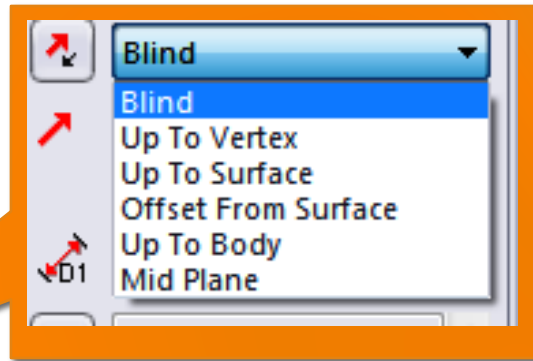
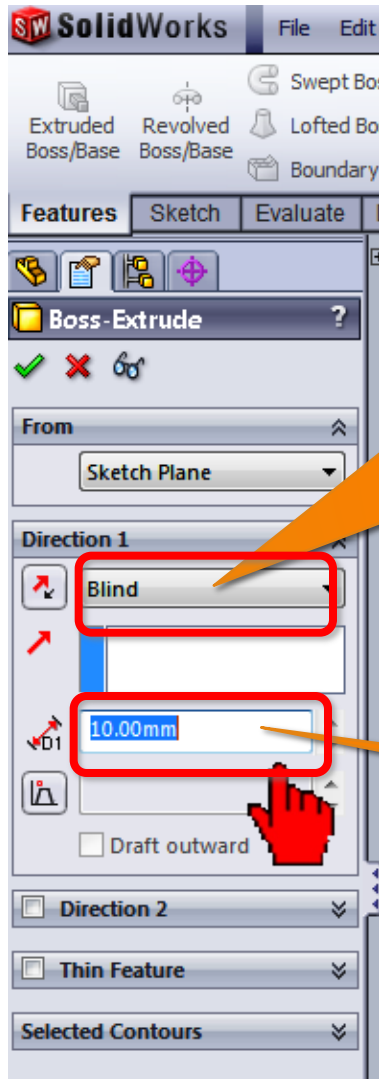


Extrude

Your wireframe curves may be a nice neat closed section with each curve starting at the end of the previous curve.



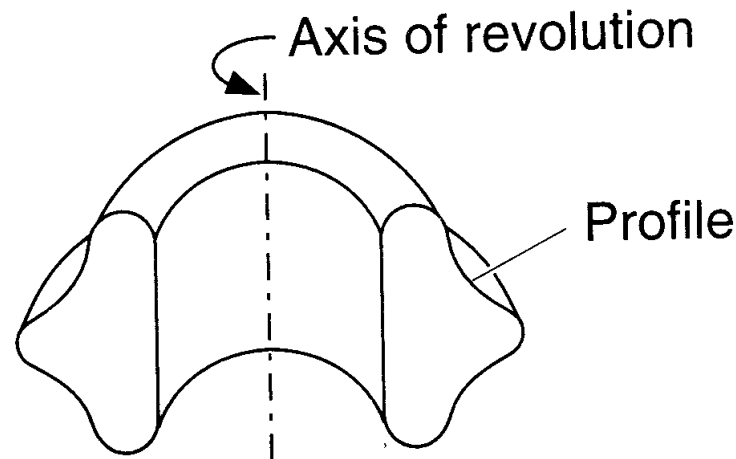
Extrude



Revolve

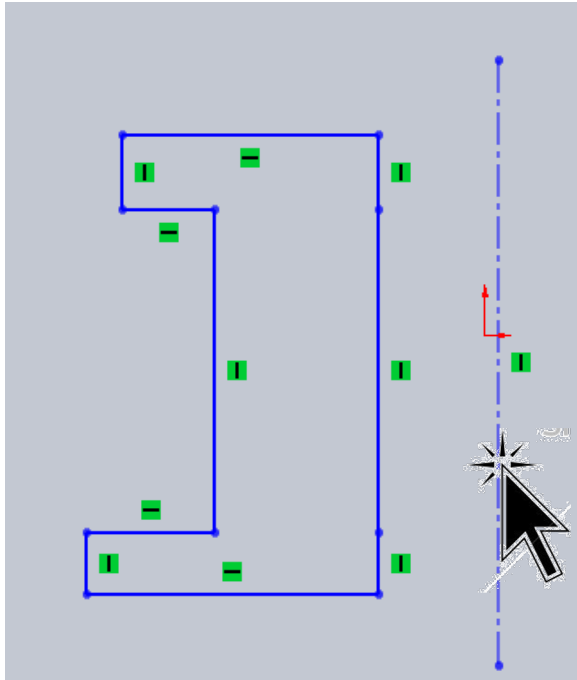
Revolution is another technique used to create a 3-D part from a two-dimensional shape.

With this technique, a given 2-D shape is revolved about an axis, sweeping a solid.

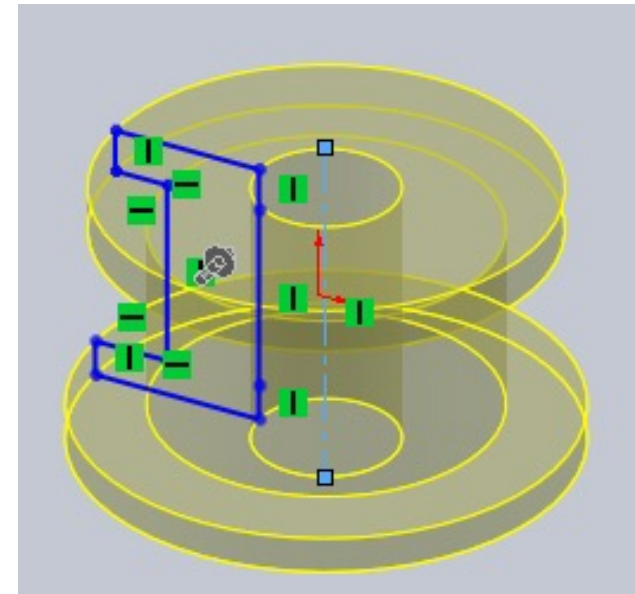
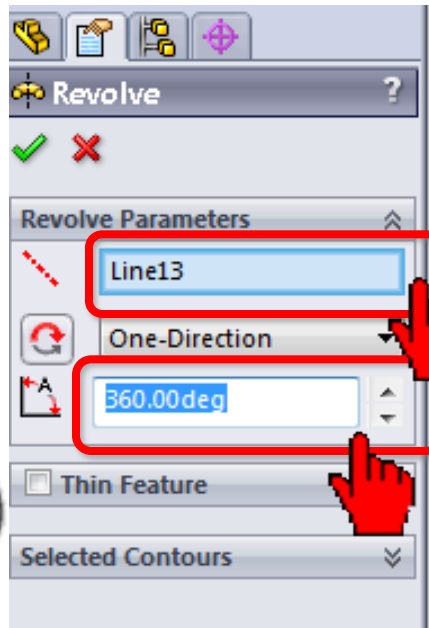
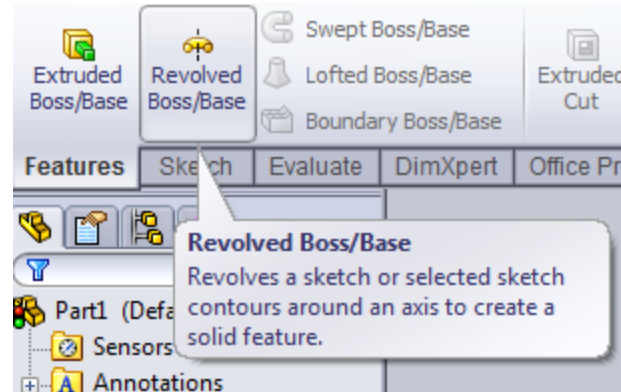


Revolution

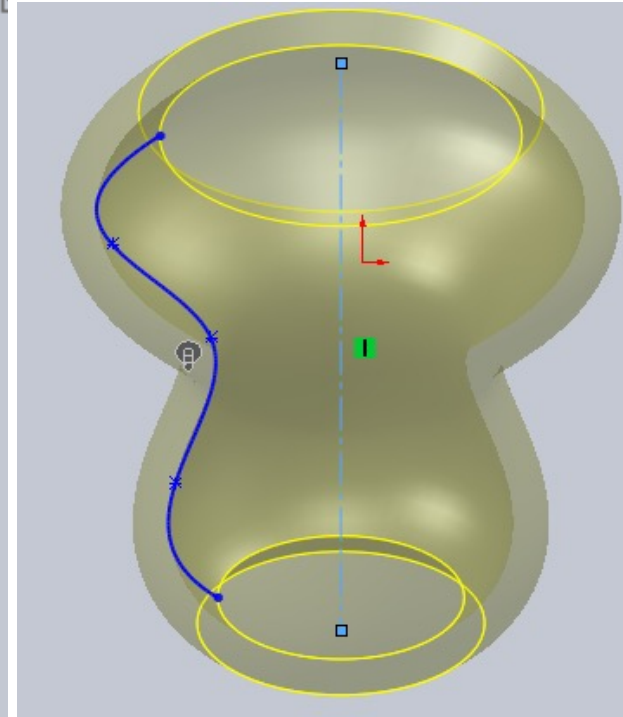
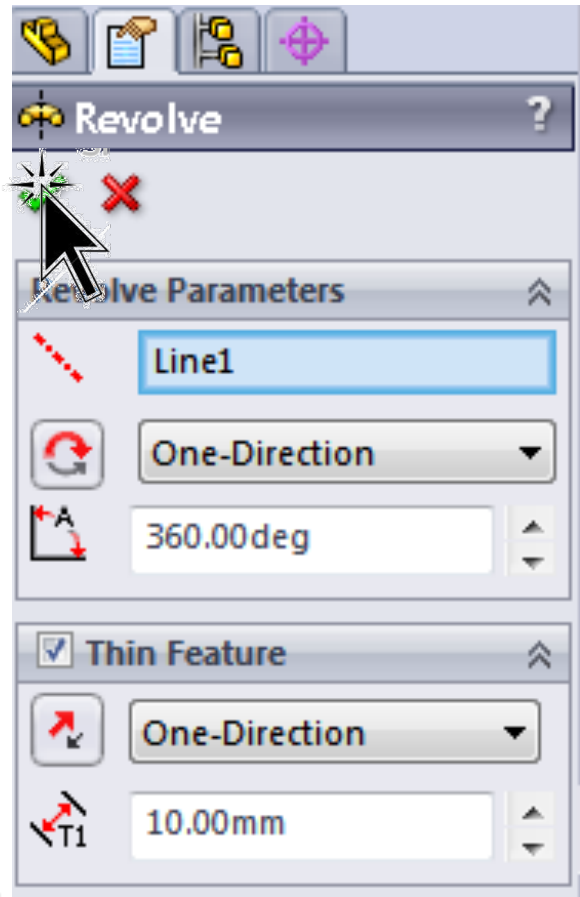
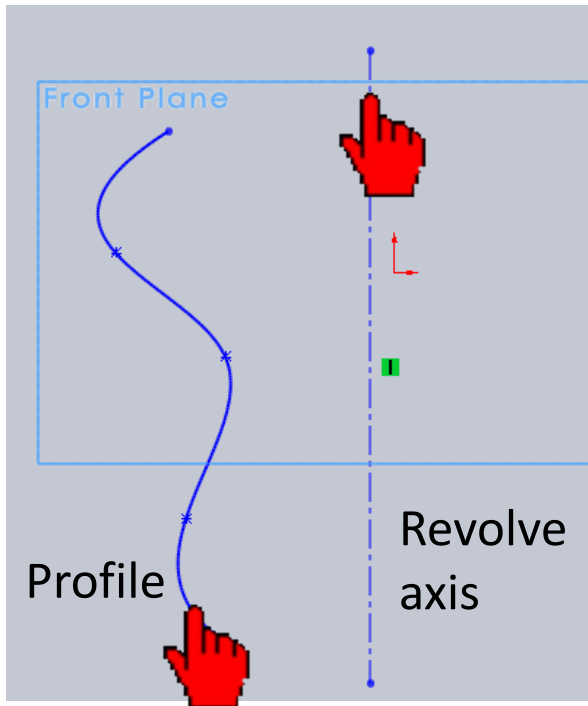
Revolve



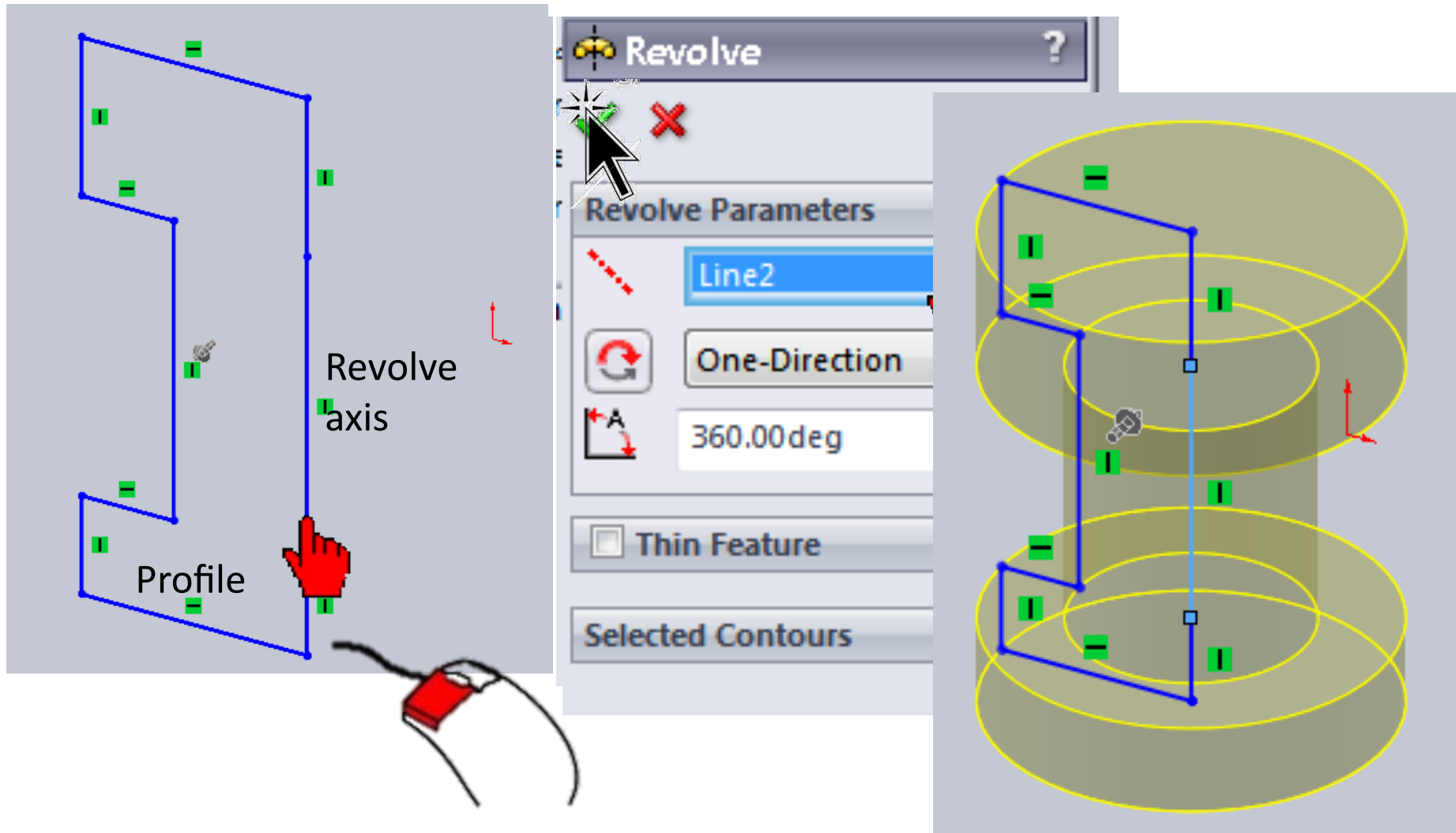
Profile



Example : Revolved Boss/Base

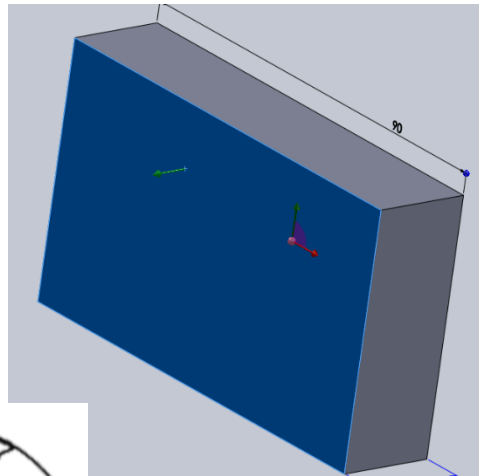


Example : Revolved Boss/Base

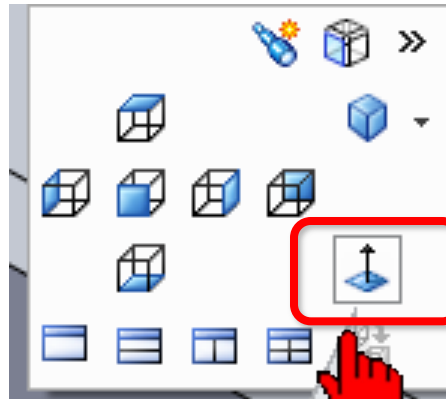


Extrude cut

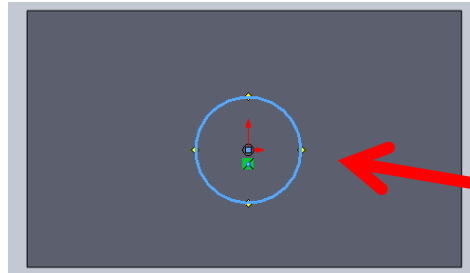
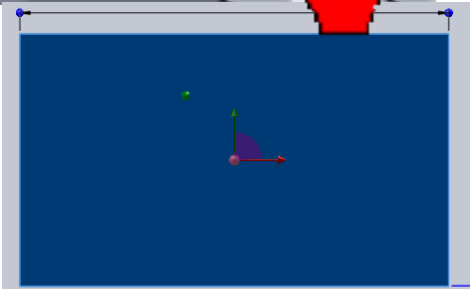
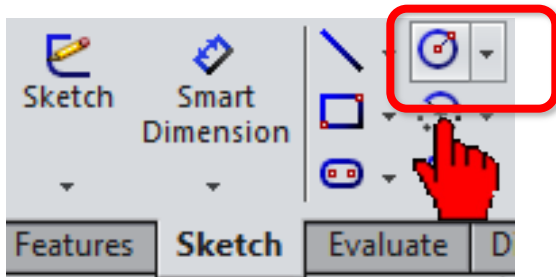
Select the face



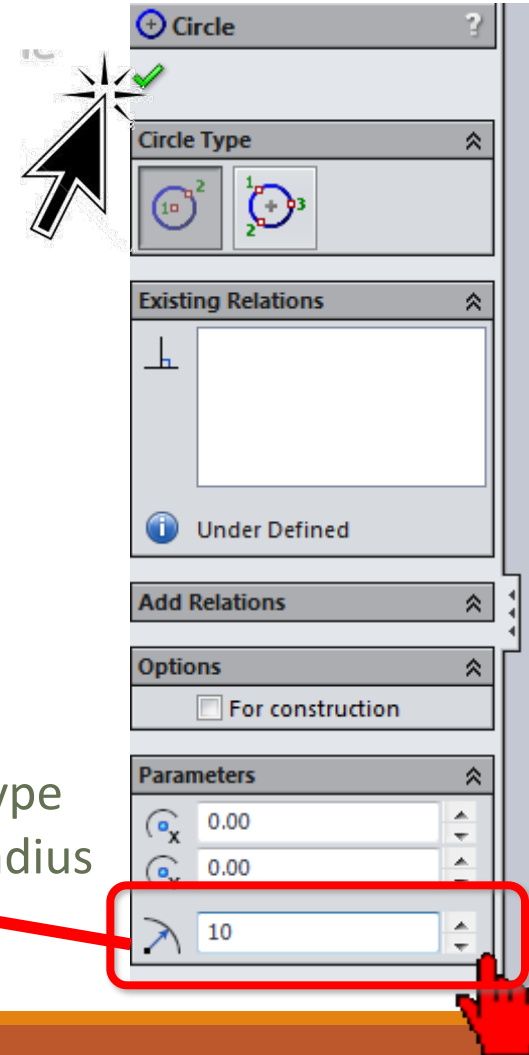
Click Normal to



Draw CIRCLE



Type radius



Extrude cut

The image shows a CAD software interface with the 'Extrude Cut' feature highlighted in the ribbon. The 'Cut-Extrude' task pane is open, showing the 'From' property set to 'Sketch Plane' and 'Direction 1' set to 'Blind' with a depth of '10.00mm'. A mouse cursor is shown clicking on the 'Blind' dropdown menu, which is open to show options like 'Through All', 'Through All - Both', etc. Two orange callout boxes are present: one labeled 'Hole type' pointing to the dropdown menu, and another labeled 'Hole depth' pointing to the '10.00mm' input field. A mouse icon is also visible at the bottom right.

Extruded Cut

Hole type

- Blind
- Through All
- Through All - Both
- Up To Next
- Up To Vertex
- Up To Surface
- Offset From Surface
- Up To Body
- Mid Plane

Hole depth

10.00mm

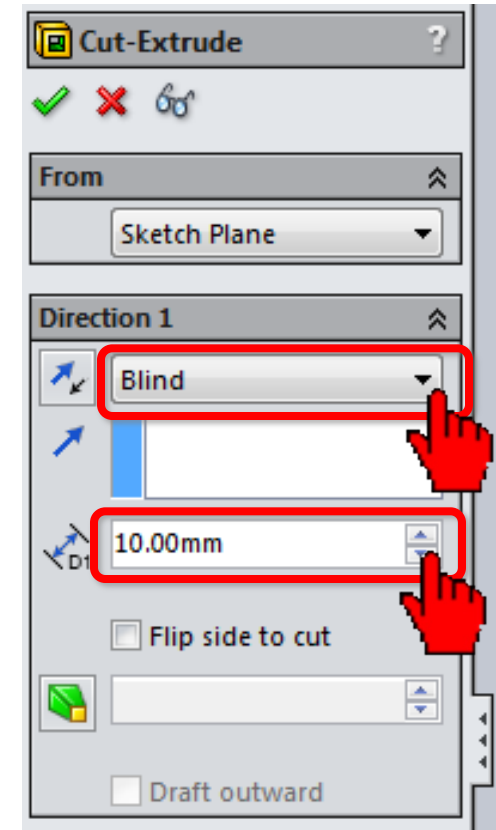
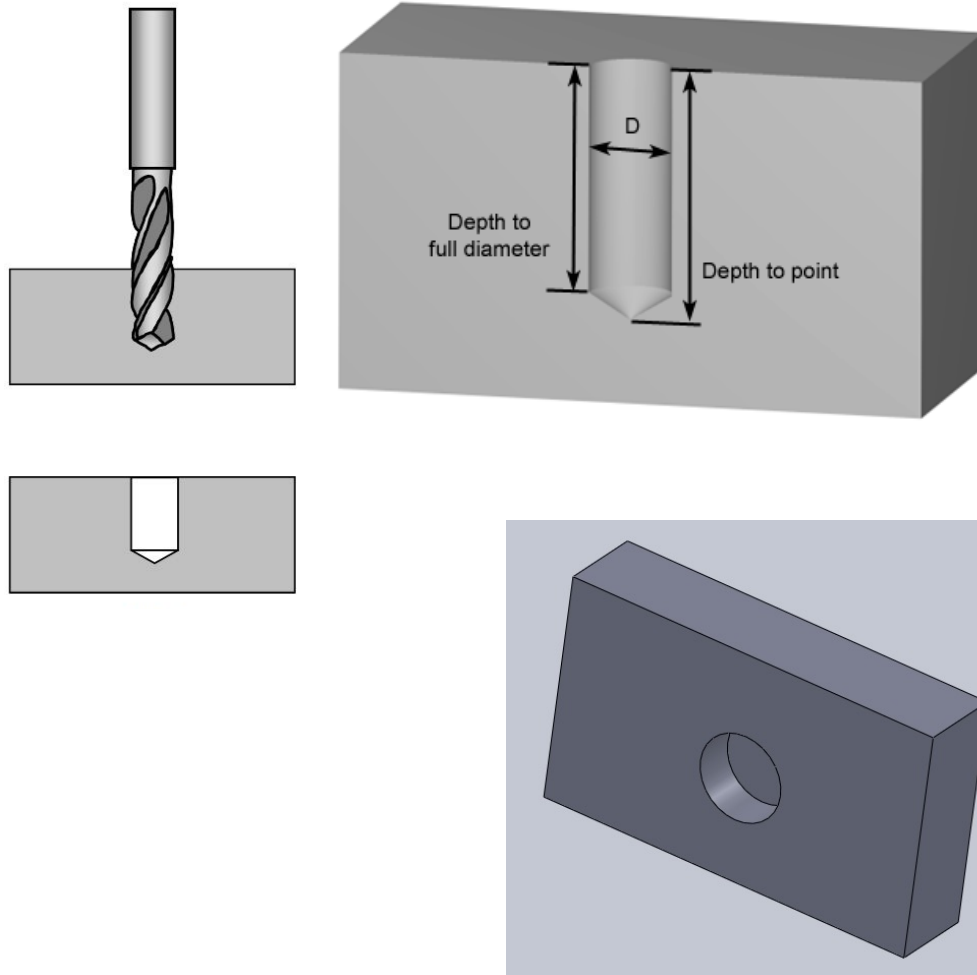
Blind

Flip side to cut

Draft outward

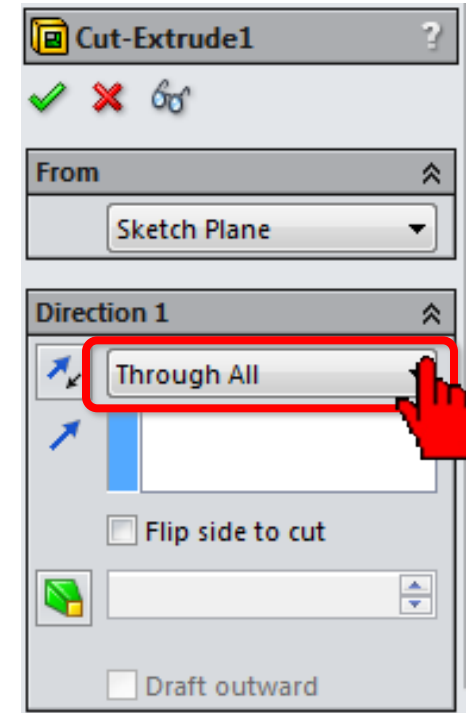
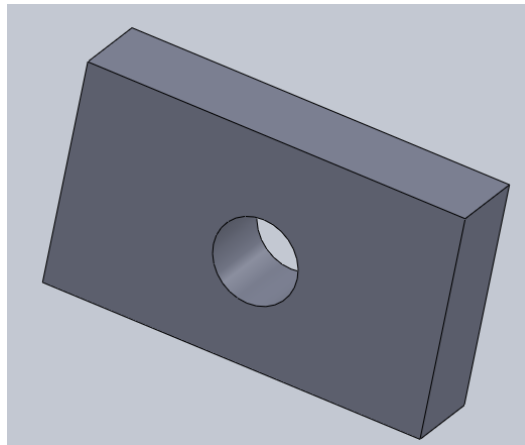
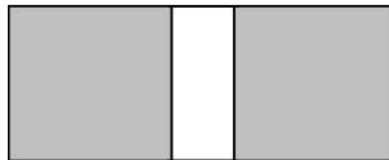
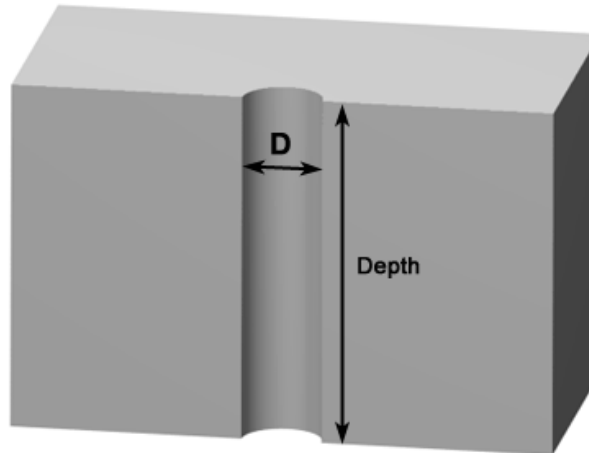
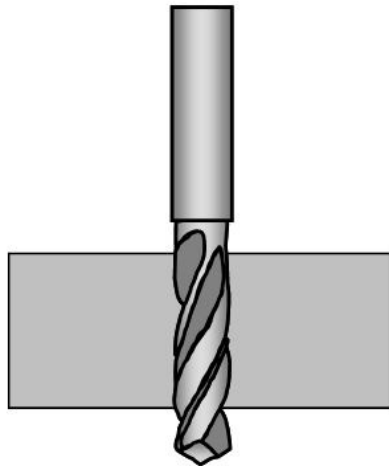
Extrude cut

Blind hole



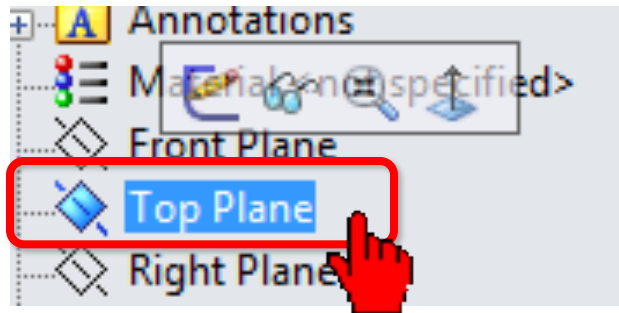
Extrude cut

Through hole

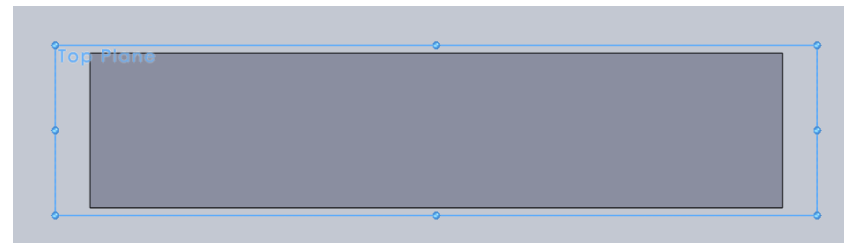
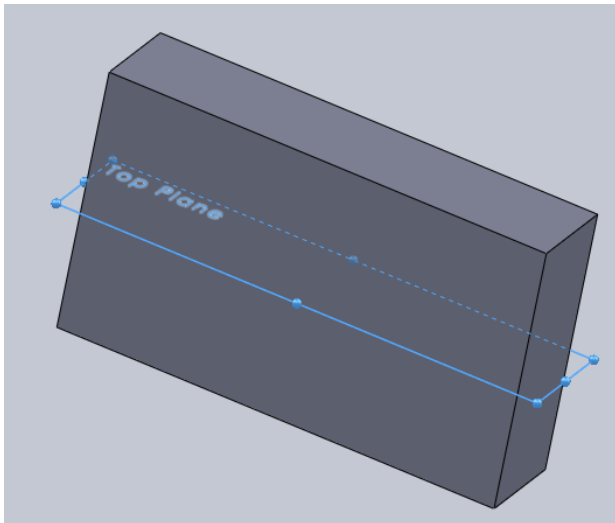
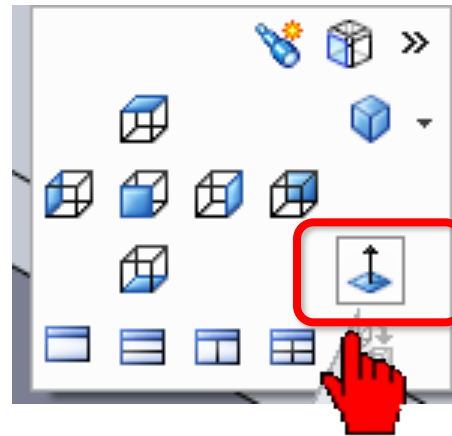


Revolve cut

Select working plane

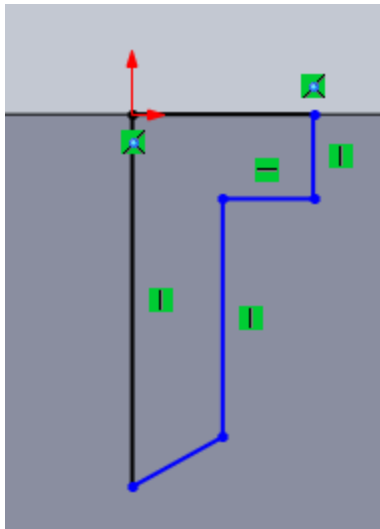
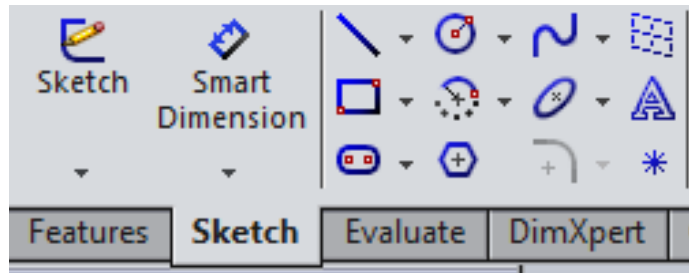


Click Normal to



Revolve cut

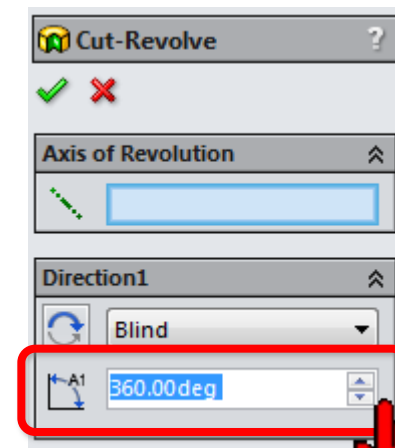
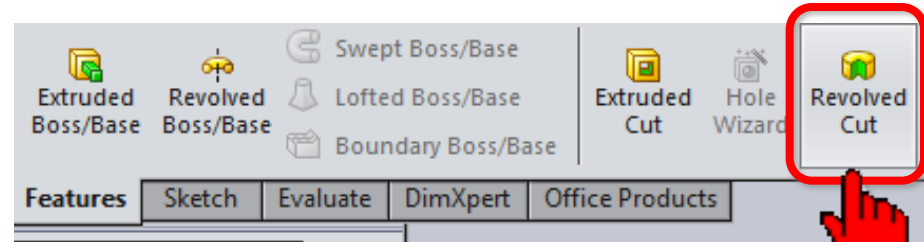
Draw hole profile



Sketch using drawing commands half view of the hole

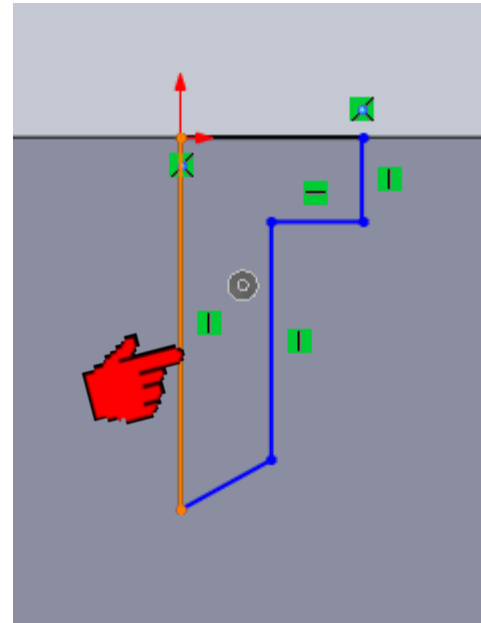
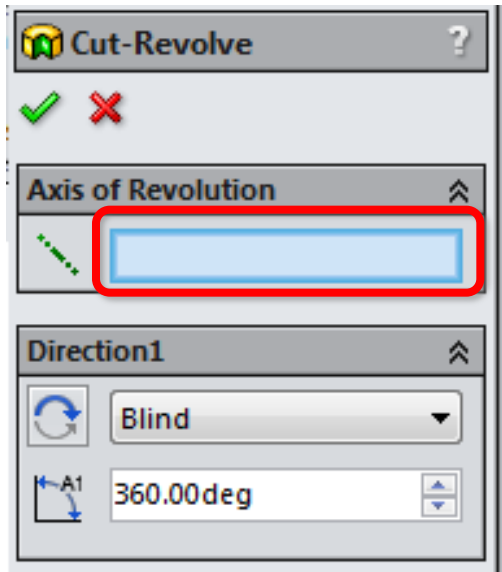
Make closed counter

Command : REVOLVED CUT

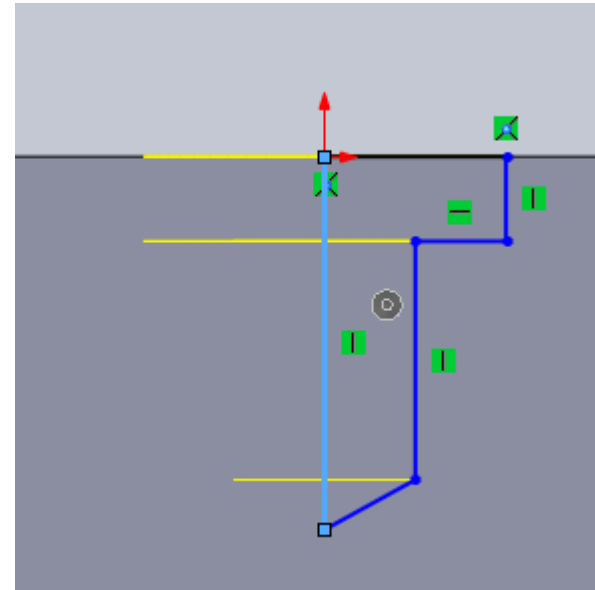
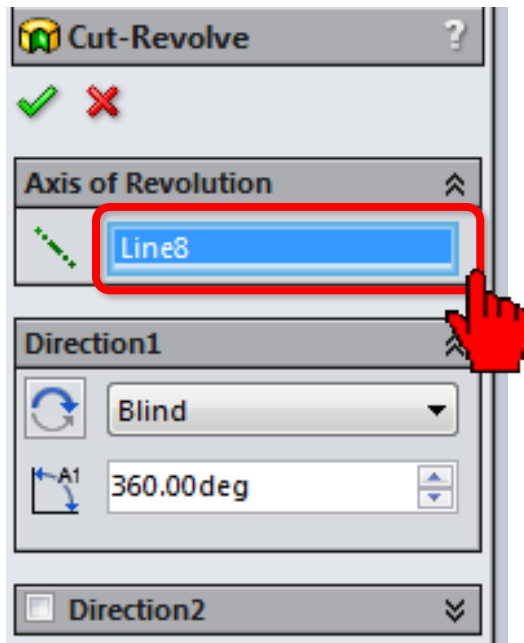


Revolve cut

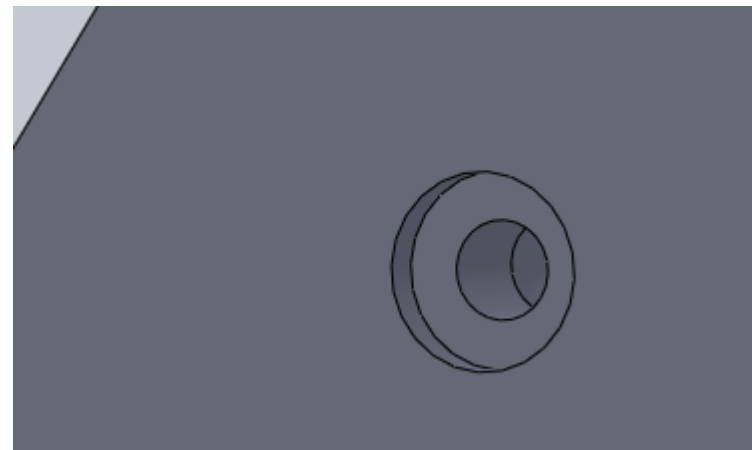
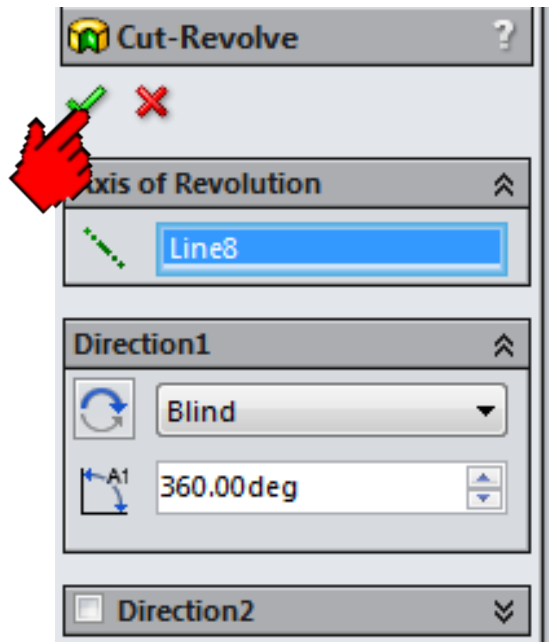
Select the axis of revolve



Revolve cut

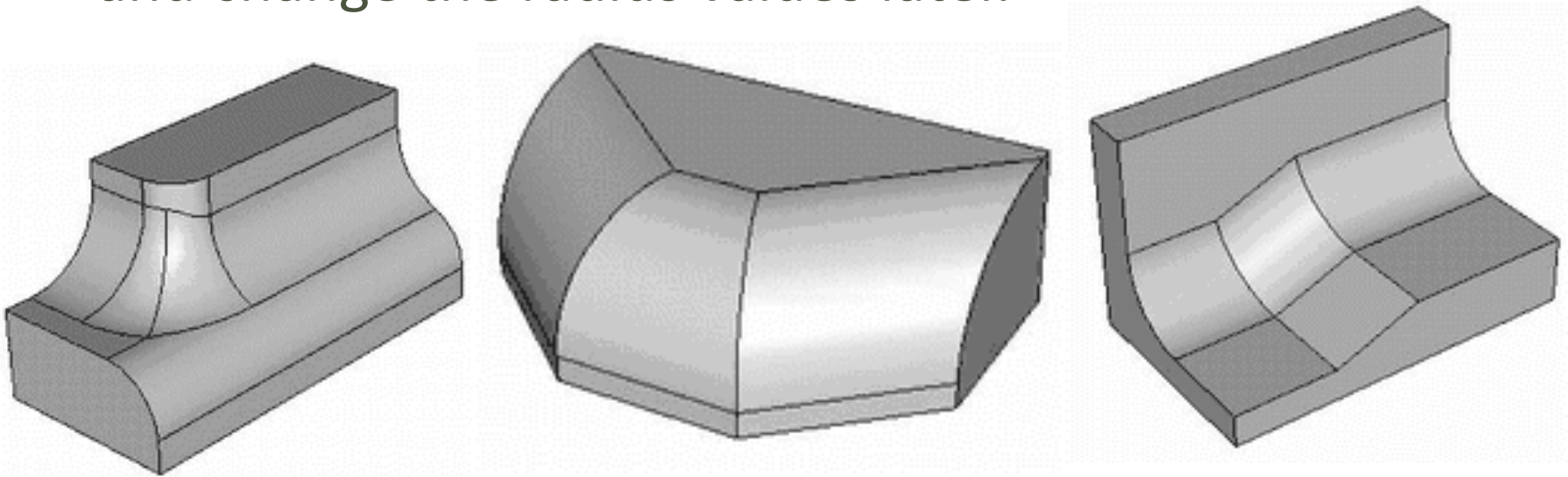


Revolve cut

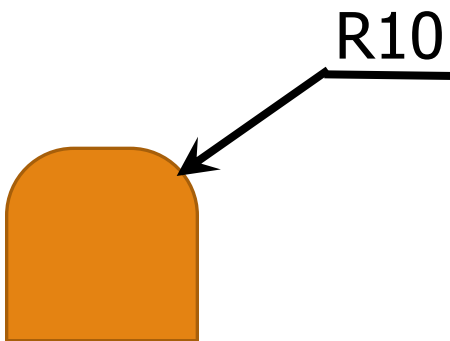
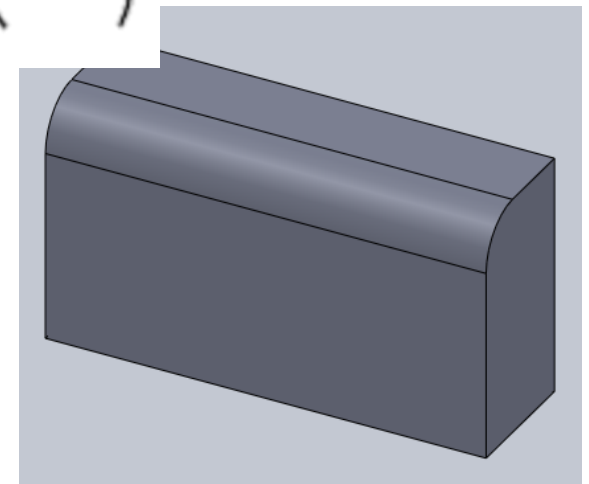
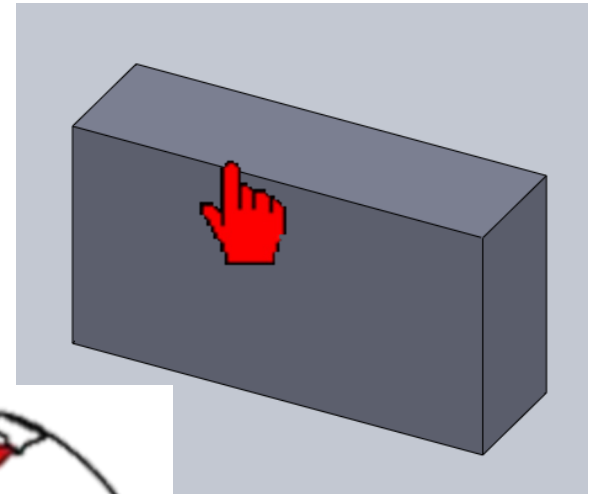
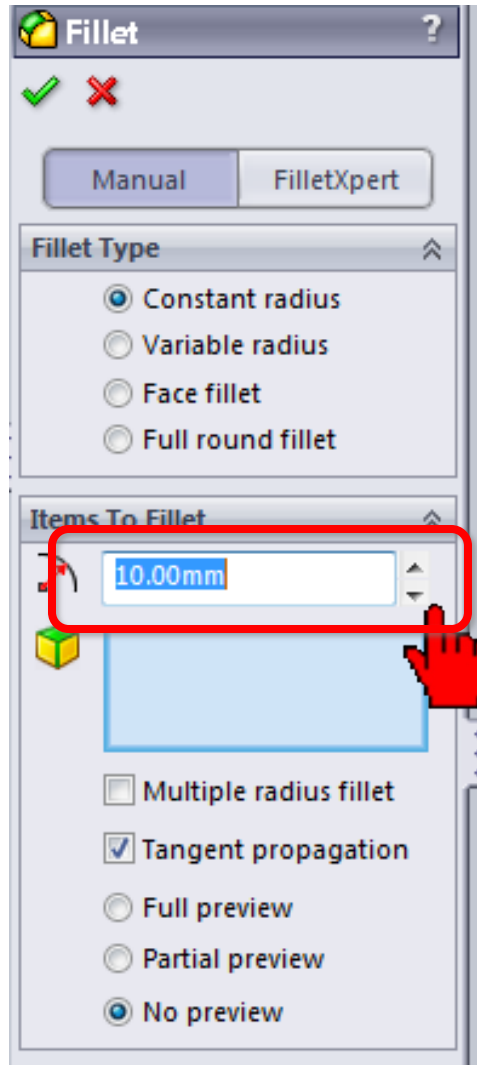
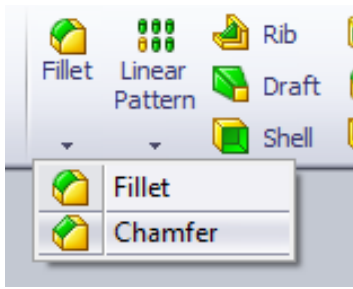


Fillet & Chamfer

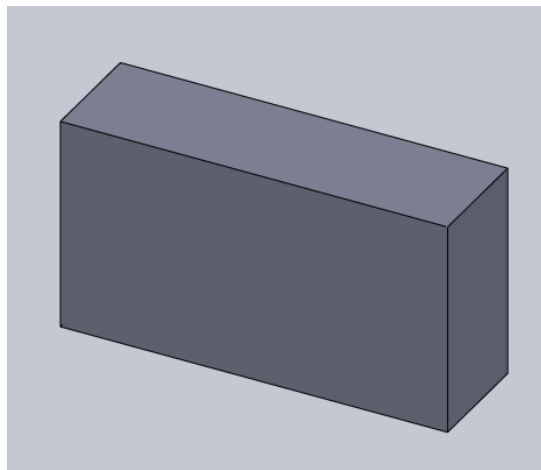
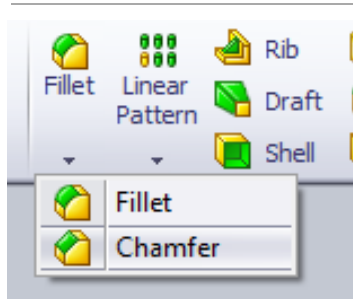
The *Fillet* and *Chamfer* commands work with part edges. To fillet edges, either select edges directly, or select associated vertices or surfaces to indirectly select edges to fillet. These operations are kept as part of the part history, so you will be able to go back and change the radius values later.



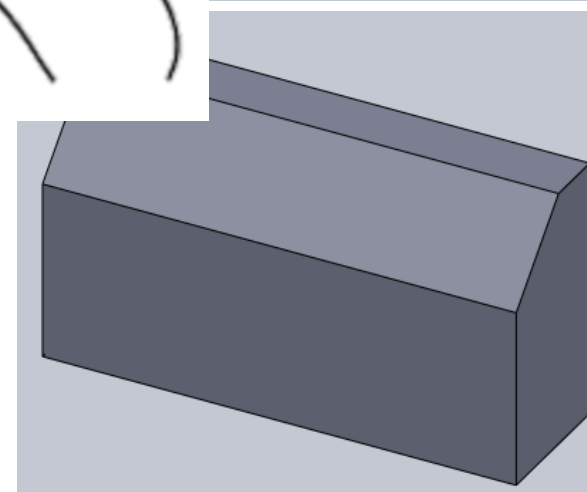
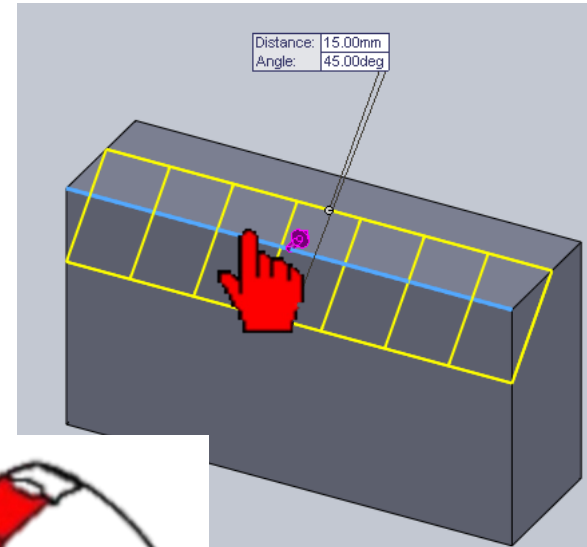
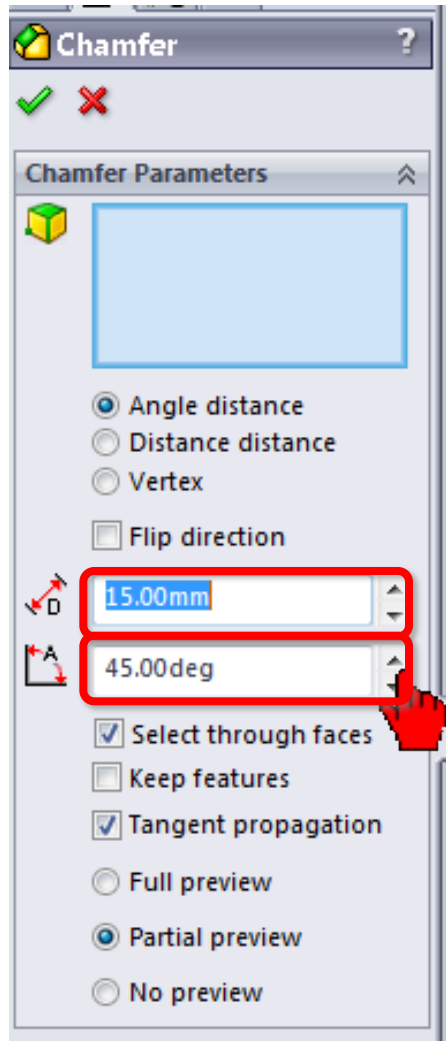
Fillet



Chamfer



15 x 45°



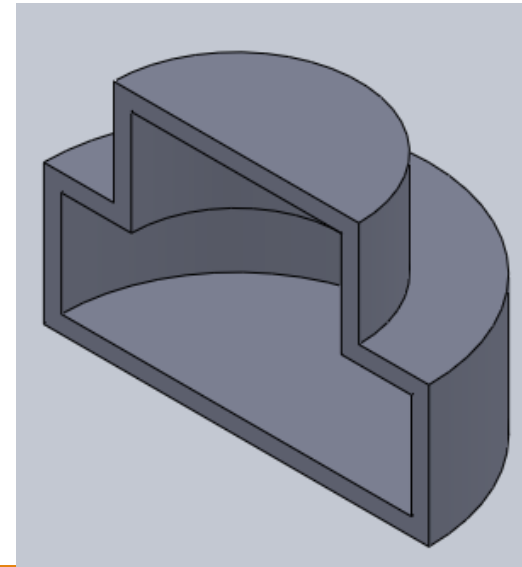
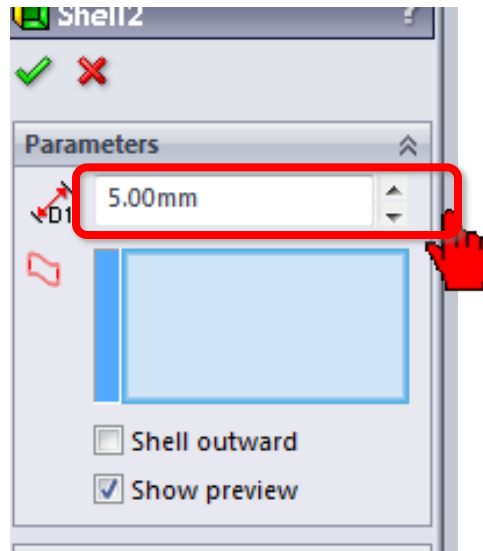
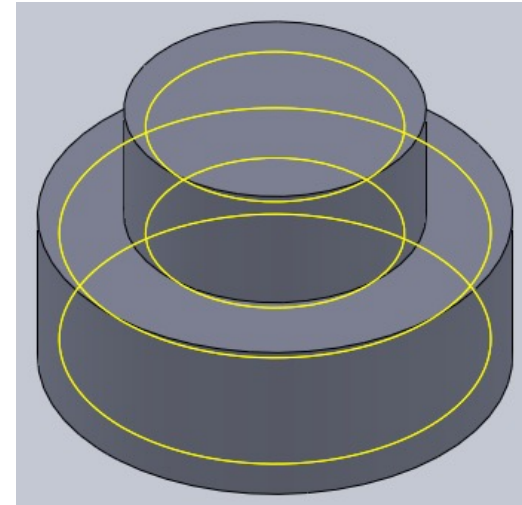
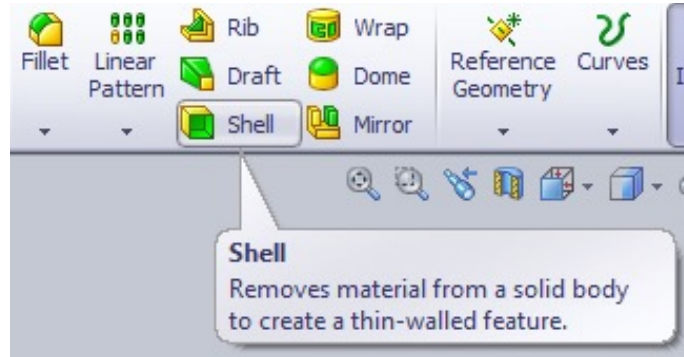
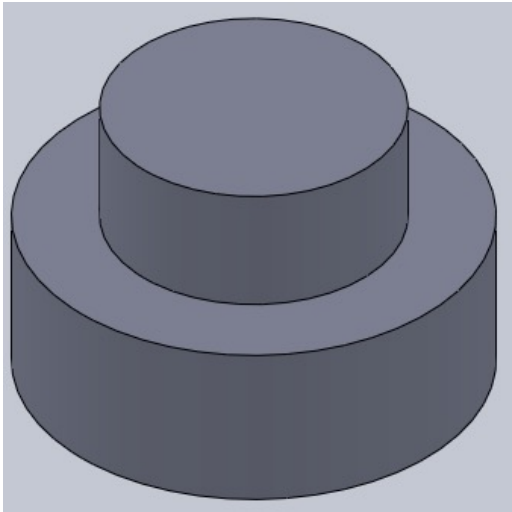
Shell

The Shell command takes surfaces and gives them thickness.

You can use this command both to start with an open surface model and create a solid, or to start with a complete solid and thicken selected surfaces, deleting others.



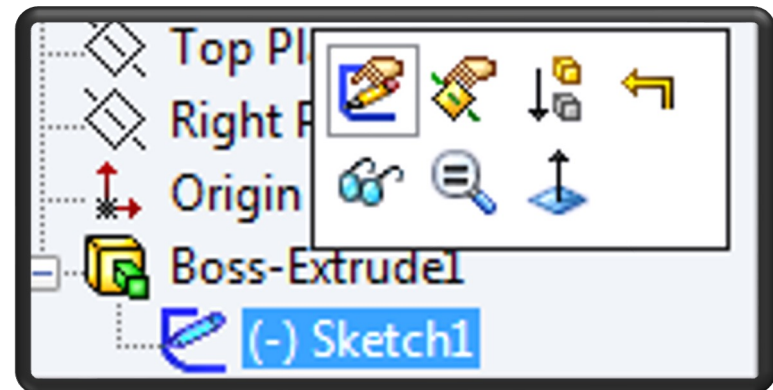
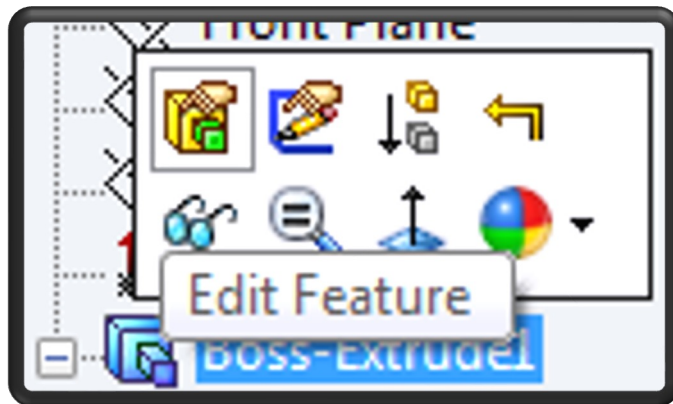
Shell



Modifying features

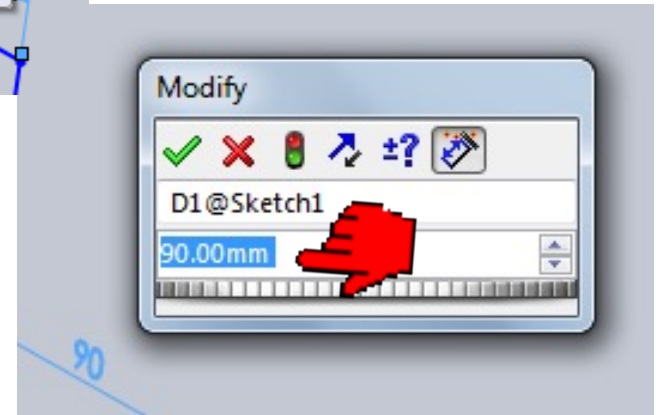
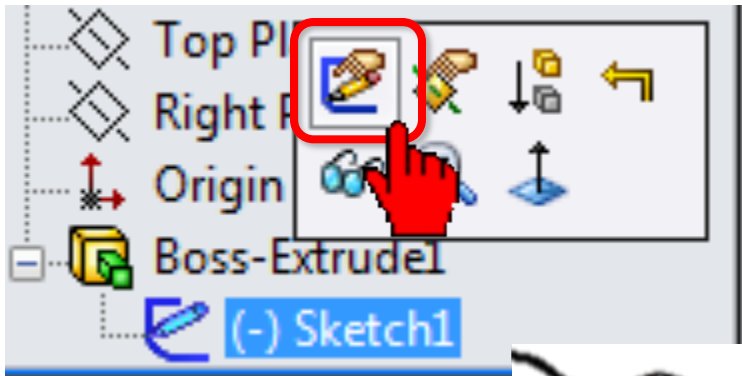
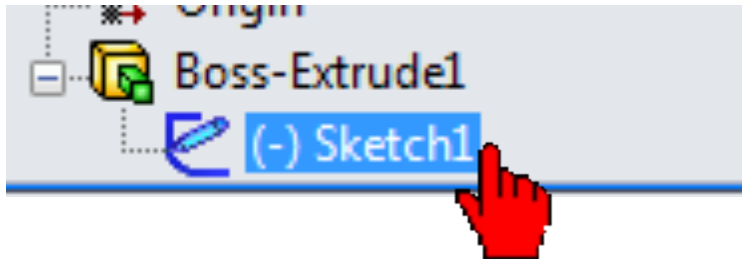
Parametric dimensions are dimensions that are used to define a feature. They can be modified at any time.

Modifying a dimension value is the most common dimension modification function, but other modification tools do exist.



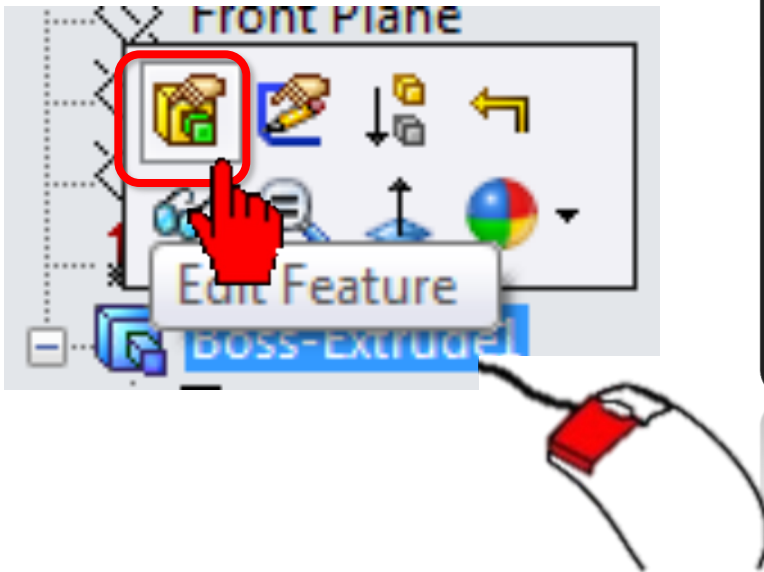
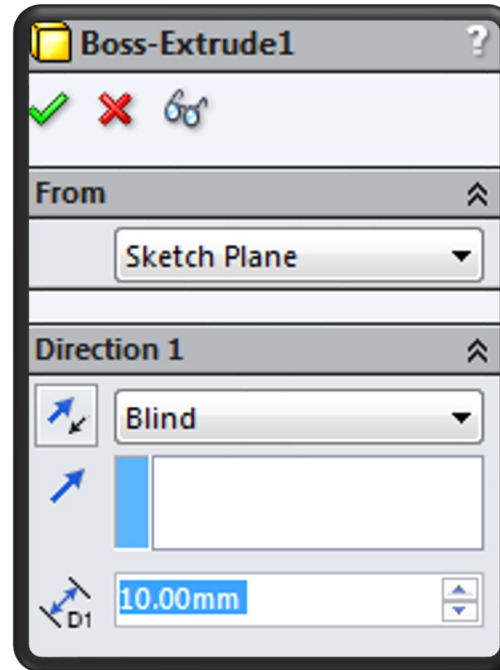
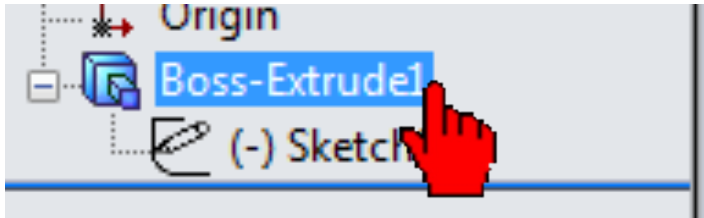
Edit features

Sketch edit



Edit features

Feature edit



The following week



MIDTERM EXAM #1

The following week ...

DON'T FORGET



To complete and submit
the
Progress Report #1

<http://transport.itu.edu.tr/dersler/lisansdersleri/mak110e/desingproject>

Assignment #5

Page

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Figure

14.53

You will generate the solid model

Submit the assignment on time

Upload file into NINOVA

