Benha Faculty of Engineering
Mechanical Engineering Department

M1382 : **Computer Aided Design CAD**
First Semester 2018, Y3

Lecture No. 03

Presented by:
Mahmoud Magdy

Week	Topics
1	Introduction
2	Introduction to CAD (Solid Modeling)
3	**Part modeling**
4	Finite element analysis (FEA)
5	Parts assembly using SolidWorks
6	Basic concepts of engineering drafting
7	Linear Static Analysis
8	Adaptive Analysis and Mesh Control
9	Modal Analysis
10	Design Optimization
11	Case study 1
12	Case study 2
13	Co-simulation SolidWorks and ADMS software
14	Project Discussion

Basics and Modeling Fundamentals

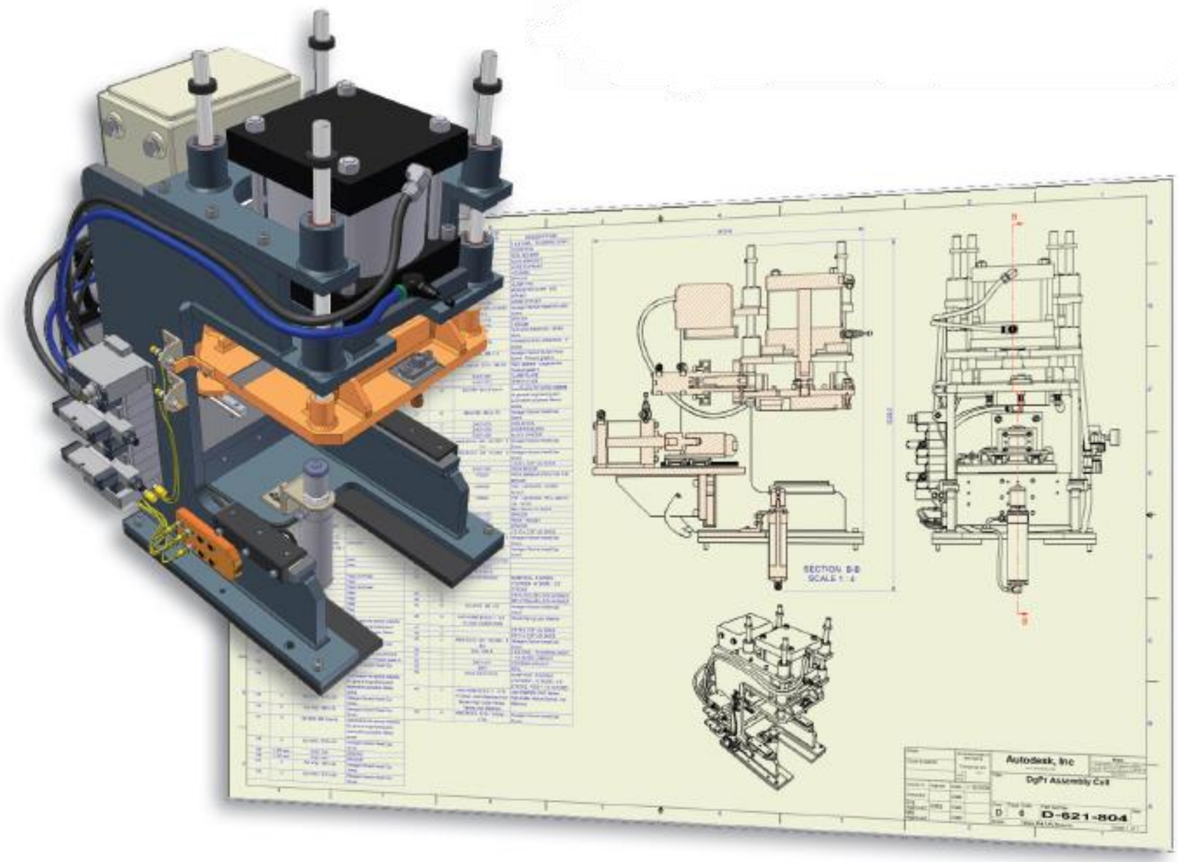
Ref : SolidWorks Teacher Guide

SolidWorks

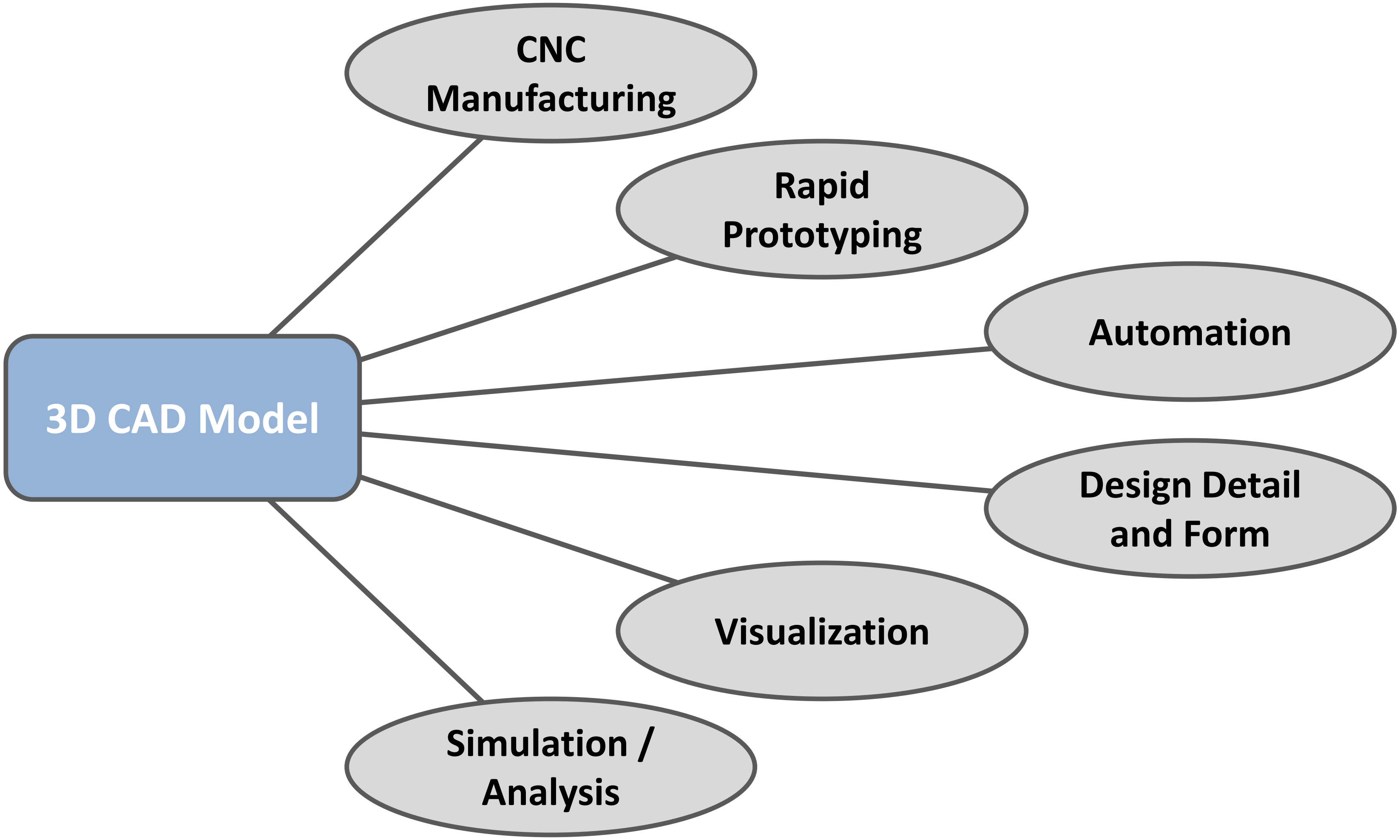
- SolidWorks is a 3D solid modeling package which allows users to develop full solid models in a simulated environment for both design and analysis.

What is 3D Modeling

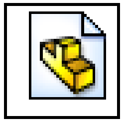
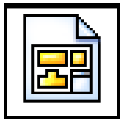
In SolidWorks, you sketch ideas and experiment with different designs to create **3D models** of the **real physical object**.



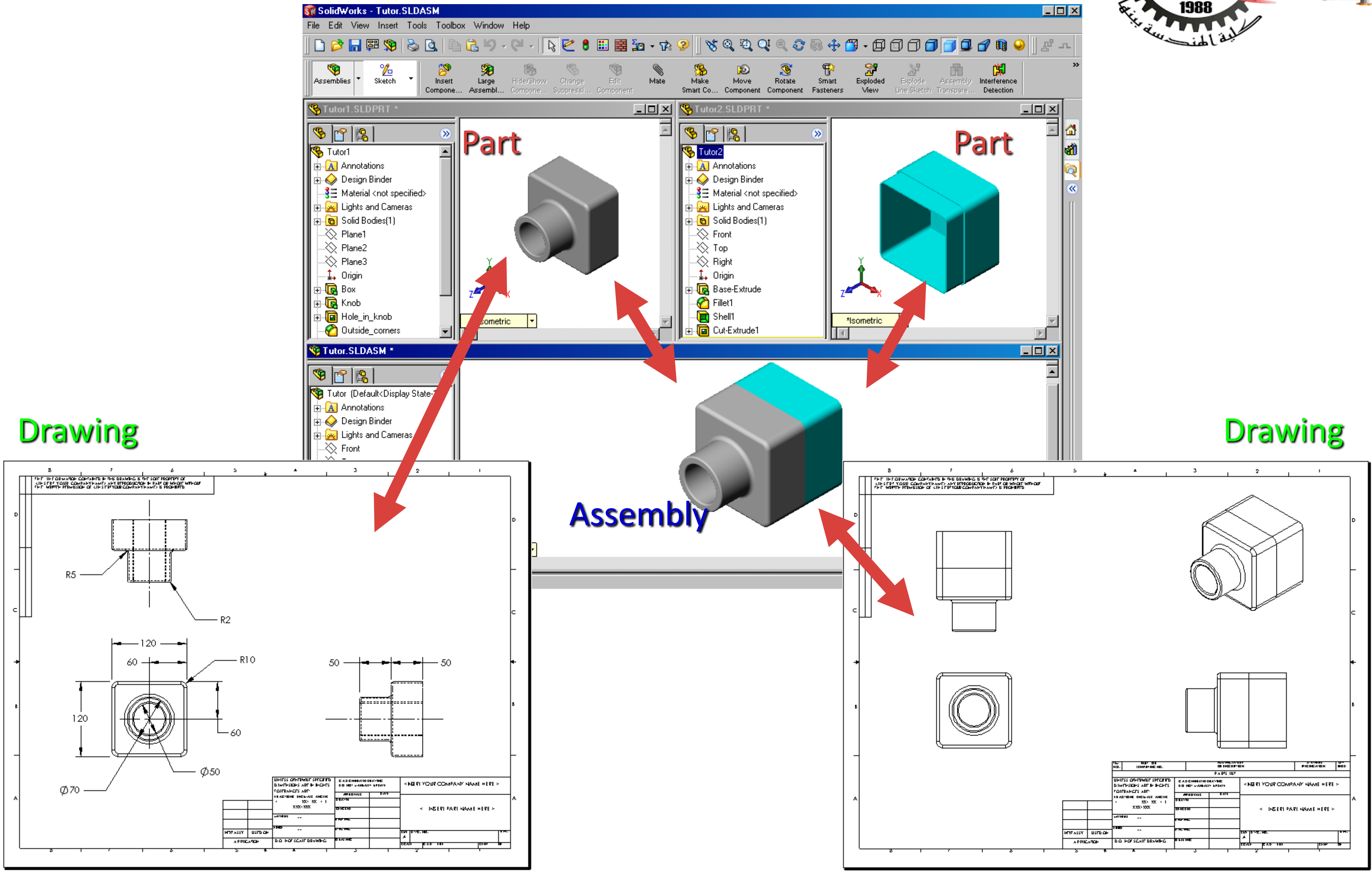
3D Design Use



The SolidWorks Model

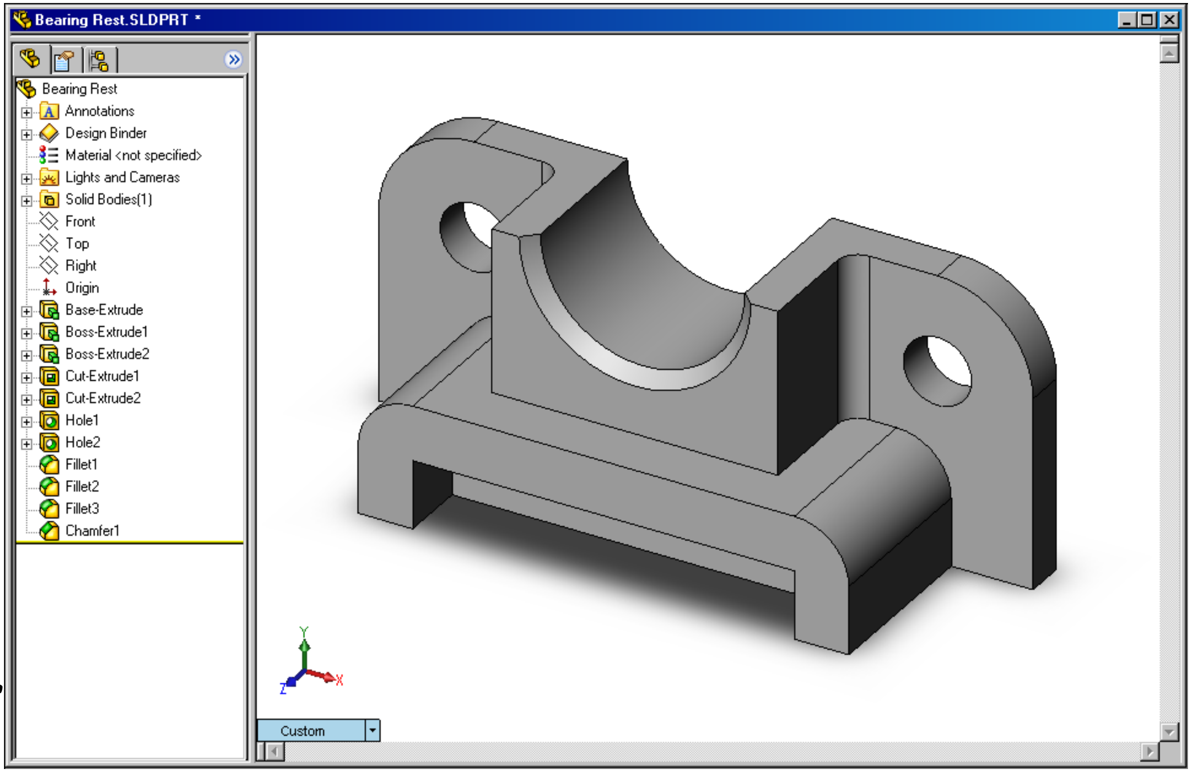
- **The SolidWorks model is made up of:**
  - **Parts** 
  - **Assemblies** 
  - **Drawings** 

The SolidWorks Model



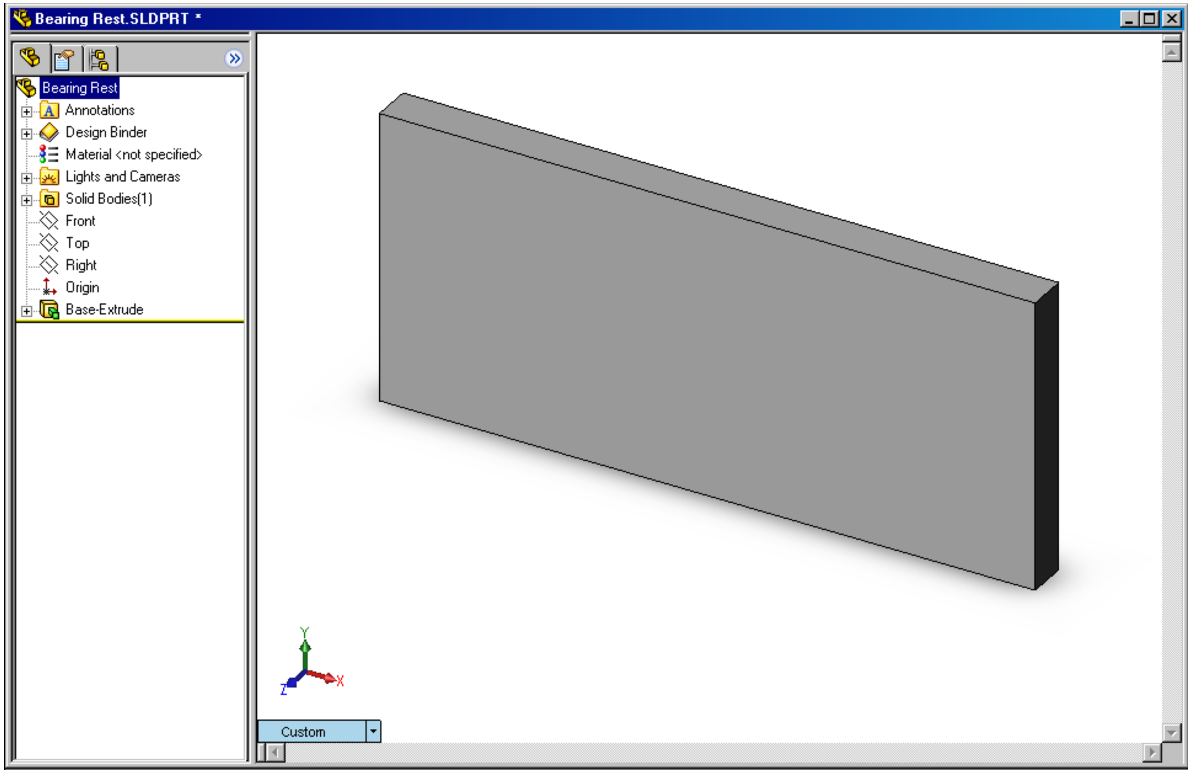
Features

- **Features are the building blocks of the part.**
- **Features are the *shapes* and *operations* that construct the part.**



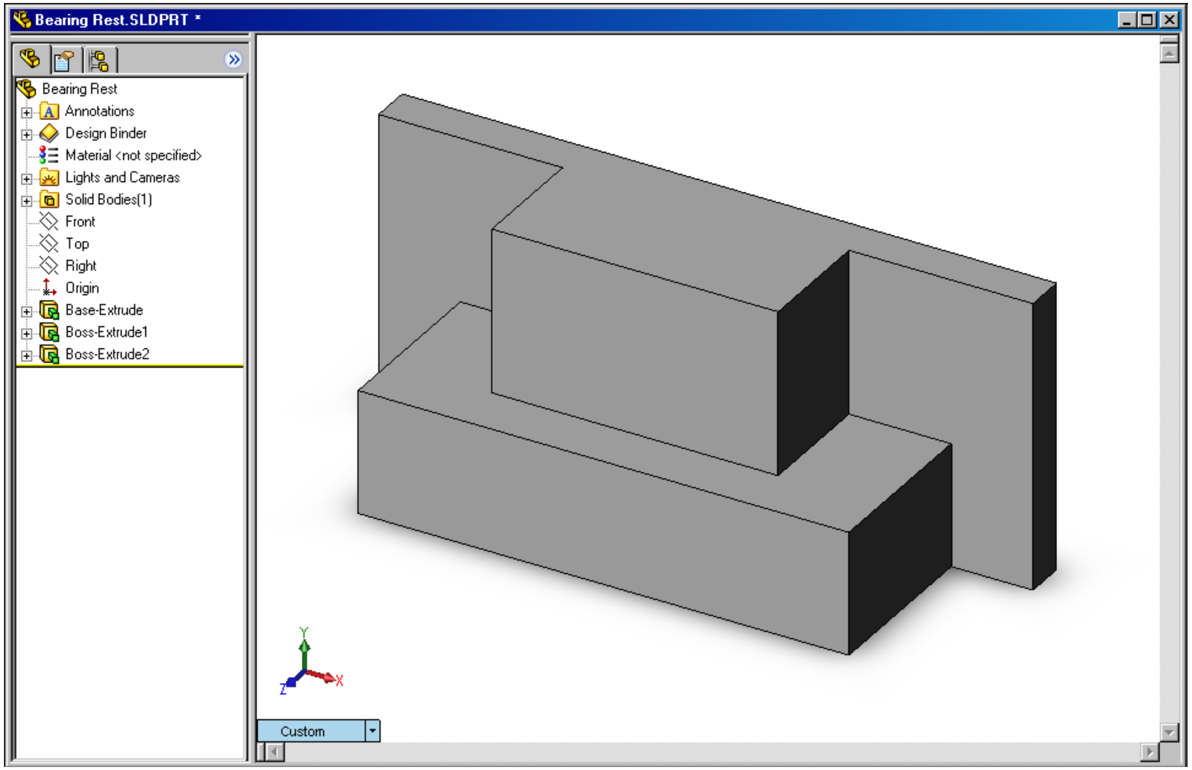
Examples of Shape Features

- **Base Feature**
  - First feature in part.
  - Created from a 2D sketch.
  - Forms the work piece to which other features are added.



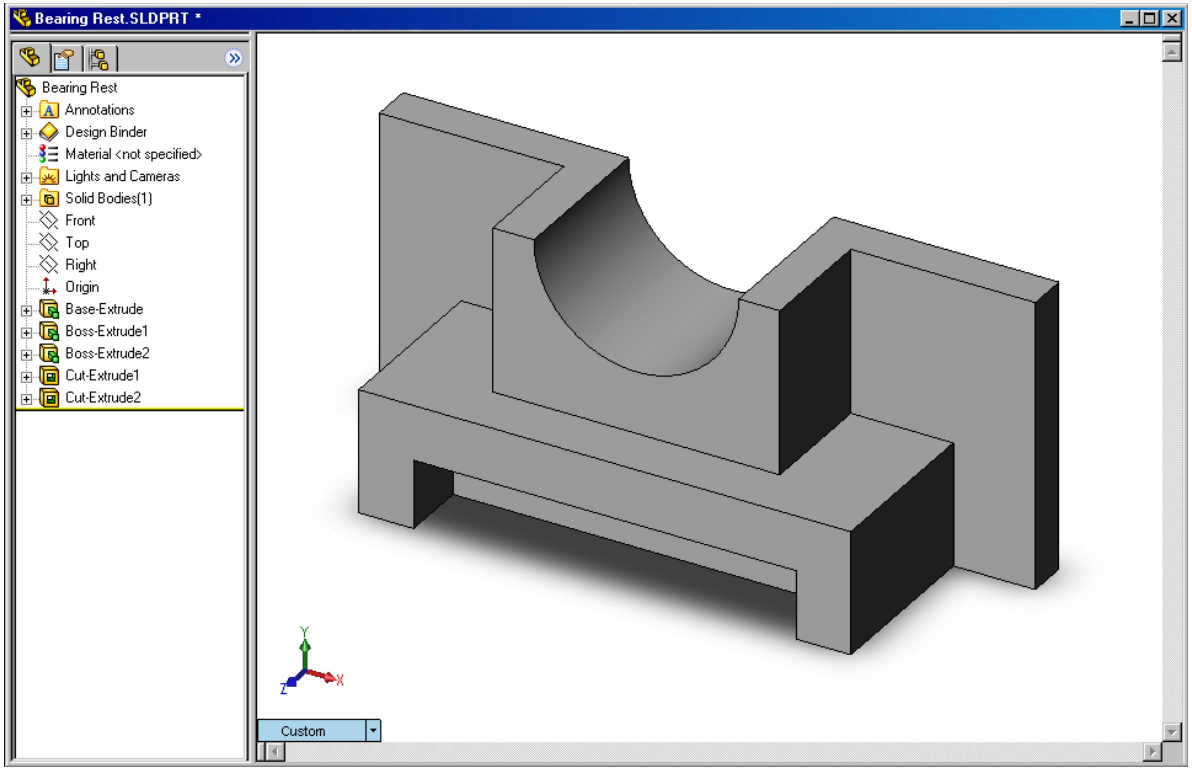
Examples of Shape Features

- **Boss feature**
  - **Adds material to part.**
  - **Created from 2D sketch.**



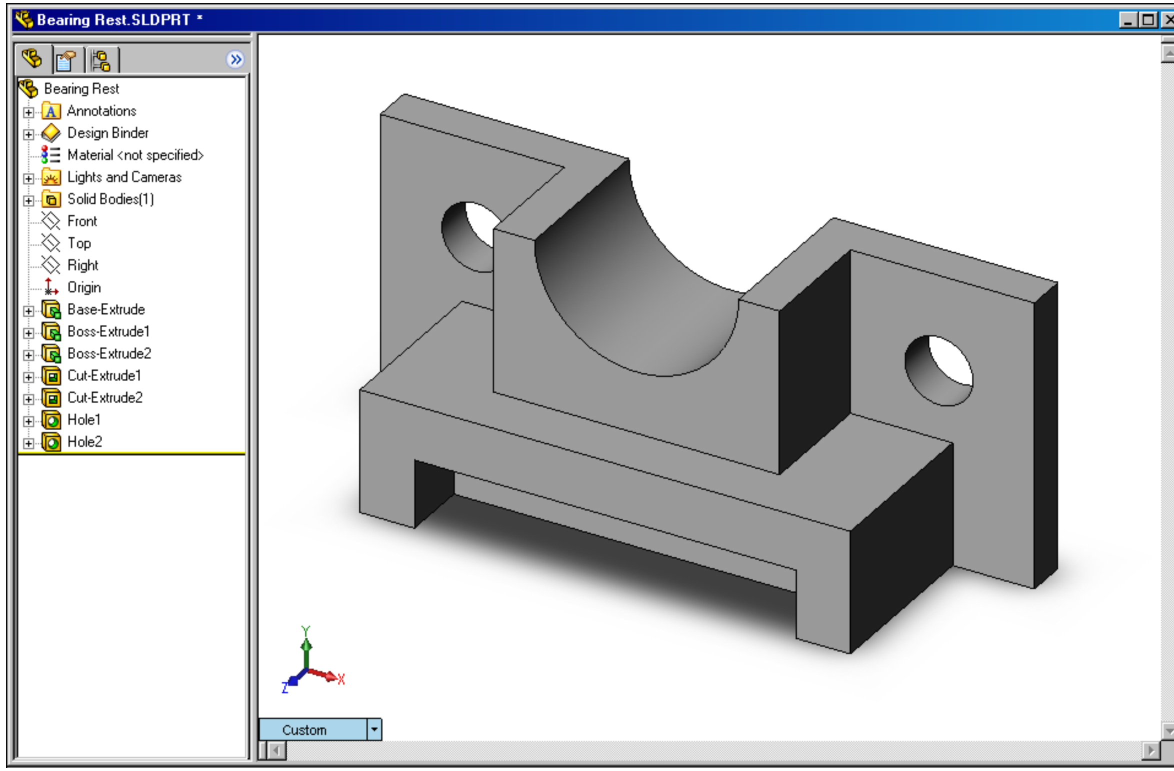
Examples of Shape Features

- **Cut feature**
  - **Removes material from part.**
  - **Created from 2D sketch.**



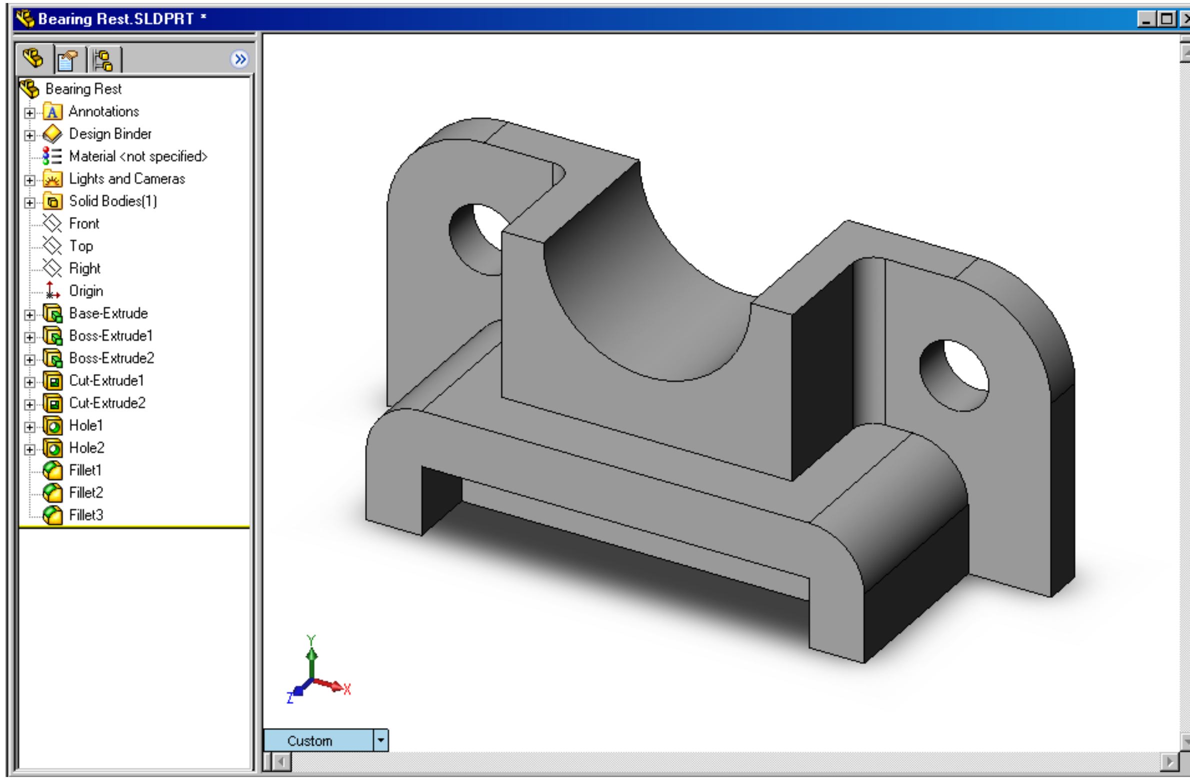
Examples of Shape Features

- **Hole feature**
  - Removes material.
  - Works like more intelligent cut feature.
  - Corresponds to process such as counter-sink, thread, counter-bore.

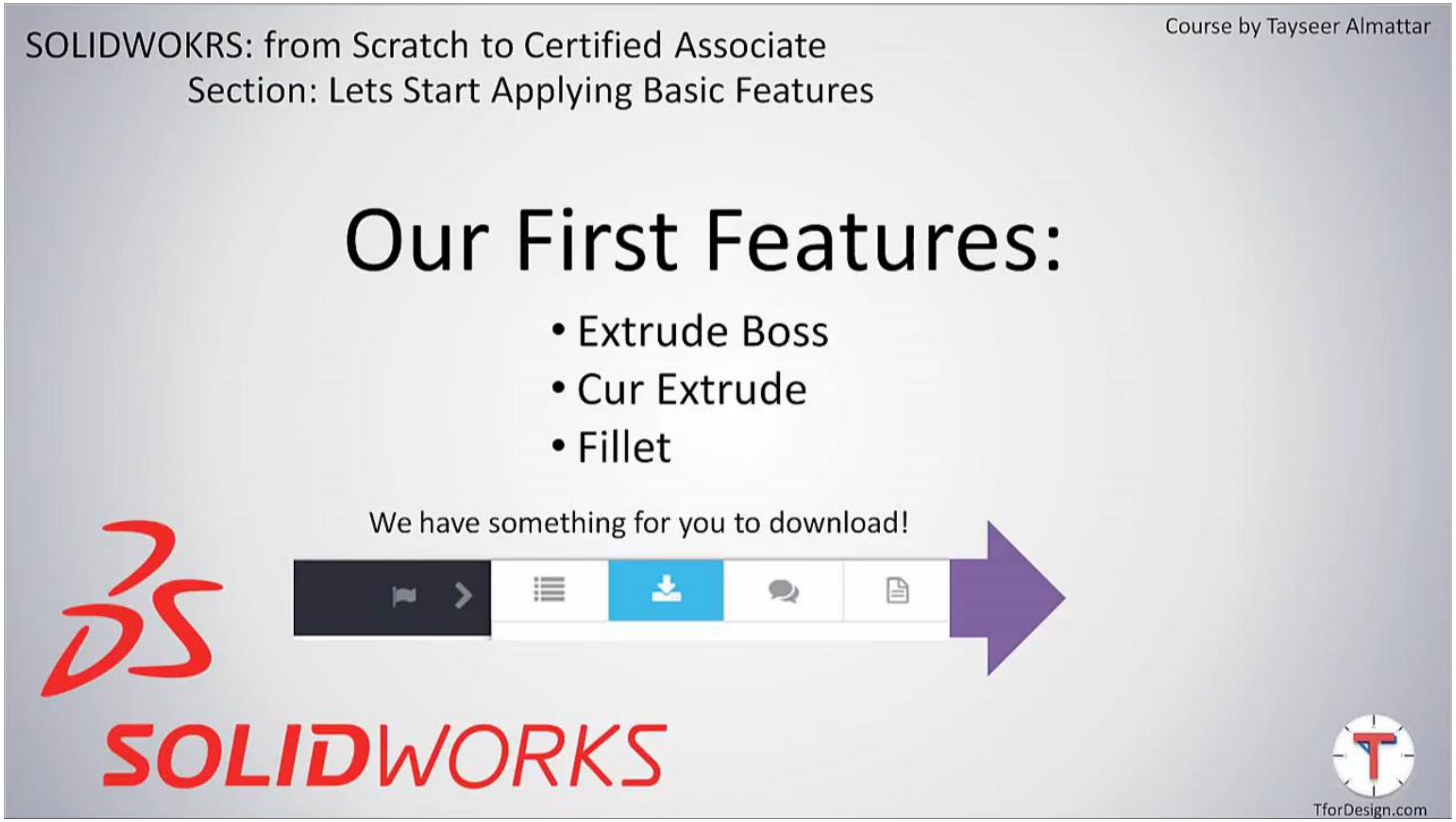


Examples of Shape Features

- **Fillet feature**
  - Used to round off sharp edges.
  - Can remove or add material.
    - Outside edge (convex fillet) removes material.
    - Inside edge (concave fillet) adds material.



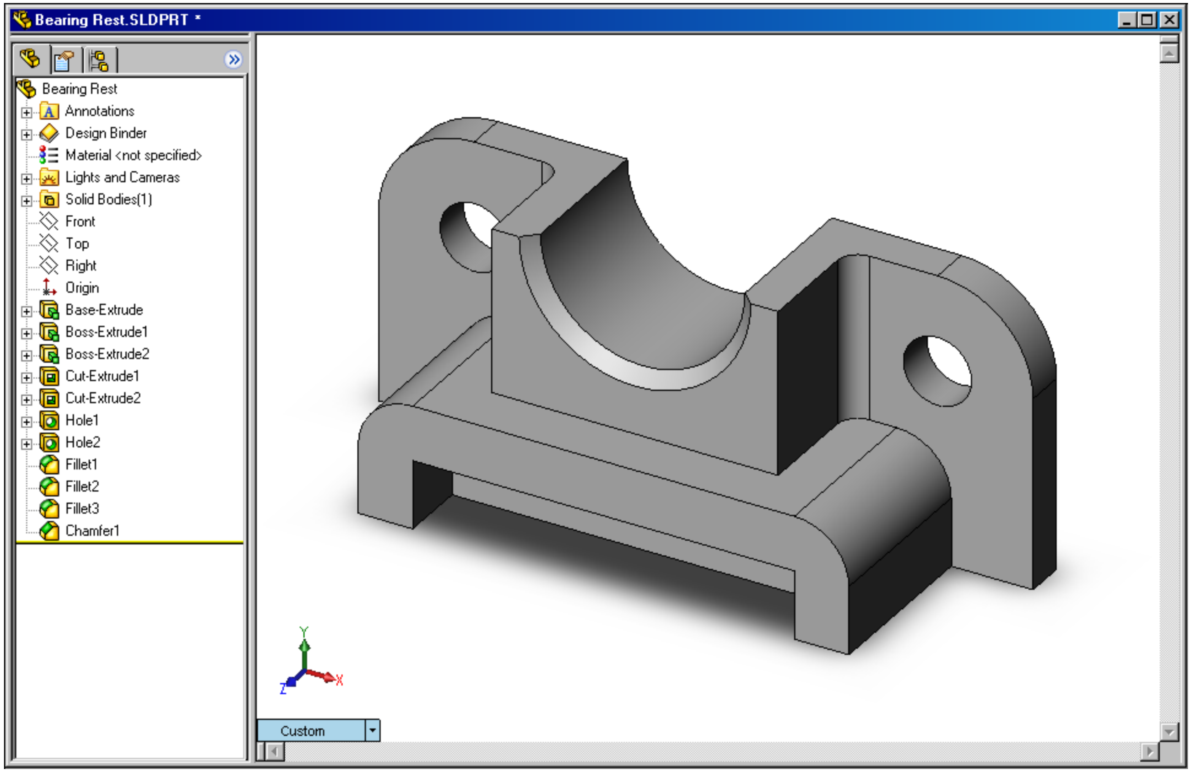
Examples of Shape Features



https://www.youtube.com/watch?v=sAy2xObj8CM&t=160s

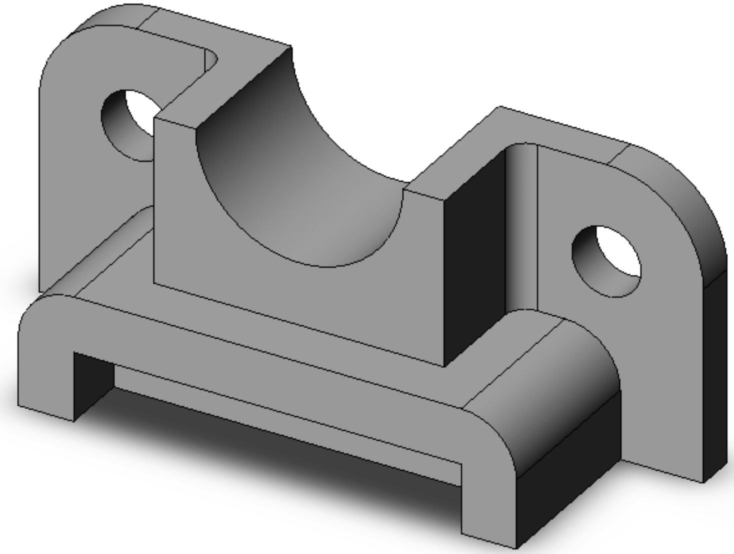
Examples of Shape Features

- **Chamfer feature**
  - Similar to a fillet.
  - Bevels an edge rather than rounding it.
  - Can remove or add material.

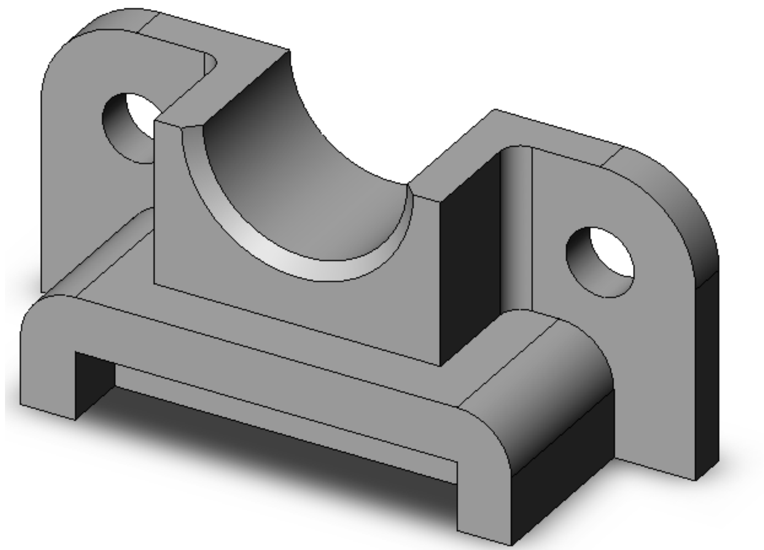


Examples of Shape Features

Fillet feature



Chamfer feature



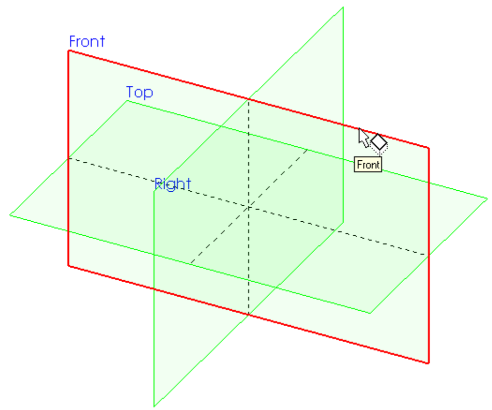
Examples of Shape Features

Sketched Features & Operation Features

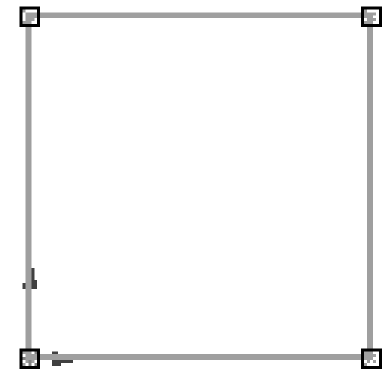
- **Sketched Features**
  - **Shape features have sketches.**
  - **Sketched features are built from 2D profiles.**
- **Operation Features**
  - **Operation features do not have sketches.**
  - **Applied directly to the work piece by selecting edges or faces.**

To Create an Extruded Base Feature:

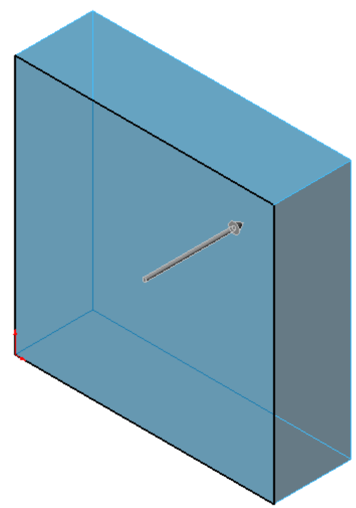
1. **Select a sketch plane.**
2. **Sketch a 2D profile.**
3. **Extrude the sketch perpendicular to sketch plane.**



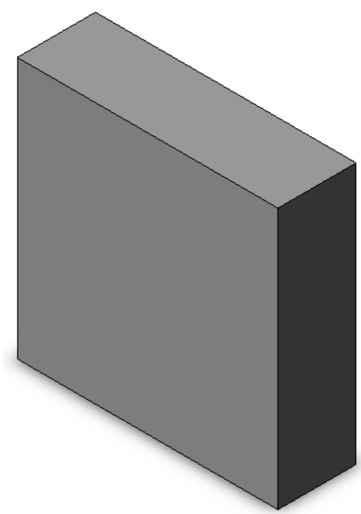
Select the sketch plane



Sketch the 2D profile



Extrude the sketch

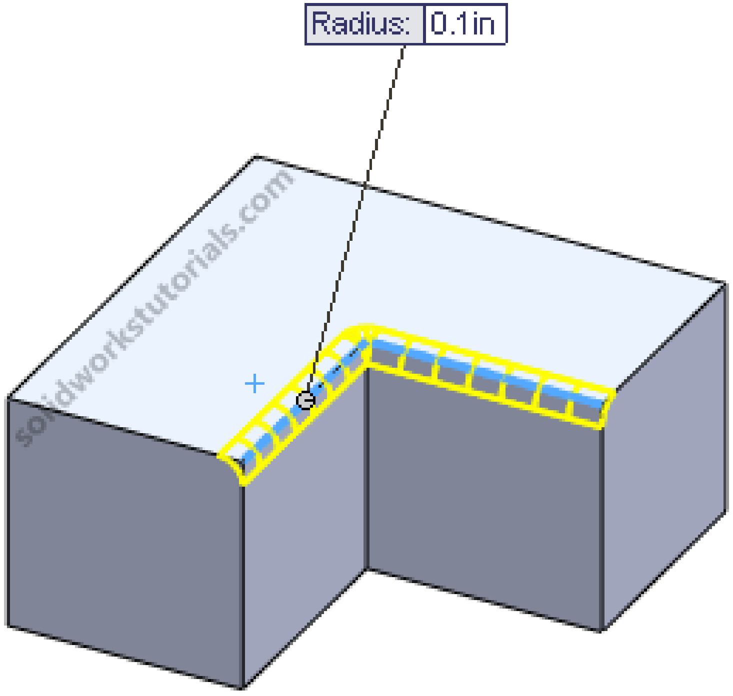


Resulting base feature

Operation Features

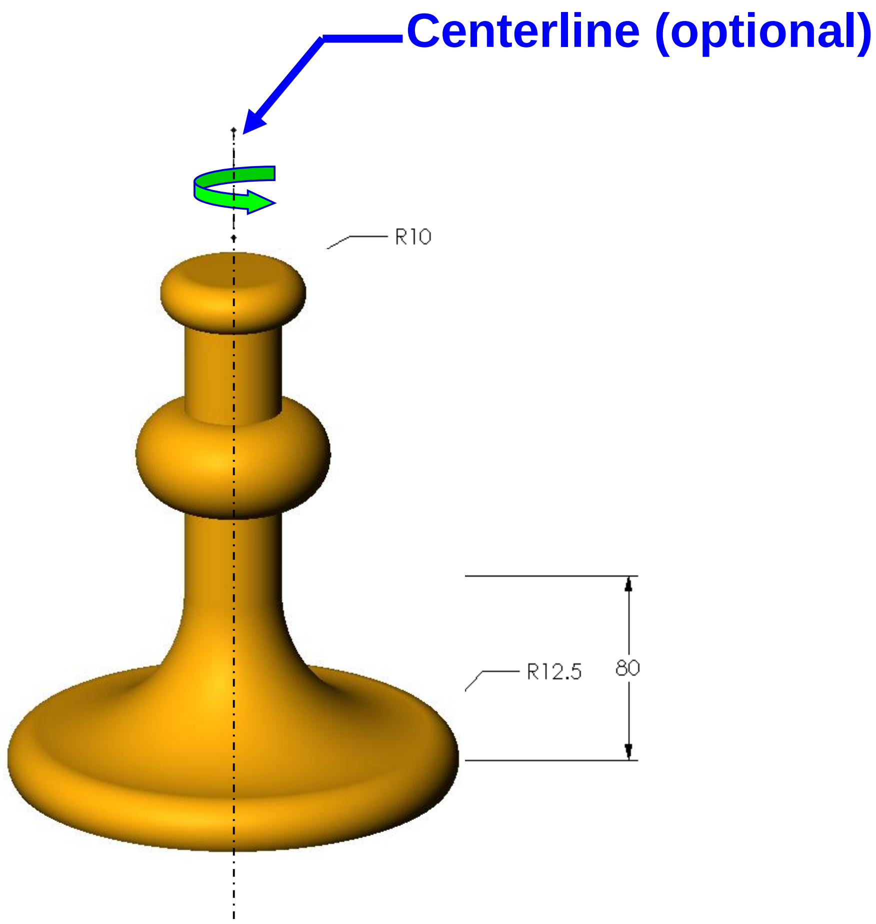
Operation features do not have sketches.

Applied directly to the work piece by selecting edges or faces.

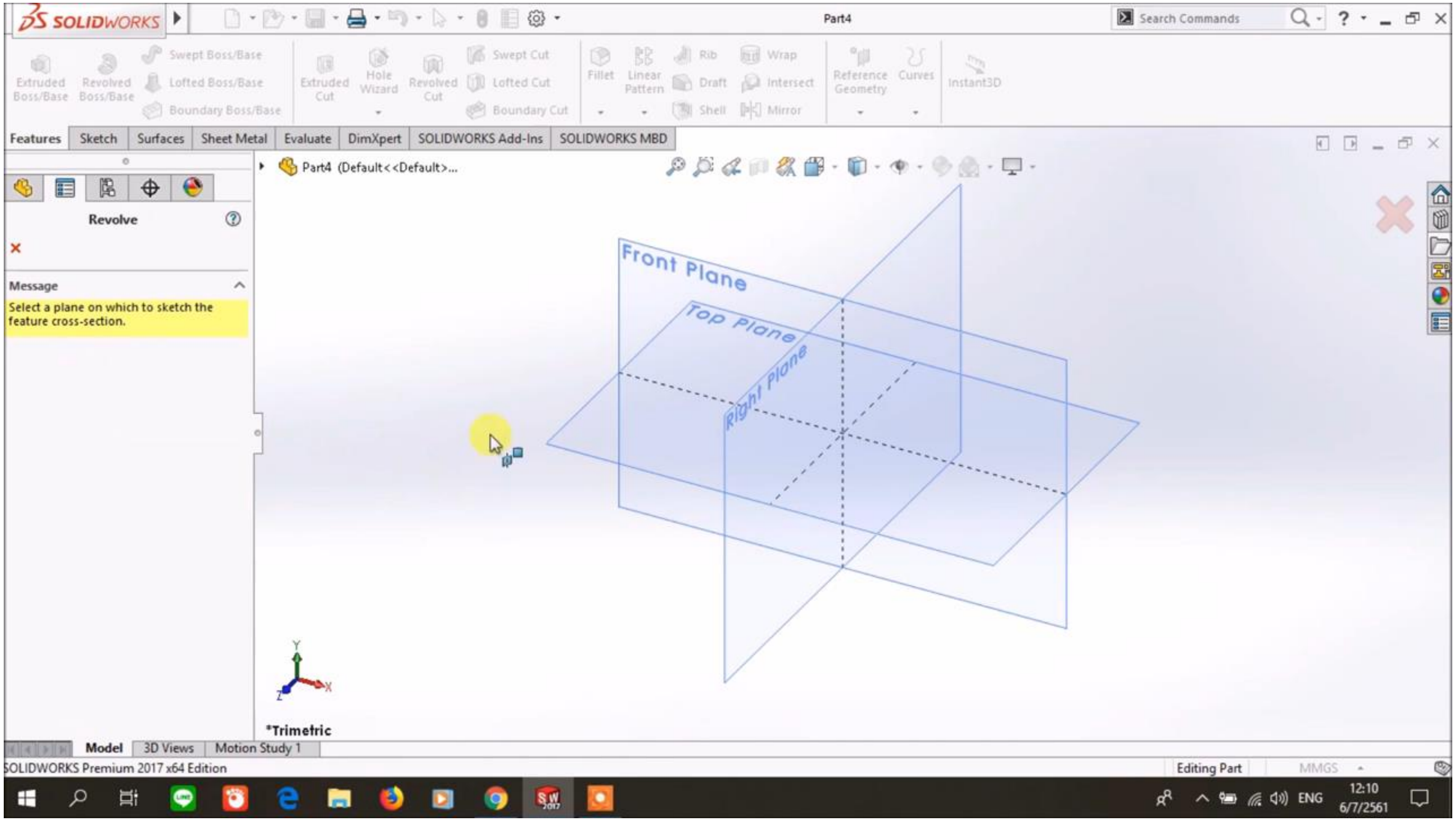


To Create a Revolved Base Feature:

1. **Select a sketch plane.**
2. **Sketch a 2D profile.**
3. **Sketch a centerline (optional).**
4. **Revolve the sketch around a sketch line or centerline.**



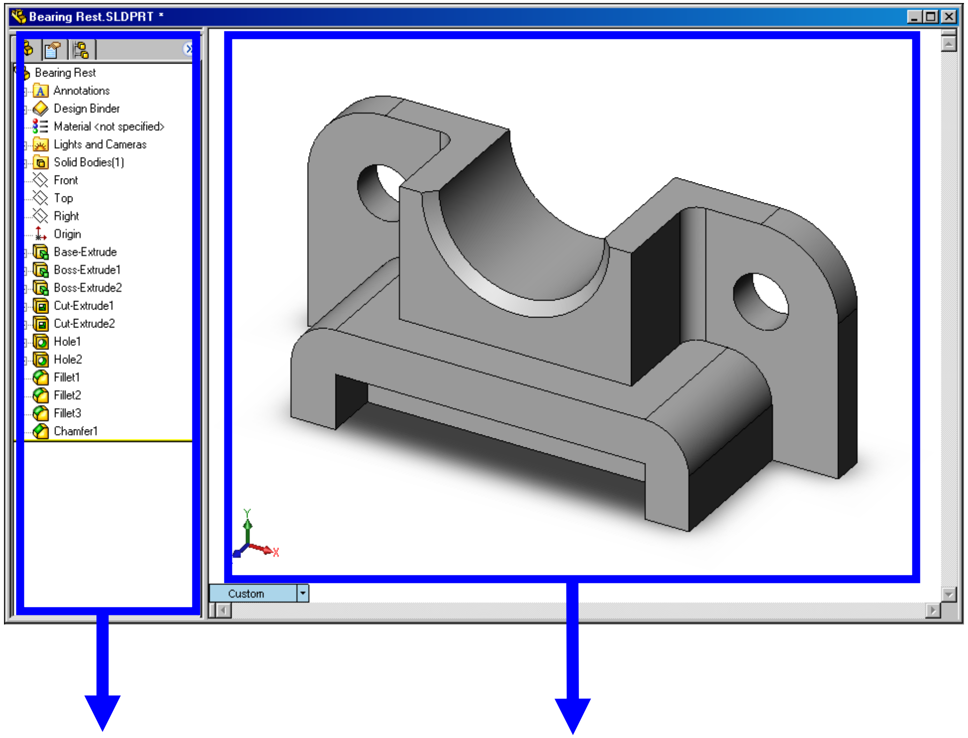
Revolved Base Feature



https://www.youtube.com/watch?v=NRqKmK6BkjQ

Terminology: Document Window

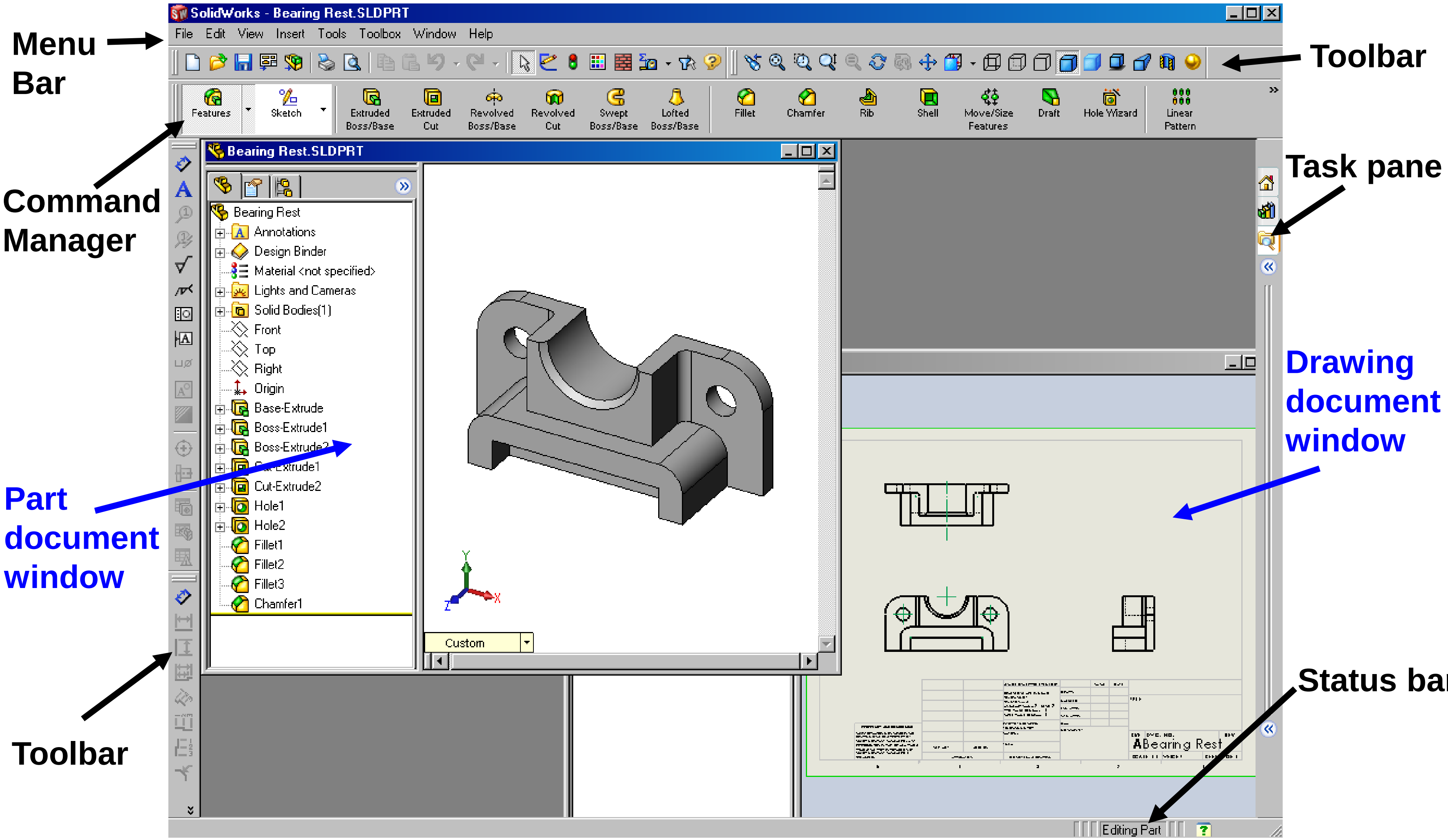
- **Divided into two panels:**
  - **Left panel contains the FeatureManager® design tree.**
    - **Lists the structure of the part, assembly or drawing.**
  - **Right panel contains the Graphics Area.**
    - **Location to display, create, and modify a part, assembly or drawing.**



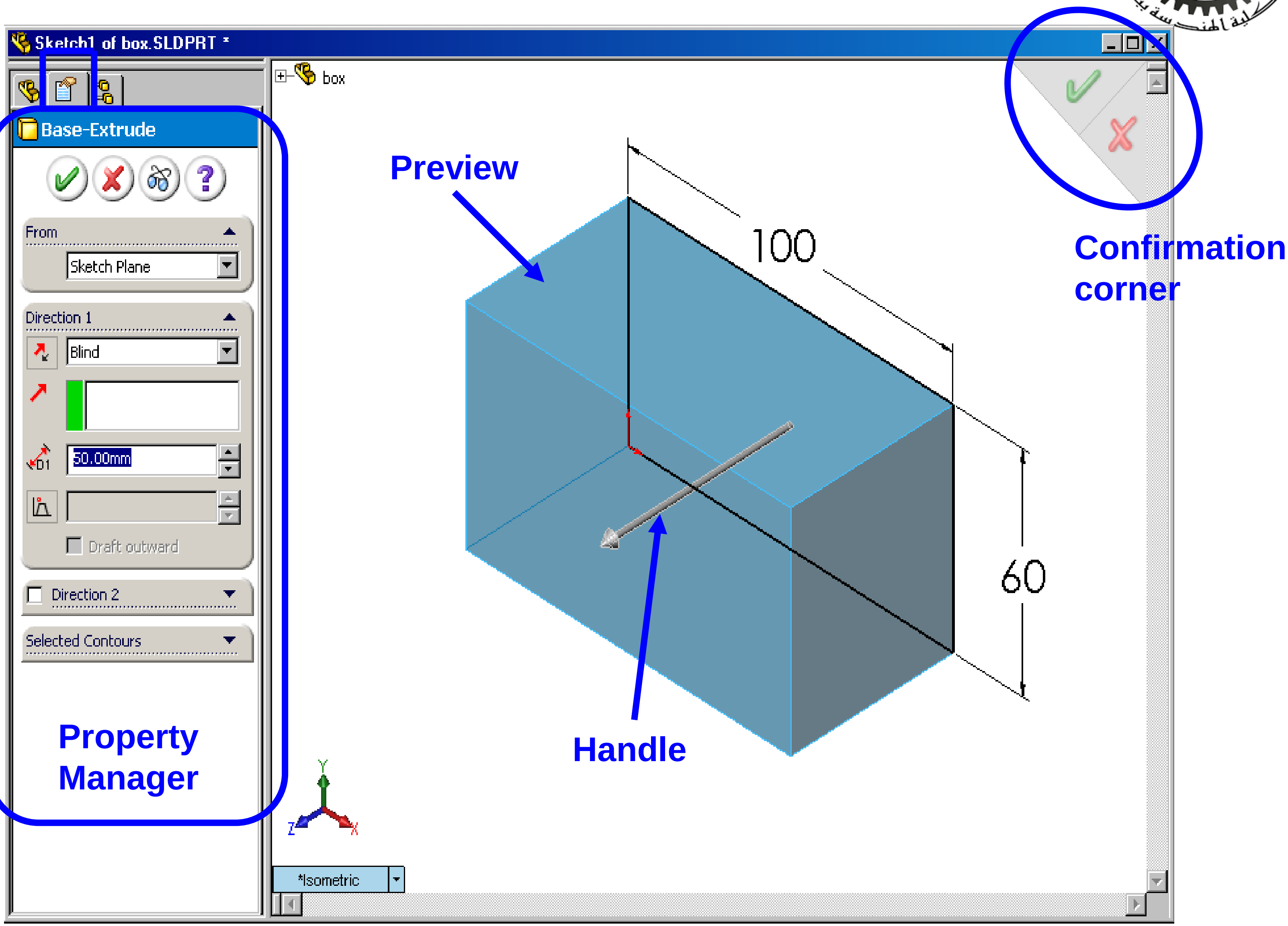
FeatureManager design tree

Graphics Area

Terminology: User Interface



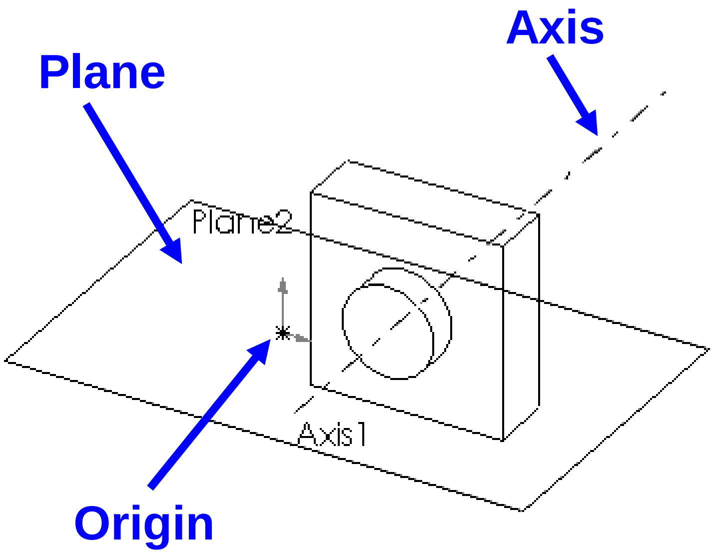
Terminology: Property Manager



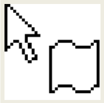
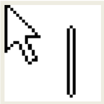
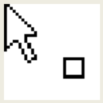
Terminology: Basic Geometry

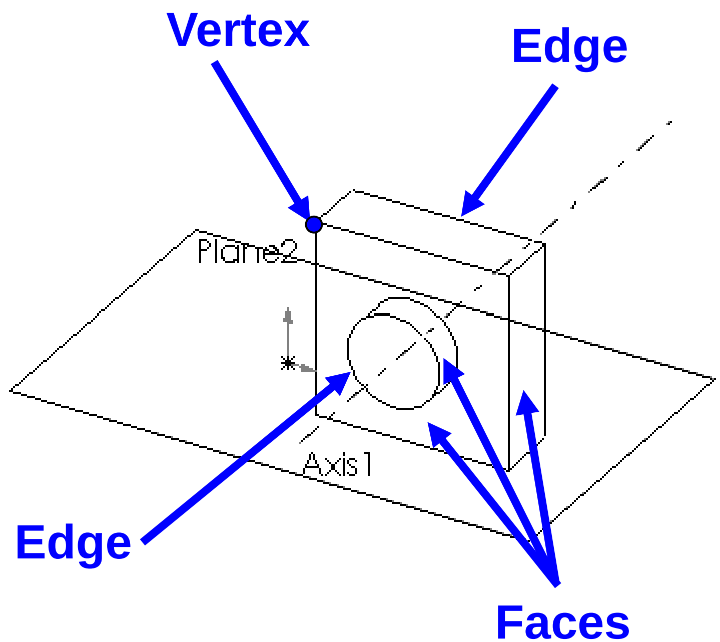
- **Axis - An implied centerline that runs through every cylindrical feature.**
- **Plane - A flat 2D surface.**
- **Origin - The point where the three default reference planes intersect. The coordinates of the origin are:**

  (x = 0, y = 0, z = 0).



Terminology: Basic Geometry

- **Face**  **–**
  The surface or "skin" of a part. Faces can be flat or curved.
- **Edge**  **–**
  The boundary of a face. Edges can be straight or curved.
- **Vertex**  **–**
  The corner where edges meet.



Features and Commands

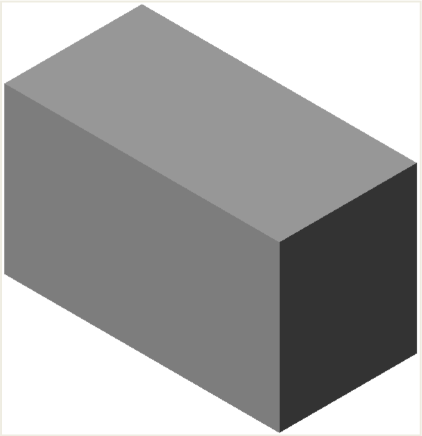
Base feature

- **The Base feature is the first feature that is created.**
- **The Base feature is the foundation of the part.**
- **The Base feature geometry for the box is an extrusion.**
- **The extrusion is named** Extrude1**.**

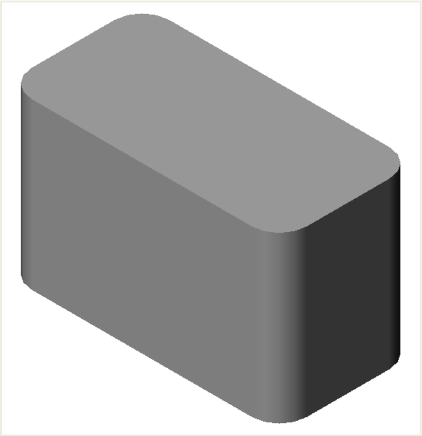
Features and Commands

Features used to build the *box* are:

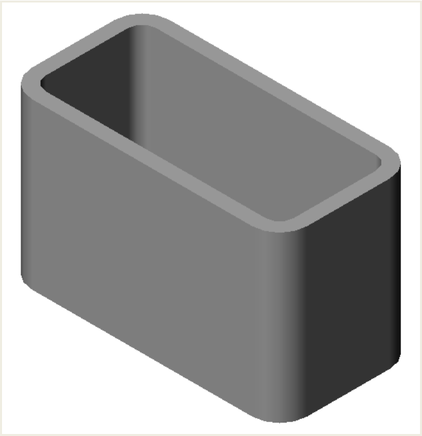
- **Extruded Base feature**
- **Fillet feature**
- **Shell feature**
- **Extruded Cut feature**



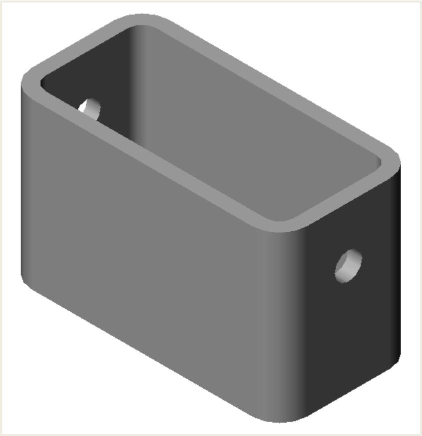
1.Base Feature



2.Fillet Feature



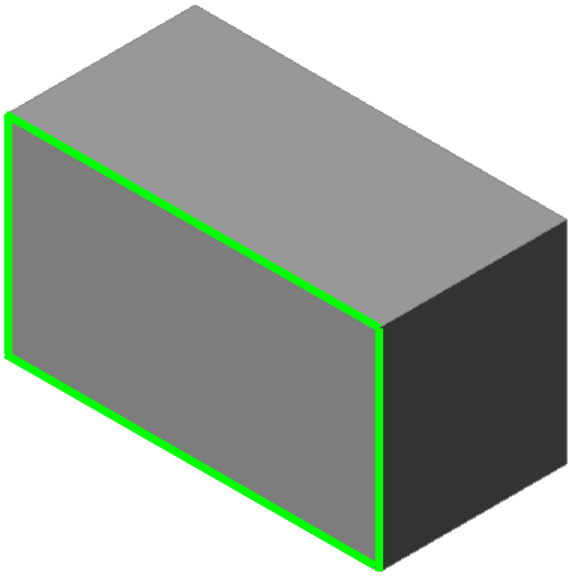
3.Shell Feature



4.Cut Feature

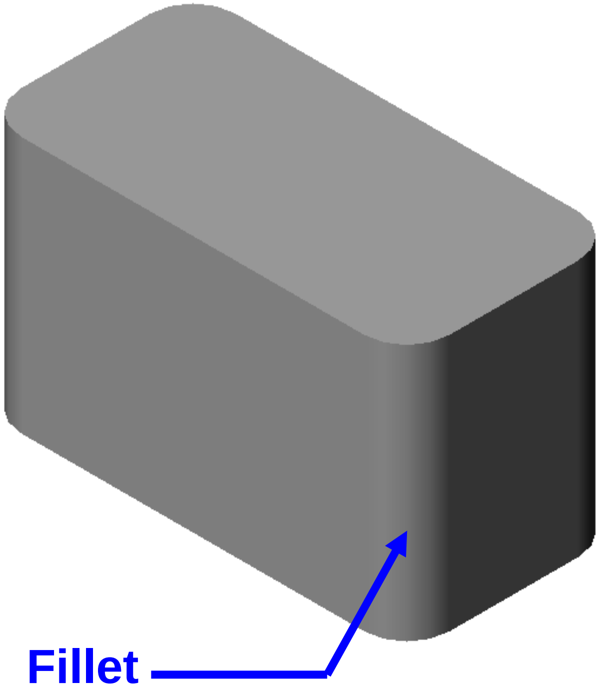
To create the extruded base feature for the *box*:

- **Sketch a rectangular profile on a 2D plane.**
- **Extrude the sketch.**
- **By default extrusions are perpendicular to the sketch plane.**



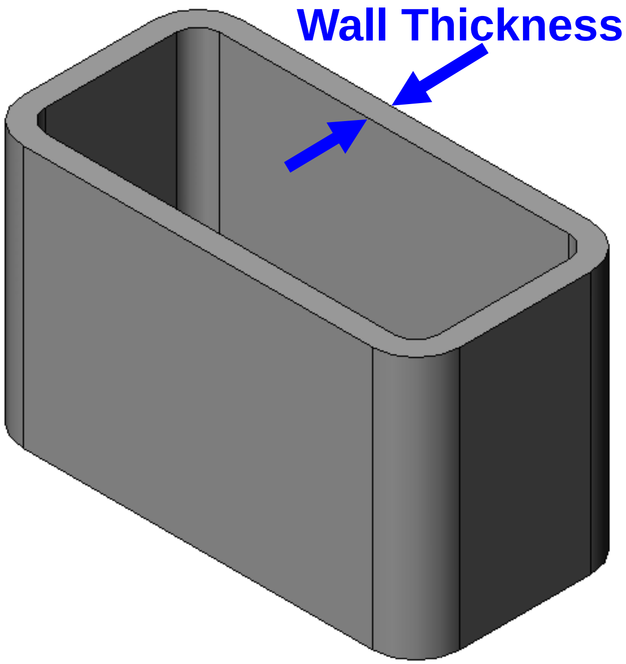
Fillet feature

- The fillet feature rounds the edges or faces of a part.
- Select the edges to be rounded. Selecting a face rounds all the edges of that face.
- Specify the fillet radius.



Shell feature

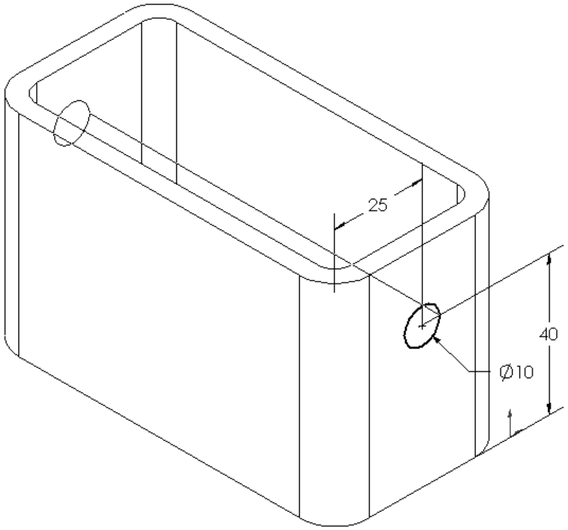
- The shell feature removes material from the selected face.
- Using the shell feature creates a hollow box from a solid box.
- Specify the wall thickness for the shell feature.

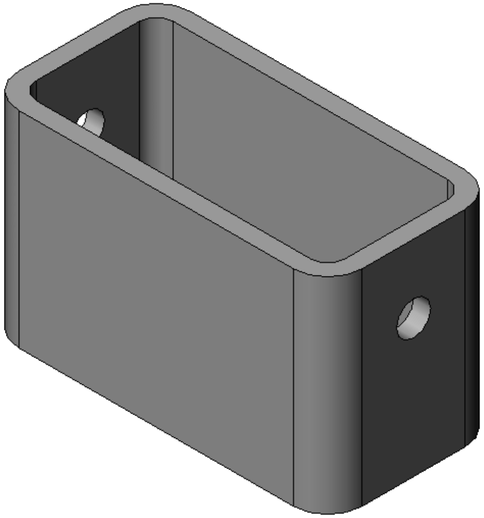


Features and Commands

To create the extruded cut feature for the *box*:

- Sketch the 2D circular profile.
- Extrude the 2D Sketch profile perpendicular to the sketch plane.
- Enter <u>Through All</u> for the end condition.
- The cut penetrates through the entire part.



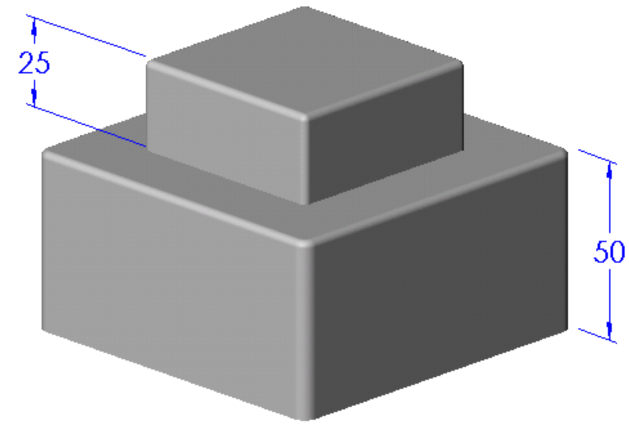


Dimensions and Geometric Relationships

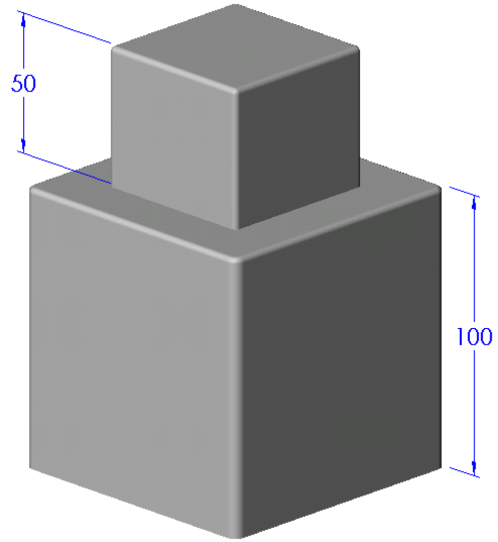
- **Specify dimensions and geometric relationships between features and sketches.**
- **Dimensions change the size and shape of the part.**
- **Mathematical relationships between dimensions can be controlled by equations.**
- **Geometric relationships are the rules that control the behavior of sketch geometry.**
- **Geometric relationships help capture design intent.**

Dimensions

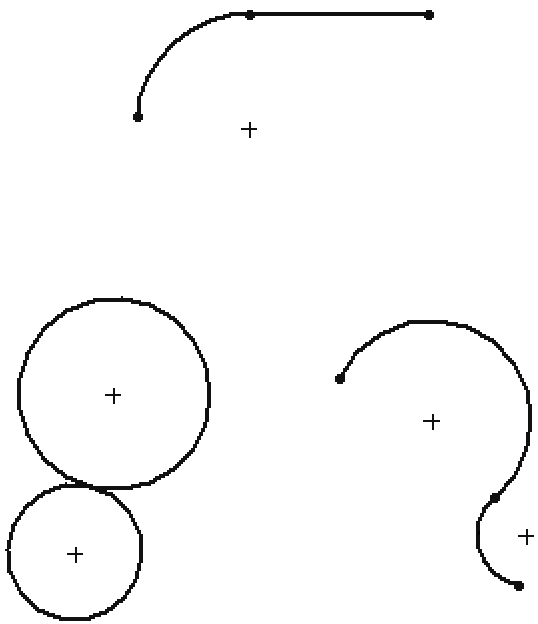
- **Dimensions**
  - **Base depth = 50 mm**
  - **Boss depth = 25 mm**



- **Mathematical relationship**
  - **Boss depth = Base depth ÷ 2**



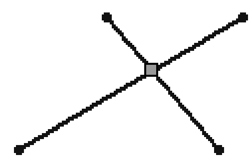
Geometric Relationships



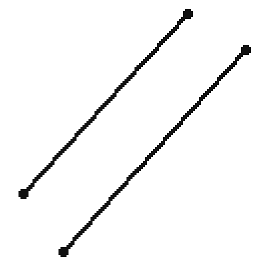
Tangent

Vertical

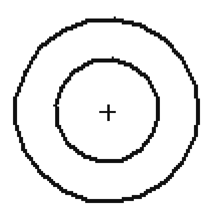
Horizontal



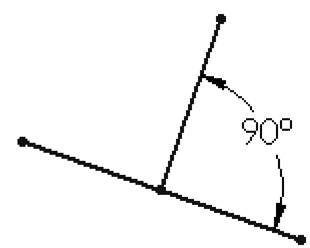
Intersection



Parallel



Concentric

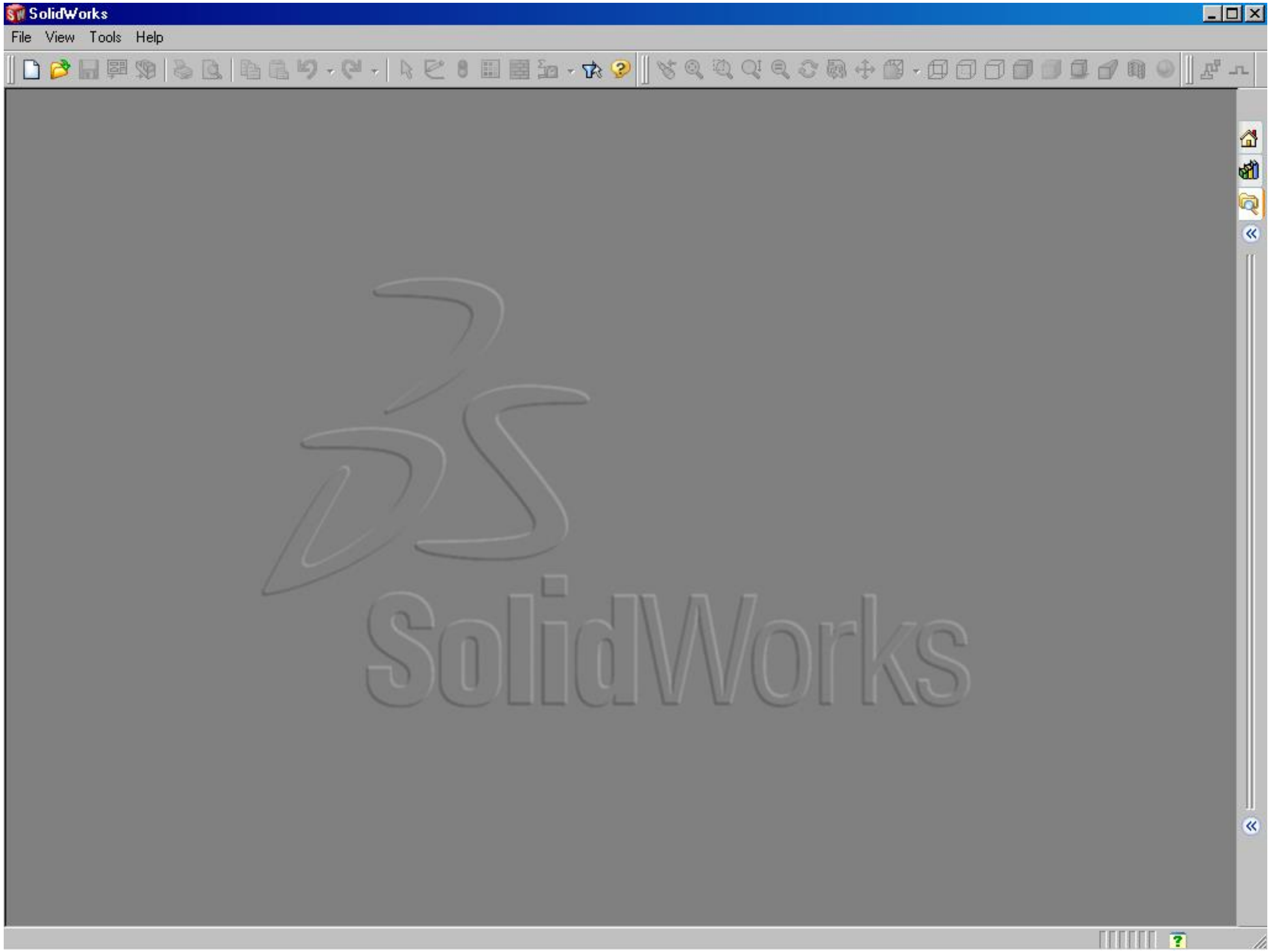


Perpendicular

To Start SolidWorks

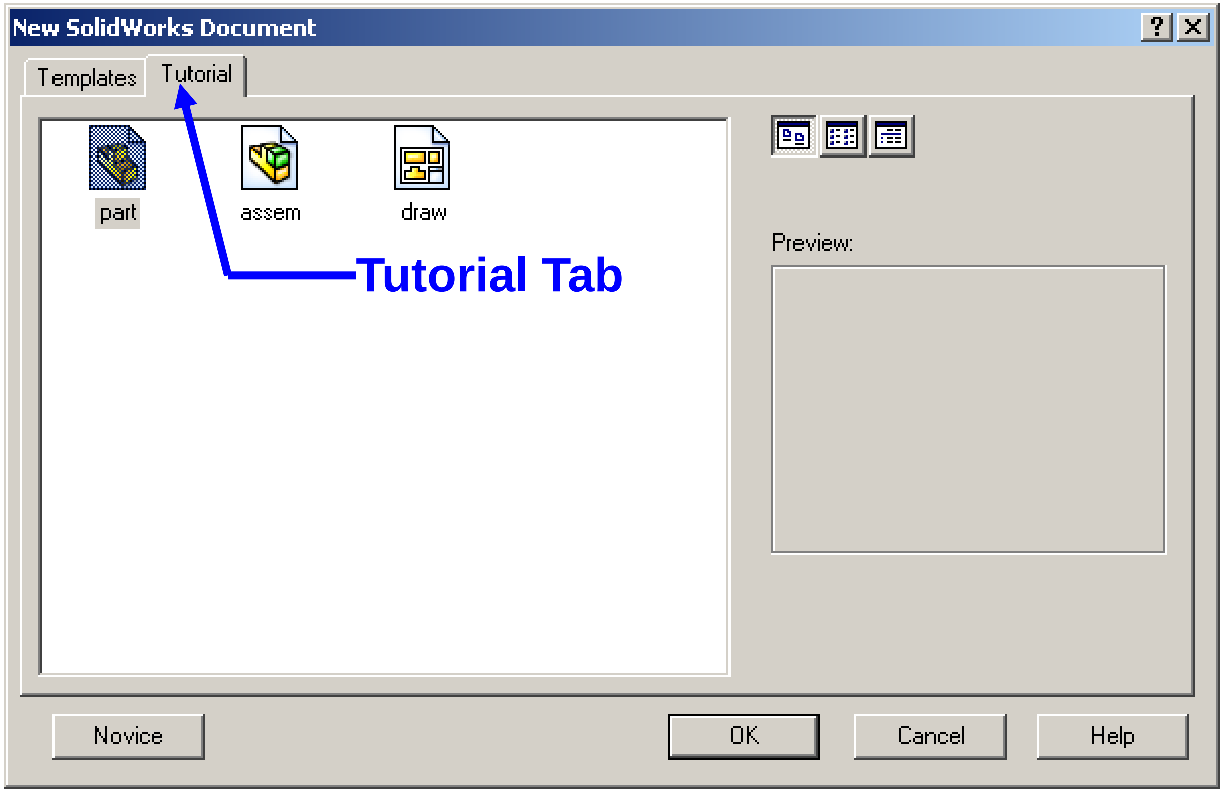
- **Click the <u>Start</u> button**  **on Windows task bar.**
  - ➢ **Click <u>Programs</u>.**
    - ➢ **Click the SolidWorks folder.**
      - ➢ **Click the SolidWorks application.**

The SolidWorks Window



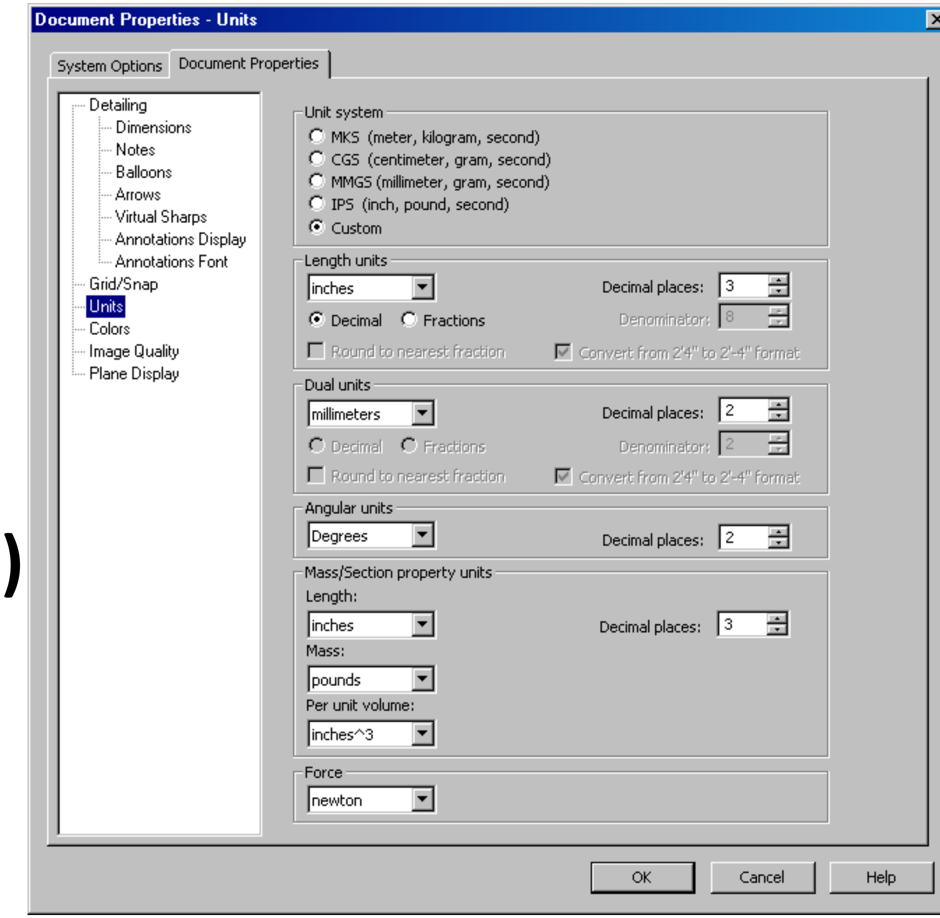
Creating New Files Using Templates

- **Click <u>New</u> on the Standard toolbar.**
- **Select a document template:**
  - Part
  - Assembly
  - Drawing



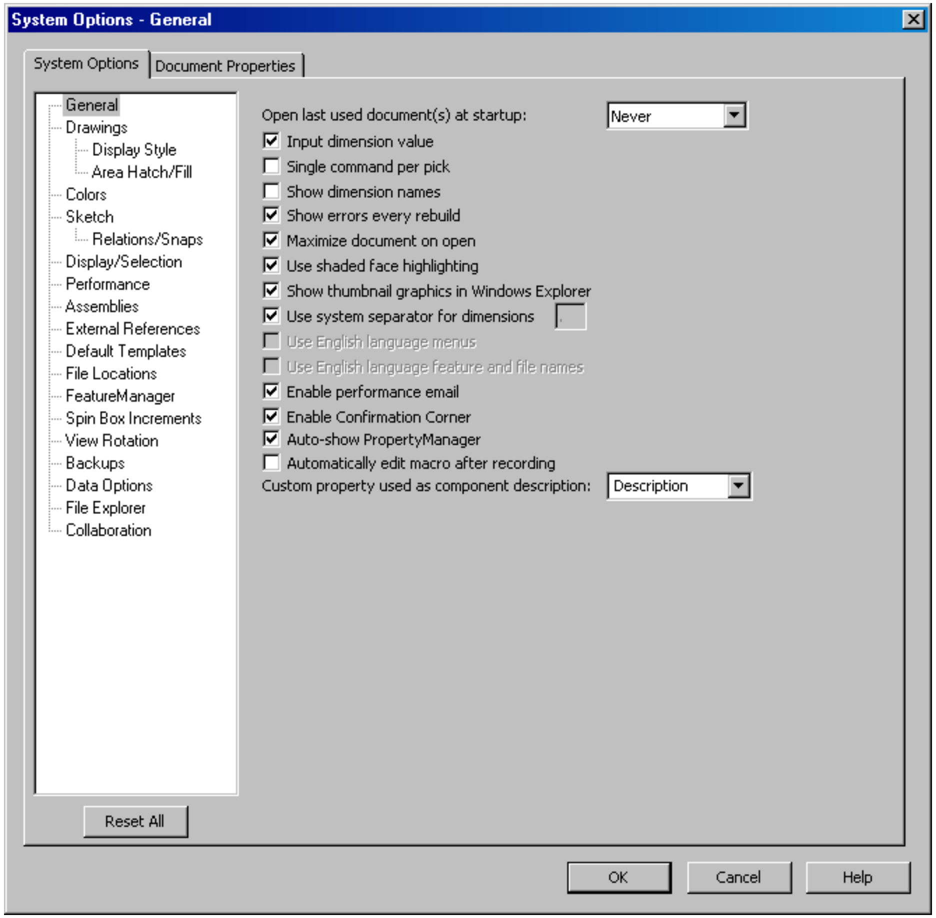
Document Properties

- **Accessed through the Tools, Options menu.**
- **Control settings like:**
  - **Units: English (inches) or Metric (millimeters)**
  - **Grid/Snap Settings**
  - **Colors, Material Properties and Image Quality**



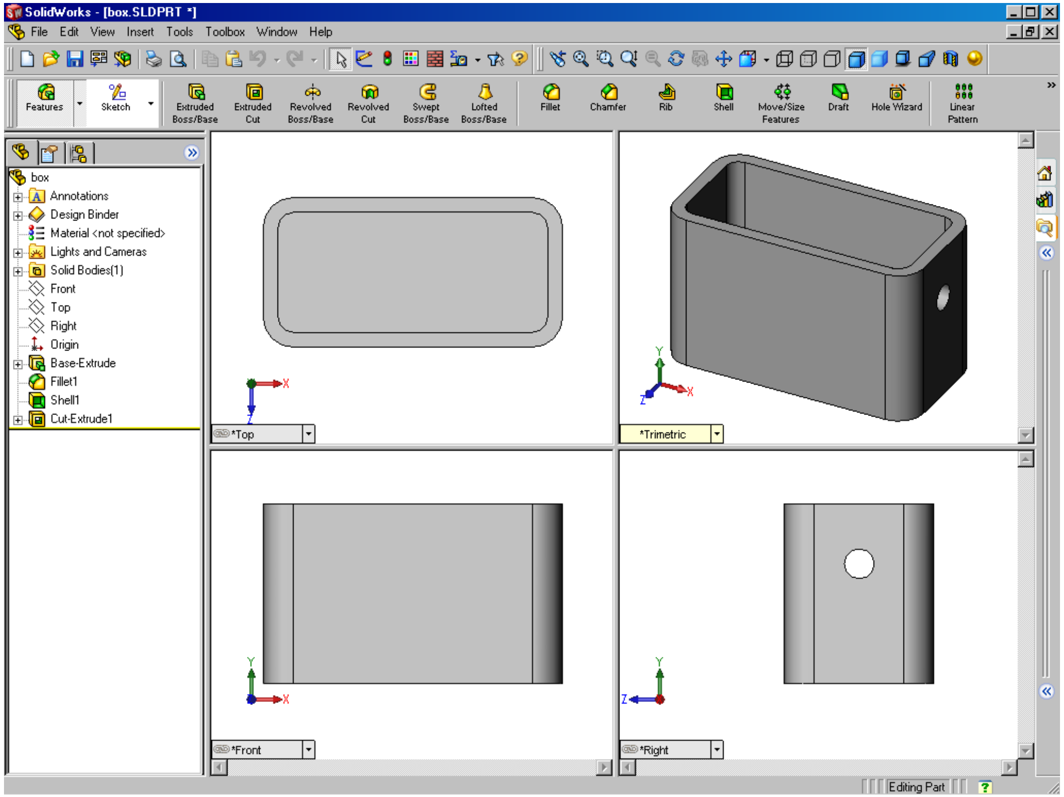
System Options

- **Accessed through the Tools, Options menu.**
- **Allow you to customize your work environment.**
- **System options control:**
  - **File locations**
  - **Performance**
  - **Spin box increments**



Multiple Views of a Document

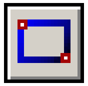
- **Click the view pop-up menu.**
- **Select an icon. The viewport icons include:**
  - Single View
  - Two View (horizontal and vertical)
  - Four View



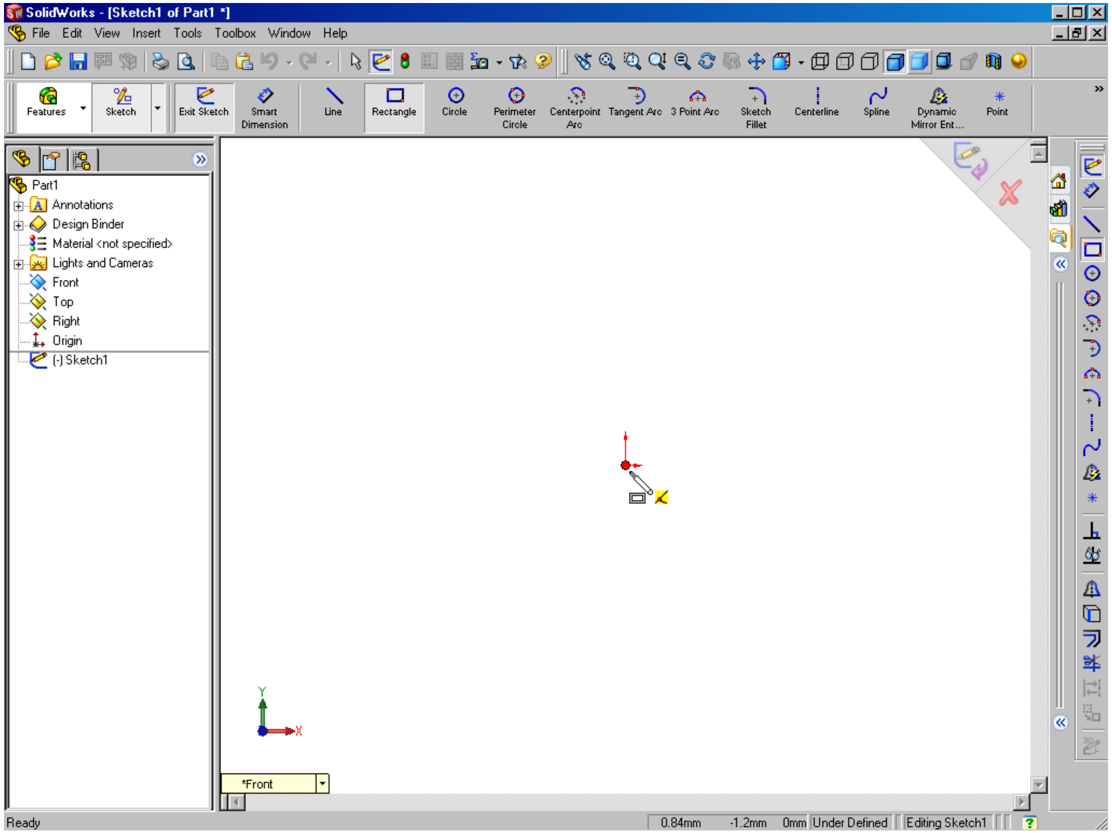
Creating a 2D Sketch

1. Click Sketch  the Sketch toolbar.

2. Select the Front plane as a sketch plane.

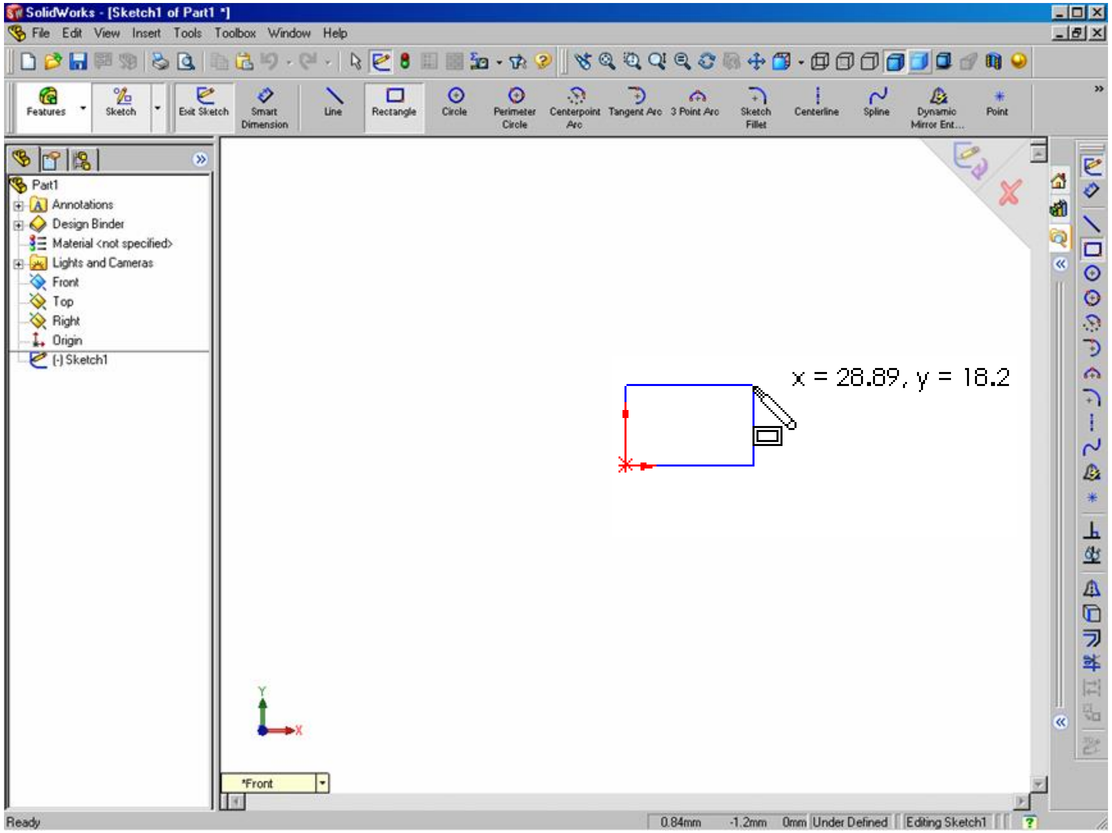
3. Click Rectangle  on the Sketch Tools toolbar.

4. Move the pointer to the Sketch Origin.



Creating a 2D Sketch

5. **Click the left mouse button.**
6. **Drag the pointer up and to the right.**
7. **Click the left mouse button again.**

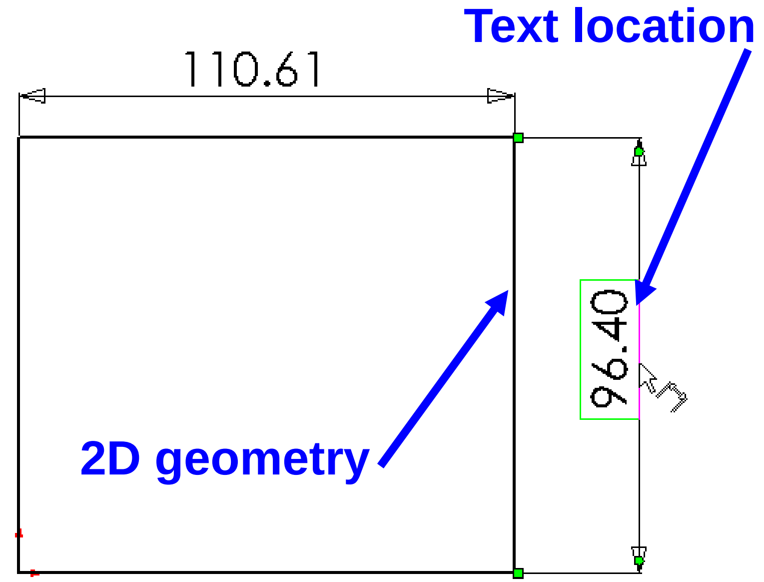


Adding Dimensions

- **Dimensions specify the size of the model.**

To create a dimension:

1. **Click Dimension on the Sketch Relations toolbar.**
2. **Click the 2D geometry.**
3. **Click the text location.**
4. **Enter the dimension value.**



Thank You for Attention !!

Any Questions