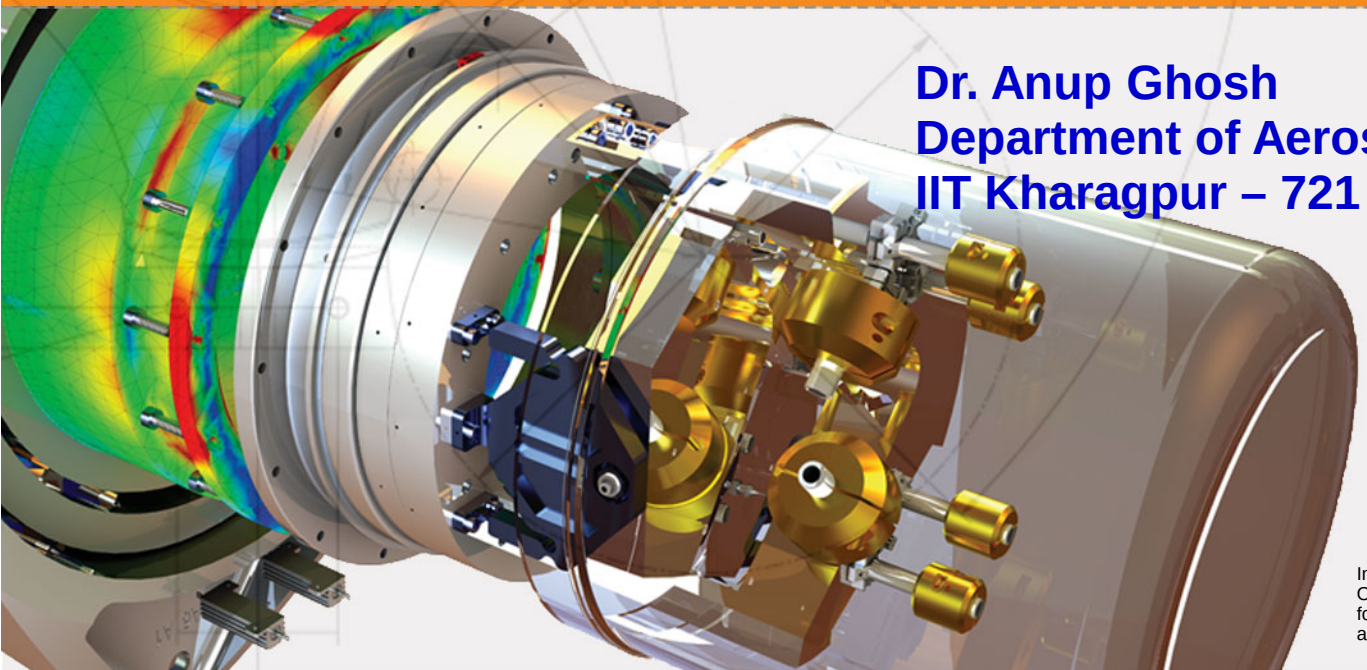
Basic Functionality



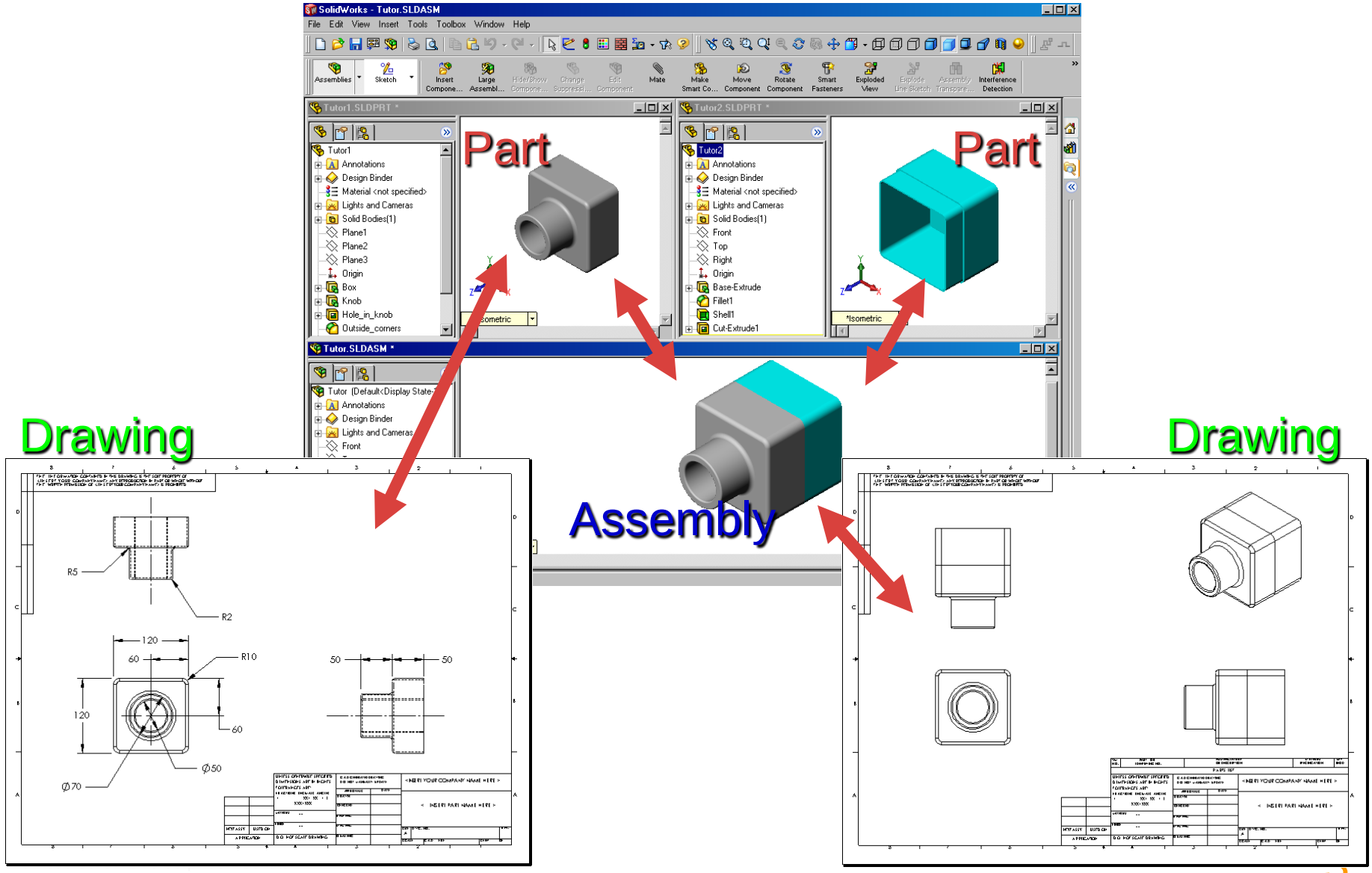
Dr. Anup Ghosh
Department of Aerospace Engineering
IIT Kharagpur – 721 302

Image courtesy of National Optical Astronomy Observatory, operated by the Association of Universities for Research in Astronomy, under cooperative agreement with the National Science Foundation.

The SolidWorks Model

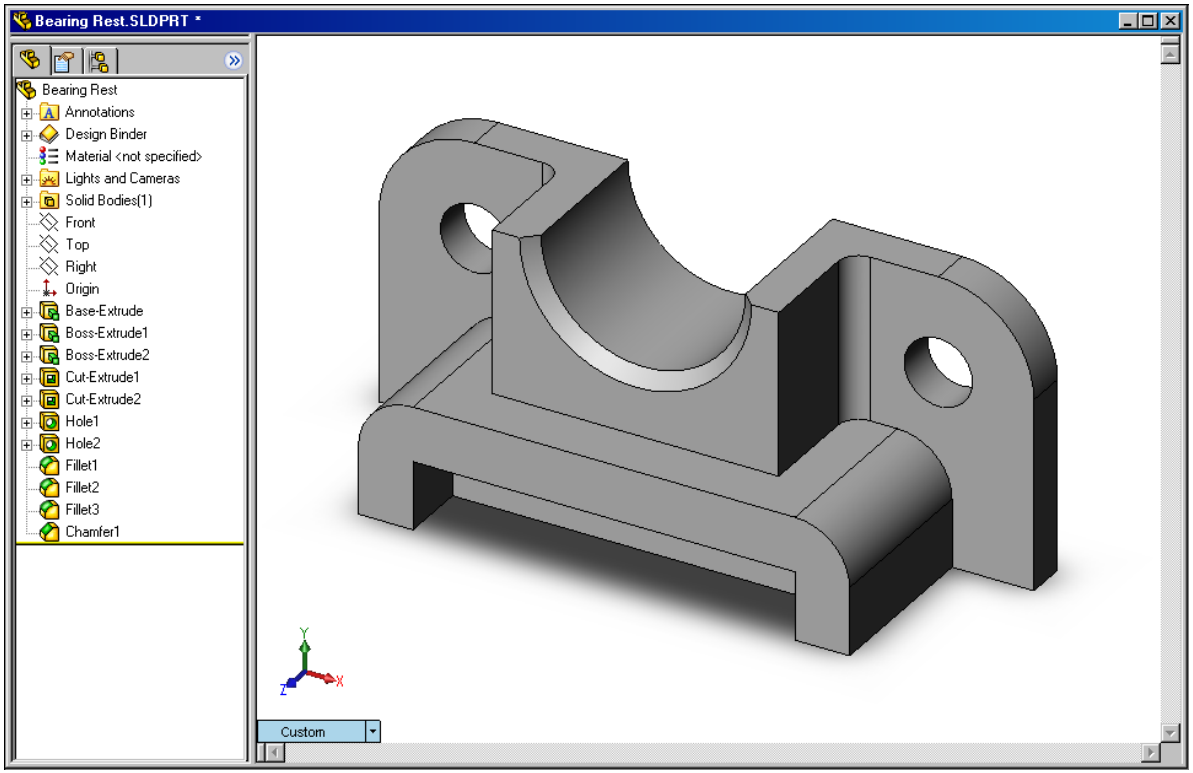
- **The SolidWorks model is made up of:**
  -  **Parts**
  -  **Assemblies**
  -  **Drawings**

The SolidWorks Model



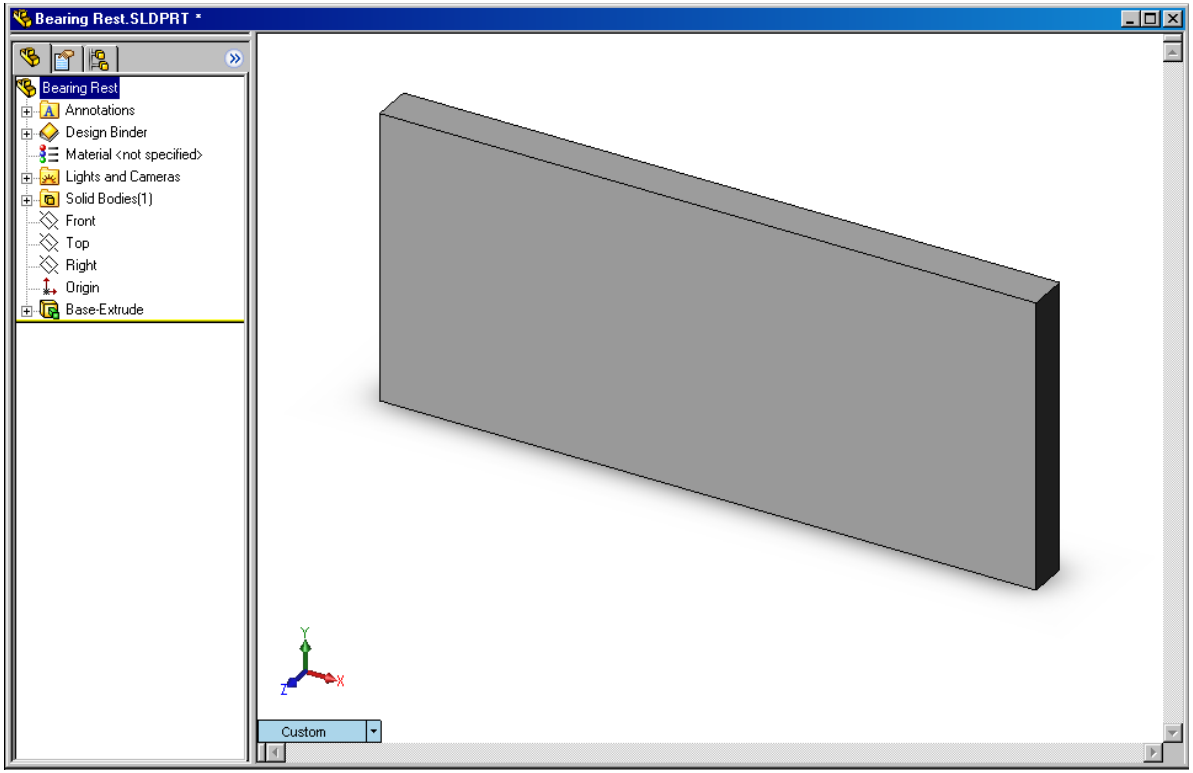
Features

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



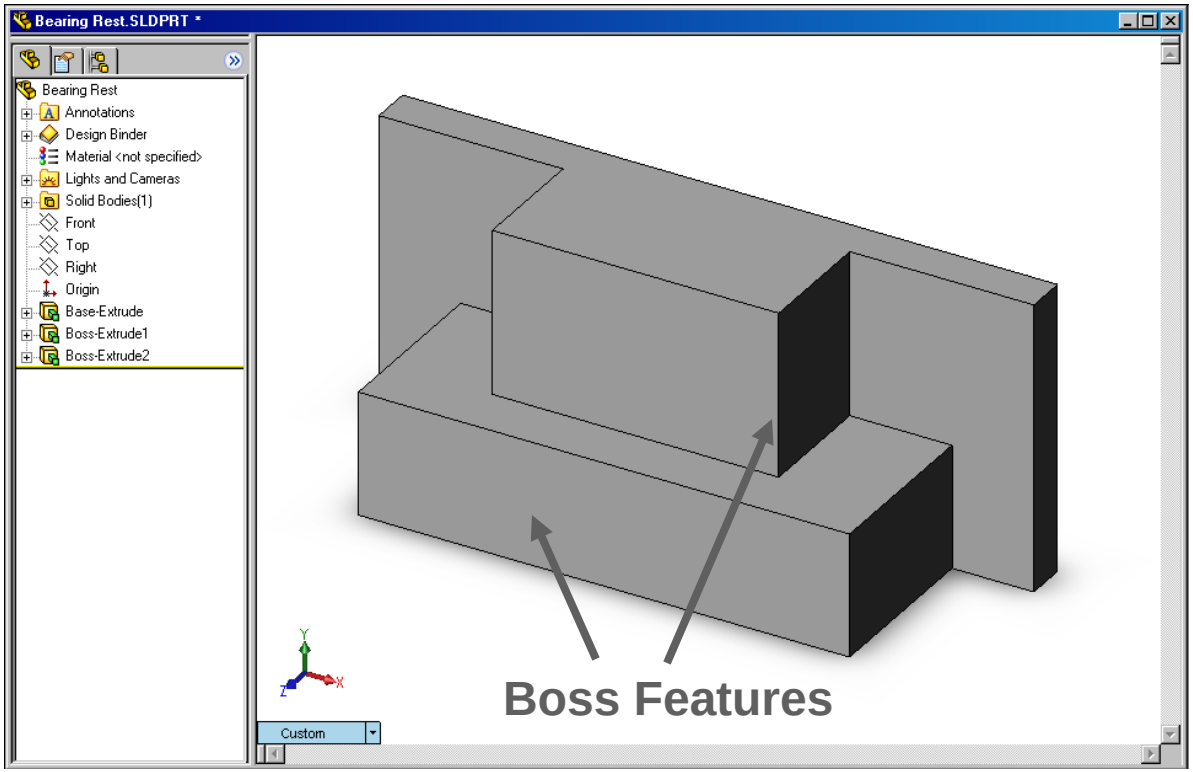
Examples of Shape Features

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



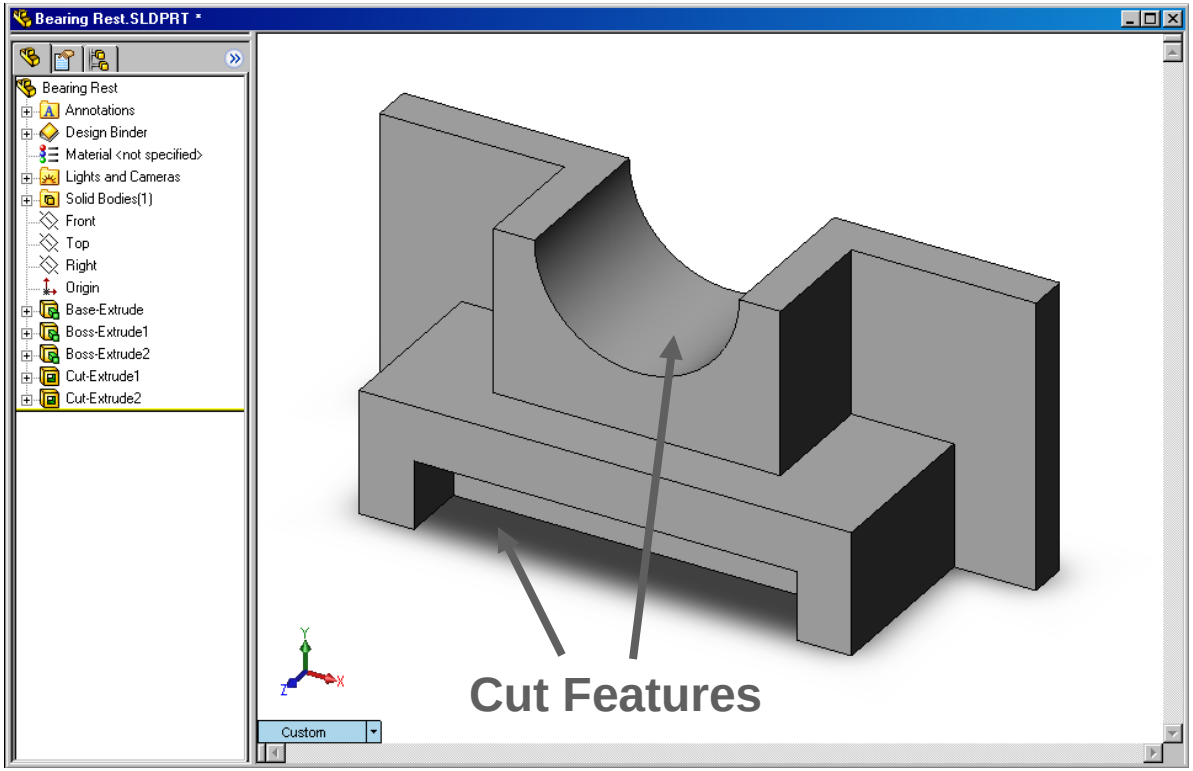
Examples of Shape Features

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



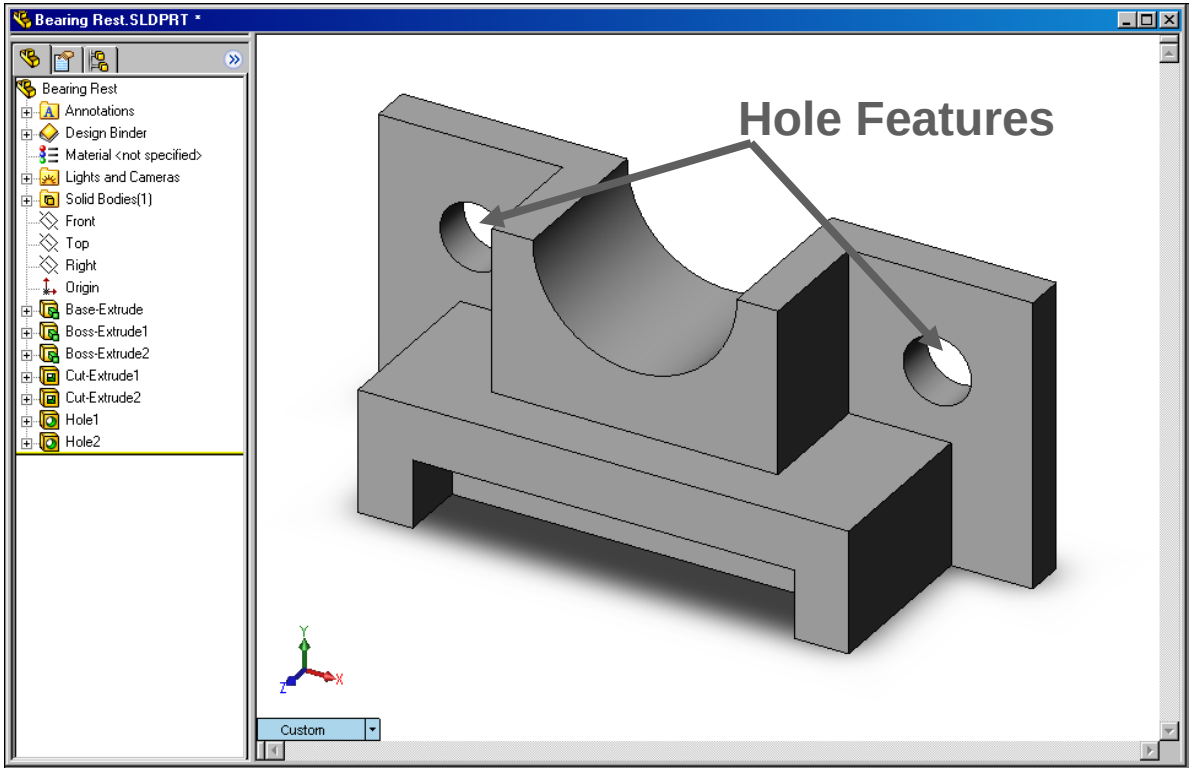
Examples of Shape Features

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



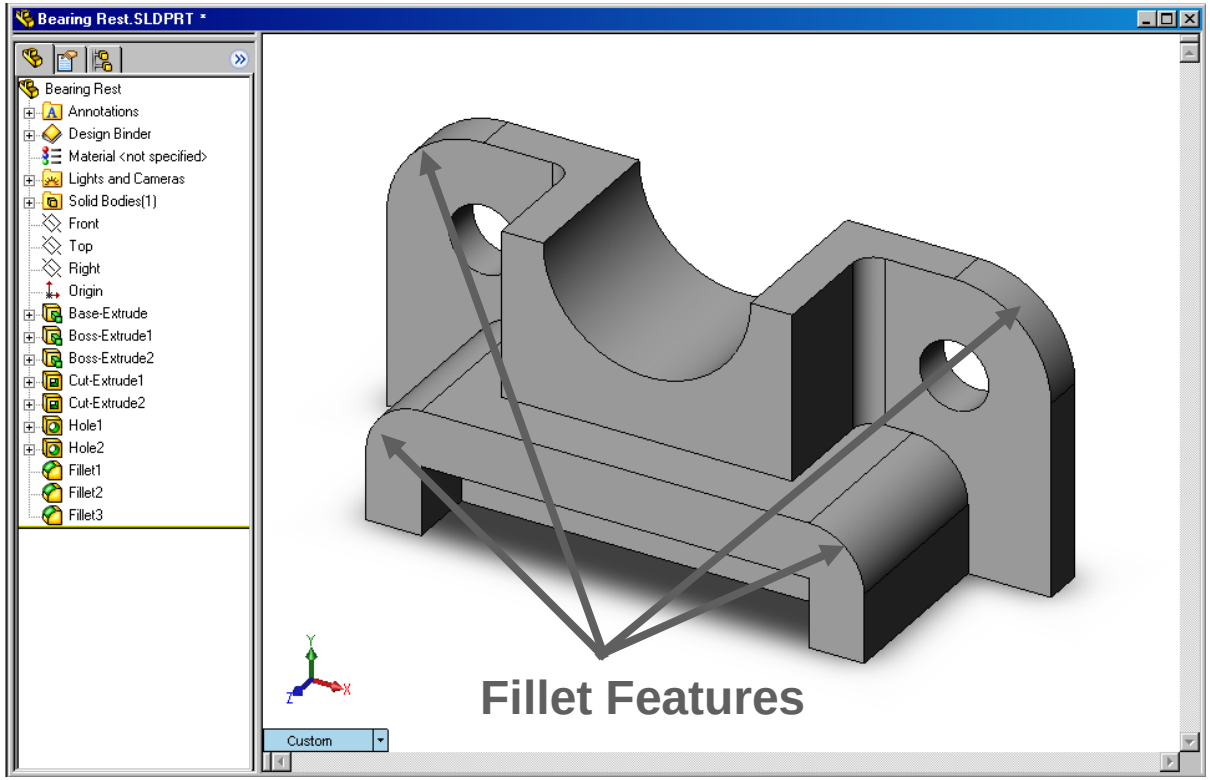
Examples of Shape Features

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



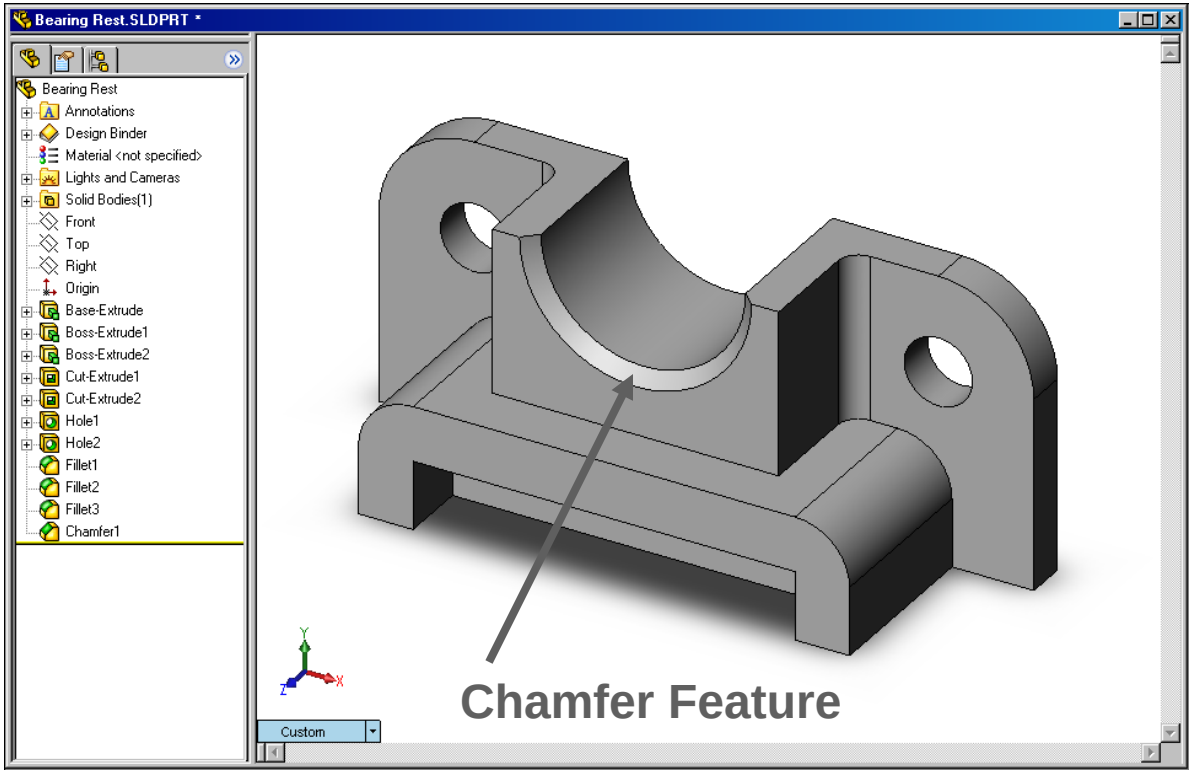
Examples of Shape Features

- **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

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



Sketched Features & Operation Features

- **Sketched Features**
  - **Shape features have sketches.**
  - **Sketched features are built from 2D profiles.**

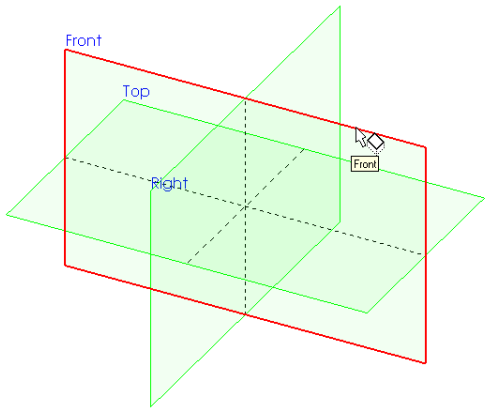
- **Operation Features**
  - **Operation features do not have sketches.**
  - **Applied directly to the work piece by selecting edges or faces.**

To Create an Extruded Base Feature:

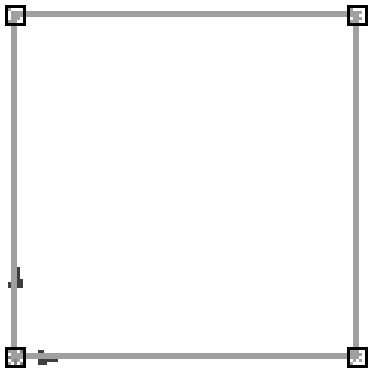
1. **Select a sketch plane.**

2. **Sketch a 2D profile.**

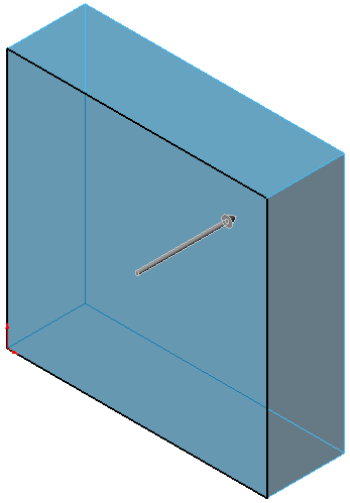
3. **Extrude the sketch perpendicular to sketch plane.**



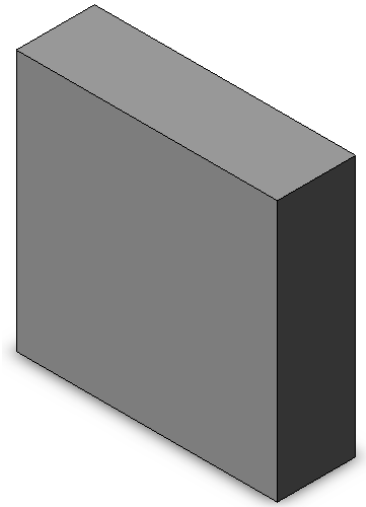
Select the sketch plane



Sketch the 2D profile



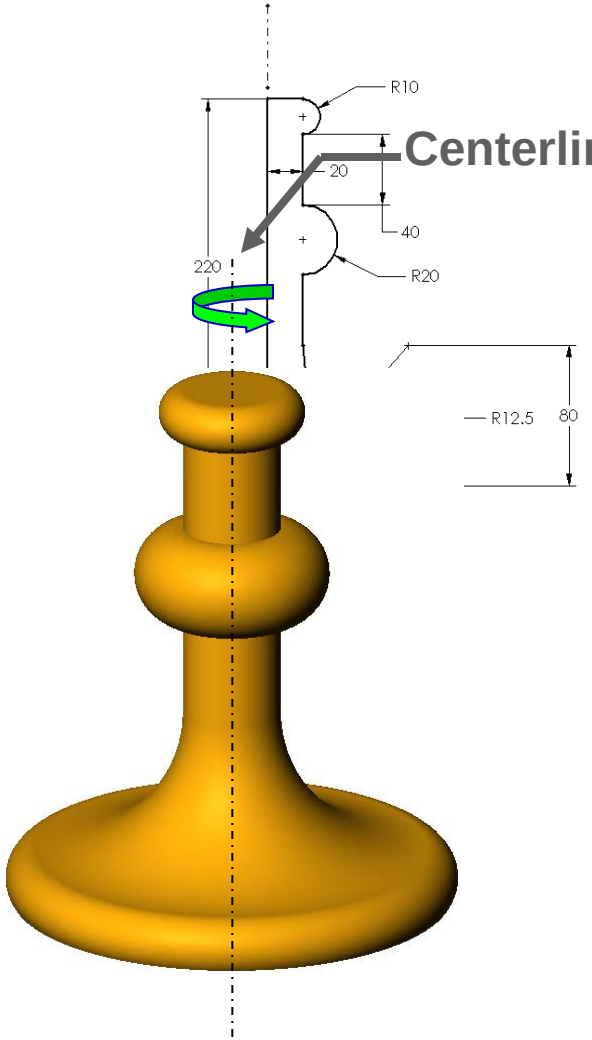
Extrude the sketch



Resulting base feature

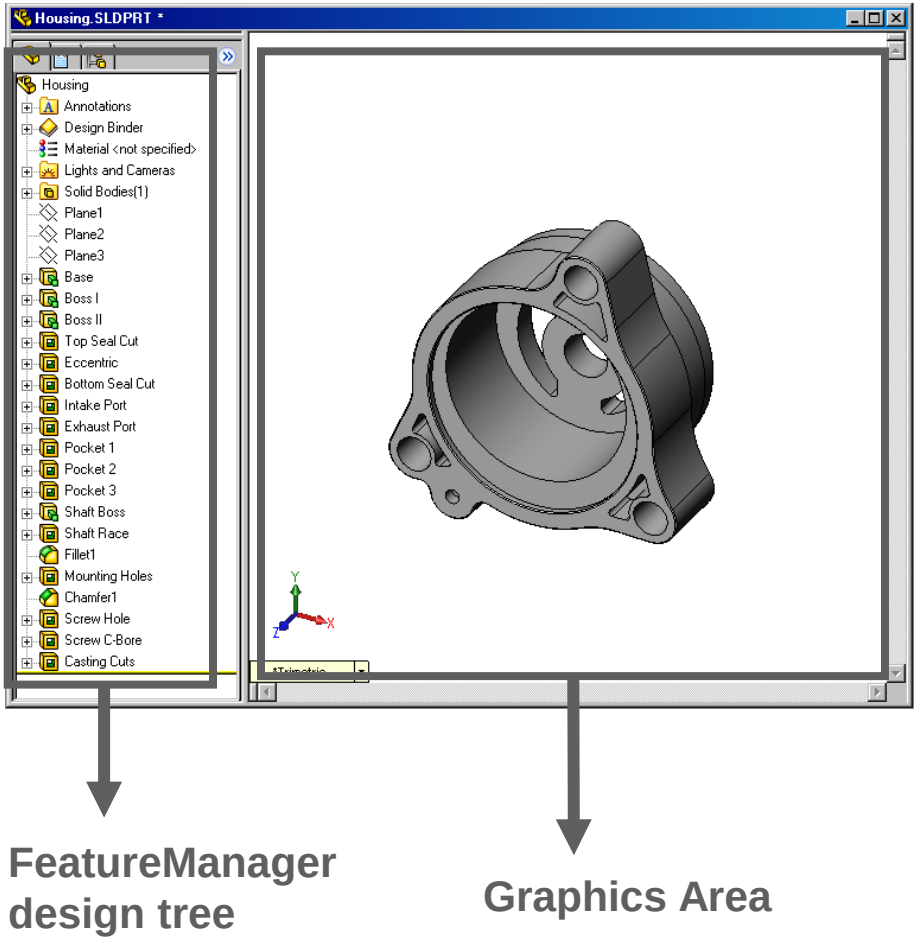
To Create a Revolved Base Feature:

1. **Select a sketch plane.**
2. **Sketch a 2D profile.**
3. **Sketch a centerline.**
4. **Revolve the sketch around the centerline.**

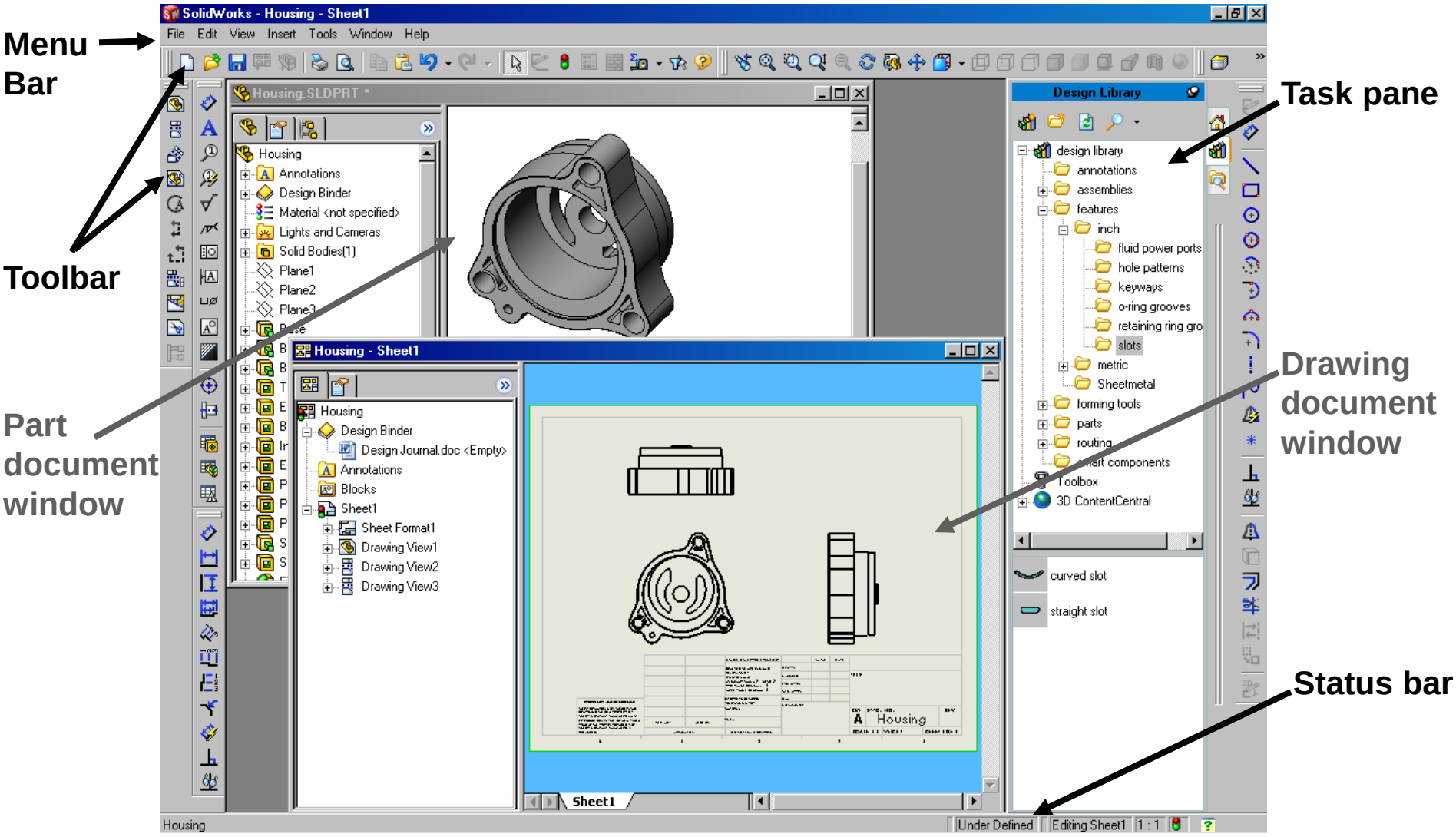


Terminology: Document Window

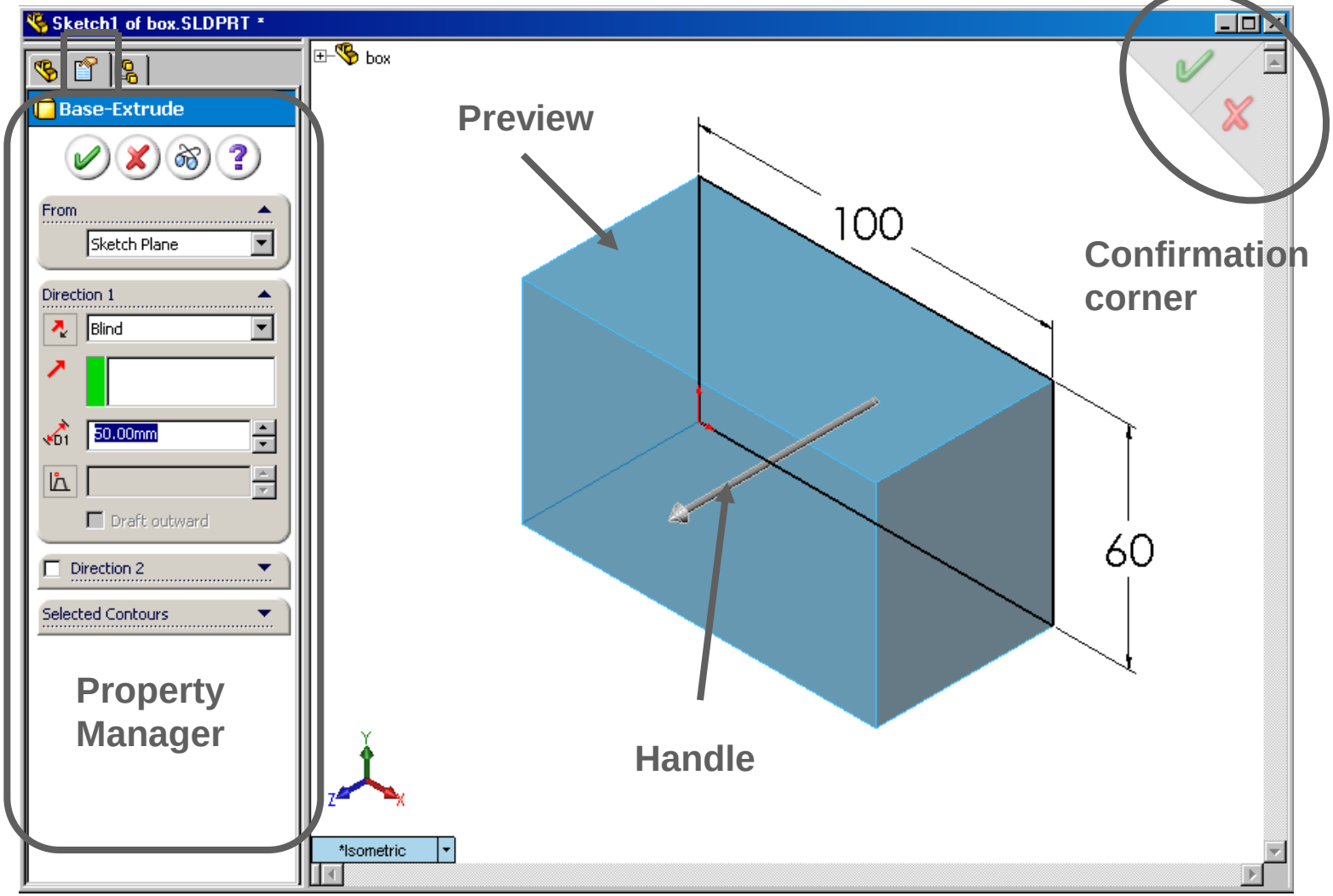
- **Divided into two panels:**
  - **Left panel - FeatureManager® design tree.**
    - **Lists the structure of the part, assembly or drawing.**
  - **Middle panel – Graphics Area.**
    - **Location to display, create, and modify a part, assembly or drawing.**



Terminology: User Interface

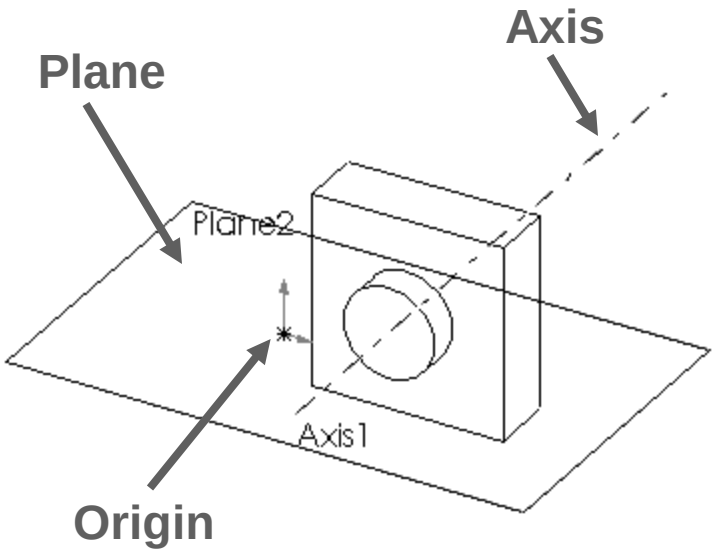


Terminology: PropertyManager



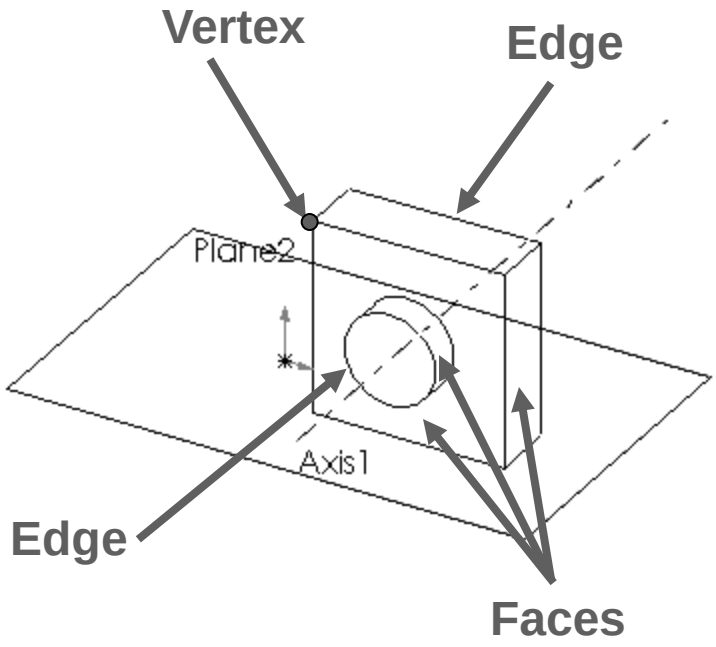
Terminology: Basic Geometry

- **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

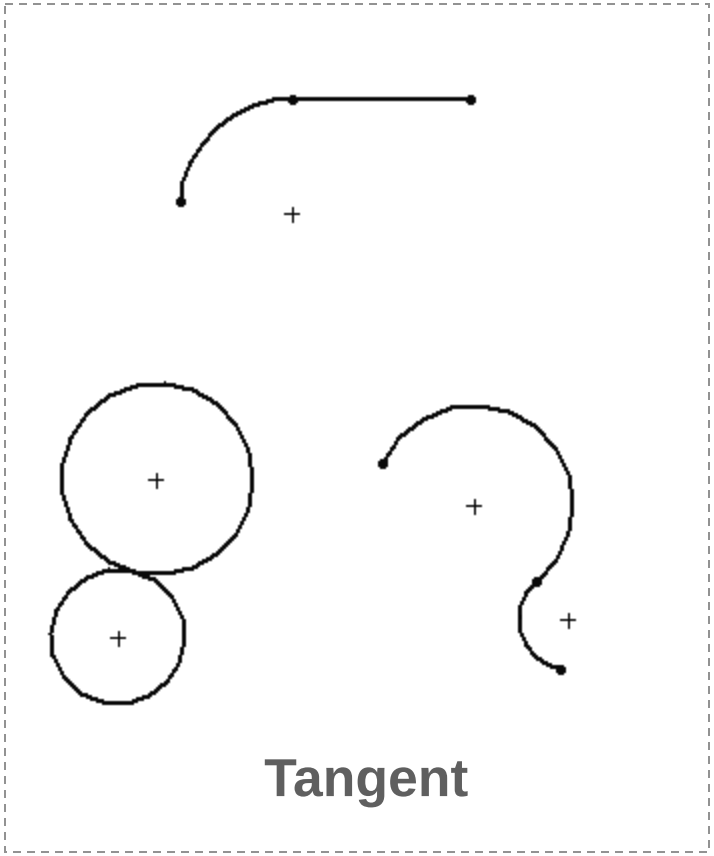
- **Face**
  The surface or "skin" of a part. Faces can be flat or curved.
- **Edge**
  The boundary of a face. Edges can be straight or curved.
- **Vertex –**
  The corner where edges meet.

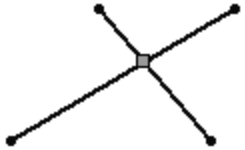


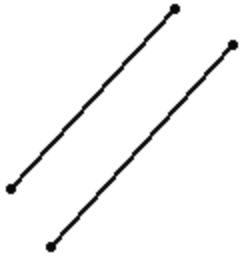
Base feature

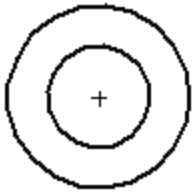
- **The Base feature is the first feature that is created.**
- **The Base feature is the foundation of the part.**
- **The Base feature geometry for the** box **is an extrusion.**
- **The extrusion is named** Extrude1**.**

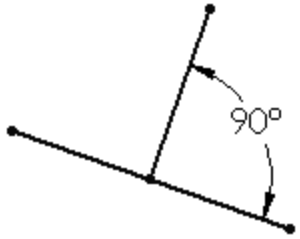
Geometric Relationships



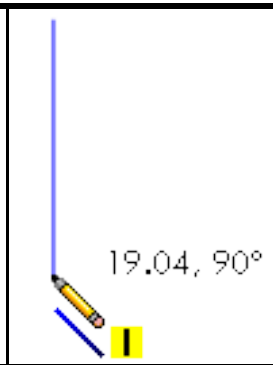
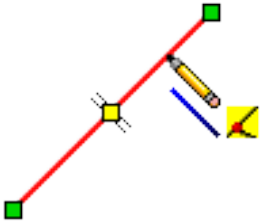
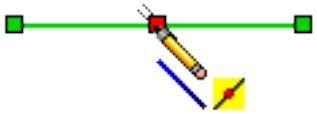
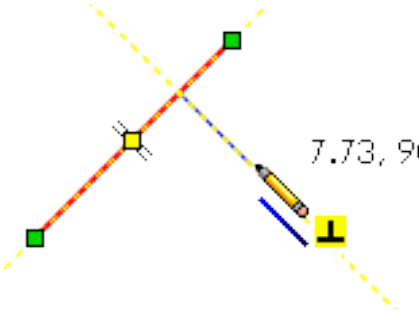
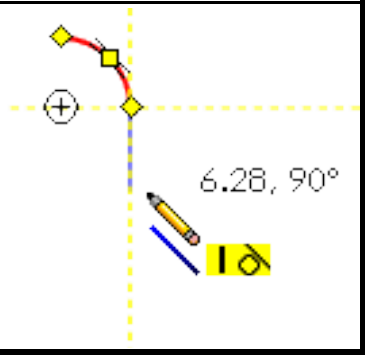






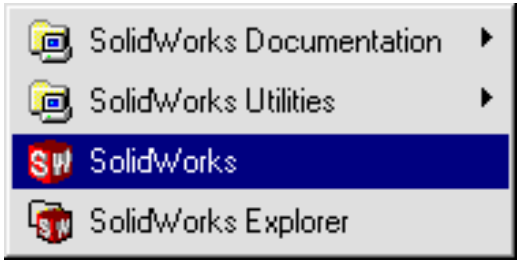


Cursor Informations

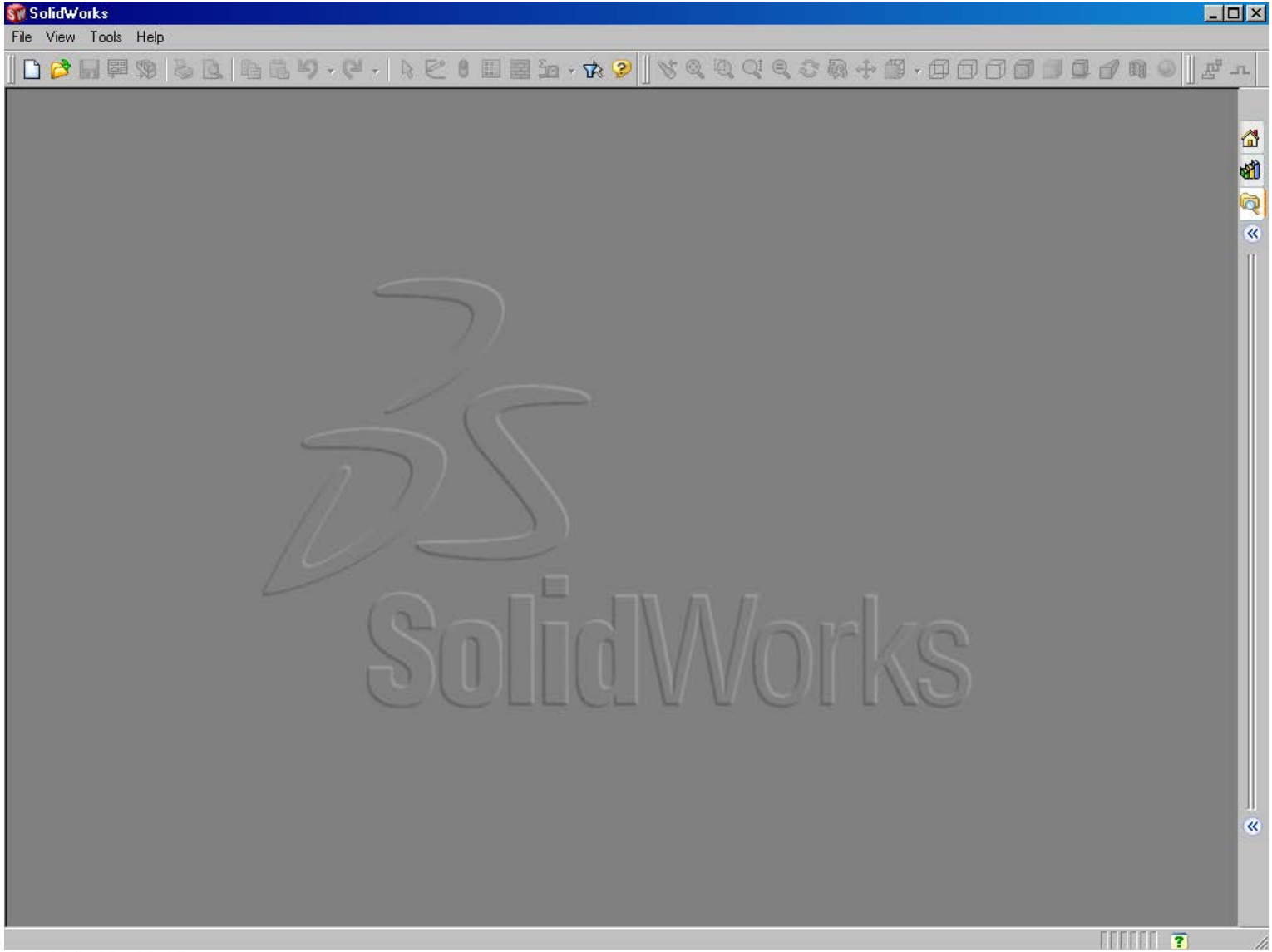
Horizontal		**Vertical**	
Coincident		**Midpoint**	
Perpendicular		**Tangent**	

To Start SolidWorks

1. **Click the Start button  on Windows task bar.**
2. **Click Programs.**
3. **Click the SolidWorks folder.**
4. **Click the SolidWorks application.**

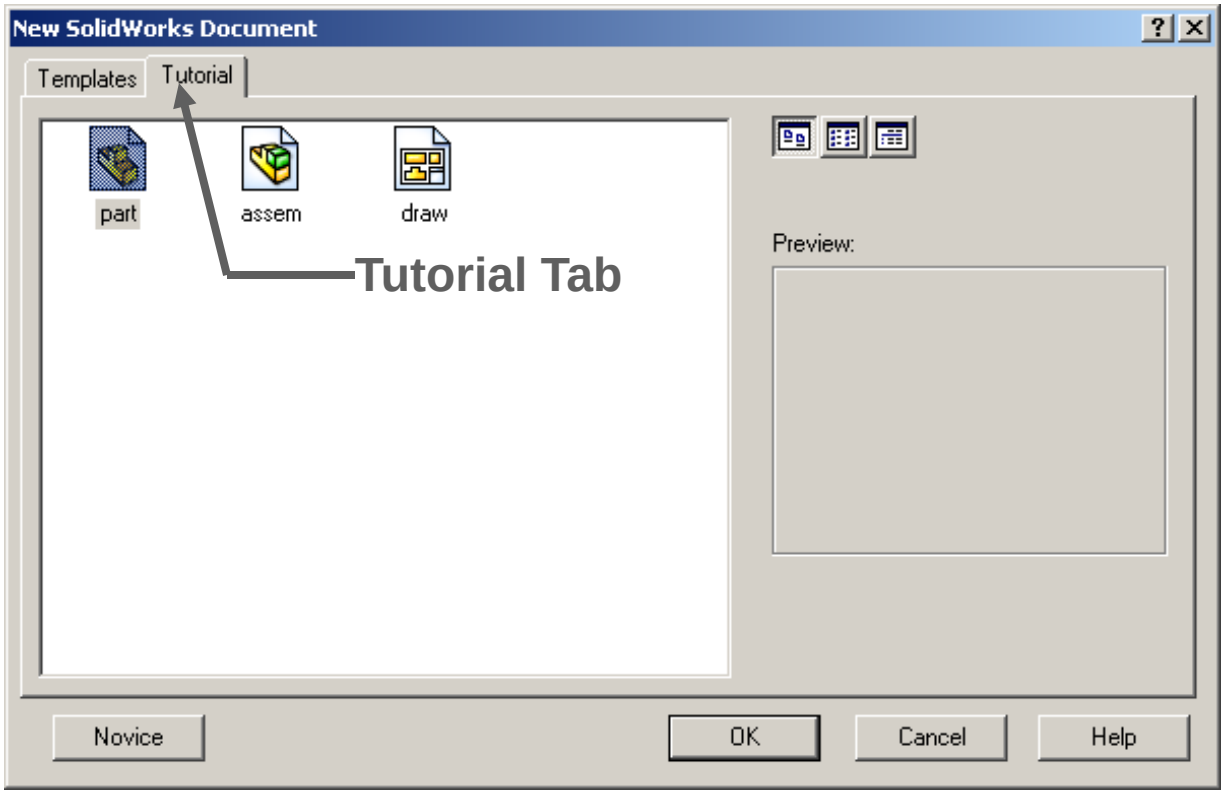


The SolidWorks Window



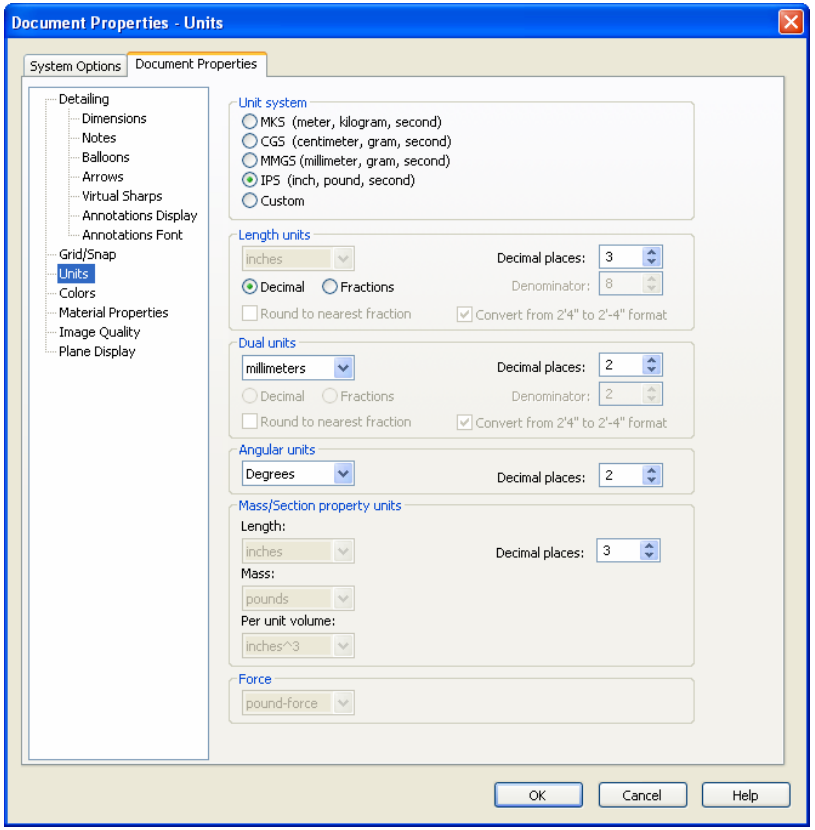
Creating New Files Using Templates

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



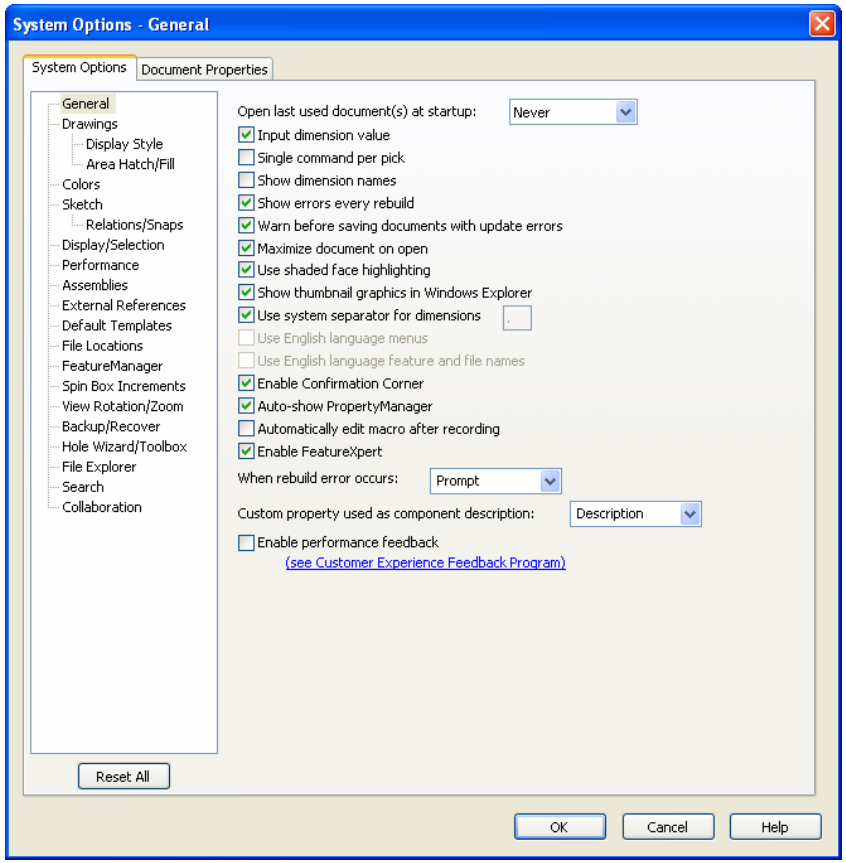
Document Properties

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



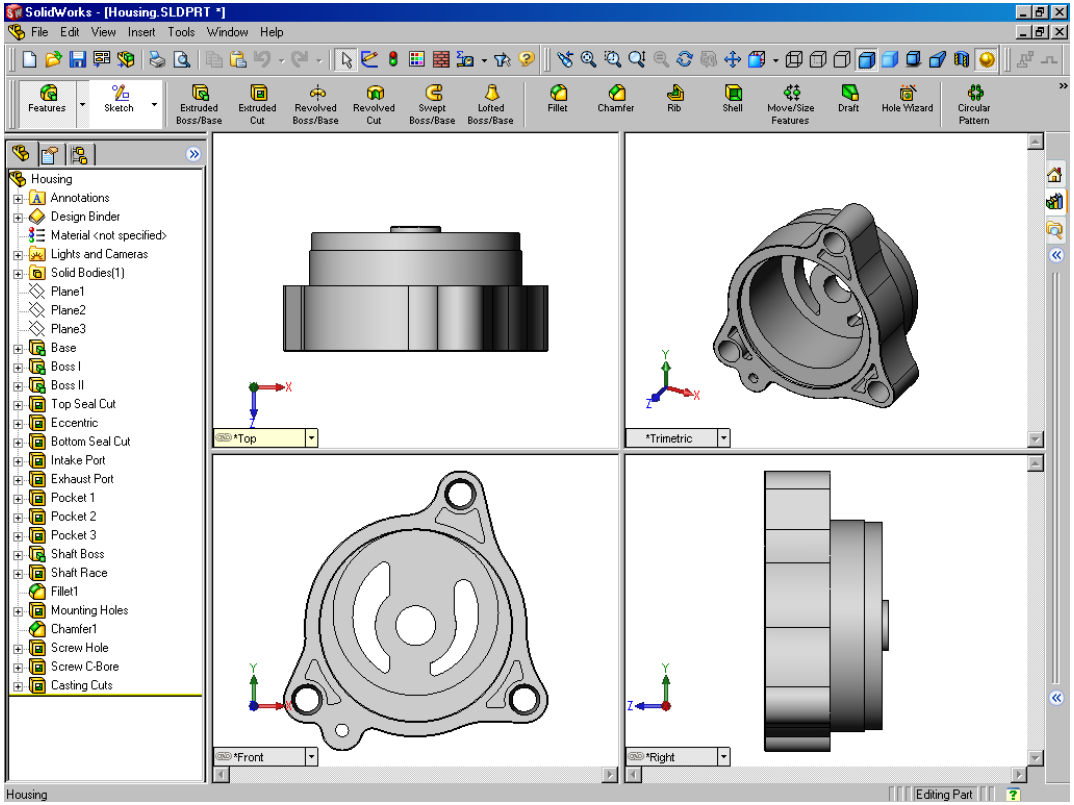
System Options

- **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



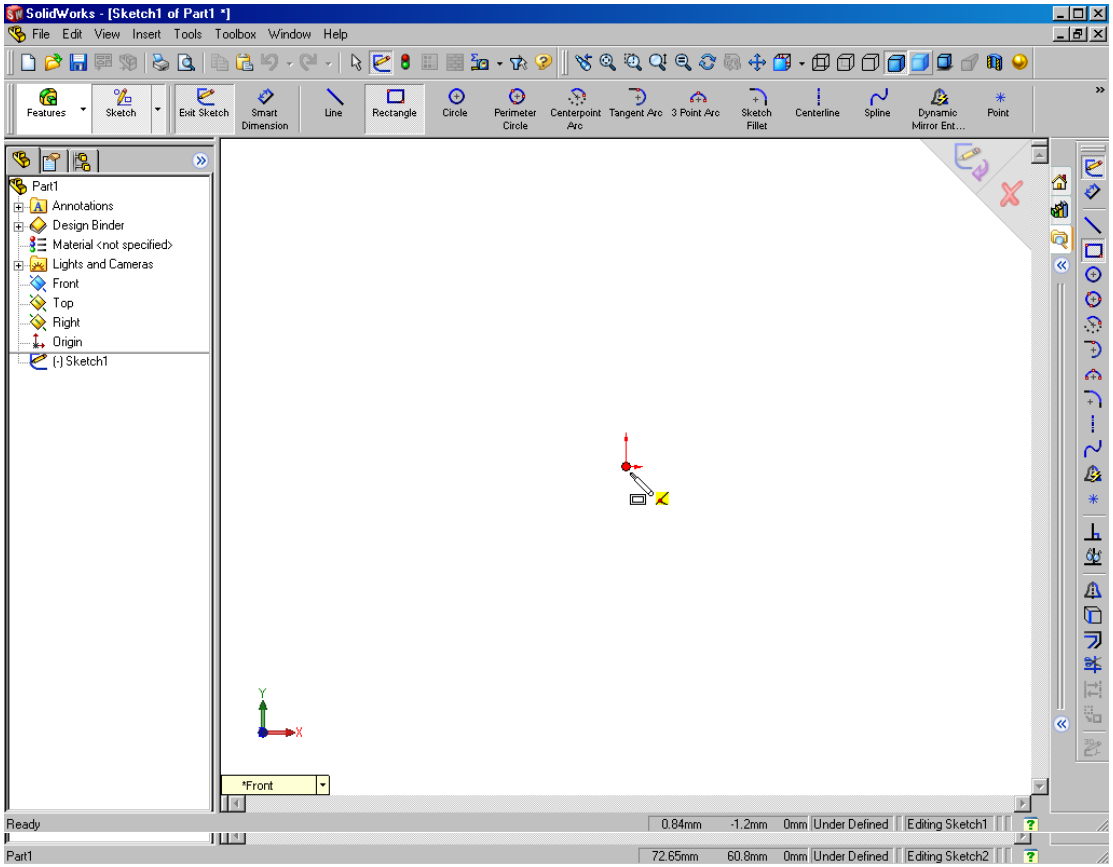
Creating a 2D Sketch

1. **Click Sketch on 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.**

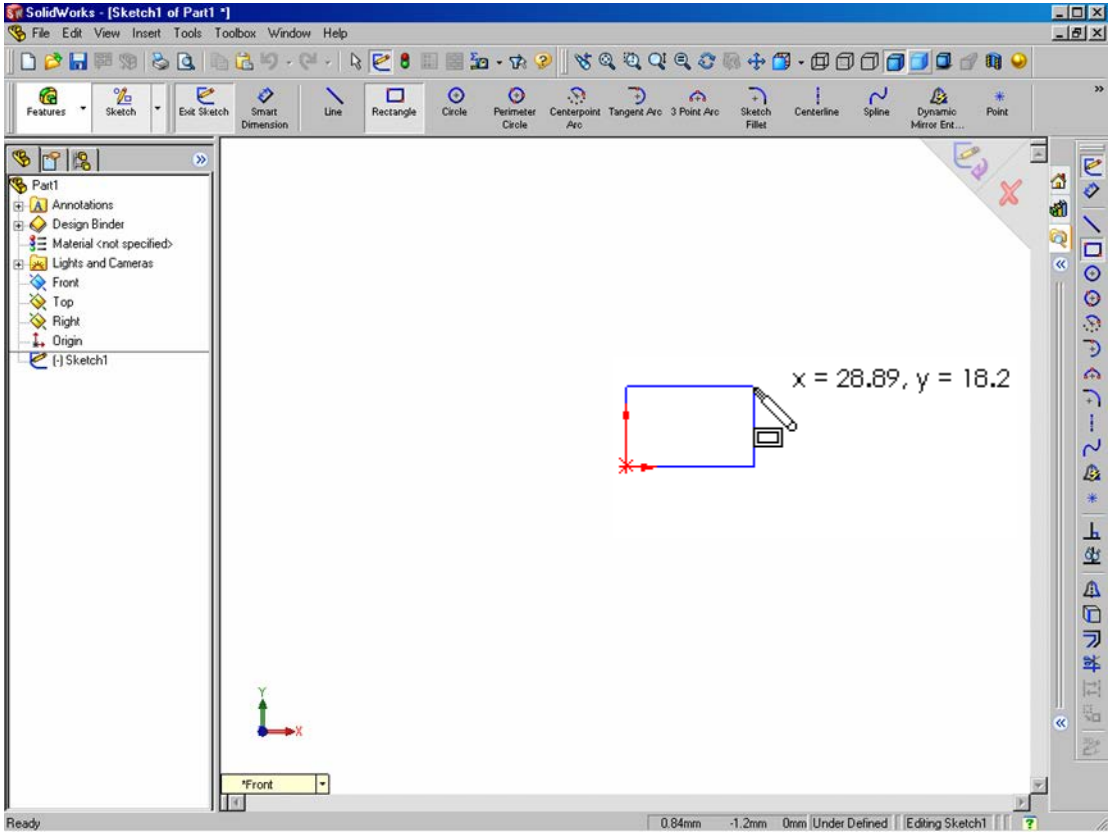


Creating a 2D Sketch

5. **Click the left mouse button.**

6. **Drag the pointer up and to the right.**

7. **Click the left mouse button again.**

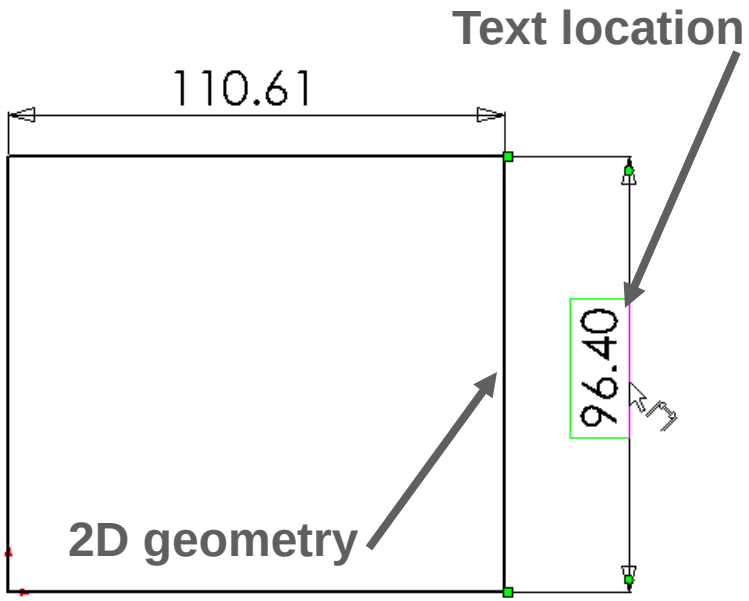


Adding Dimensions

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

To create a dimension:

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



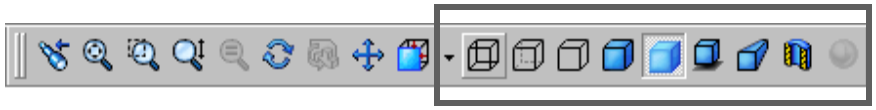
View Control

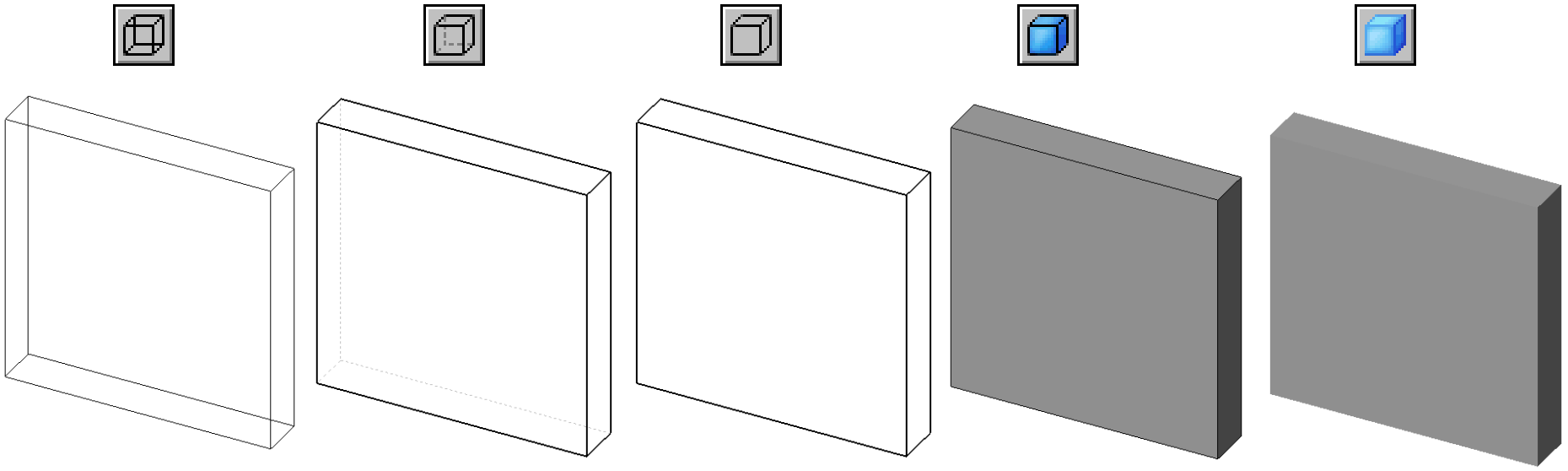
Magnify or reduce the view of a model in the graphics area.

- **Zoom to Fit – displays the part so that it fills the current window.**
- **Zoom to Area – zooms in on a portion of the view that you select by dragging a bounding box.**
- **Zoom In/Out – drag the pointer upward to zoom in. Drag the pointer downward to zoom out.**
- **Zoom to Selection – the view zooms so that the selected object fills the window.**

Display Modes

- **Illustrate the part in various display modes.**





Wireframe

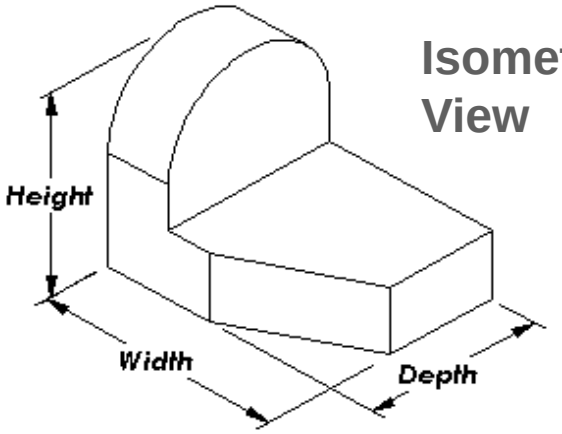
Hidden lines Visible

Hidden Lines Removed

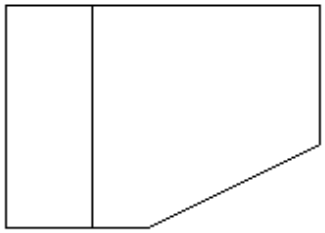
Shaded With Edges

Shaded

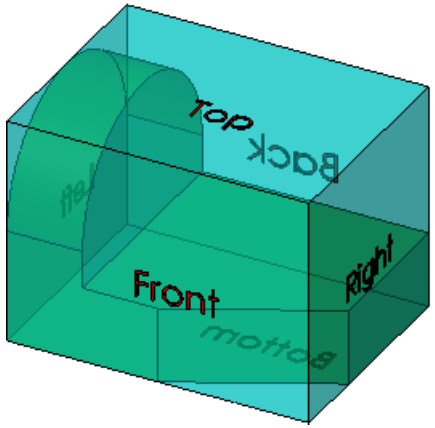
Standard Views

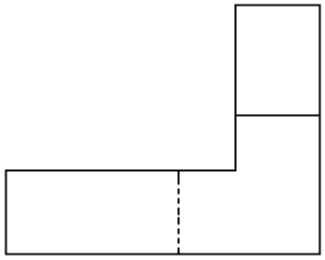


Isometric View

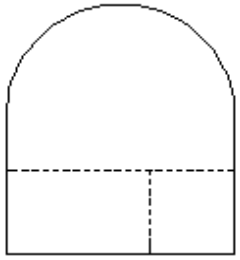


Top View

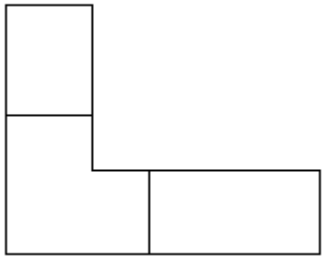




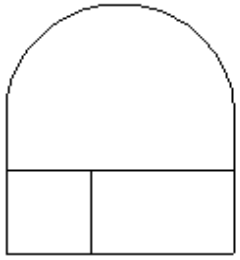
Back View



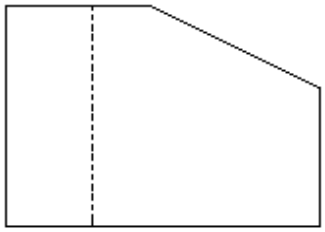
Left View



Front View



Right View



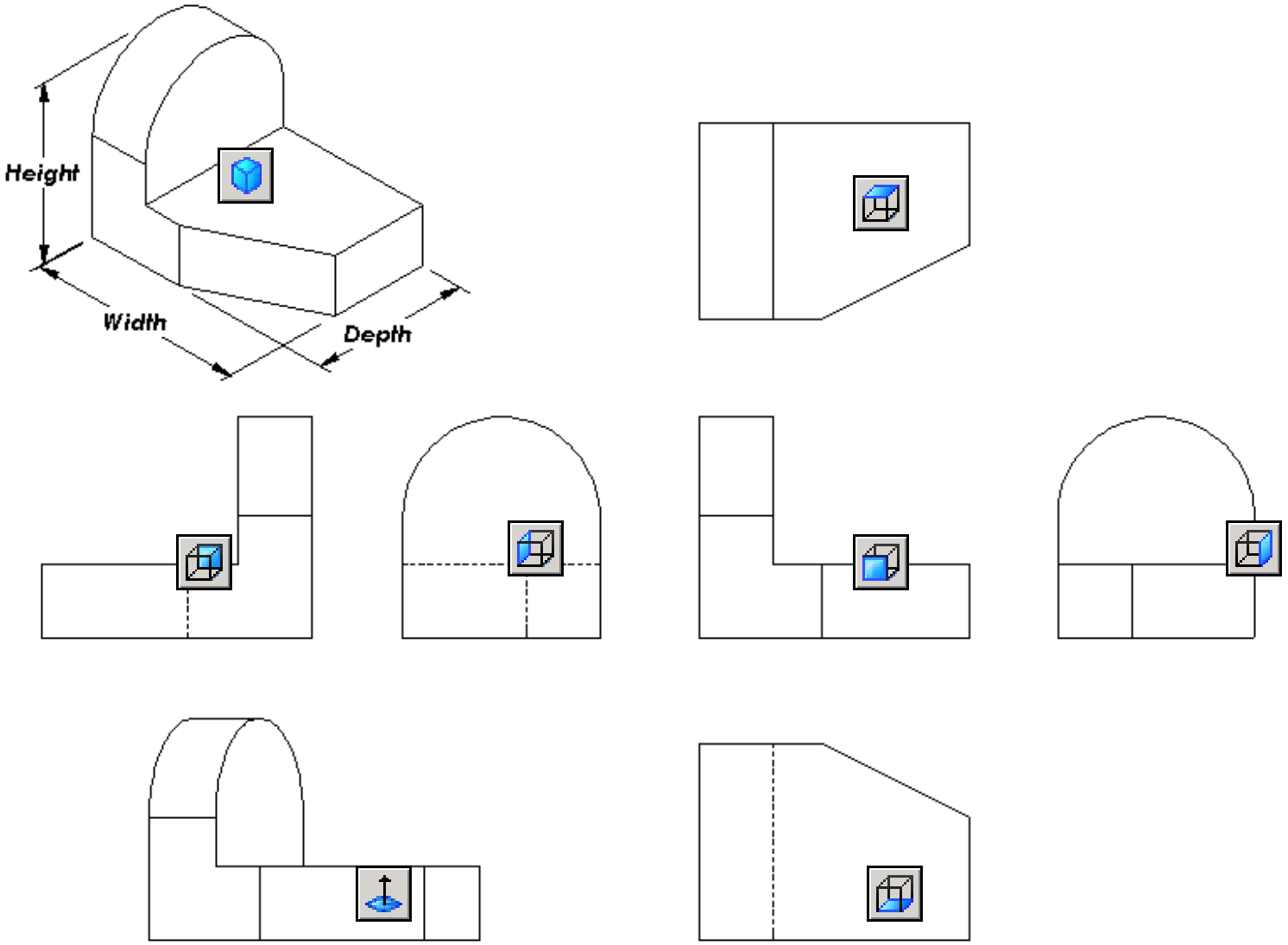
Bottom View

View Orientation

Changes the view display to correspond to one of the standard view orientations.

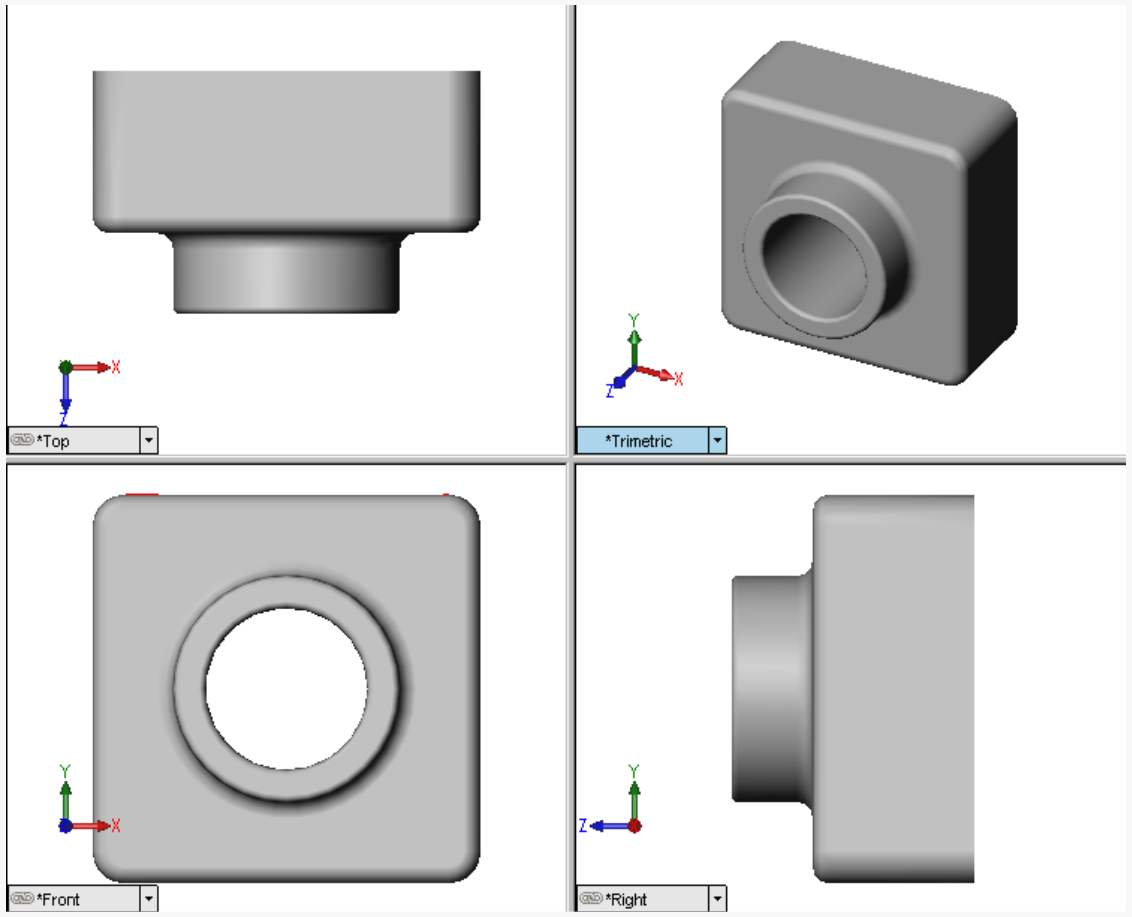
- Front
- Right
- Bottom
- Isometric
- Trimetric
- Top
- Left
- Back
- Dimetric
- Normal To (selected plane or planar face)

Standard Views



View Orientation

- **The views most commonly used to describe a part are:**
  - **Top View**
  - **Front View**
  - **Right View**
  - **Isometric View**

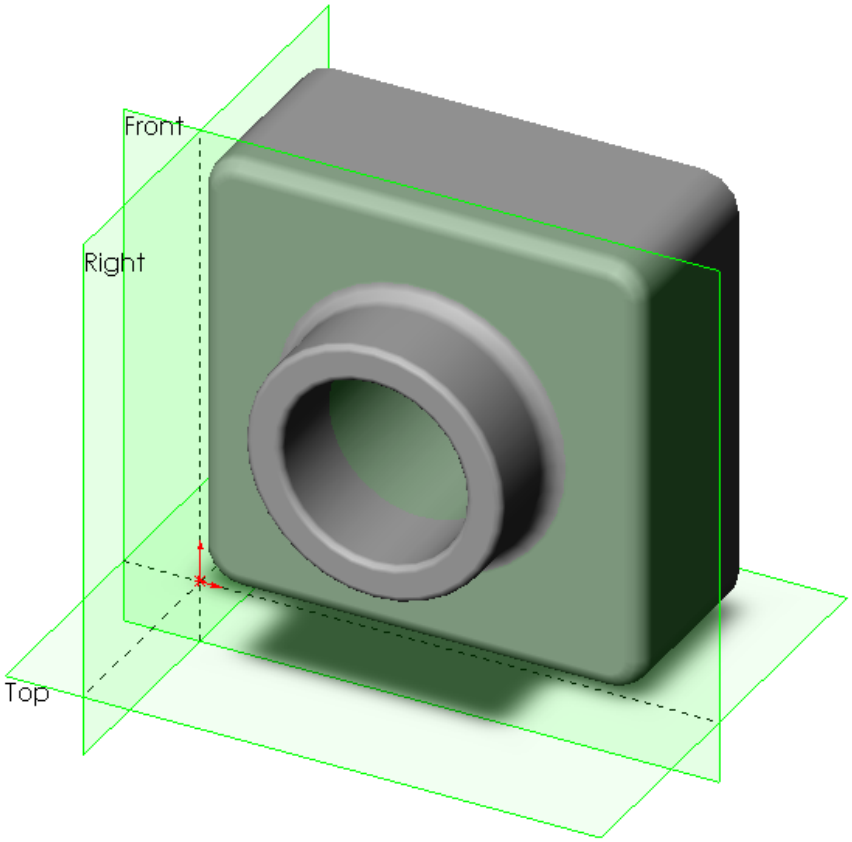


Default Planes

- **Default Planes**
  - Front, Top, **and** Right

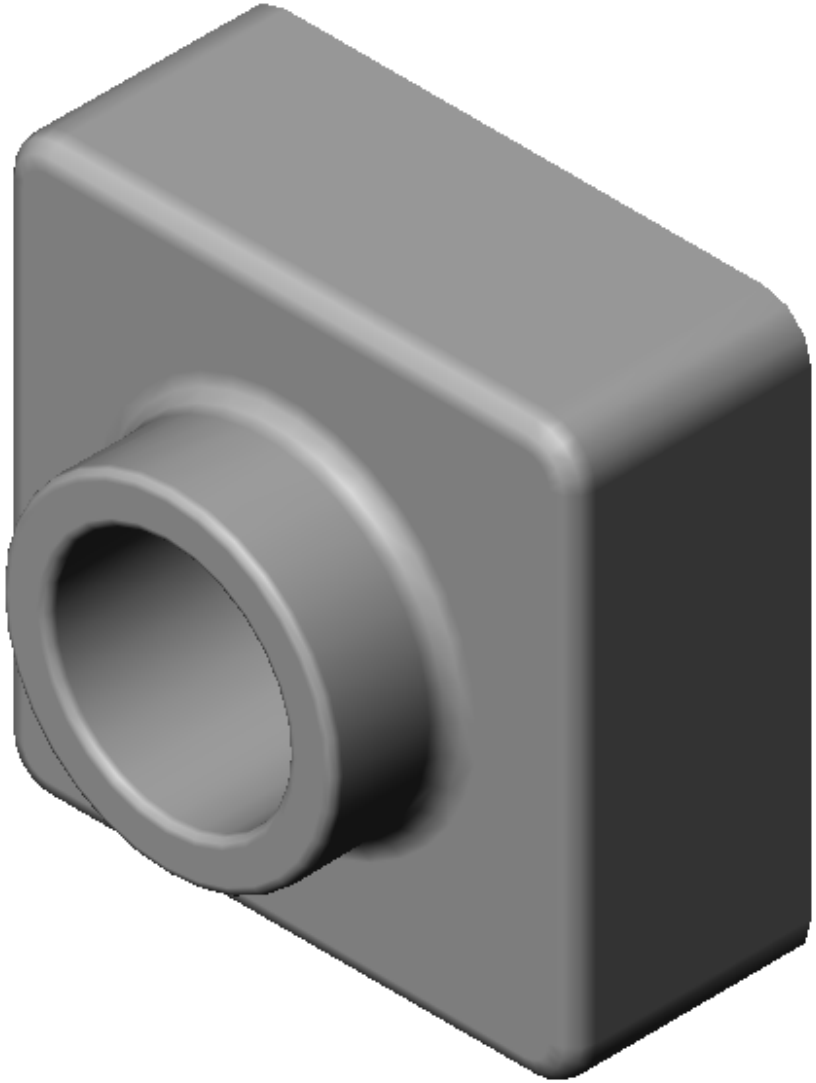
Correspond to the standard principle drawing views:

- Front **= Front or Back view**
- Top **= Top or Bottom view**
- Right **= Right or Left view**



Isometric View

- **Displays the part with height, width, and depth equally foreshortened.**
  - **Pictorial rather than orthographic.**
  - **Shows all three dimensions – height, width, and depth.**
  - **Easier to visualize than orthographic views.**



Determining the Weight

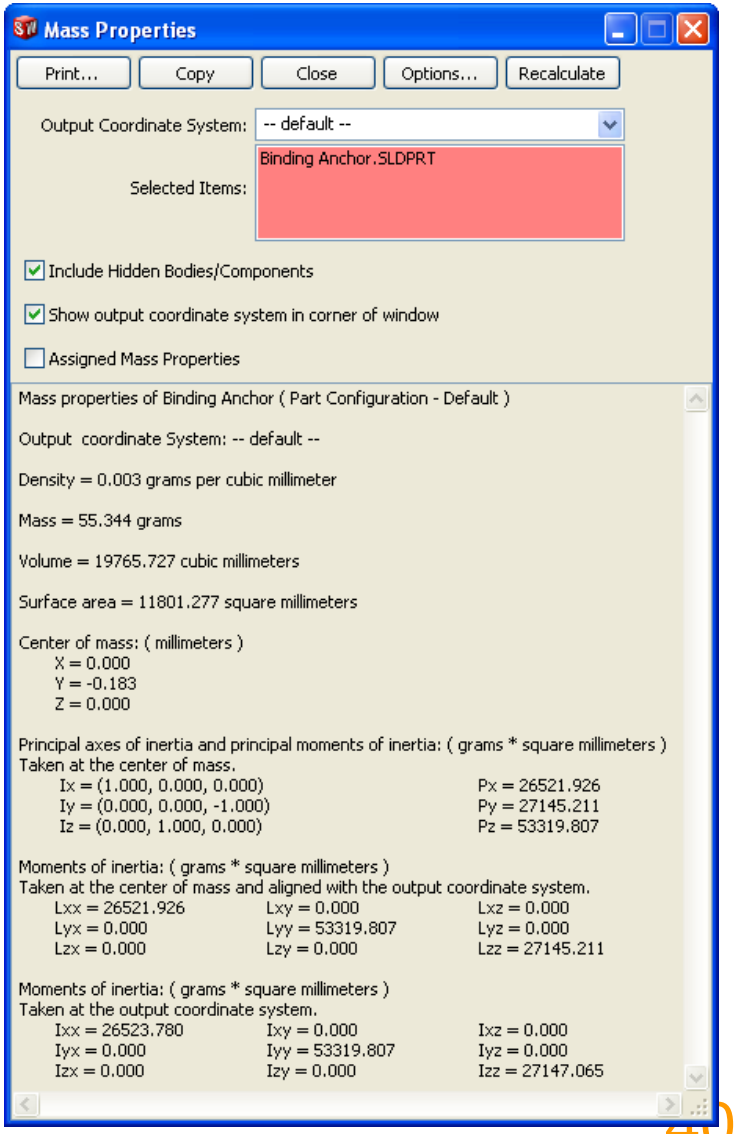
Weight = Volume X Density

- **Volume can be calculated from the geometry of the model**
- **Density can be obtained from handbooks, data sheets or online**

Mass Properties

The Mass Properties tool can calculate:

- **Volume**
- **Mass**
- **Surface Area**
- **Center of Mass**
- **Moments of Inertia**



Submission requirements:

- Stop Drawing at 5:15pm and start submission.
- File name should include the following information:
- a) Session Code like 1cad / 2cad / 3cad / examcad
- b) Roll Number
- c) Problem Number
- d) Name
- Example: (use lowercase letters)
  2cad12ae10099prob1anupghosh.SLDPRT4
- Collect your work and delete it from the computer.