

# Solidworks Lesson 2 – Basic Part Features



University of Pittsburgh  
MEMS 0024

# The SolidWorks Model

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- **The SolidWorks model is made up of:**



**Parts**



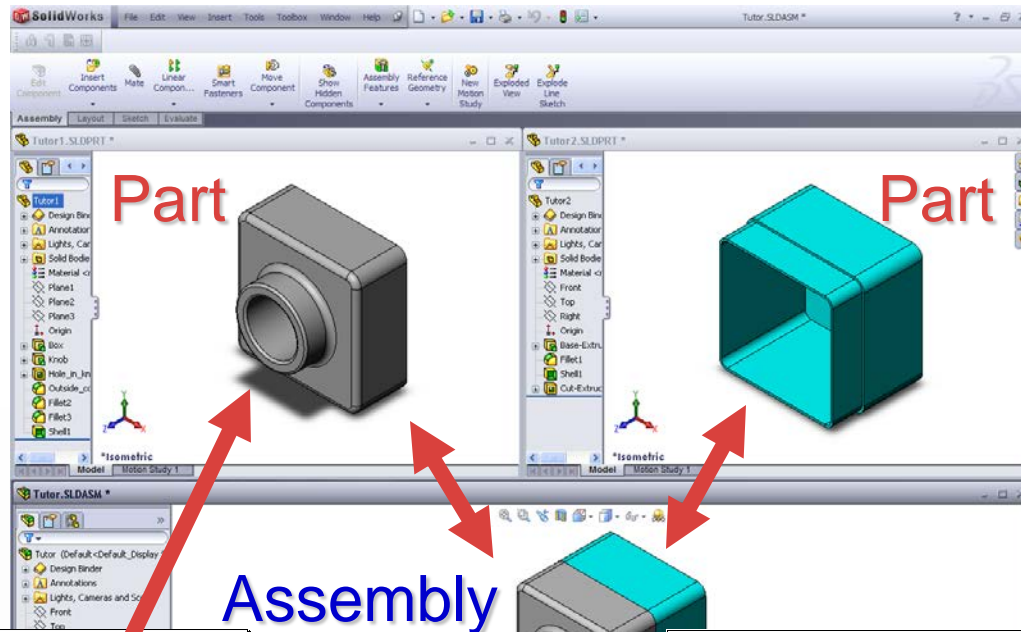
**Assemblies**



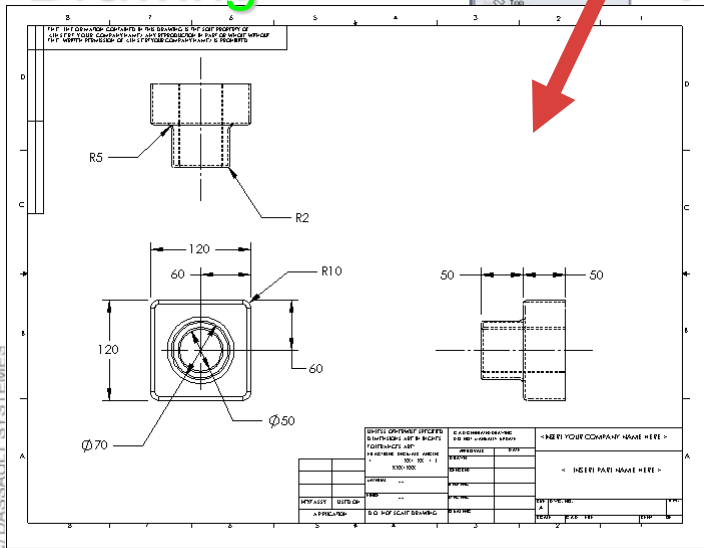
**Drawings**



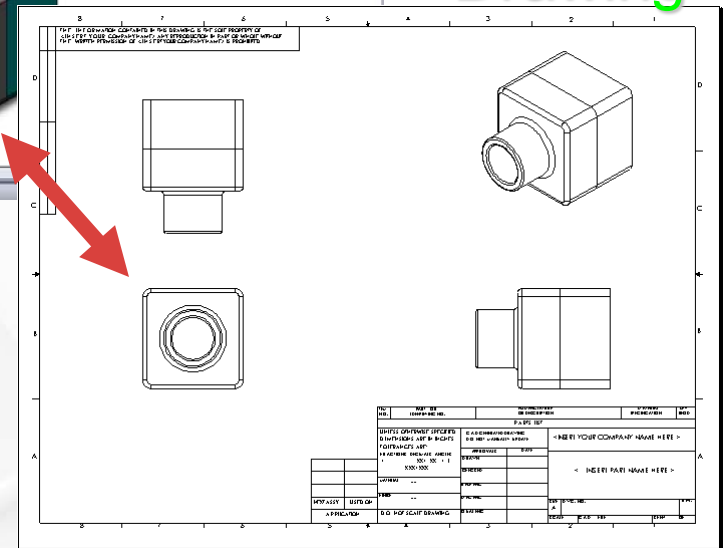
# The SolidWorks Model



Drawing

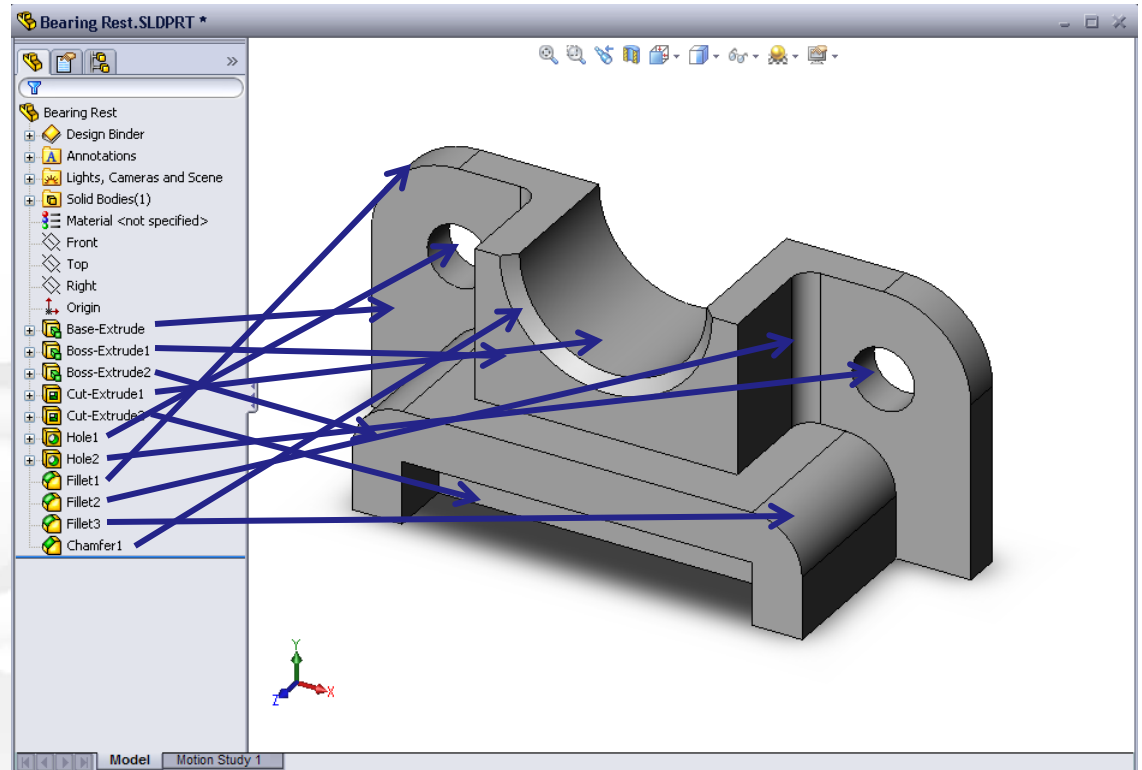


Drawing



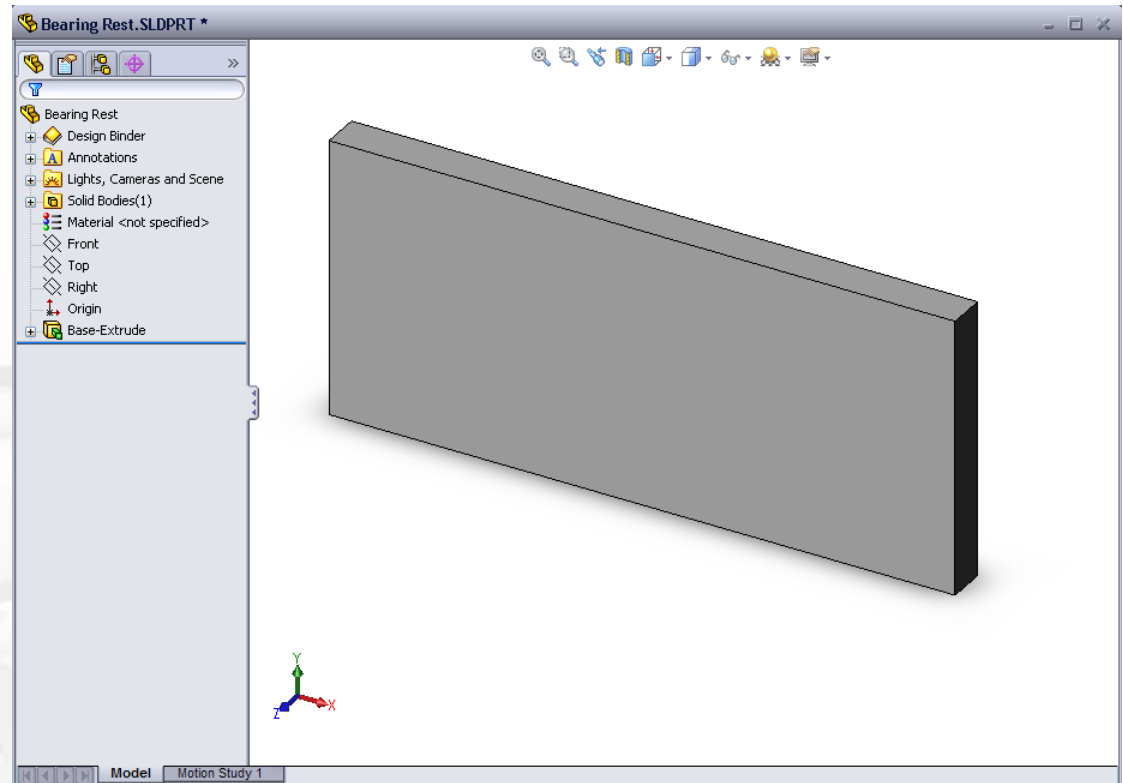
# Features

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



# Examples of Shape Features

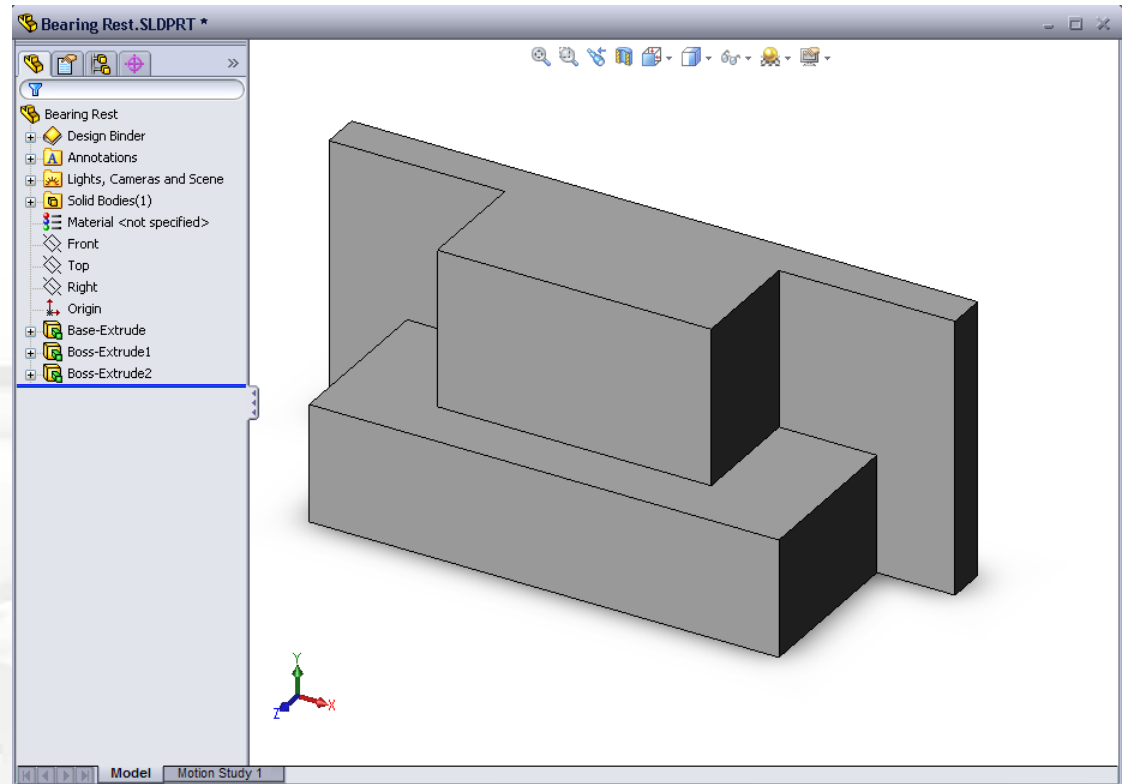
- **Base Feature**
  - First feature in part.
  - Created from a 2D sketch.
  - Forms the work piece to which other features are added.



# Examples of Shape Features

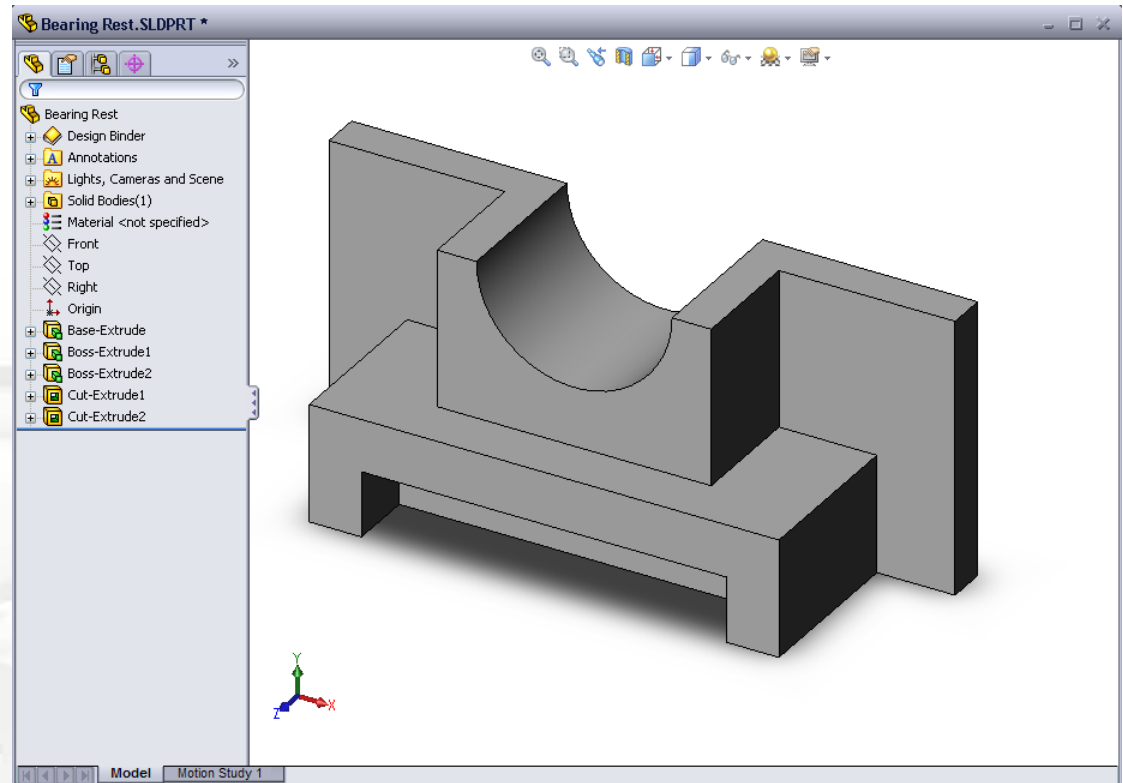
- **Boss feature**

- Adds material to part.
- Created from 2D sketch.



# Examples of Shape Features

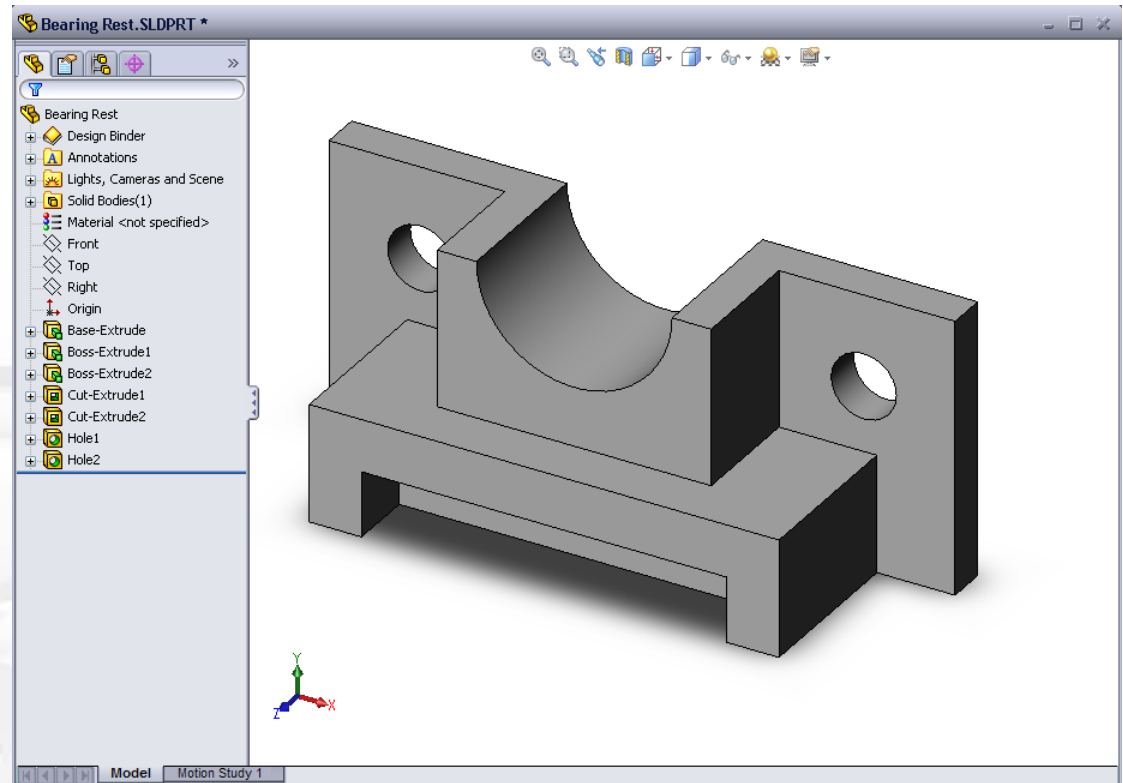
- **Cut feature**
  - Removes material from part.
  - Created from 2D sketch.



# Examples of Shape Features

- **Hole feature**

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

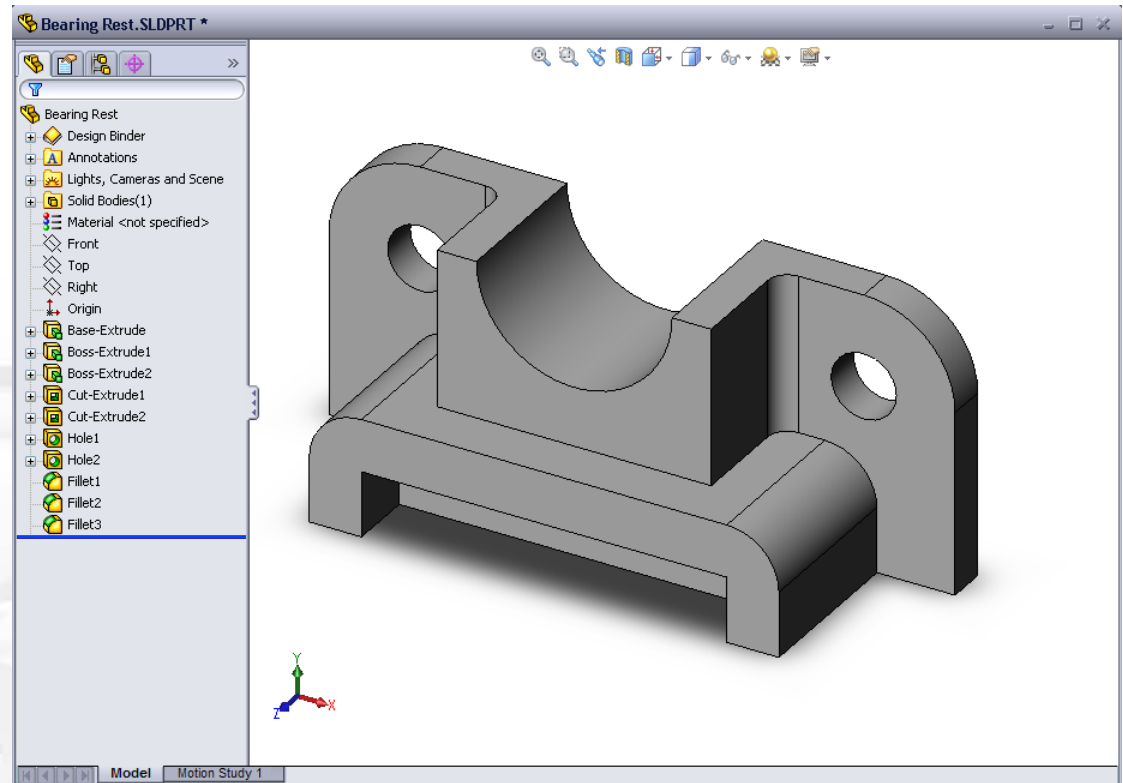




# Examples of Shape Features

## ● Fillet feature

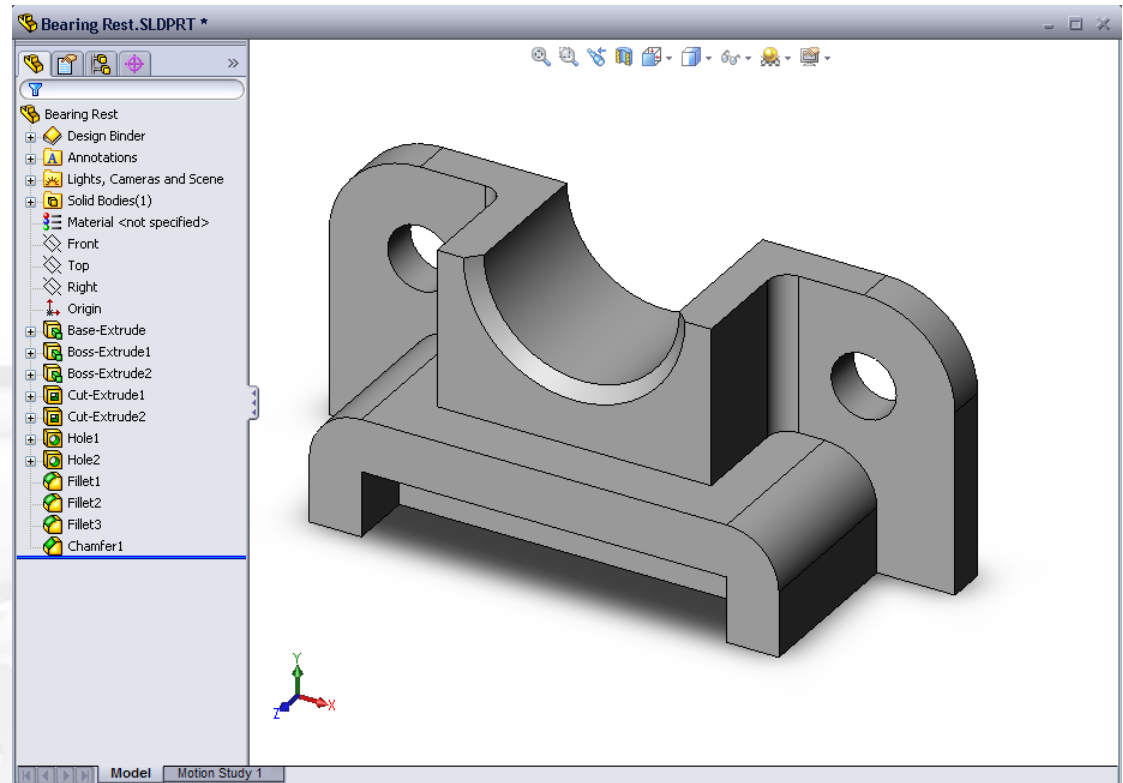
- Used to round off sharp edges.
- Can remove or add material.
  - Outside edge (convex fillet) removes material.
  - Inside edge (concave fillet) adds material.



# 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

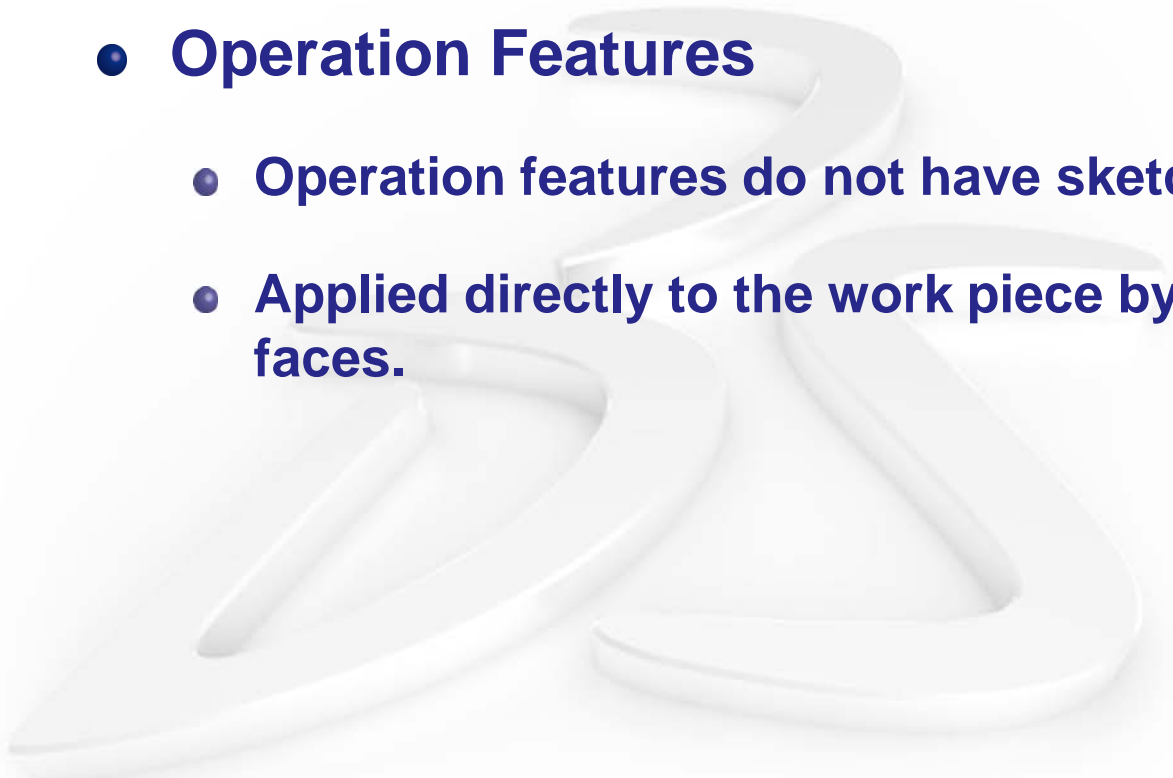
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- **Sketched Features**

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

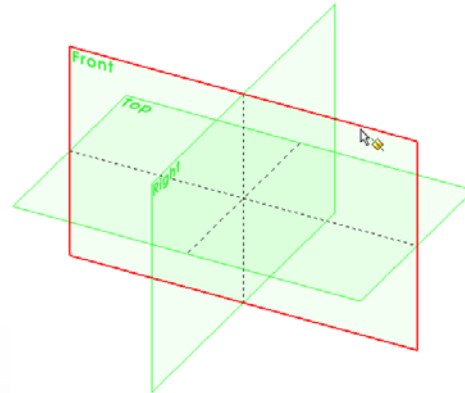
- **Operation Features**

- 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.



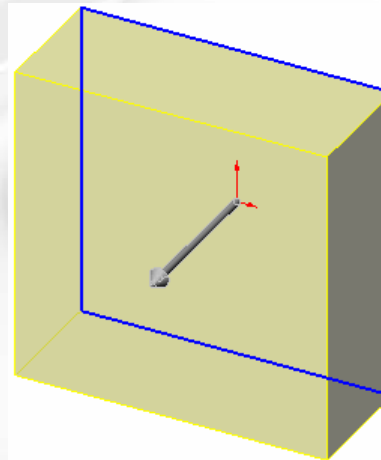
Select the sketch plane

2. Sketch a 2D profile.

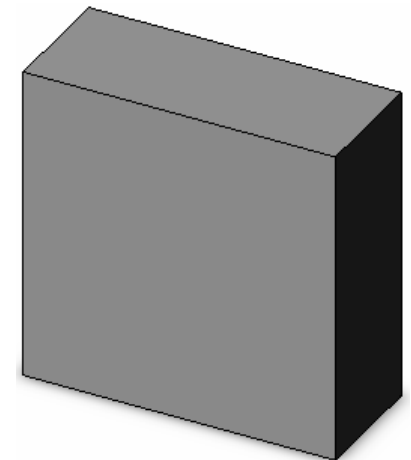


Sketch the 2D profile

3. Extrude the sketch perpendicular to sketch plane.



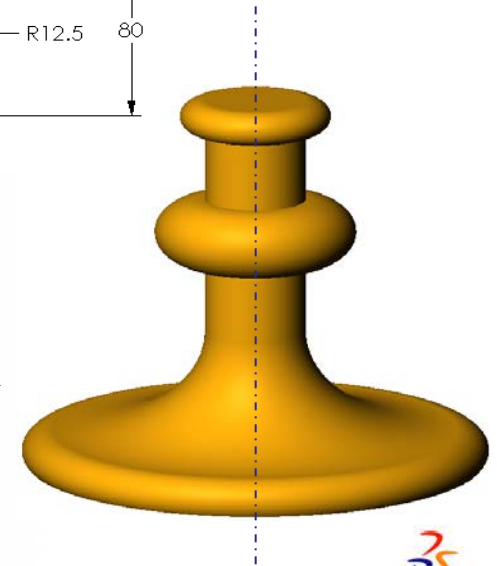
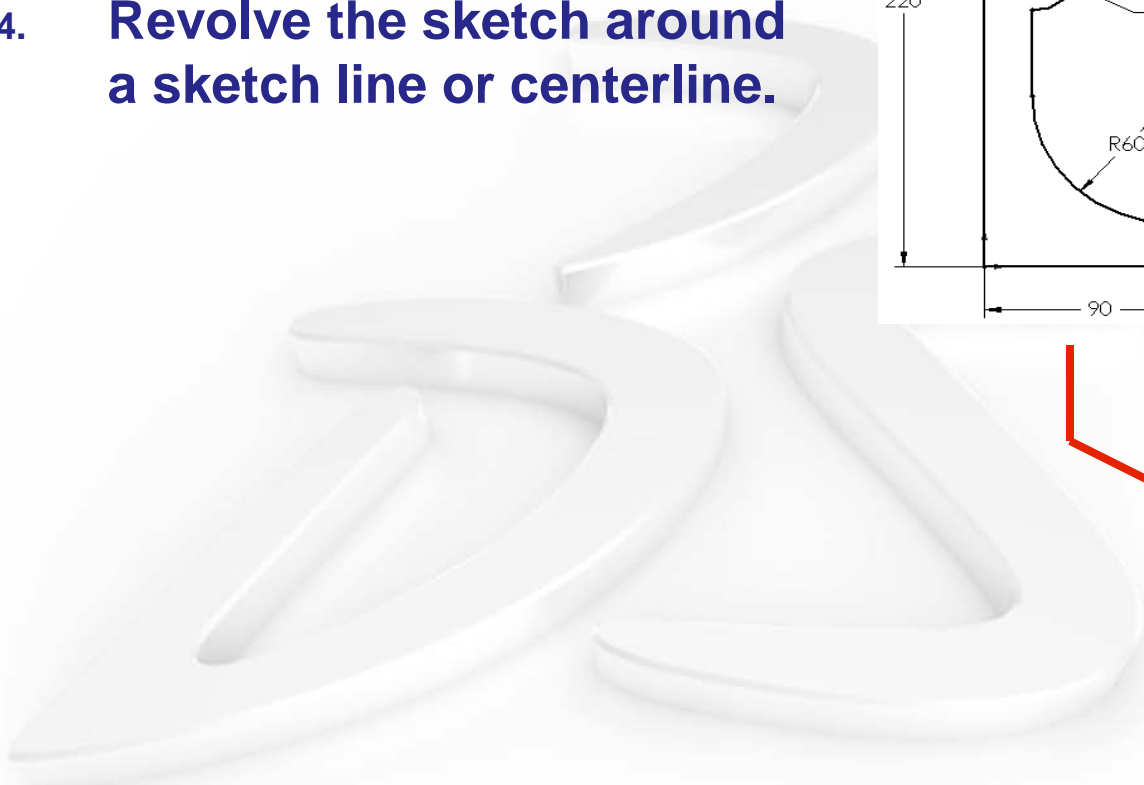
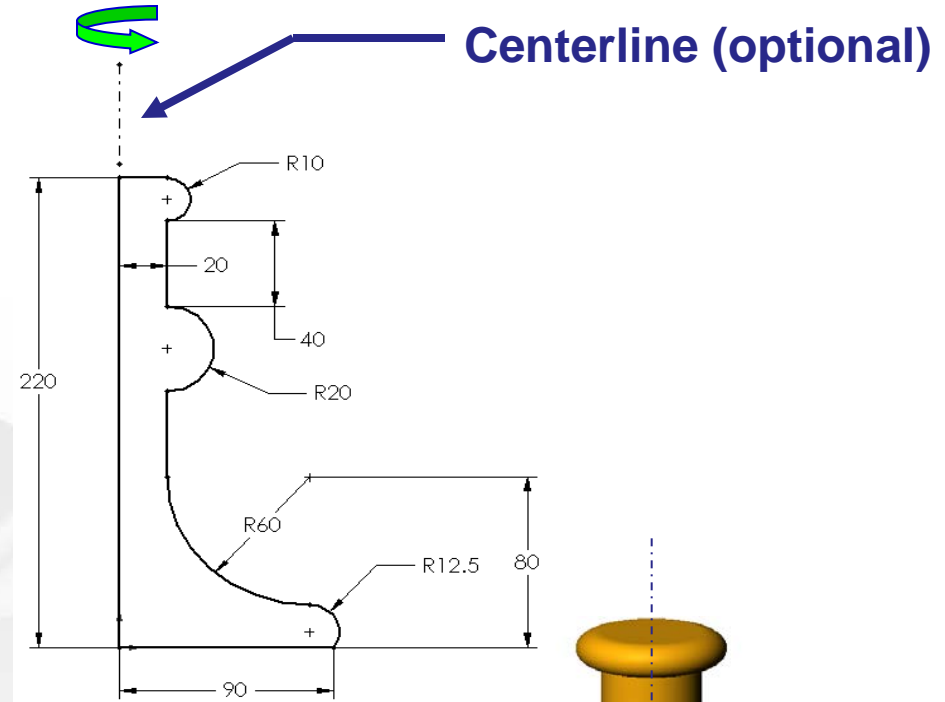
Extrude the sketch



Resulting base feature

# To Create a Revolved Base Feature:

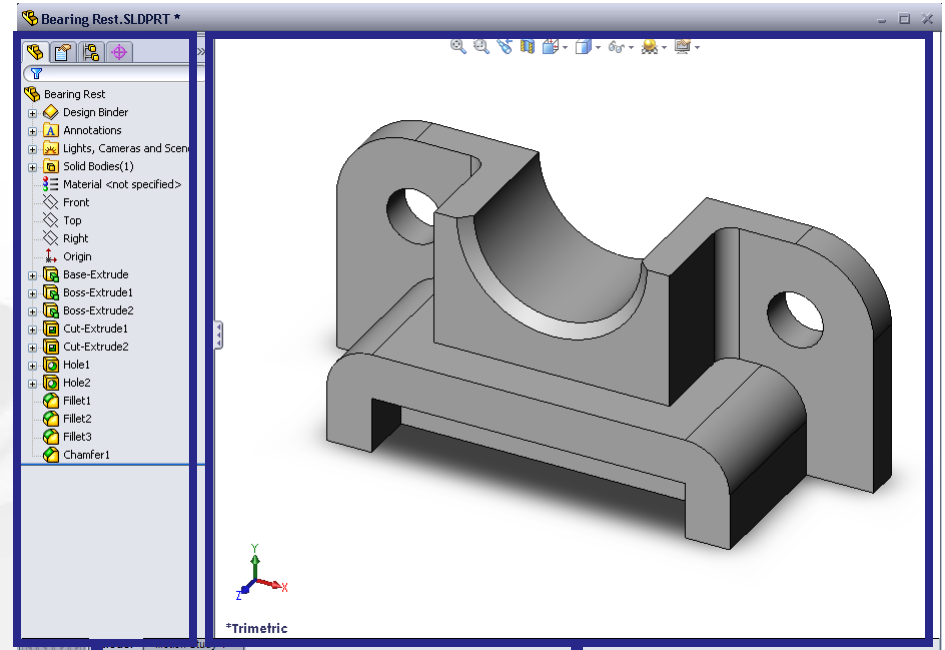
1. **Select a sketch plane.**
2. **Sketch a 2D profile.**
3. **Sketch a centerline (optional).**
4. **Revolve the sketch around a sketch line or centerline.**



# Terminology: Document Window

- Divided into two panels:

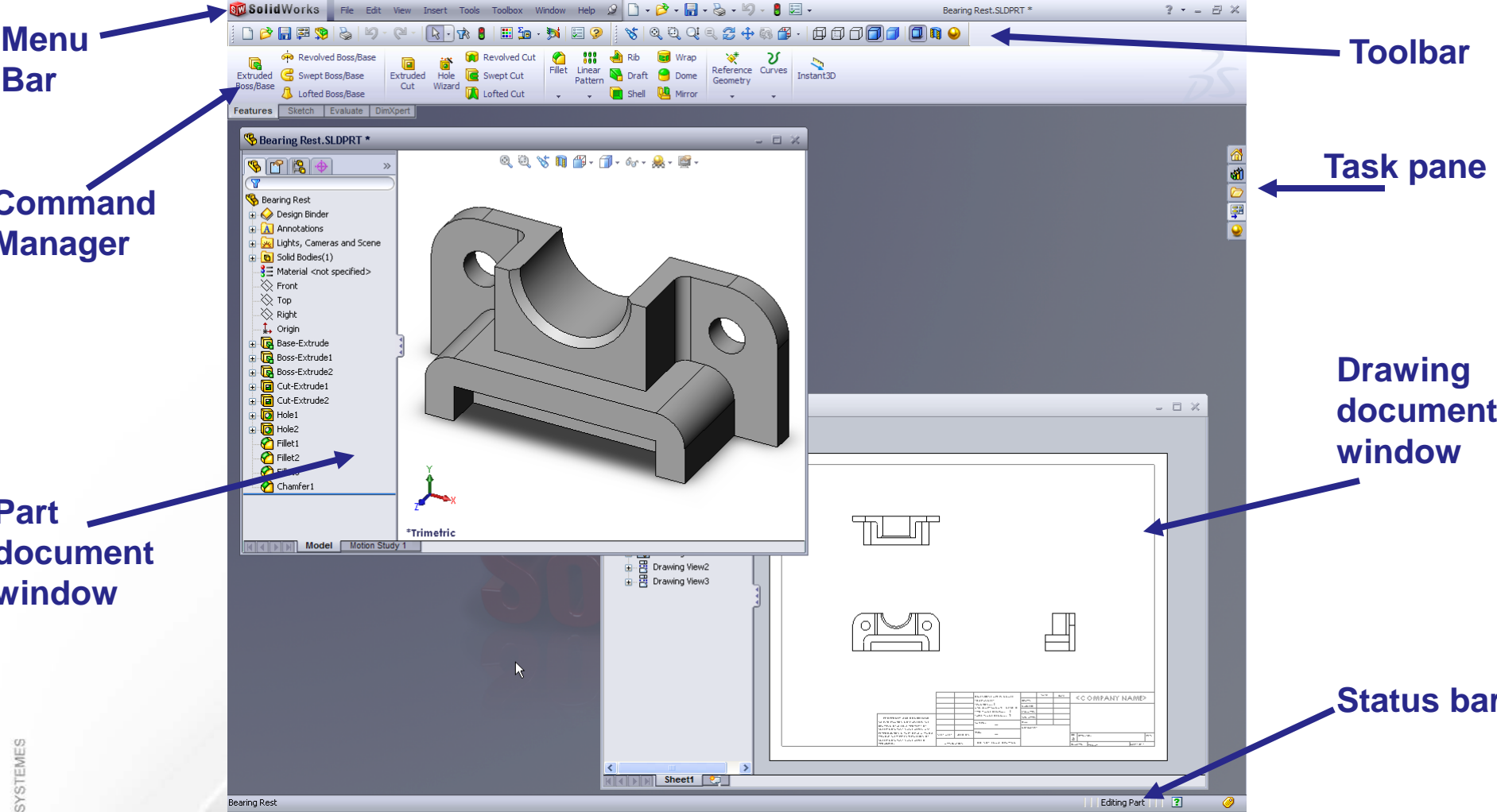
- Left panel contains the **FeatureManager®** design tree.
  - Lists the structure of the part, assembly or drawing.
- Right panel contains the **Graphics Area**.
  - Location to display, create, and modify a part, assembly or drawing.



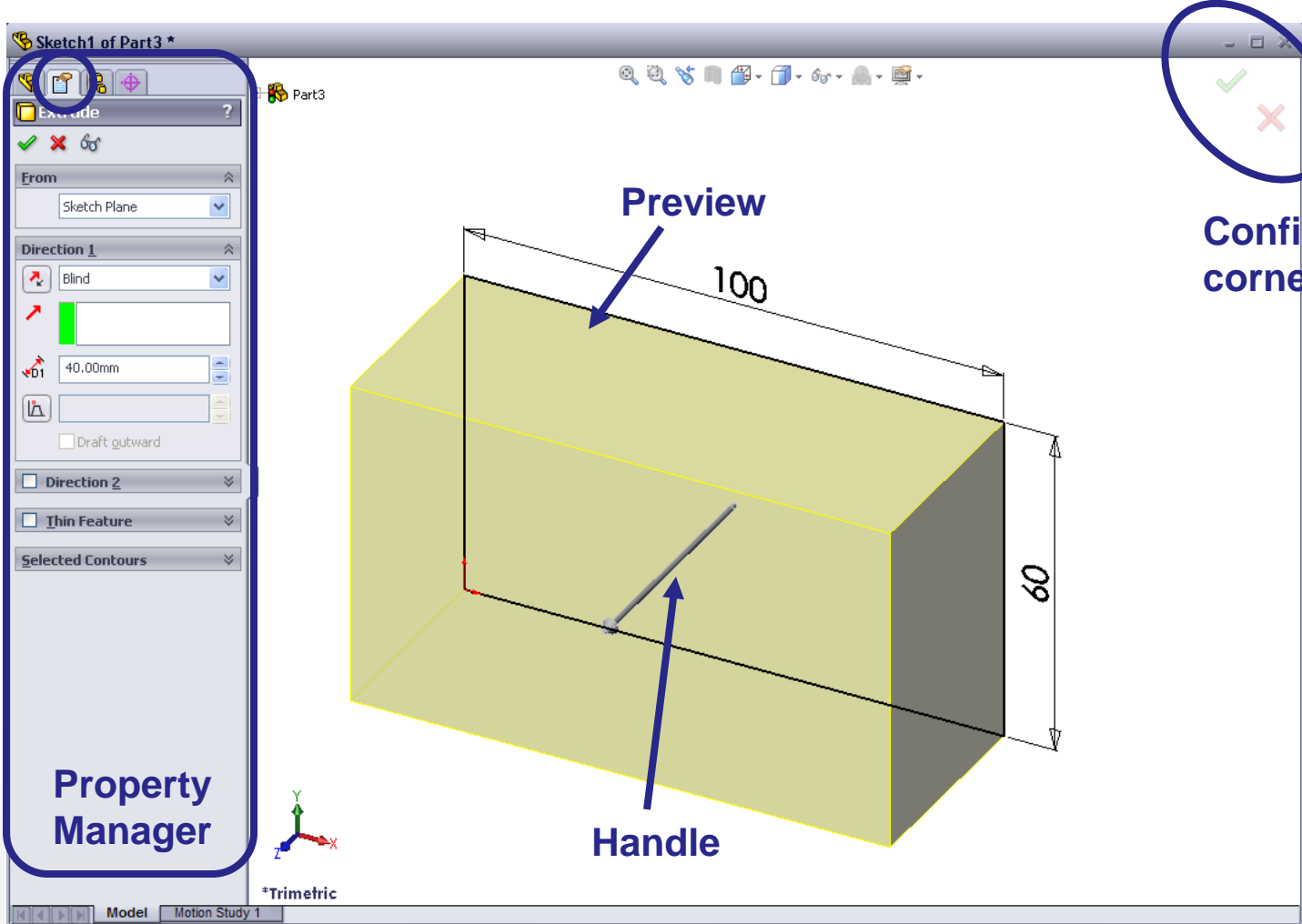
↓  
**FeatureManager  
design tree**

↓  
**Graphics Area**

# Terminology: User Interface



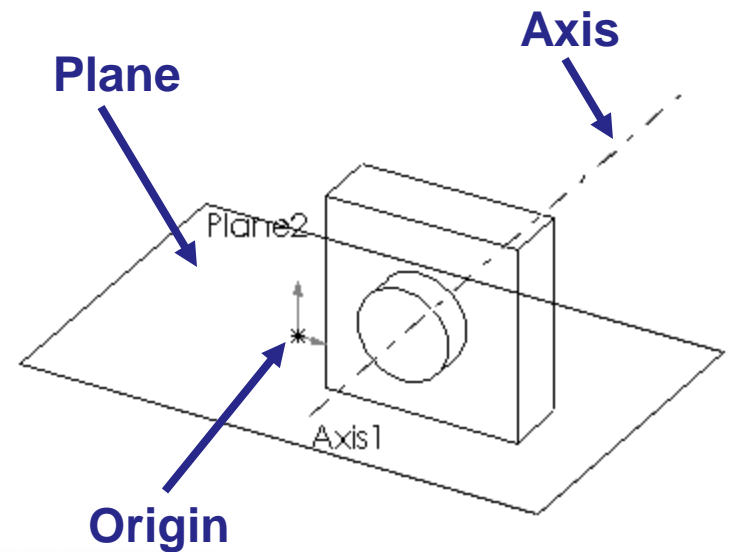
# Terminology: PropertyManager








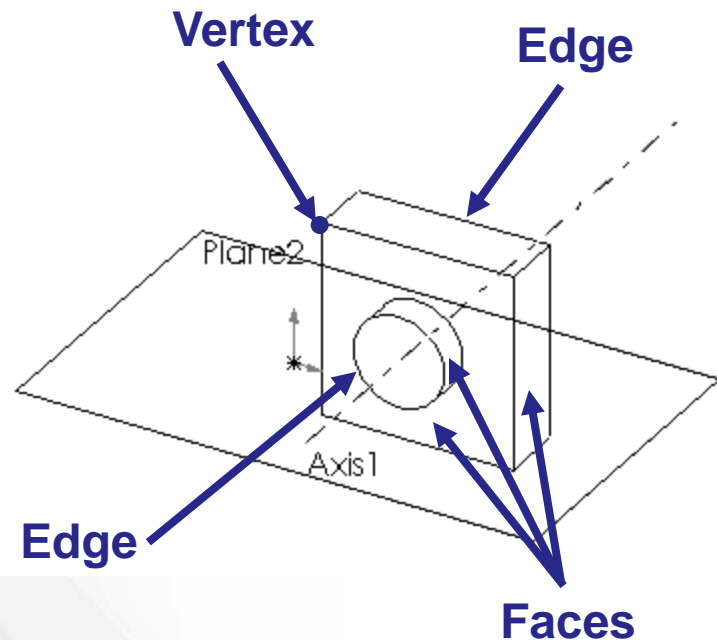
# Terminology: Basic Geometry

- **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.

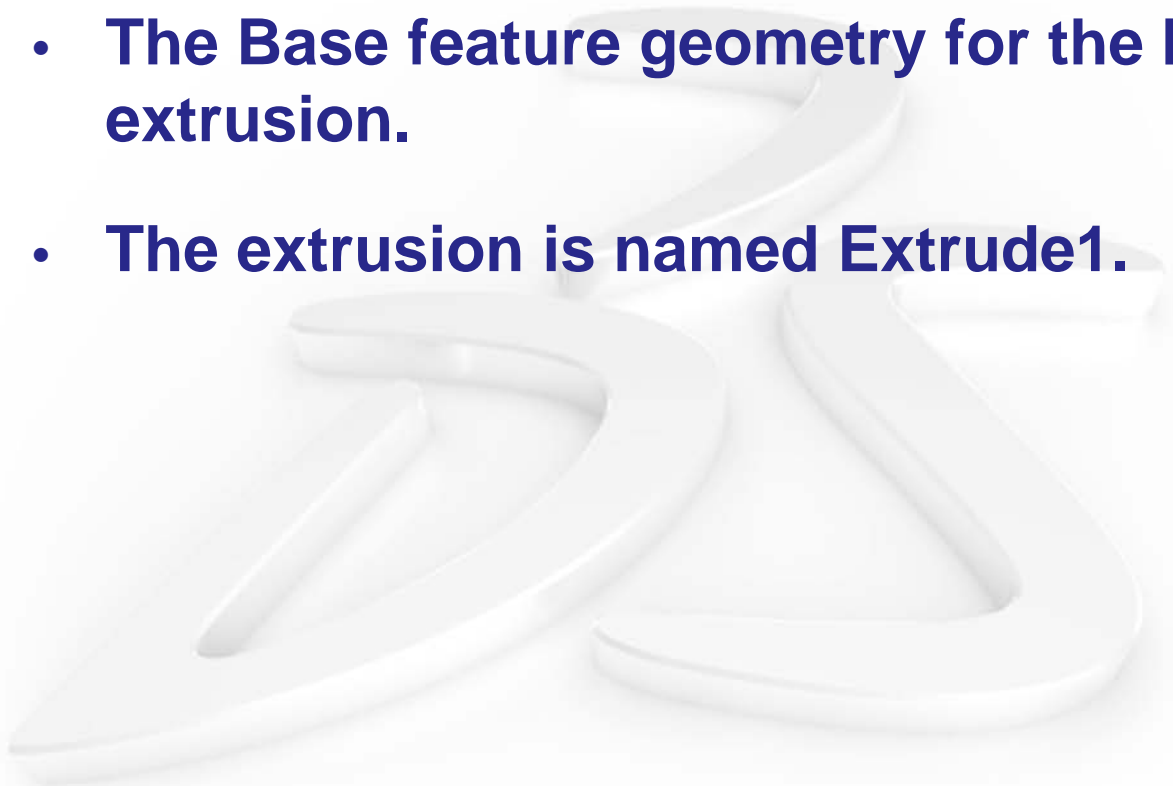


# Features and Commands

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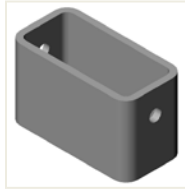
## Base feature

- **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.**



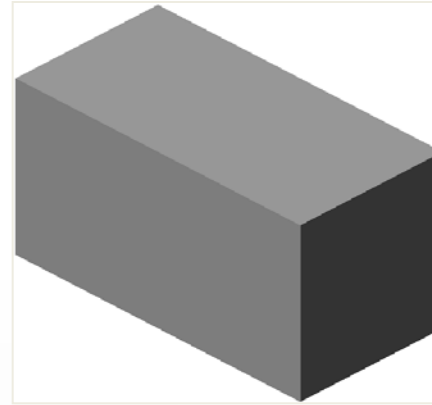
# Features and Commands

Example:

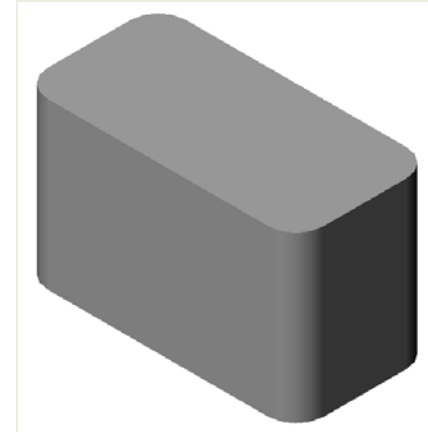


Features used to build the *box* are:

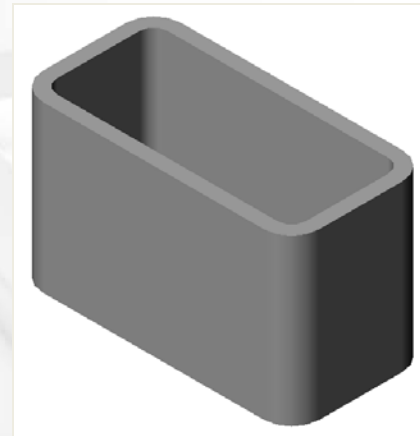
- Extruded Base feature
- Fillet feature
- Shell feature
- Extruded Cut feature



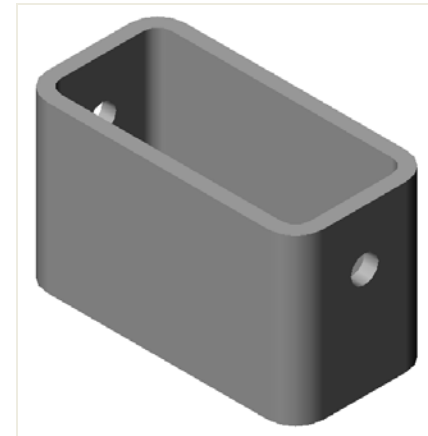
1. Base Feature



2. Fillet Feature





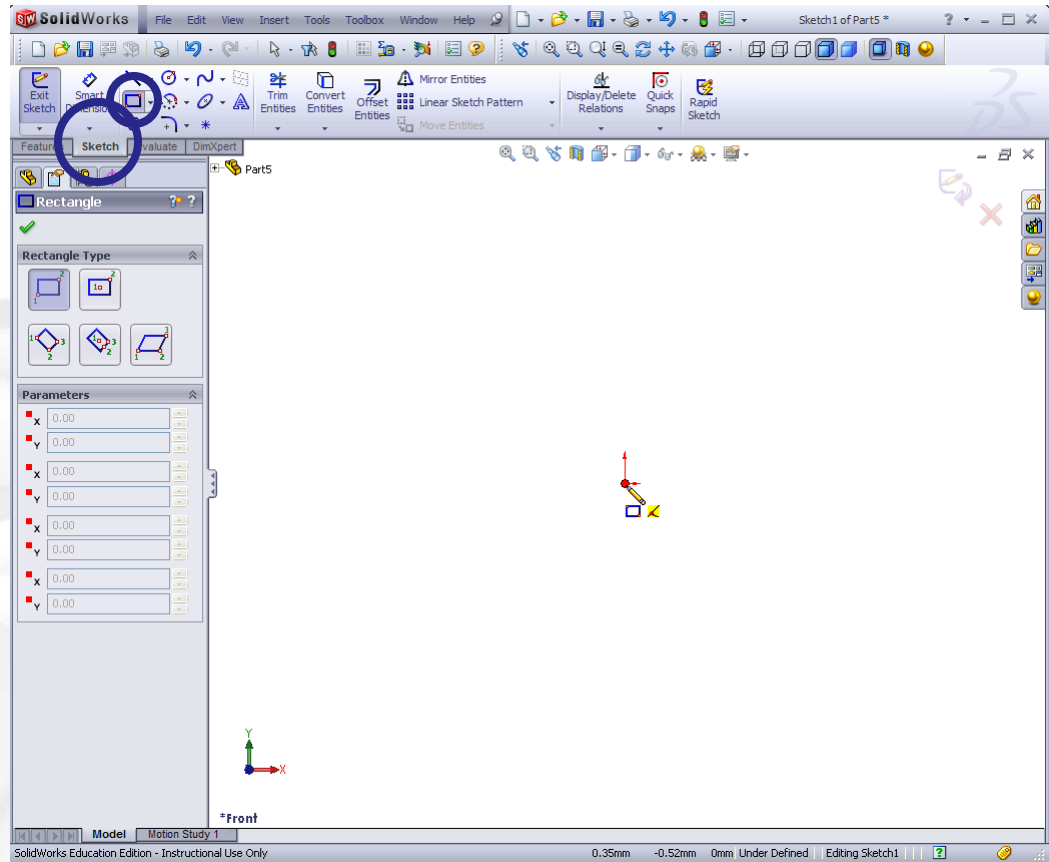
3. Shell Feature



4. Cut Feature

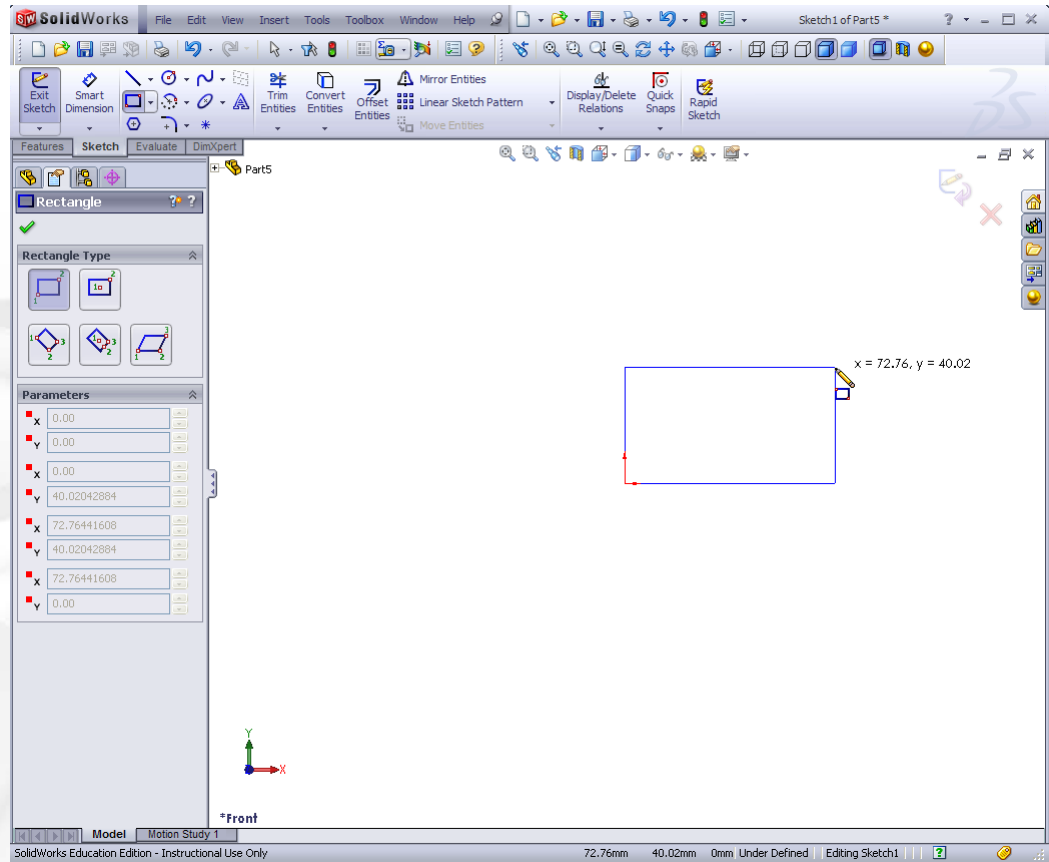
# Creating a 2D Sketch

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

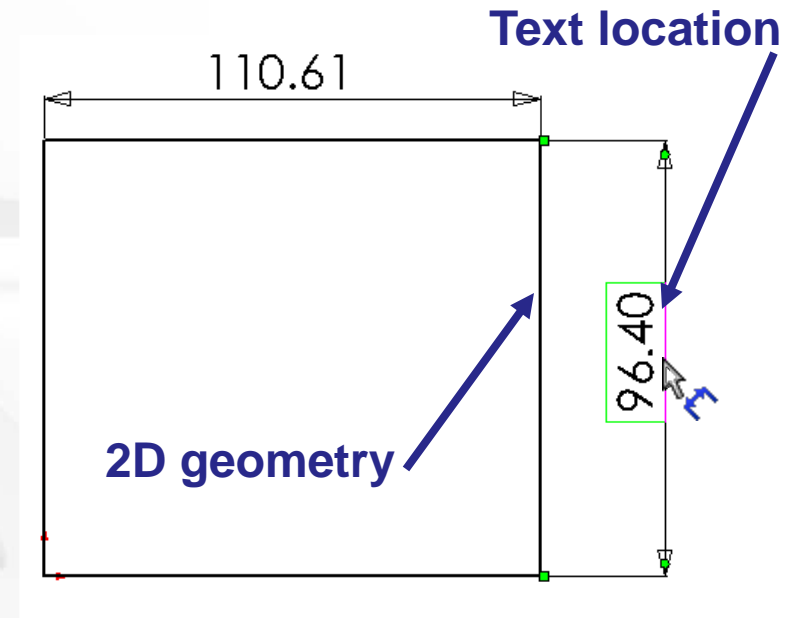


# Adding Dimensions

- Dimensions specify the size of the model.

To create a dimension:

1. Click Smart Dimension  on the Dimensions/Relations toolbar.
2. Click the 2D geometry.
3. Click the text location.
4. Enter the dimension value.

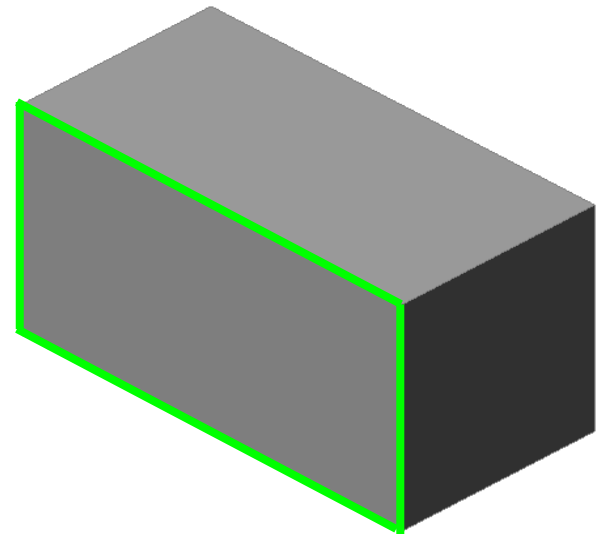


# Features and Commands

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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.

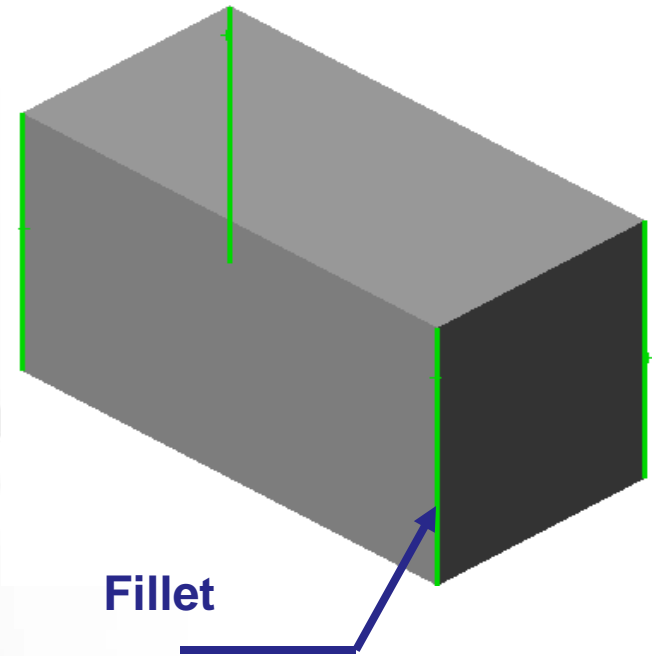




# Features and Commands

## Fillet feature

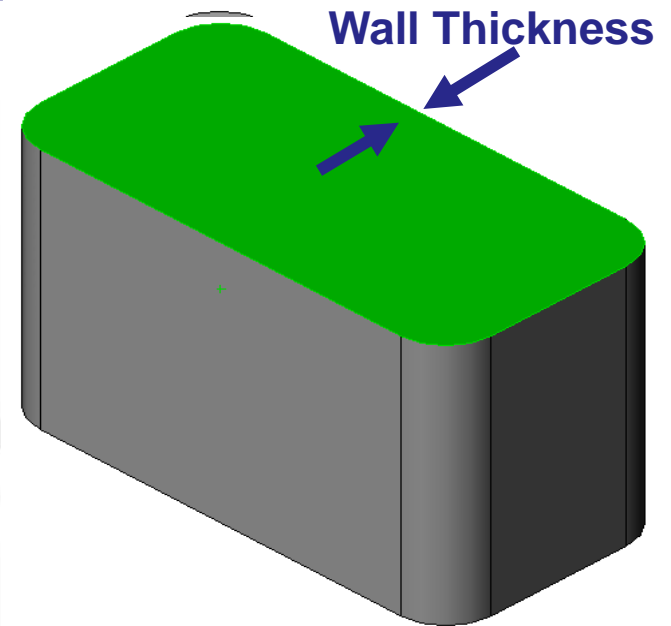
- The fillet feature rounds the edges or faces of a part.
- Select the edges to be rounded. Selecting a face rounds all the edges of that face.
- Specify the fillet radius.



# Features and Commands

## Shell feature

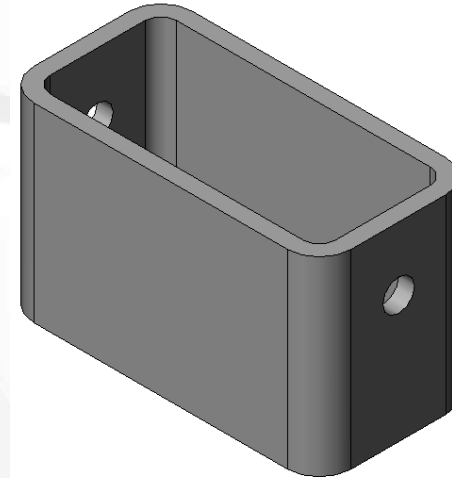
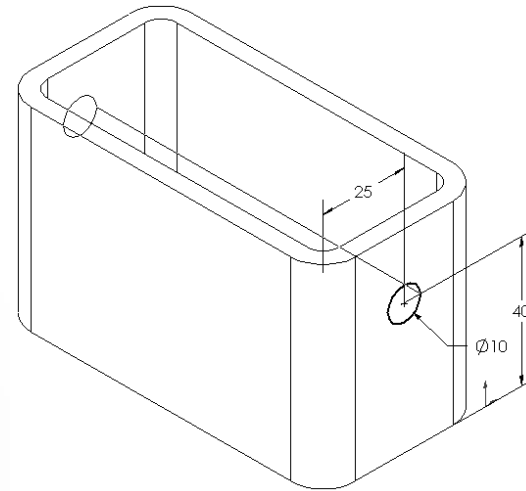
- The shell feature removes material from the selected face.
- Using the shell feature creates a hollow box from a solid box.
- Specify the wall thickness for the shell feature.



# 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 Through All for the end condition.
- The cut penetrates through the entire part.



# Dimensions and Geometric Relationships

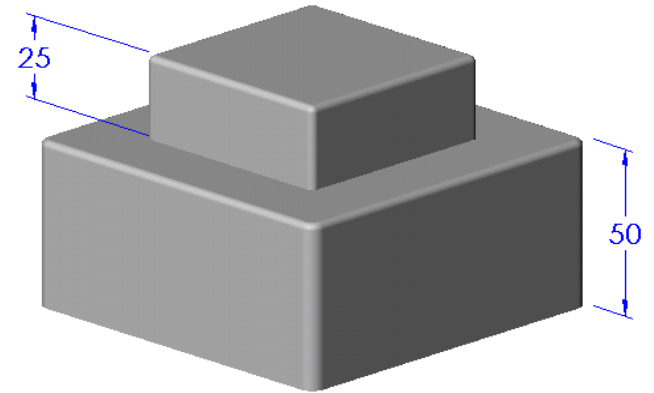
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- **Specify dimensions and geometric relationships between features and sketches.**
- **Dimensions change the size and shape of the part.**
- **Mathematical relationships between dimensions can be controlled by equations.**
- **Geometric relationships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

# Dimensions

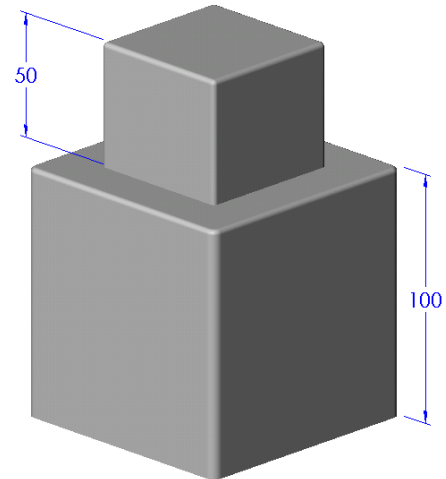
- **Dimensions**

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

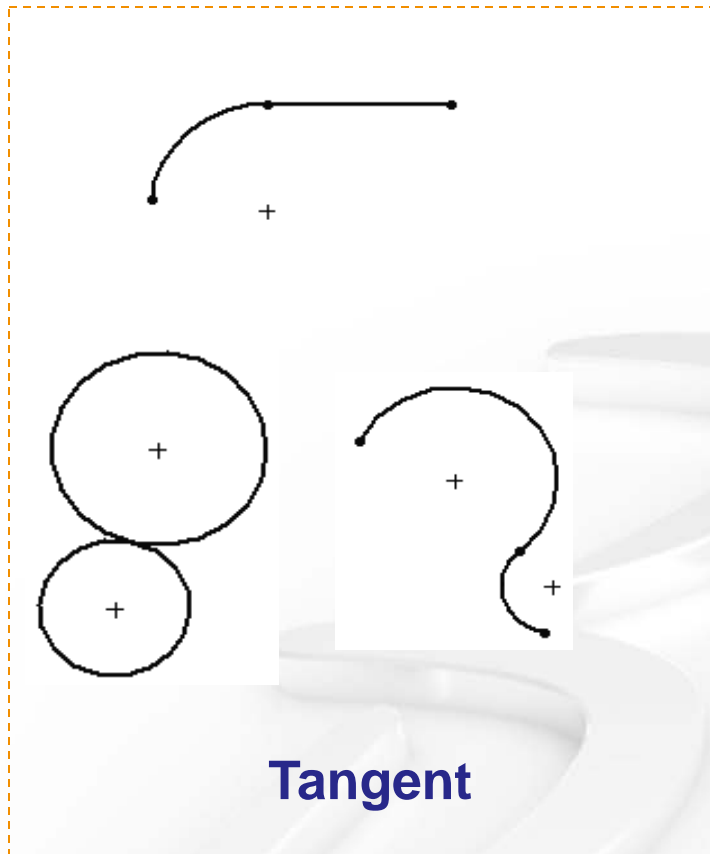


- **Mathematical relationship**

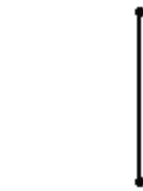
- **Boss depth = Base depth  $\div$  2**



# Geometric Relationships



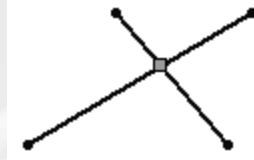
**Tangent**



**Vertical**



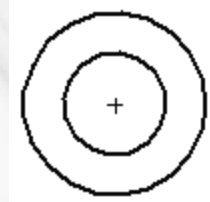
**Horizontal**



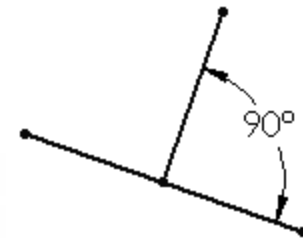
**Intersection**



**Parallel**




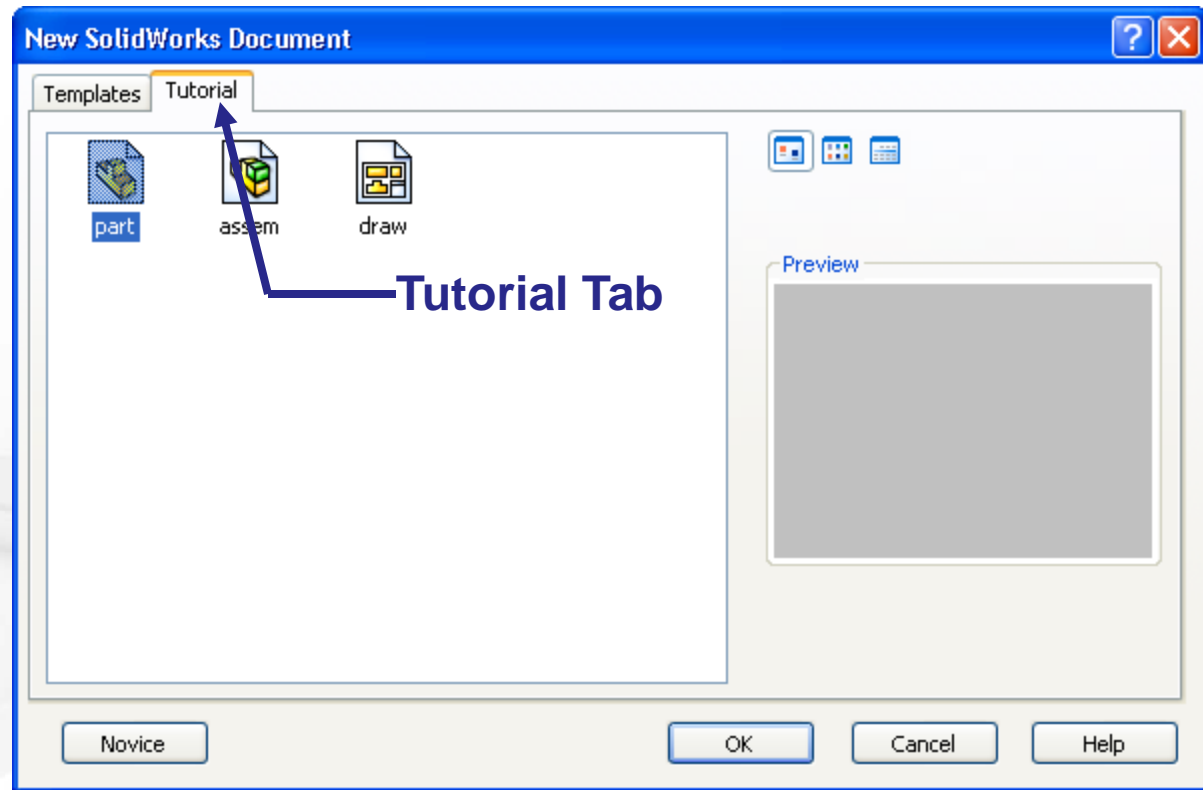
**Concentric**



**Perpendicular**

# Creating New Files Using Templates

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



# Document Templates

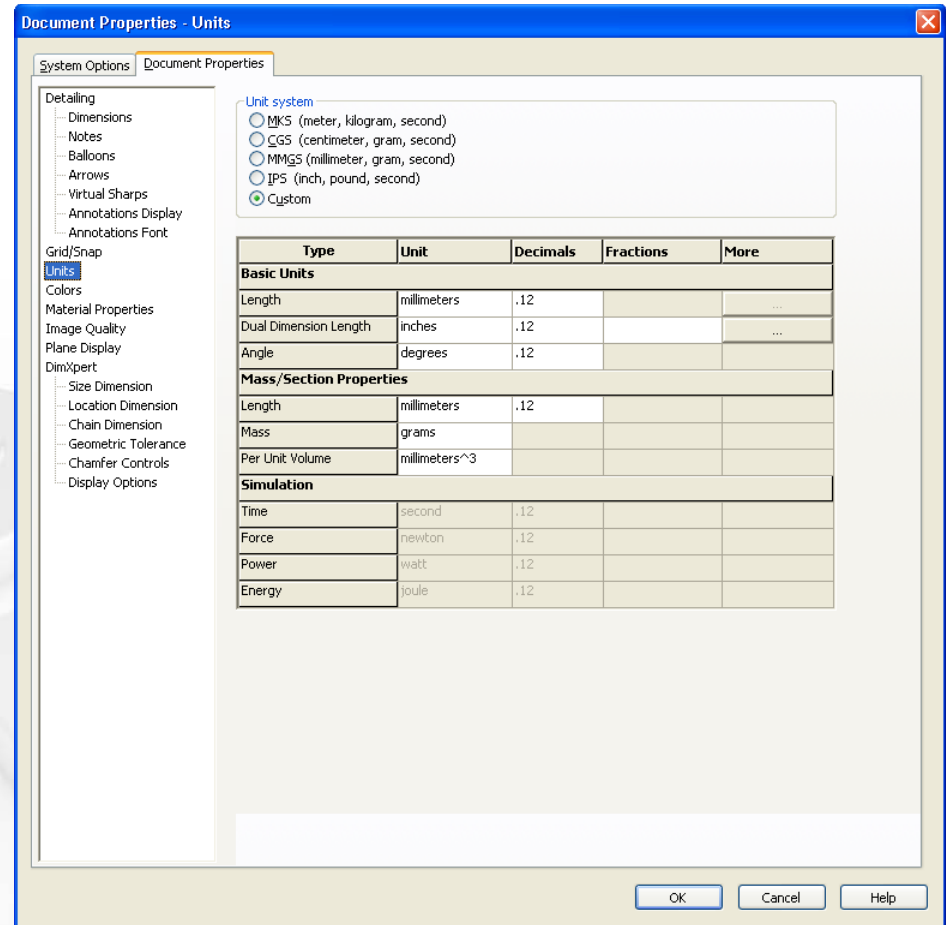
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- Document Templates control the units, grid, text, and other settings for the model.
- The Tutorial document templates are required to complete the exercises in the *Online Tutorials*.
- The templates are located in the Tutorial tab on the New SolidWorks Document dialog box.
- Document properties are saved in templates.



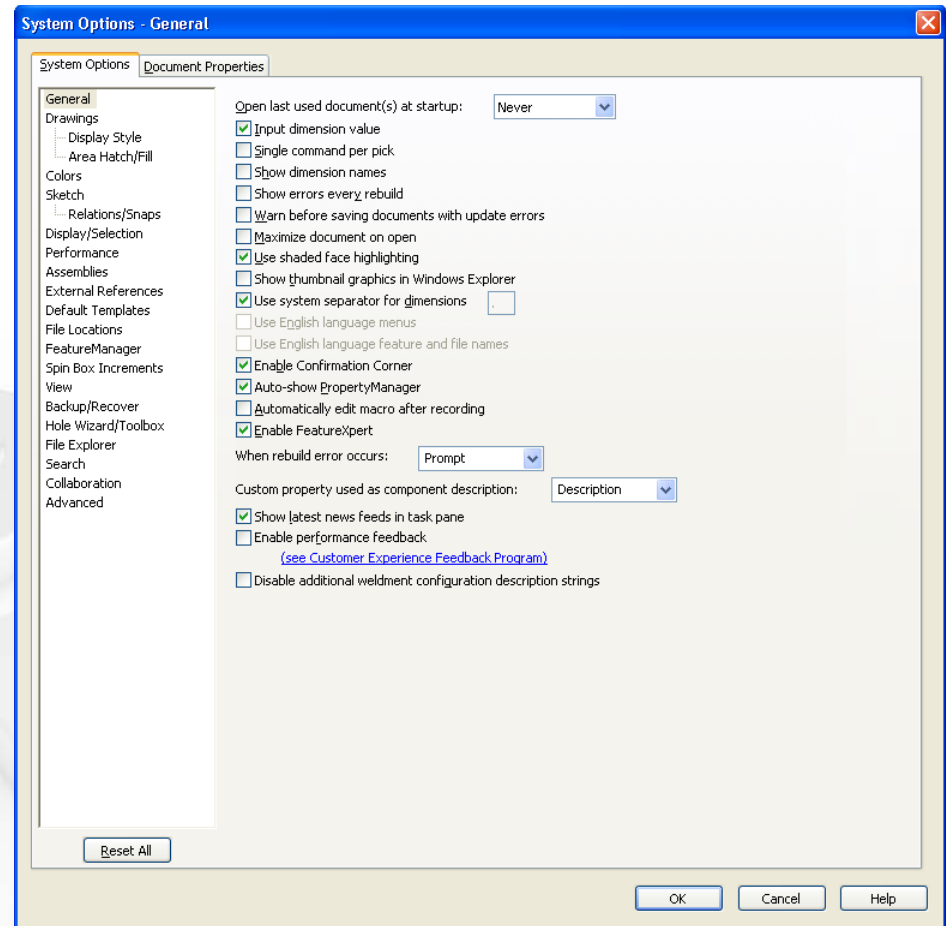
# Document Properties

- 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

