

3DEXPERIENCE

CAD Instructor Guide

Lesson 3

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Features and Commands

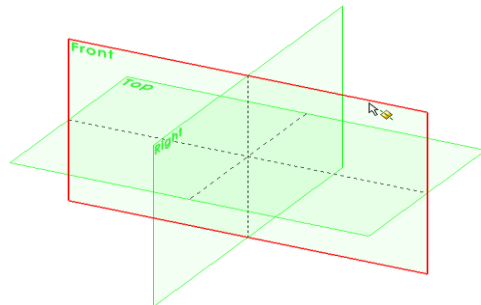
Base Feature

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

Tip: Keep the base feature simple.

To Create an Extruded Base Feature:

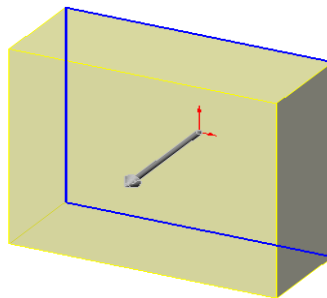
1. Select a sketch plane.
2. Sketch a 2D profile.
3. Extrude the sketch perpendicular to sketch plane.



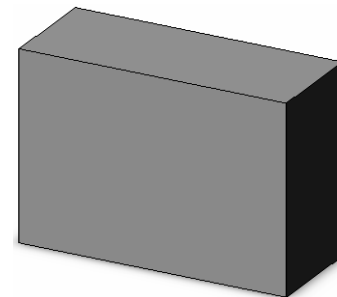
Select the sketch plane



Sketch the 2D profile



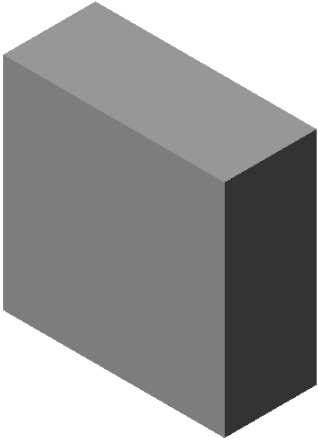
Extrude the sketch



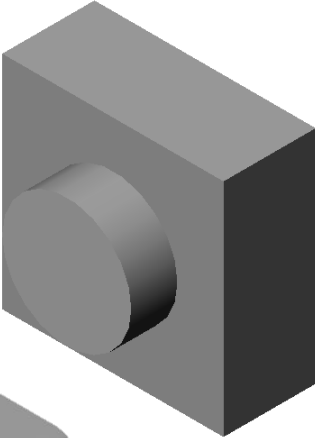
Resulting base feature

Features Used to Build Tutor1

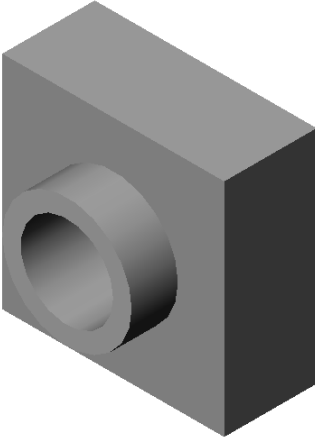
1. Base Extrude



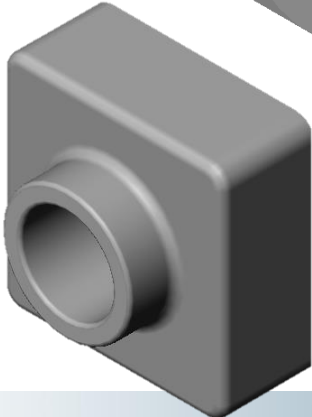
2. Boss Extrude



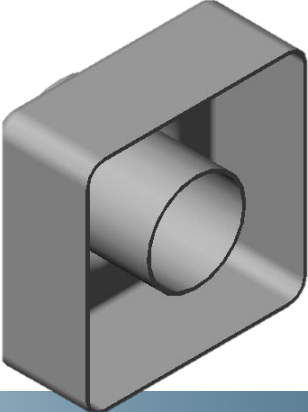
3. Cut Extrude



4. Fillets

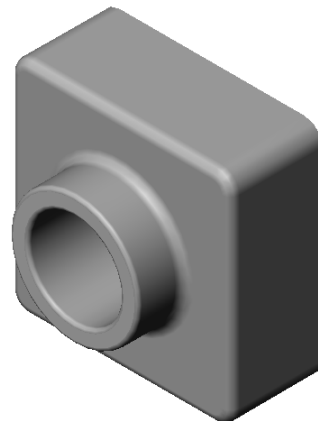


5. Shell



Features Used to Build Tutor1

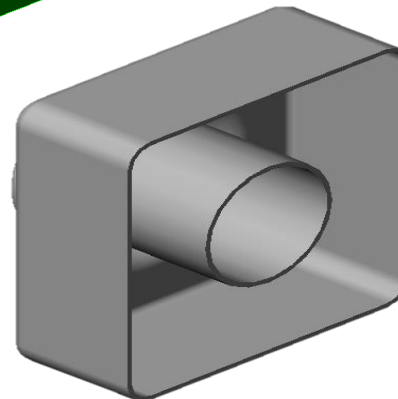
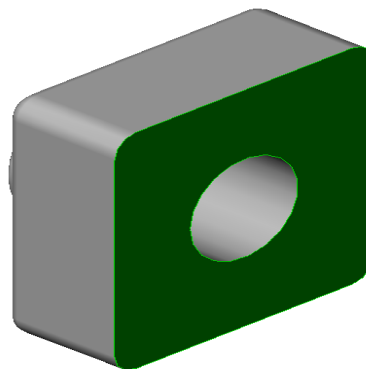
- ▶ Extruded Boss Feature
 - ▷ Adds material to the part.
 - ▷ Requires a sketch.
- ▶ Extruded Cut Feature
 - ▷ Removes material from the part.
 - ▷ Requires a sketch.
- ▶ Fillet Feature
 - ▷ Rounds the edges or faces of a part to a specified radius.



Features Used to Build Tutor1

Shell Feature



- ▷ Removes material from the selected face.
- ▷ Creates a hollow block from a solid block.
- ▷ Very useful for thin-walled, plastic parts.
- ▷ You are required to specify a wall thickness when using the shell feature.



View Control

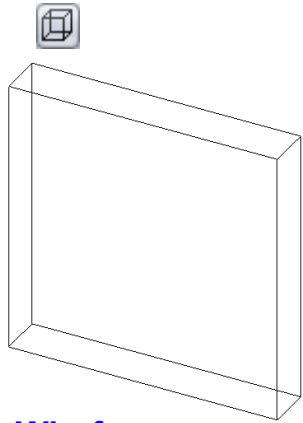
Magnify or reduce the view of a model in the graphics area.



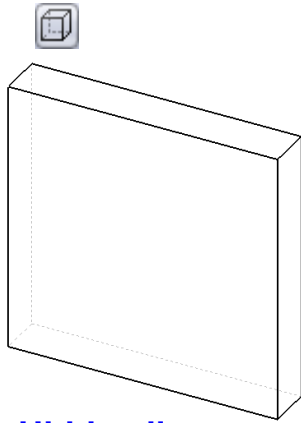
- ▷  Zoom to Fit – displays the part so that it fills the current window.
- ▷  Zoom to Area – zooms in on a portion of the view that you select by dragging a bounding box.
- ▷ Zoom In/Out – roll the wheel on the mouse to zoom in or out.

Display Modes

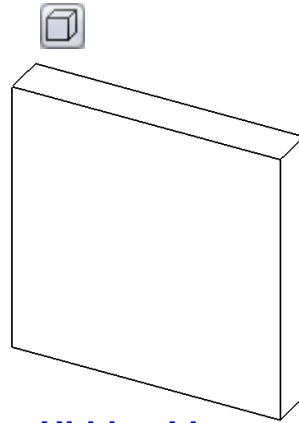
- ▶ Illustrate the part in various display modes.



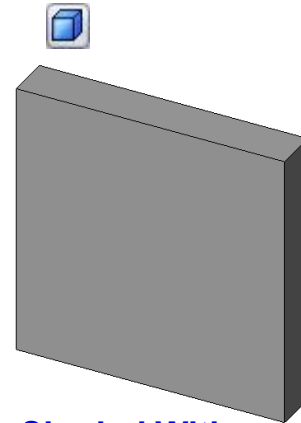
Wireframe



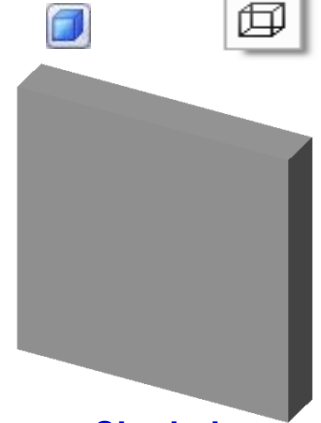
**Hidden lines
Visible**



**Hidden Lines
Removed**



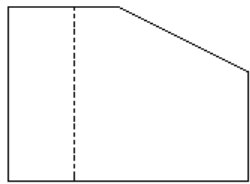
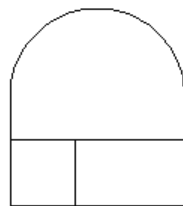
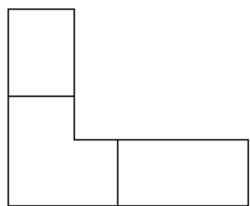
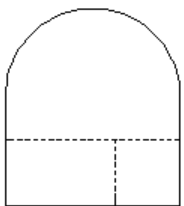
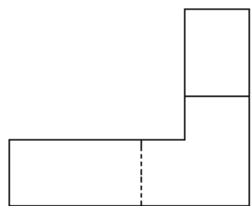
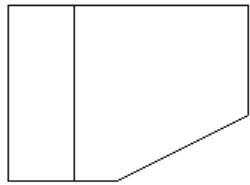
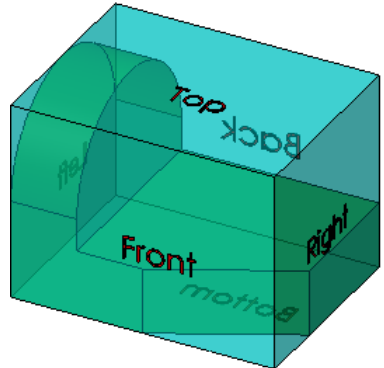
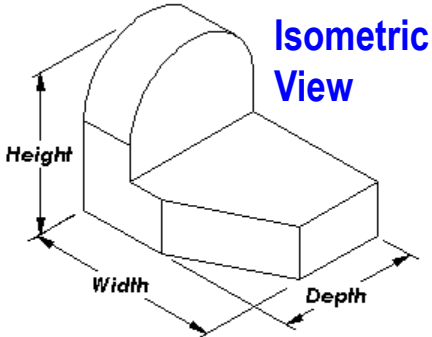
**Shaded With
Edges**



Shaded



Standard Views



Back View

Left View


Front View

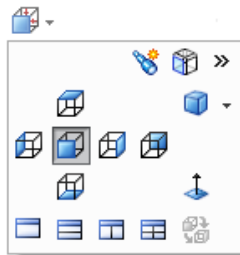
Right View

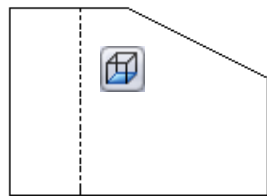
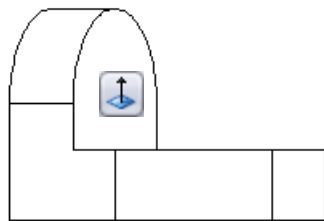
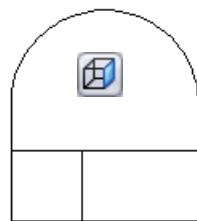
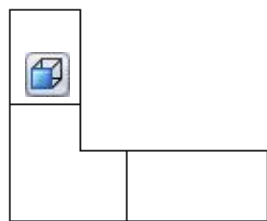
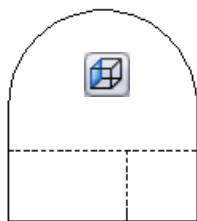
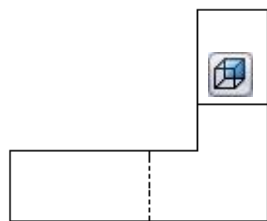
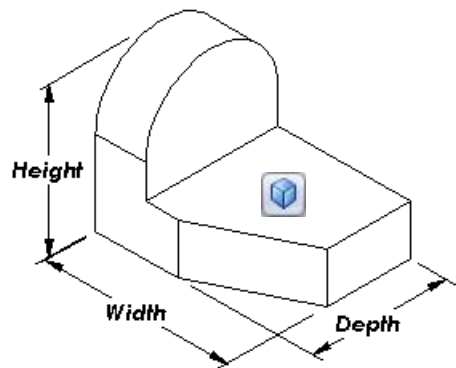
Bottom View

View Orientation

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

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

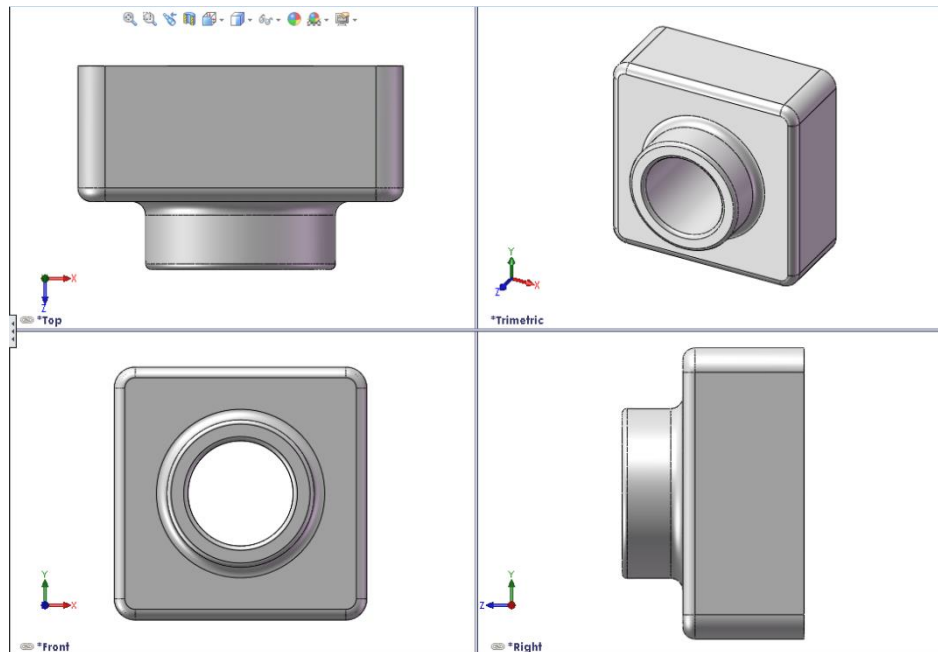




View Orientation

► The views most commonly used to describe a part are:

- ▷ Top View
- ▷ Front View
- ▷ Right View
- ▷ Isometric or Trimetric View



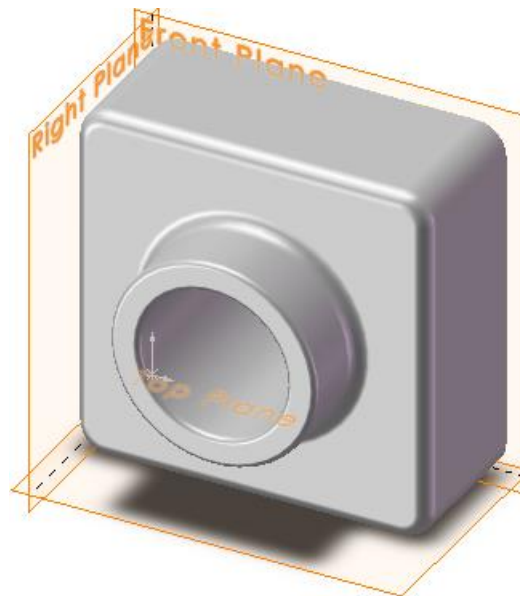
Default Planes

Default Planes

- ▷ Front, Top, and Right

Correspond to the standard principle drawing views:

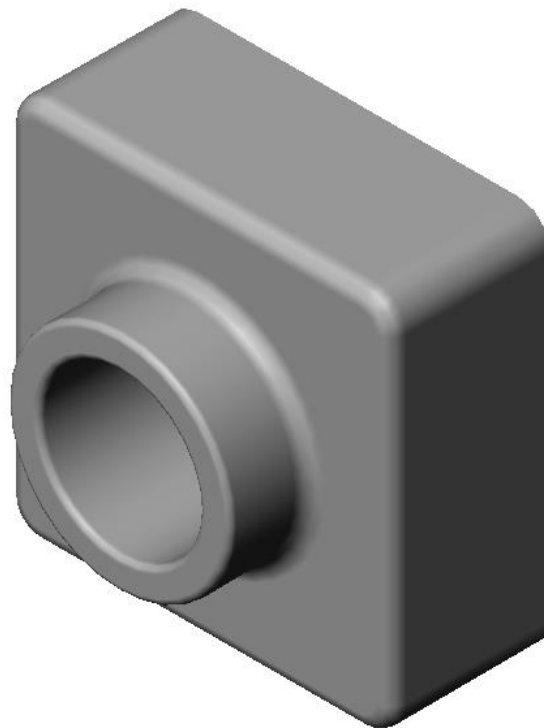
- ▷ Front = **Front or Back view**
- ▷ Top = **Top or Bottom view**
- ▷ Right = **Right or Left view**



Isometric View

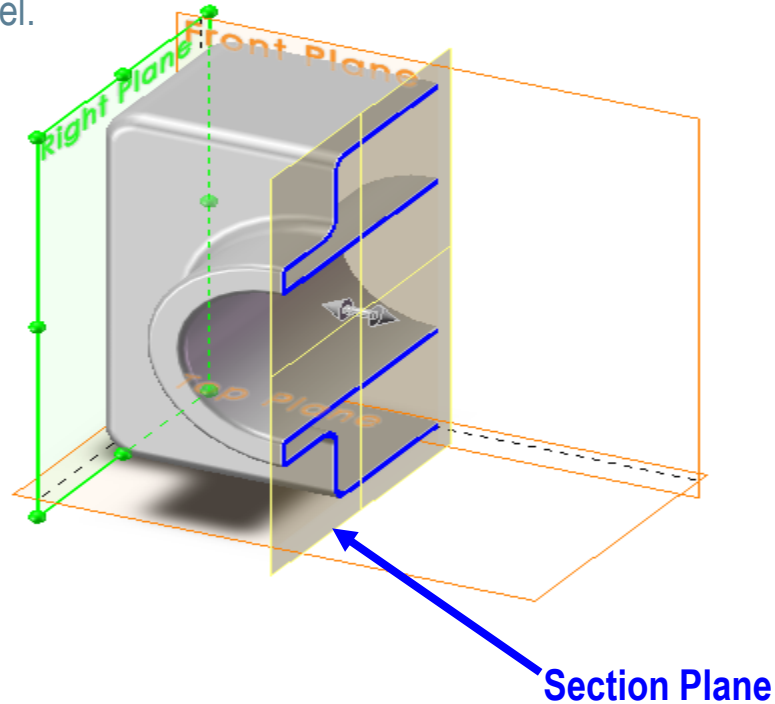
Displays the part with height, width, and depth equally foreshortened.

- ▷ Pictorial rather than orthographic.
- ▷ Shows all three dimensions – height, width, and depth.
- ▷ Easier to visualize than orthographic views.



Section View

- ▷ Displays the internal structure of a model.
- ▷ Requires a section cutting plane.



Mouse over

The Status of a Sketch

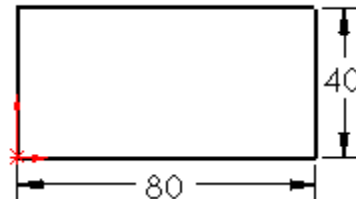
► Under defined

- ▷ Additional dimensions or relations are required.
- ▷ Under defined sketch entities are *blue* (by default).



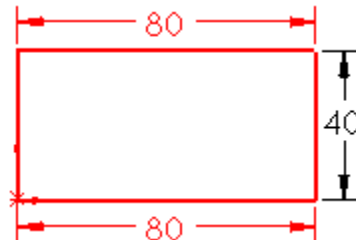
► Fully defined

- ▷ No additional dimensions or relationships are required.
- ▷ Fully defined sketch entities are *black* (by default).



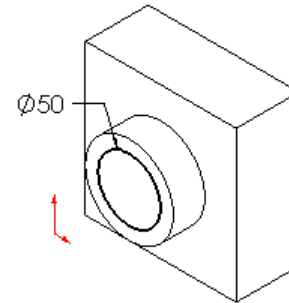
► Over defined

- ▷ Contains conflicting dimensions or relations, or both.
- ▷ Over defined sketch entities are *red* (by default).



Geometric Relations

- ▶ Geometric relations are the rules that control the behavior of sketch geometry.
- ▶ Geometric relations help capture design intent.
- ▶ **Example:** The sketched circle is concentric with the circular edge of the extruded boss feature.
- ▶ In a concentric relation, selected entities have the same center point.



Geometric Relations

- ▶ The SolidWorks default name for circular geometry is an Arc#.
- ▶ SolidWorks treats circles as 360° arcs.

