

Instructor's Guide to Teaching SolidWorks Software Lesson 3

School's Name
Teacher's Name
Date



Features and Commands

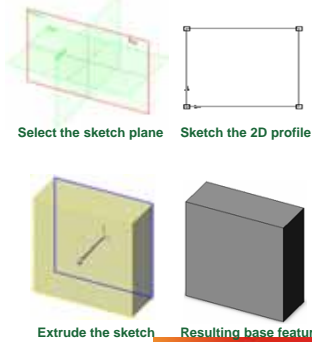
Base Feature

- 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.

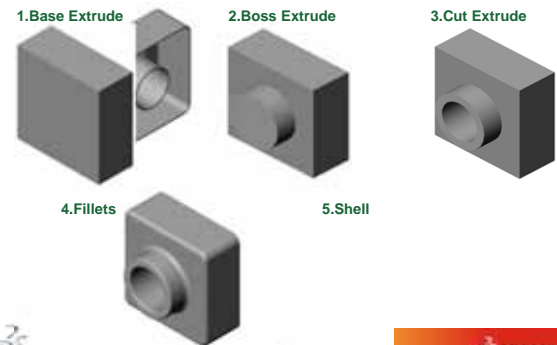
Tip: Keep the base feature simple.

To Create an Extruded Base Feature:

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

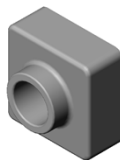


Features Used to Build Tutor1



Features Used to Build Tutor1

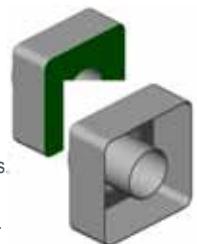
- Extruded Boss Feature
 - Adds material to the part.
 - Requires a sketch.
- Extruded Cut Feature
 - Removes material from the part.
 - Requires a sketch.
- Fillet Feature
 - Rounds the edges or faces of a part to a specified radius.



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.

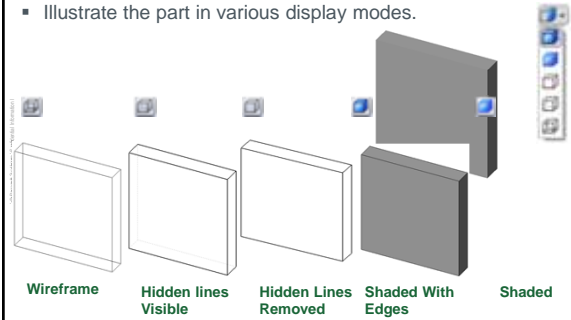


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Display Modes

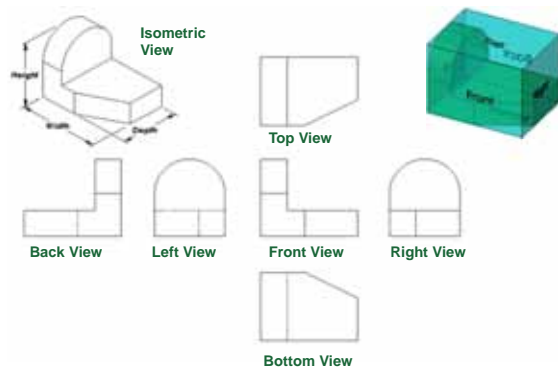
- Illustrate the part in various display modes.



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Standard Views



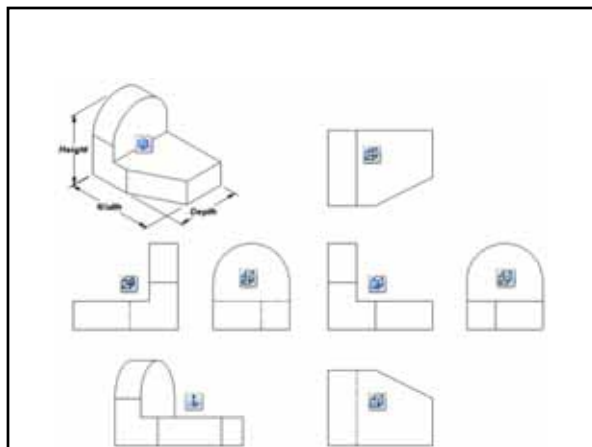
View Orientation

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)



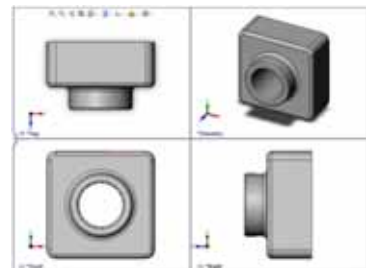
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View Orientation

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

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



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Default Planes

Default Planes

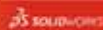
- Front, Top, and Right

Correspond to the standard principle drawing views:

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



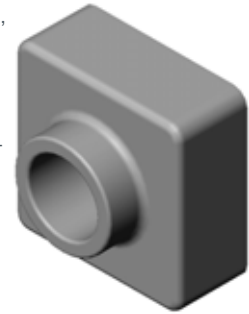
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Isometric View

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.

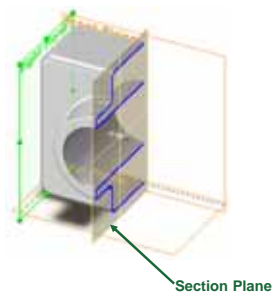


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Section View

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

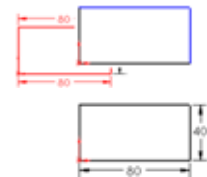


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The Status of a Sketch

- 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).

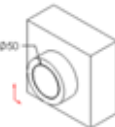


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Geometric Relations

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

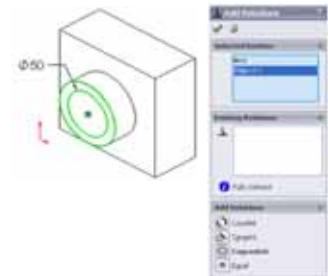


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Geometric Relations

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



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