

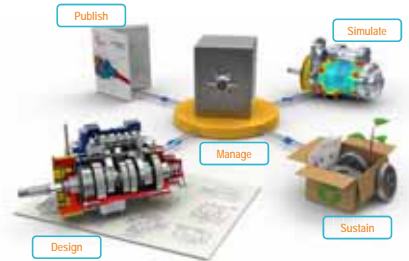
Instructor's Guide to Teaching SolidWorks Software Lesson 2

School's Name
Teacher's Name
Date



What is SolidWorks?

Intuitive solutions for every stage of your design.



SolidWorks – For Everything You Design

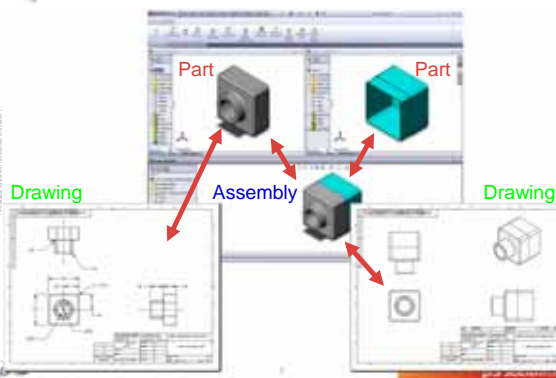


The SolidWorks Model

- The SolidWorks model is made up of:

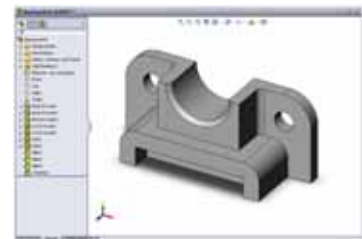
- Parts
- Assemblies
- Drawings

The SolidWorks Model



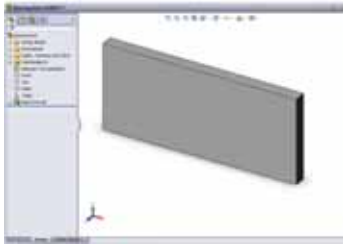
Features

- 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.

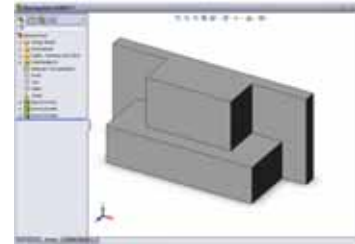


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Examples of Shape Features

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

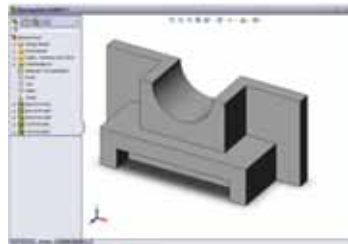


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Examples of Shape Features

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

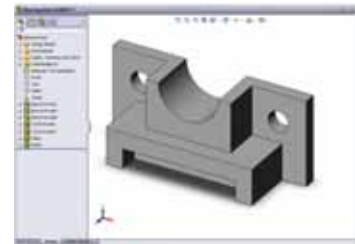


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Examples of Shape Features

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



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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.



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Examples of Shape Features

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



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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.

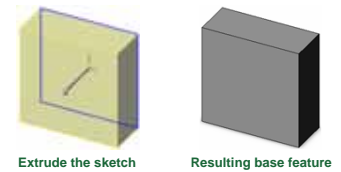
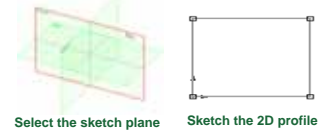


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To Create an Extruded Base Feature:

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

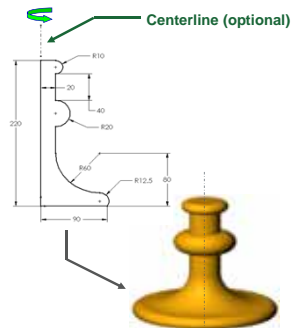


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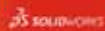


To Create a Revolved Base Feature:

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.

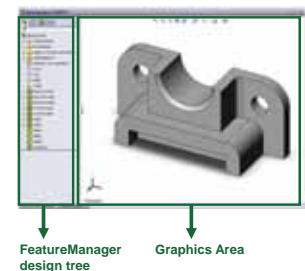


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Terminology: Document Window

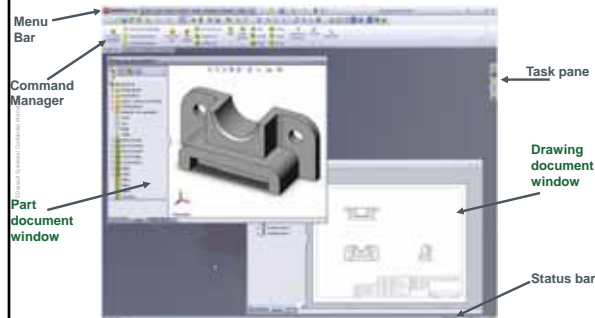
- Divided into two panels:
 - Left panel contains the FeatureManager® design tree.
 - Lists the structure of the part, assembly or drawing.
 - Right panel contains the Graphics Area.
 - Location to display, create, and modify a part, assembly or drawing.



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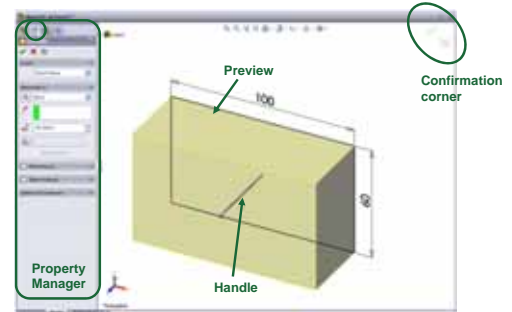
Terminology: User Interface



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Terminology: PropertyManager

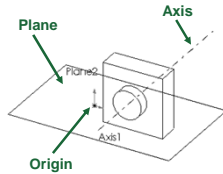


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Terminology: Basic Geometry

- 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: $(x = 0, y = 0, z = 0)$.

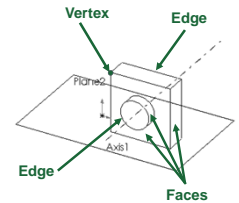


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Terminology: Basic Geometry

- 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.



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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.



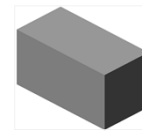
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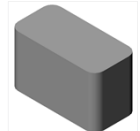
Features and Commands

Features used to build the box are:

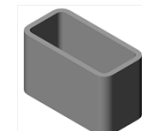
- Extruded Base feature
- Fillet feature
- Shell feature
- Extruded Cut feature



1.Base Feature



2.Fillet Feature



3.Shell Feature



4.Cut Feature



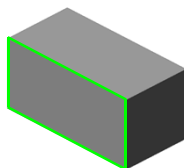
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Features and Commands

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.



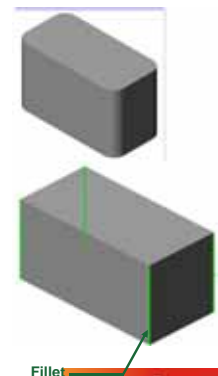
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Features and Commands

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.



Fillet



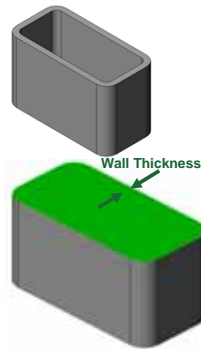
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Features and Commands

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.



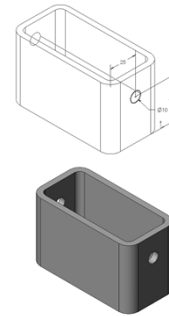
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Features and Commands

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.



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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.

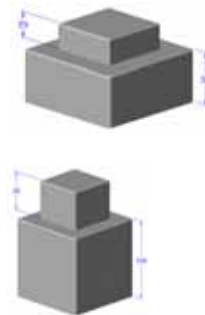


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Dimensions

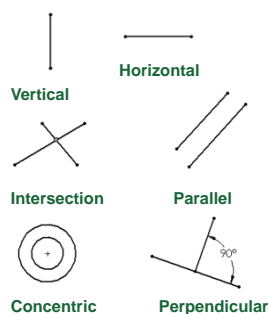
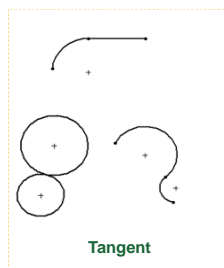
- Dimensions
 - Base depth = 50 mm
 - Boss depth = 25 mm
- Mathematical relationship
 - Boss depth = Base depth ÷ 2



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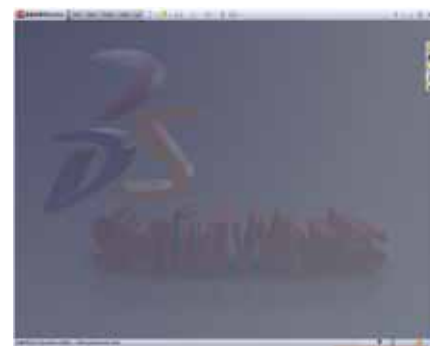
Geometric Relationships



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The SolidWorks Window

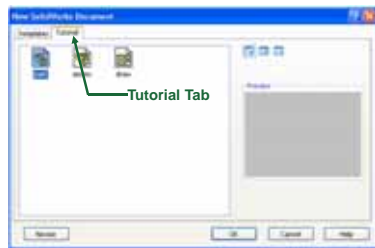


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Creating New Files Using Templates

- Click New  on the Menu Bar.
- Select a document template:
 - Part
 - Assembly
 - Drawing



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Document Templates

- 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.

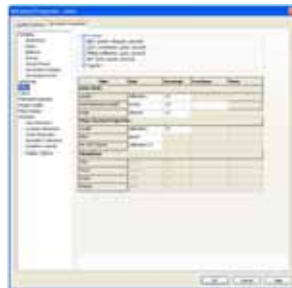


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Document Properties

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



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System Options

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



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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

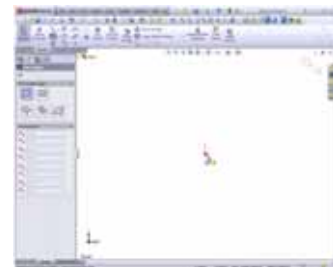


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Creating a 2D Sketch

- Click Sketch > Sketch .
- Select the Front plane as a sketch plane.
- Click Sketch > Rectangle .
- Move the pointer to the Sketch Origin.

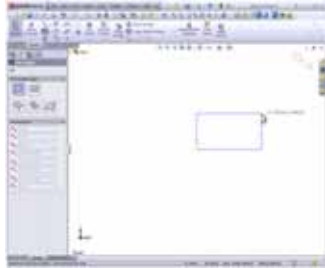


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Creating a 2D Sketch

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



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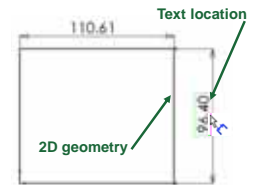
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Adding Dimensions

- Dimensions specify the size of the model.

To create a dimension:

1. Click Sketch > Smart Dimension .
2. Click the 2D geometry.
3. Click the text location.
4. Enter the dimension value.



BS

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