# SolidWorks®

## Engineering Design Project
### The Mountainboard

**Teacher Resources**


Dassault Systèmes SolidWorks Corporation
300 Baker Avenue
Concord, Massachusetts 01742 USA
Phone +1-800-693-9000

Outside the U.S.: +1-978-371-5011
Fax: +1-978-371-7303
Email: info@solidworks.com
Web: http://www.solidworks.com/education

Document Number: PME0718-ENG

# Contents

## 5 Minute Assessment – #2 Answer Key

1. How do you start a SolidWorks session?
   **Answer:** Click **Start**, Click **All Programs**. Click the SolidWorks folder. Click the SolidWorks application.
2. Why do you create and use Document Templates?
   **Answer:** Document Templates contain the units, grid and text setting for the model. You can create Metric and English templates each with different settings.
3. How do you start a new Part Document?
   **Answer:** Click the **New** icon. Select a part template.
4. What features did you use to create the Binding Anchor?
   **Answer:** Extruded Boss, Fillet, and Extruded Cut.
5. True or False. SolidWorks is used by designers and engineers.
   **Answer:** True.
6. A SolidWorks 3D model consists of _____ _____ _____.
   **Answer:** Parts, assemblies and drawings.
7. How do you open a sketch?
   **Answer:** Select a plane or planar face, then click the Sketch icon on the Sketch toolbar.
8. What does the Fillet feature do?
   **Answer:** The Fillet feature rounds sharp edges.
9. What tool calculates the volume of a part?
   **Answer:** The Mass Properties tool.
10. What does the Cut-Extrude feature do?
    **Answer:** The Cut-Extrude feature removes material.
11. How do you change an existing feature?
    **Answer:** Right-click on the feature and select **Edit Feature**.

## Lesson 2 Vocabulary Worksheet – Answer Key

1. The corner or point where edges meet: **vertex**
2. The intersection of the three default reference planes: **origin**
3. A feature used to round off sharp corners: **fillet**
4. The three types of documents that make up a SolidWorks model: **parts, assemblies, drawings**
5. Controls the units, grid, text, and other settings of the document: **template**
6. Forms the basis of all extruded features: **sketch**
7. Two lines that are at right angles (90°) to each other are: **perpendicular**
8. The first feature in a part is called the **base** feature.
9. The outside surface or skin of a part: **face**
10. A mechanical design automation software application: **SolidWorks**
11. The boundary of a face: **edge**
12. Two straight lines that are always the same distance apart are: **parallel**
13. Two circles or arcs that share the same center are: **concentric**
14. The shapes and operations that are the building blocks of a part: **features**
15. A feature that adds material to a part: **boss**
16. A feature that removes material from a part: **cut**
17. An implied centerline that runs through the center of every cylindrical feature: **temporary axis**

## Lesson 2 Quiz — Answer Key

1. You build parts from features. What are features?

   **Answer:** Features are the shapes (bosses, cuts and holes) and the operations (fillets, chamfers and shells) that are use to build a part.

2. Name the features that are used to create the `Binding Anchor` in Lesson 2.

   **Answer:** Extruded Boss, Fillet and Extruded Cut.

3. How do you begin a new part document?

   **Answer:** Click the **New** tool or click **File, New**. Select a part template.

4. Give two examples of shape features that require a sketched profile.

   **Answer:** Shape features are Extruded Boss, Extruded Cut, and Hole.

5. Give an example of an operation feature that requires a selected edge or face.

   **Answer:** Operation features are Fillet or Chamfer.

6. Name the three documents that make up a SolidWorks model.

   **Answer:** Parts, assemblies and drawings

7. What is the default sketch plane?

   **Answer:** The default sketch plane is `Front`.

8. What is a plane?

   **Answer:** A plane is a flat 2D surface.

9. How do you create an extruded boss feature?

   **Answer:** Select a sketch plane. Open a new sketch. Sketch the profile. Extrude the profile perpendicular to the sketch plane.

10. Why do you create and use document templates?

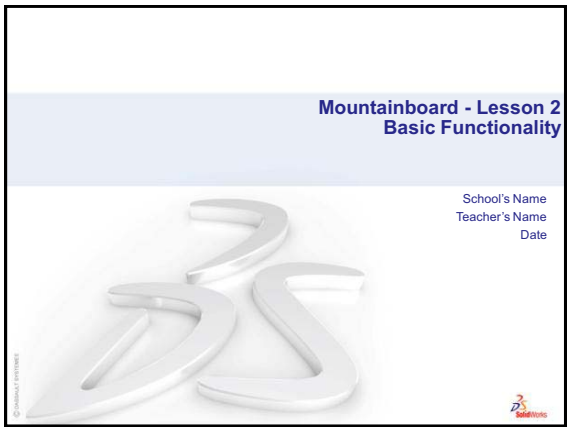
    **Answer:** Document templates contain the units, grid and text setting for the model. You can create Metric and English templates, each with different settings.

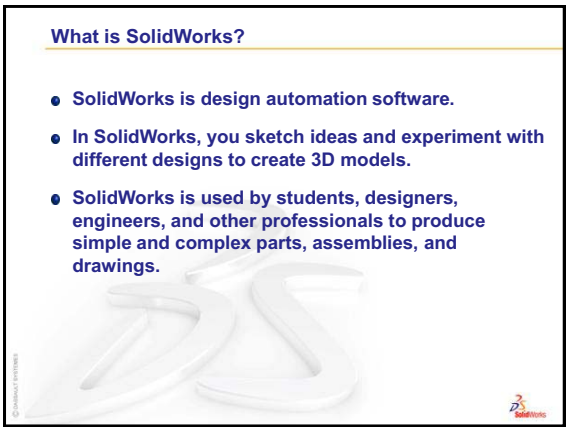
11. What is a section view?

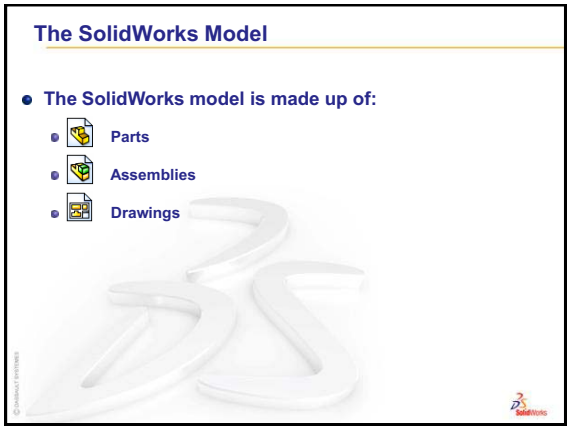
    **Answer:** A section view shows the part as if it were cut into two pieces. This displays the internal structure of the model.

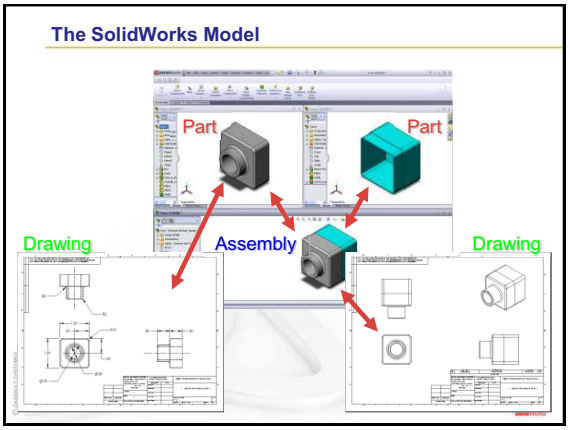
## Thumbnail Images of PowerPoint Slides

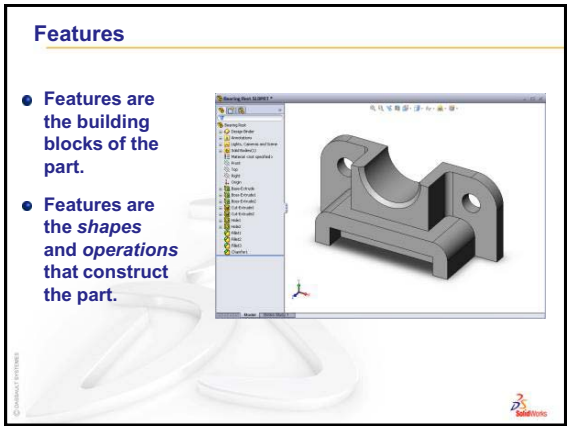
The following thumbnail images, arranged left to right, show the PowerPoint slides provided with this lesson.

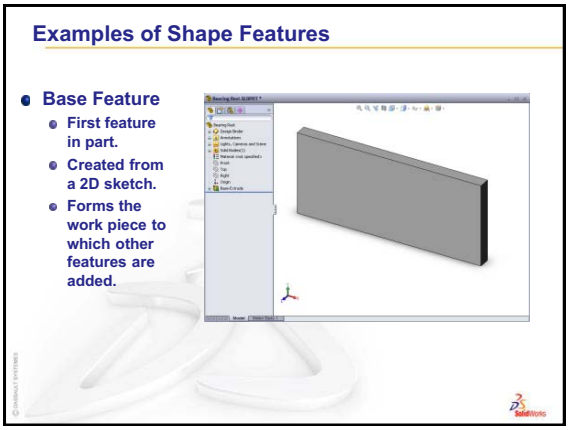


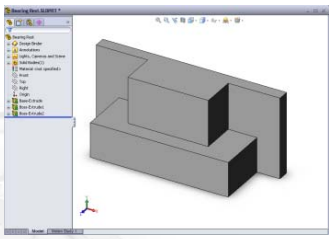



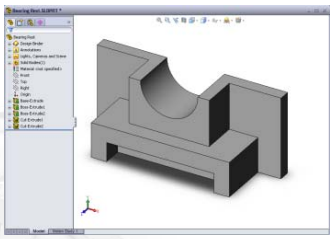
## Examples of Shape Features

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



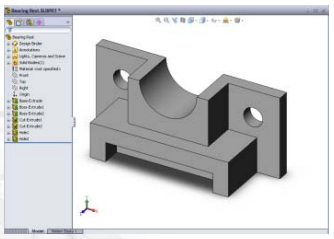
## Examples of Shape Features

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



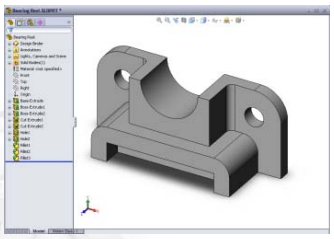
## Examples of Shape Features

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



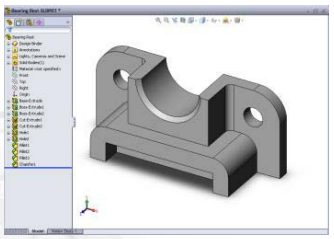
## Examples of Shape Features

- **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.

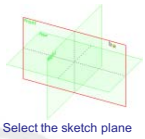


## Sketched Features & Operation Features

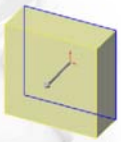
- **Sketched Features**
  - Shape features have sketches.
  - Sketched features are built from 2D profiles.
- **Operation Features**
  - Operation features do not have sketches.
  - Applied directly to the work piece by selecting edges or faces.

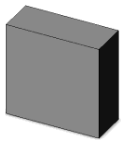
## To Create an Extruded Base Feature:

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


Select the sketch plane

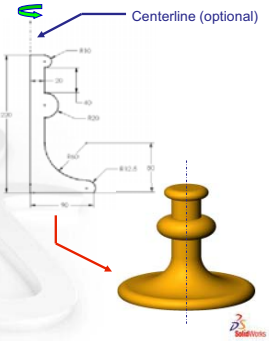

Sketch the 2D profile


Extrude the sketch


Resulting base feature

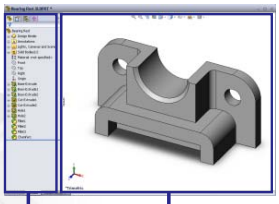
## To Create a Revolved Base Feature:

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

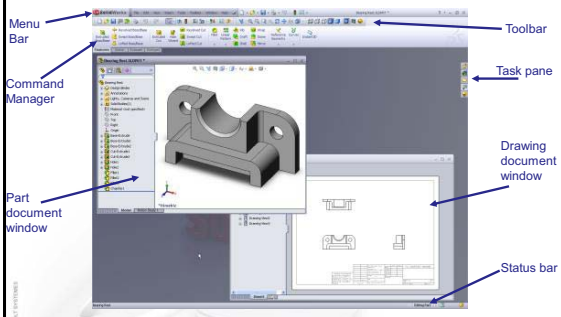



## Terminology: Document Window

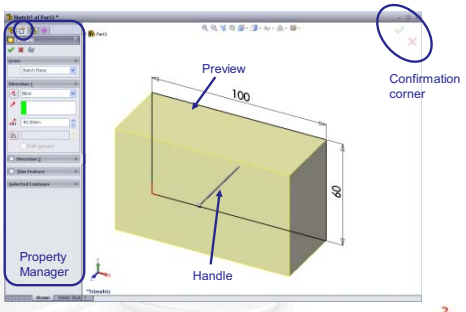
- **Divided into two panels:**
  - **Left panel contains the FeatureManager® design tree.**
    - **Lists the structure of the part, assembly or drawing.**
  - **Right panel contains the Graphics Area.**
    - **Location to display, create, and modify a part, assembly or drawing.**




## Terminology: User Interface

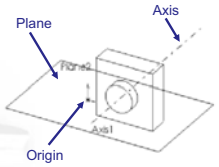



## Terminology: PropertyManager



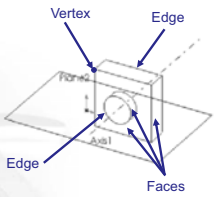

## Terminology: Basic Geometry

- **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

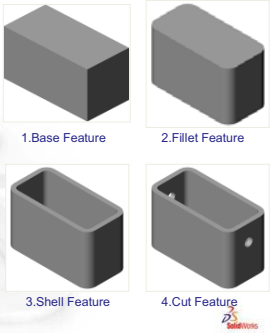
**Base feature**

- 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

**Features used to build the *box* are:**

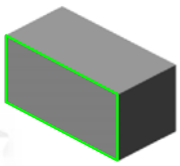
- Extruded Base feature
- Fillet feature
- Shell feature
- Extruded Cut feature




## Features and Commands

**To create the extruded base feature for the *box*:**

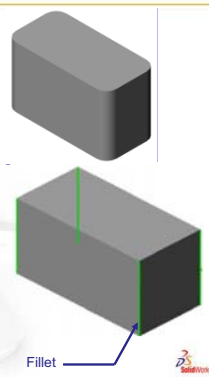
- Sketch a rectangular profile on a 2D plane.
- Extrude the sketch.
- By default extrusions are perpendicular to the sketch plane.



## Features and Commands

**Fillet feature**

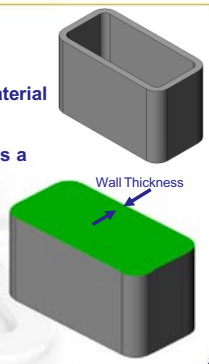
- The fillet feature rounds the edges or faces of a part.
- Select the edges to be rounded. Selecting a face rounds all the edges of that face.
- Specify the fillet radius.




## Features and Commands

**Shell feature**

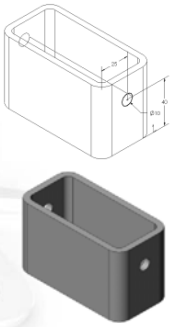
- The shell feature removes material from the selected face.
- Using the shell feature creates a hollow box from a solid box.
- Specify the wall thickness for the shell feature.



## 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.

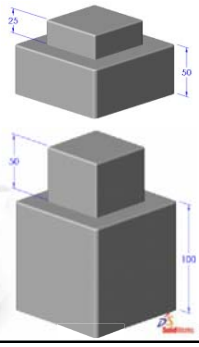


## Dimensions and Geometric Relationships

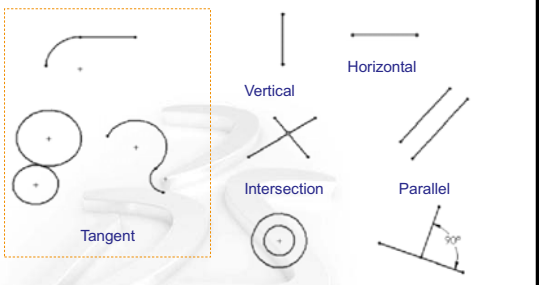
- Specify dimensions and geometric relationships between features and sketches.
- Dimensions change the size and shape of the part.
- Mathematical relationships between dimensions can be controlled by equations.
- Geometric relationships are the rules that control the behavior of sketch geometry.
- Geometric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

## Dimensions

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

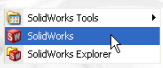



## Geometric Relationships

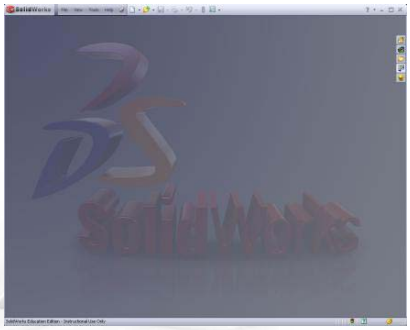



## To Start SolidWorks

- Click the Start button on Windows task bar.
  - ➢ Click Programs.
    - ➢ Click the SolidWorks folder.
      - ➢ Click the SolidWorks application.

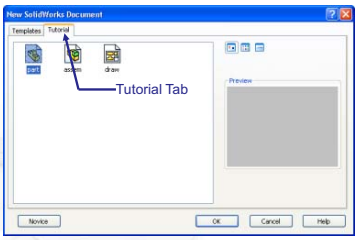



## The SolidWorks Window



## Creating New Files Using Templates

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

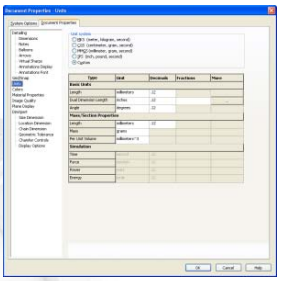



## Document Templates

- 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.

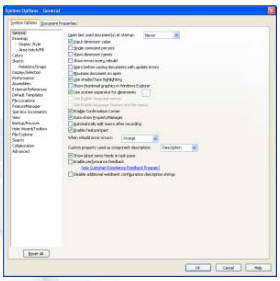
## Document Properties

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



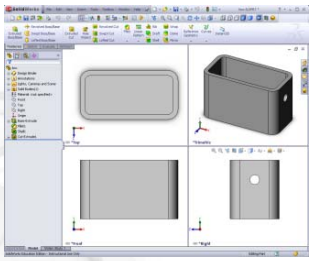
## System Options

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



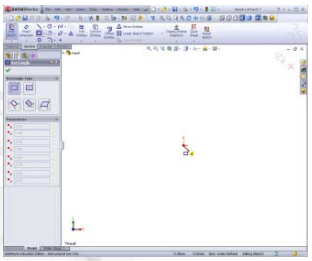
## Multiple Views of a Document

- Click the view pop-up menu.
- Select an icon. The viewport icons include:
  - Single View
  - Two View (horizontal and vertical)
  - Four View



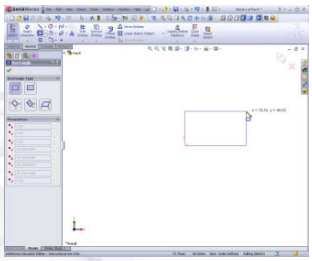
## Creating a 2D Sketch

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.

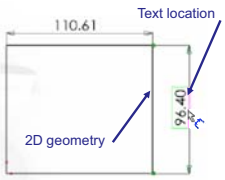


## Adding Dimensions

- Dimensions specify the size of the model.

To create a dimension:

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




## The Mountainboard

## Design Process

- Project goals.
  - Design Intent
    - How does the model respond to change
    - Anticipated changes in the design
  - Analysis
    - Estimation of the lifecycle of the product
    - How will we insure that the parts are strong enough
  - Output
    - Engineering reports
    - Presentations
    - Marketing material

## Key parts

- The Binding
  - Right and Left versions
  - Adjusts position
    - Rotation
    - Along Deck

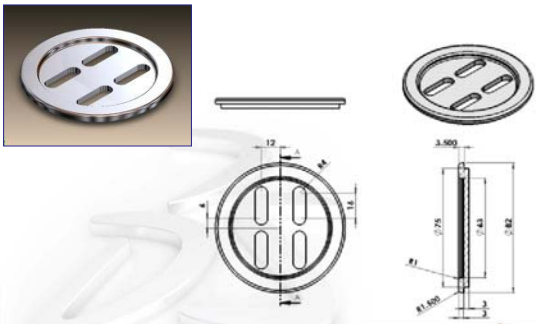
## Key parts

- The Deck
  - Flexible
  - Mounting for Binding and Truck
  - Non-slip surface
  - Support a 100 kg rider
- Truck and Axle
  - Adjustable suspension
  - Mounting for optional braking system

## Key parts

- Wheel Assembly
  - Easy to assemble
  - Tire and Tube are purchased parts
  - Sealed bearings
  - Mounting for optional braking system

## Binding Anchor



## Design Intent - Binding Anchor

- Clamps and Positions the Binding on the Deck
- Positioning
  - Along centerline
  - At angle to centerline
- No sharp edges to injure a rider

## Determining the Weight

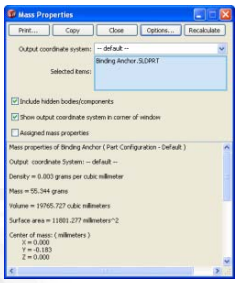
Weight = Volume X Density

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

## Mass Properties

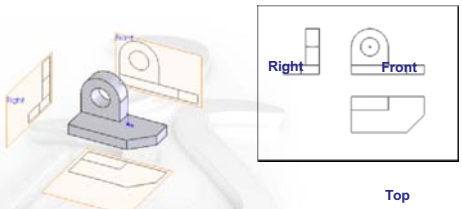
The Mass Properties tool can calculate:

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



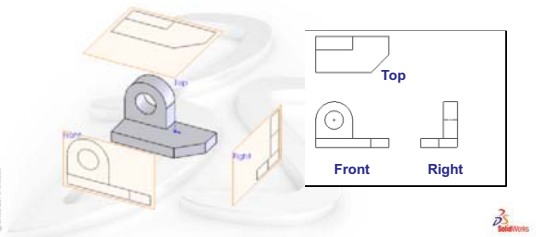

## First Angle Projection

- Like projecting the model on a screen behind the model.
- Used in Europe.



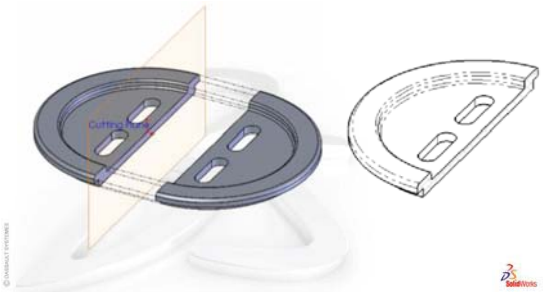
## Third Angle Projection

- Like looking at the part inside of a glass box.
- Used in the United States.




## Section View

- Cuts the model to reveal detail.



# Basic Parts — The Binding

## Review of Lesson 2: Basics

### Questions for Discussion

1. A SolidWorks 3D model consists of three documents. Name the three documents.

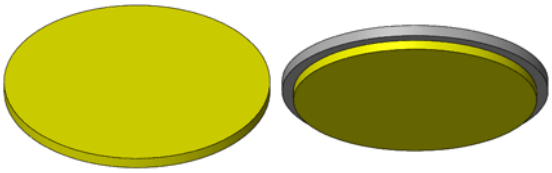
   **Answer:** Part, Assembly and Drawing.

2. Parts are built from features. What are features?

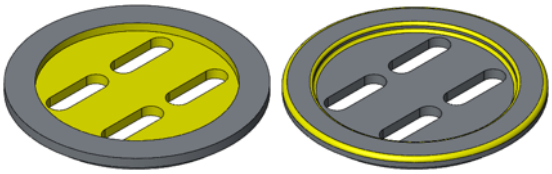
   **Answer:** Features are the shapes (bosses, cuts and holes) and the operations (fillets, chamfers and shells) that you use to build a part.

3. Name the features that are used to create the Binding Anchor in Lesson 2.

   **Answer:** Extruded Boss, Extruded Cut, and Fillet.



1. Base Feature 2. Extruded Boss



3. Extruded Cuts 4. Fillets

4. What is the base feature of the Binding Anchor?

   **Answer:** The base feature is the first feature of the Binding Anchor. The base feature is the foundation of the part. The base feature geometry for the Binding Anchor is an extrusion. The extrusion is named Extrude1. The base feature represents the general shape of the Binding Anchor.

5. Why did you use the Fillet feature?

   **Answer:** The fillet feature rounds the sharp edges and faces. The result of using the fillet feature created the rounded edges of the Binding Anchor.

6. How did you create the Base feature?

   **Answer:** To create a solid Base feature:

   - Sketch a circular profile on a flat 2D plane.
   - Extrude the profile perpendicular to the sketch plane.