Introduction to SolidWorks Software

Marine Advanced Technology Education
Design Tools
What is SolidWorks?

- SolidWorks is design automation software.
- In SolidWorks, you sketch ideas and experiment with different designs to create 3D models.
- SolidWorks is used by students, designers, engineers, and other professionals to produce simple and complex parts, assemblies, and drawings.
The SolidWorks Model

The SolidWorks model is made up of:

- Parts
- Assemblies
- Drawings
The SolidWorks Model

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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.
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.
To Create an Extruded Base Feature:

1. Select a sketch plane.

2. Sketch a 2D profile.

3. Extrude the sketch perpendicular to sketch plane.
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

- **Menu Bar**
- **Toolbar**
- **Task pane**
- **Command Manager**
- **Part document window**
- **Drawing document window**
- **Status bar**

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Toolbars

Buttons for frequently used commands.

- You can select the toolbars to display.
- Toolbars are displayed at the top and sides of the window.
- You can also access the toolbars from the CommandManager.
Left Side of SolidWorks Window

- FeatureManager® design tree

- Property Manager

- Configuration Manager

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Right Side of SolidWorks Window

The Task Pane

- **SolidWorks Resources**
- **Design Library**

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Terminology: PropertyManager

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Using the Mouse for 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. **Double click the scroll wheel**

- **Zoom In/Out** – drag the pointer upward to zoom in. Drag the pointer downward to zoom out. **Rotate the scroll wheel**

- **Rotate** – drag the pointer upward to zoom in. Drag the pointer downward to zoom out. **Click the left mouse button & Drag**

- **Pan** – drag the pointer upward to zoom in. Drag the pointer downward to zoom out. **Click the scroll wheel & Drag**
Display Modes

Illustrate the part in various display modes.

- Wireframe
- Hidden lines Visible
- Hidden Lines Removed
- Shaded With Edges
- Shaded

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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)
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

Features used to build the box are:

- Extruded Base feature
- Fillet feature
- Shell feature
- Extruded 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.
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.

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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.

Wall Thickness

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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 $\div 2$

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Geometric Relationships

- **Tangent**
- **Parallel**
- **Horizontal**
- **Vertical**
- **Intersection**
- **Concentric**
- **Perpendicular**

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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
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.

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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).