Name: ______________________________ Department__________

Unit: Manuals / How-to

Activity 1: Exploring a Software Manual

Instructions: Read a manual on using SolidWorks and identify key technical vocabulary by scanning for specific information.

Technical Term / Function	Visual Reference (Icon/Image)	Definition or Function
1.________ Revolve		A. The area on the left that lists all sketches and features in chronological order.
2. ________Under-defined		B. A feature that connects multiple profiles on different planes to create complex shapes.
3. ________Extruded Boss/Base		C. Constraints like Parallel or Tangent that maintain the sketch's behavior.
4. ________FeatureManager		D. The specific file extension for a SOLIDWORKS Part document.
5. ________Loft		E. A tool that removes material to create holes or slots in a solid body.
6. ________.SLDPRT		F. A feature that creates a solid by rotating a profile around a centerline.
7. ________Geometric Relations		G. The central 0,0,0 point where a design should ideally begin.
8. ________Smart Dimension		H. A modeling system where changing one value updates all related geometry.
9. ________Chamfer		I. The state of a sketch indicated by blue lines, meaning it can still move.
10. ________Origin		J. The primary tool used to set the length, radius, or angle of a sketch entity.
11. ________Parametric		K. A feature used to round off sharp edges for safety or aesthetics.
12. ________Cut-Extrude		L. A feature that adds "depth" to a 2D sketch to make it a 3D solid.
13. ________Front, Top, Right		M. The state of a sketch indicated by black lines, meaning it is fully locked.
14. ________Fully Defined		N. A feature that creates a beveled or angled edge.
15. ________Fillet		O. The three standard reference planes used to start a new sketch.

Introduction

SOLIDWORKS is a computer-aided design (CAD) software application widely used in engineering, manufacturing, and industrial design. Engineers and technicians use the software to create accurate three-dimensional (3D) models that can later be developed into assemblies, technical drawings, and manufactured products. In industrial environments, even a small design mistake can affect production quality, material usage, or machine performance. For this reason, users must understand not only how to create a model, but also how to avoid common problems during the design process. This manual explains the basic procedures involved in creating a simple 3D part in SOLIDWORKS, including sketching, dimensioning, applying features, editing geometry, and saving completed models.

1. Starting a New Part Document

The first step in creating a 3D model is opening a new part document. After launching SOLIDWORKS, the user selects **File > New** and chooses **Part** as the document type. The graphics area then appears with an empty workspace and three default reference planes: Front Plane, Top Plane, and Right Plane. These planes determine the orientation of the model and serve as surfaces for sketching.

Selecting an incorrect plane at the beginning of the design process may create difficulties later when additional features or assemblies are added. Some beginner users choose a plane without considering the final orientation of the object, which may require unnecessary modifications later. Designers therefore usually examine the intended shape carefully before deciding which plane to use.

At the center of the workspace is the Origin point. Experienced users often begin sketches at the origin because sketches created far from the origin may become difficult to align or edit accurately. The FeatureManager Design Tree on the left side of the screen records all sketches and features created during modeling. If sketches and features are not named or organized properly, complex models may become difficult to manage later.

2. Creating a 2D Sketch

After selecting a reference plane, the user enters Sketch Mode to create a two-dimensional profile. A sketch forms the foundation of most 3D models in SOLIDWORKS. The software provides sketch tools such as Line, Rectangle, Circle, Arc, and Spline for creating geometry.

During sketching, users should focus on creating clean and simple geometry. Overly complicated sketches may increase the chance of modeling errors and make editing more difficult later. Some users attempt to include too many details in a single sketch instead of building the model gradually with multiple features.

SOLIDWORKS automatically displays visual guides such as horizontal and vertical relations, midpoint symbols, and tangent indicators to help users maintain accuracy. However, users sometimes ignore these visual references and create geometry that is slightly misaligned. Even small inaccuracies may later affect feature creation or assembly alignment.

Construction lines may also be added to help create symmetrical geometry or define reference axes. If construction geometry is missing in symmetrical designs, users may need to repeat editing procedures later to correct alignment problems.

3. Adding Dimensions and Geometric Relations

Once the sketch is complete, dimensions and geometric relations are applied to define the size and behavior of the sketch entities. Dimensions specify values such as length, width, diameter, radius, and angle. In SOLIDWORKS, dimensions are usually added using the Smart Dimension tool.

Applying accurate dimensions is important because incorrect measurements may produce parts that do not fit properly in assemblies or manufacturing processes. In industrial production, even small dimensional errors may result in wasted materials or equipment problems. For this reason, engineers usually review dimensions carefully before creating 3D features.

Geometric relations such as Parallel, Perpendicular, Horizontal, Vertical, Equal, and Tangent help maintain sketch stability during editing. Without sufficient relations, sketches may move unexpectedly when dimensions are modified. Some beginner users apply dimensions but forget to add proper geometric relations, which can cause the shape to distort during later adjustments.

SOLIDWORKS displays fully defined sketches in black and under-defined sketches in blue. Although under-defined sketches may still function temporarily, they often create instability during feature creation or model updates. Designers therefore usually attempt to fully define sketches before continuing to the next stage.

4. Creating a 3D Feature

After the sketch has been fully defined, the user converts the two-dimensional profile into a three-dimensional model using a feature tool. One of the most common features is Extruded Boss/Base, usually called Extrude. The Extrude feature extends the sketch perpendicular to the sketch plane to create depth and generate a solid body.

Before accepting the feature, SOLIDWORKS displays a preview of the extrusion. Users should inspect this preview carefully because selecting the wrong extrusion direction may create geometry opposite to the intended design. In some cases, users accidentally extrude the sketch in both directions or enter an incorrect extrusion depth, producing parts that are too thick or too thin for their intended application.

Simple extrusion is commonly used to create blocks, plates, and basic mechanical parts. However, complex industrial components usually require additional features and editing procedures. Designers often create the model gradually instead of attempting to complete all geometry in a single operation.

5. Modifying the Model with Additional Features

After the basic 3D shape has been created, users can modify the model by adding or removing material. One commonly used feature is Cut-Extrude, which removes material from the existing solid body. Engineers frequently use this feature to create holes, slots, and internal cavities.

Before applying Cut-Extrude, users should confirm that the sketch is positioned correctly on the selected face. Incorrect sketch placement may remove material from unintended areas of the model. In manufacturing situations, misplaced holes or openings may weaken the part or affect assembly performance.

Fillet and Chamfer are also commonly used modification features. The Fillet feature rounds sharp edges, while Chamfer creates beveled edges. These features improve appearance and may also improve safety by reducing sharp corners. However, excessive use of fillets may increase manufacturing difficulty or create unnecessary complexity in the model. Some designers apply decorative features too early in the modeling process, which may complicate future edits.

Unlike Extrude and Cut-Extrude, fillets and chamfers are usually applied directly to existing geometry without additional sketches. Designers often leave these finishing features until later stages of the design process to maintain flexibility while editing the model.

6. Using Advanced Modeling Features

SOLIDWORKS provides advanced modeling tools that allow users to create more complex geometry. One of these tools is Loft, which creates a solid by connecting multiple sketch profiles located on different planes. Loft is useful for producing smooth transitions between shapes, but poorly aligned profiles may generate twisted or unstable geometry.

Another advanced tool is Sweep, which creates geometry by moving a sketch profile along a selected path. Sweep features are commonly used for pipes, tubes, handles, and curved industrial components. However, if the profile or path contains errors, the feature may fail to generate properly. Designers therefore usually inspect both sketches carefully before applying the sweep operation.

The Revolve feature creates a solid by rotating a sketch around a centerline. This method is commonly used to create cylindrical parts such as shafts and handles. If the centerline is positioned incorrectly, the revolved feature may become asymmetrical or produce unexpected geometry. For this reason, accurate sketch preparation remains important even when advanced features are used.

7. Inspecting and Editing the Model

During the modeling process, users should inspect the model regularly to identify errors or incorrect geometry. The graphics area allows the model to be rotated, zoomed, and viewed from multiple angles. Careful inspection helps users detect problems such as overlapping geometry, incorrect feature placement, or missing details before the design becomes more complicated.

The FeatureManager Design Tree displays all sketches and features in sequence. If the modeling process is not organized properly, editing later features may become difficult because many operations depend on earlier sketches and dimensions. Experienced users therefore often rename important features and maintain a logical modeling sequence.

SOLIDWORKS uses parametric modeling, which means that changing one dimension may automatically affect other related features. Although this capability improves efficiency, unexpected changes may occur if relationships between features are not managed carefully. Users should therefore review the model after making major edits to confirm that all geometry updates correctly.

8. Saving the Completed Part

After the model has been completed and inspected, the user saves the part file by selecting **File > Save**. SOLIDWORKS part files use the extension .SLDPRT. Proper file naming and organization are important because parts are often used later in assemblies, simulations, and technical drawings.

In industrial environments, poor file organization may create confusion between different versions of a design. Accidentally modifying or replacing the wrong file can lead to production delays or incorrect manufacturing data. For this reason, engineers often follow standardized naming systems and organize files carefully within project folders.

Saving work regularly during the modeling process is also recommended because unexpected software or system problems may cause data loss if the file has not been updated recently.

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