Al-Mustaqbal University / College of Engineering	
Prosthetics & Orthotics Eng. Department	
Third Class	
Subject (Computer Application)	
Code (UOMU)	
Asst. Lec. Ghadeer Haider	
١st term – Lecture ٣	

INTRODUCING SOLIDWORKS

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The SOLIDWORKS Software

The SOLIDWORKS CAD software is a mechanical design automation application that lets designers quickly sketch out ideas, experiment with features and dimensions, and produce models and detailed drawings.

Concepts

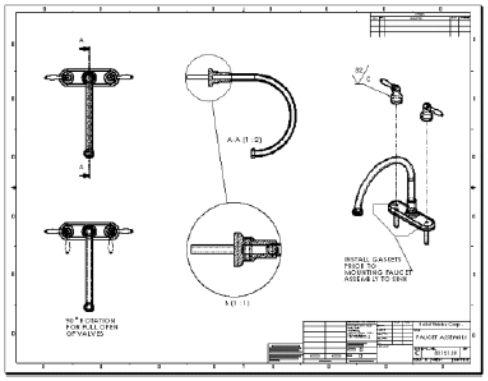
Parts are the basic building blocks in the SOLIDWORKS software. Assemblies contain parts or other assemblies, called subassemblies.
A SOLIDWORKS model consists of 3D geometry that defines its edges, faces, and surfaces.
The SOLIDWORKS software lets you design models quickly and precisely.

٣D Design

SOLIDWORKS uses a 3D design approach. As you design a part, from the initial sketch to the final result, you create a 3D model. From this model, you can create 2D drawings or mate components consisting of parts or subassemblies to create 3D assemblies. You can also create 2D drawings of 3D assemblies.

SOLIDWORKS 3D part

SOLIDWORKS 3D assembly



SOLIDWORKS 2D drawing generated from 3D model

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The difference between (Part) and (Assembly) in SolidWorks is:

Part

A ٣D model of a single component, such as a bolt, shaft, or base. It includes features like Extrude, Cut, and Fillet used to shape the object.

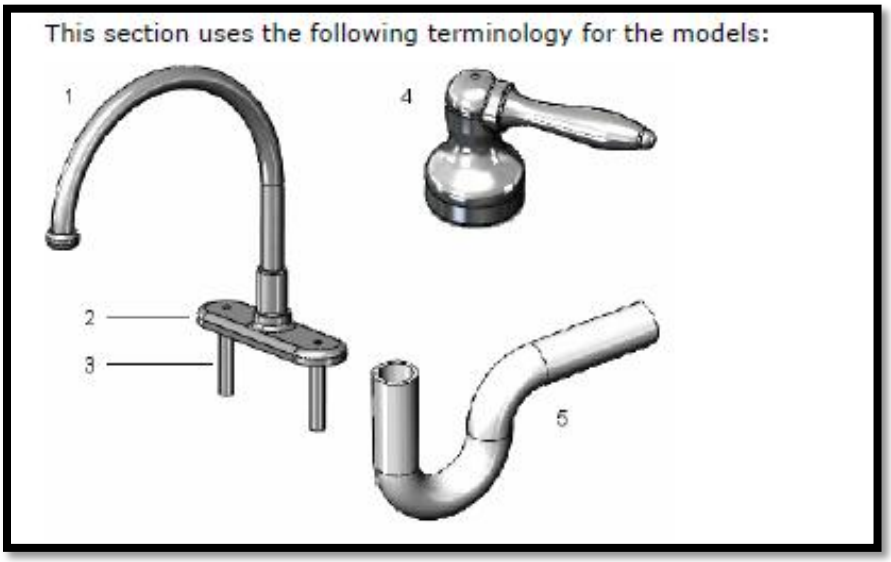
Assembly

Used to combine multiple Parts together to form a complete product, such as an engine or a machine. In an assembly, you define relationships between parts (called Mates) to control how they fit and move together.

In short:

Part = a single component

Assembly= a combination of multiple parts forming one product



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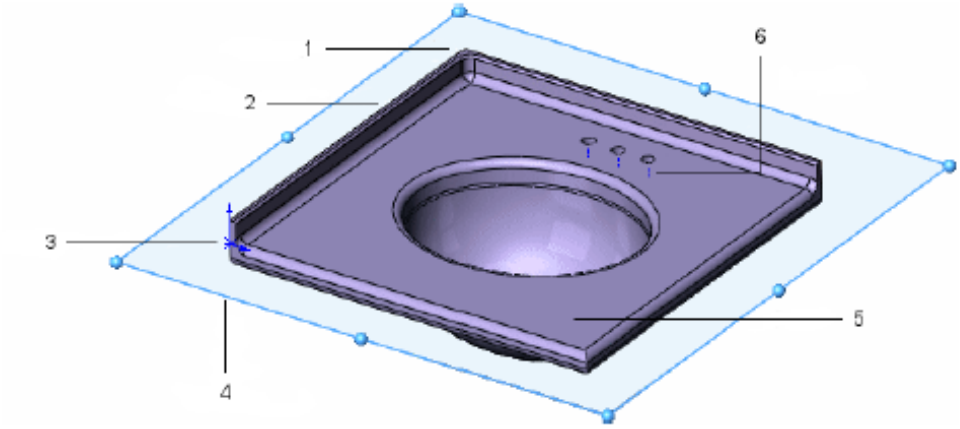
Terminology

These terms appear throughout the SOLIDWORKS software and documentation.



1	Vertex	Point at which two or more lines or edges intersect. You can select vertices for sketching and dimensioning, for example.
2	Edge	Location where two or more faces intersect and are joined together. You can select edges for sketching and dimensioning, for example.
3	Origin	Appears as two blue arrows and represents the (0,0,0) coordinate of the model. When a sketch is active, a sketch origin appears in red and represents the (0,0,0) coordinate of the sketch. You can add dimensions and relations to a *model* origin, but not to a sketch origin.
4	Plane	Flat construction geometry. You can use planes for adding a 2D sketch, section view of a model, or a neutral plane in a draft feature, for example.
5	Face	Boundaries that help define the shape of a model or a surface. A face is a selectable area (planar or nonplanar) of a model or surface. For example, a rectangular solid has six faces.
6	Axis	Straight line used to create model geometry, features, or patterns. You can create an axis in different ways, including intersecting two planes. The SOLIDWORKS application creates temporary axes implicitly for every conical or cylindrical face in a model.

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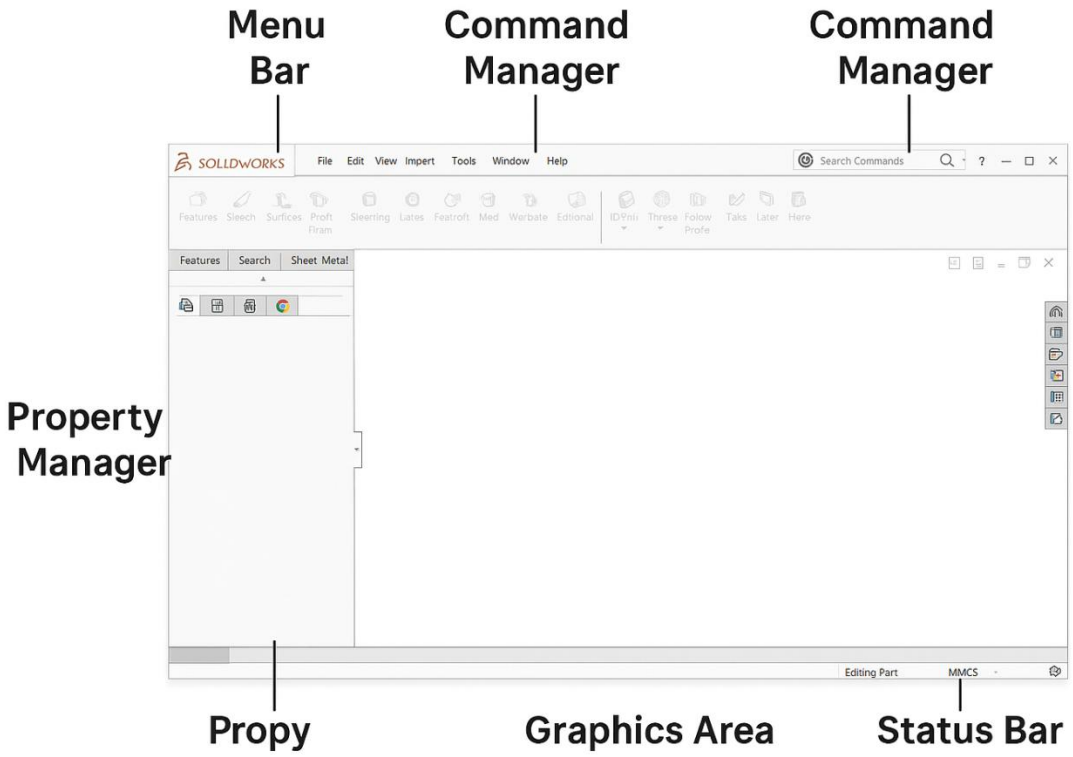

Menu Bar

- Located at the top of the screen.
- Contains drop-down menus such as (File, Edit, View, Insert, Tools, Window, Help)
- Used to access (commands and settings) for creating, editing, and managing your model or drawing.

Command Manager

- A toolbar that provides easy access to (commonly used commands).
- It changes depending on what you are working on (Part, Assembly, Drawing).
- Contains tabs like Features, Sketch, Evaluate, Sheet Metal, etc.

Property Manager

- Appears on the (left side) of the screen when you select a tool or feature.
- Displays options and settings for the current command.
- Used to (edit parameters), dimensions, or constraints of features.

Graphics Area

- The (main workspace) where you view and create models.
- Shows sketches, parts, assemblies, and drawings in ٢D or ٣D.
- You can rotate, zoom, and pan the view.

Status Bar

- Located at the bottom of the window.
- Displays information about the current state of the model or command.
- Shows units, selection info and messages or hints.

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Design Method

Before you actually design the model, it is helpful to plan out a method of how to create the model.
After you identify needs and isolate the appropriate concepts, you can develop the model: Create the sketches and decide how to dimension and where to apply relations.

Sketches

Select the appropriate features, such as extrudes and fillets, determine the best features to apply, and decide in what order to apply those features.

Features Assemblies

Select the components to mate and the types of mates to apply.

Step by step

Tool	Icon	Menu	Toolbar	CommandManager
New		**File > New**	Standard	Menu Bar
Save		**File > Save**	Standard	Menu Bar
Options		**Tools > Options**	Standard	Menu Bar
Sketch		**Insert > Sketch**	Sketch	Sketch
Smart Dimension		**Tools > Dimensions > Smart**	Sketch	Sketch
Rectangle		**Tools > Sketch Entities > Rectangle**	Sketch	Sketch
Extruded Boss/Base		**Insert > Boss/Base > Extrude**	Features	Features
Shell		**Insert > Features > Shell**	Features	Features
Insert Components		**Insert > Component > Existing Part/Assembly**	Assembly	Assembly
Mate		**Insert > Mate**	Assembly	Assembly

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