



جامعة الفرات الاوسط التقنية
الكلية التقنية الهندسية / النجف الاشرف

Lecture-05

SOLIDWORKS Fundamentals

Dr. Dhafer Manea Hachim AL-HASNAWI

Assist Prof

Al-Furat Al-Awsat Technical University

Engineering Technical College / Najaf

email: coj.dfr@atu.edu.iq

Concepts

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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. SOLIDWORKS models are:

- Defined by 3D design
- Based on components

3D 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.

When designing a model using SOLIDWORKS, you can visualize it in three dimensions, the way the model exists once it is manufactured.

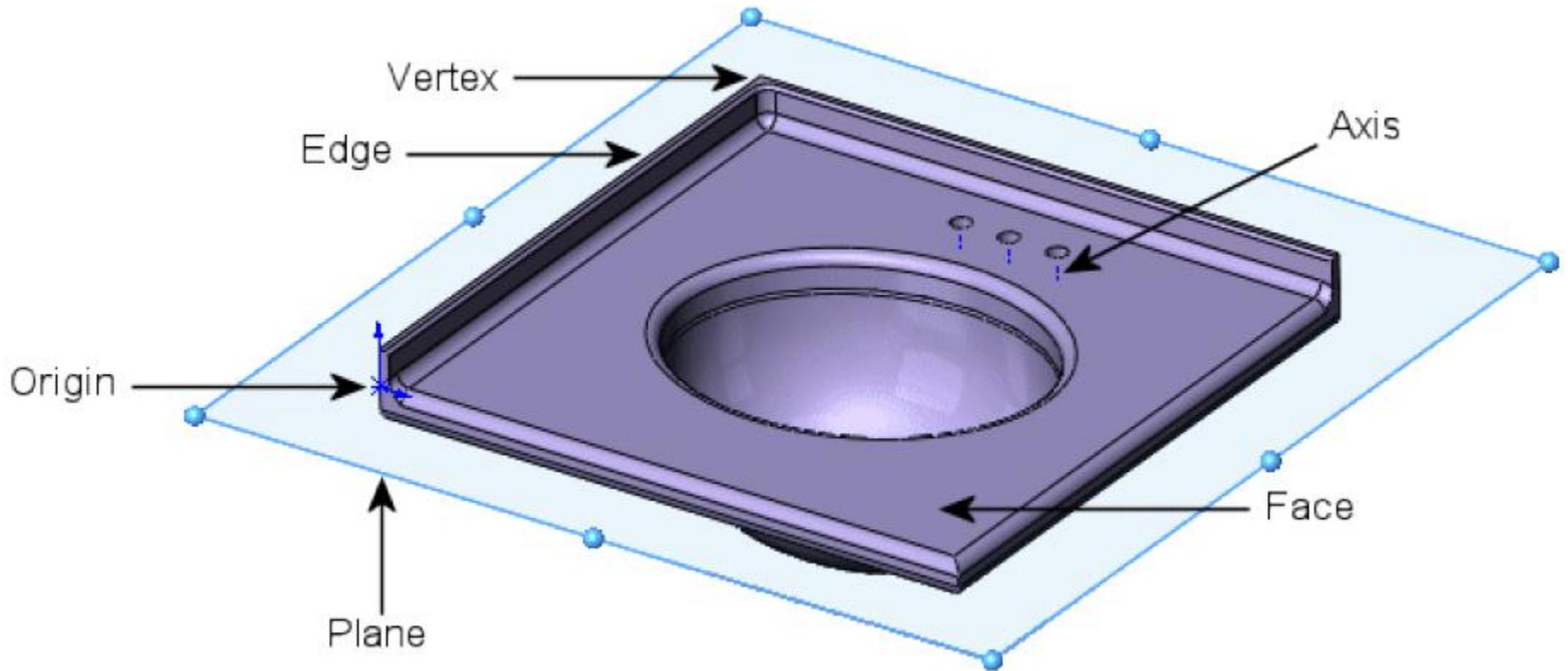
Terminology

Terminology

These terms appear throughout the SOLIDWORKS software and documentation.

- 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.
- 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.
- 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.
- 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.
- Edge** Location where two or more faces intersect and are joined together. You can select edges for sketching and dimensioning, for example.
- Vertex** Point at which two or more lines or edges intersect. You can select vertices for sketching and dimensioning, for example.

Terminology



User Interface

The SOLIDWORKS application includes user interface tools and capabilities to help you create and edit models efficiently, including:

Windows Functions

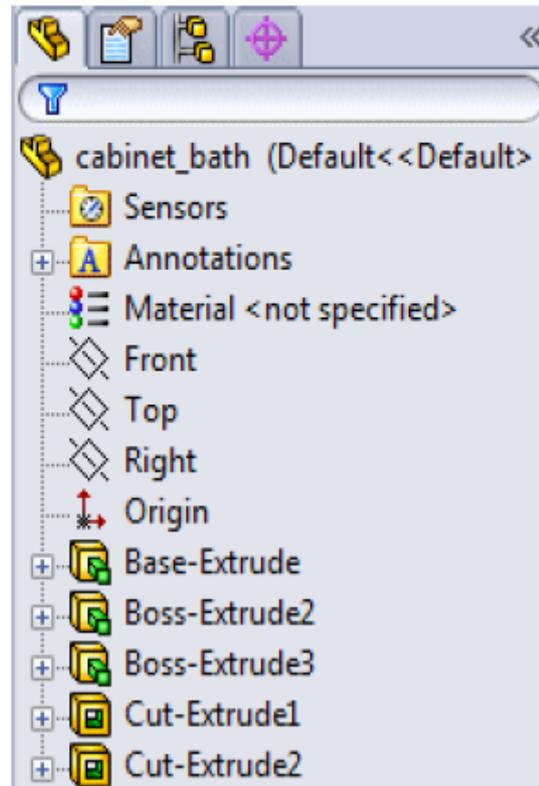
The SOLIDWORKS application includes familiar Windows functions, such as dragging and resizing windows. Many of the same icons, such as print, open, save, cut, and paste are also part of the SOLIDWORKS application.

SOLIDWORKS Document Windows

SOLIDWORKS document windows have two panels. The left panel, or Manager Pane, contains:

FeatureManager® design tree

Displays the structure of the part, assembly, or drawing. Select an item from the FeatureManager design tree to edit the underlying sketch, edit the feature, and suppress and unsuppress the feature or component, for example.



PropertyManager

Provides settings for many functions such as sketches, fillet features, and assembly mates.



ConfigurationManager

Lets you create, select, and view multiple configurations of parts and assemblies in a document. Configurations are variations of a part or assembly within a single document. For example, you can use configurations of a bolt to specify different lengths and diameters.

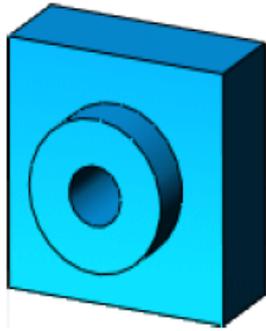


Design Process

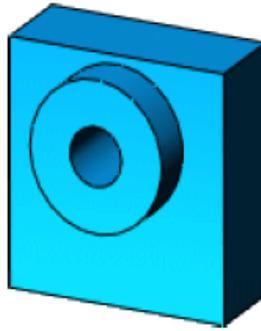
- The design process usually involves the following steps:
 - Identify the model requirements.
 - Conceptualize the model based on the identified needs.
 - Develop the model based on the concepts.
 - Analyze the model.
 - Prototype the model.
 - Construct the model.
 - Edit the model, if needed.

Design Intent

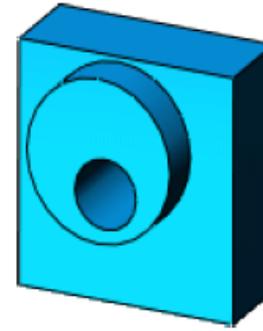
- Design intent determines how you want your model to react as a result of the changes you need to make to the model. For example, if you make a boss with a hole in it, the hole should move when the boss moves:



Original part



Design intent maintained
when boss moves



Design intent not maintained
when boss moves

Design intent is primarily about planning. How you create the model determines how changes affect it. The closer your design implementation is to your design intent, the greater the integrity of the model.

Various factors contribute to the design process, including:

Current needs

Understand the purpose of the model to design it efficiently.

Future considerations

Anticipate potential requirements to minimize redesign efforts.

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:

Sketches

Create the sketches and decide how to dimension and where to apply relations.

Features

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

Assemblies

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

Sketches

- The sketch is the basis for most 3D models.

Creating a model usually begins with a sketch. From the sketch, you can create features. You can combine one or more features to make a part. Then, you can combine and mate the appropriate parts to create an assembly. From the parts or assemblies, you can then create drawings.

A sketch is a 2D profile or cross section. To create a 2D sketch, you use a plane or a planar face. In addition to 2D sketches, you can also create 3D sketches that include a Z axis, as well as the X and Y axes.

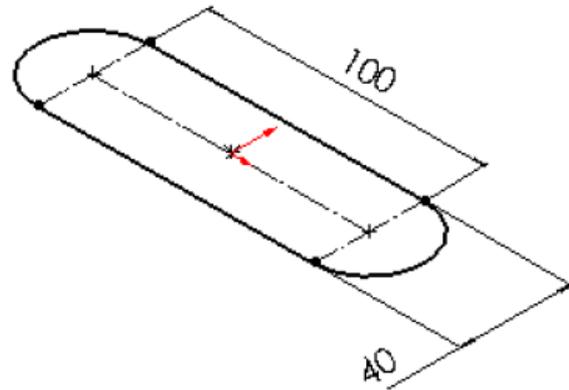
There are various ways of creating a sketch. All sketches include the following elements:

Features

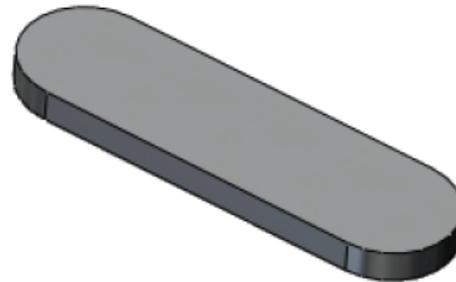
- Once you complete the sketch, you can create a 3D model using features such as an extrude (the base of the faucet) or a revolve (the faucet handle).



Create the sketch



Dimension the sketch



Extrude the sketch 10mm

Assemblies

You can combine multiple parts that fit together to create assemblies.

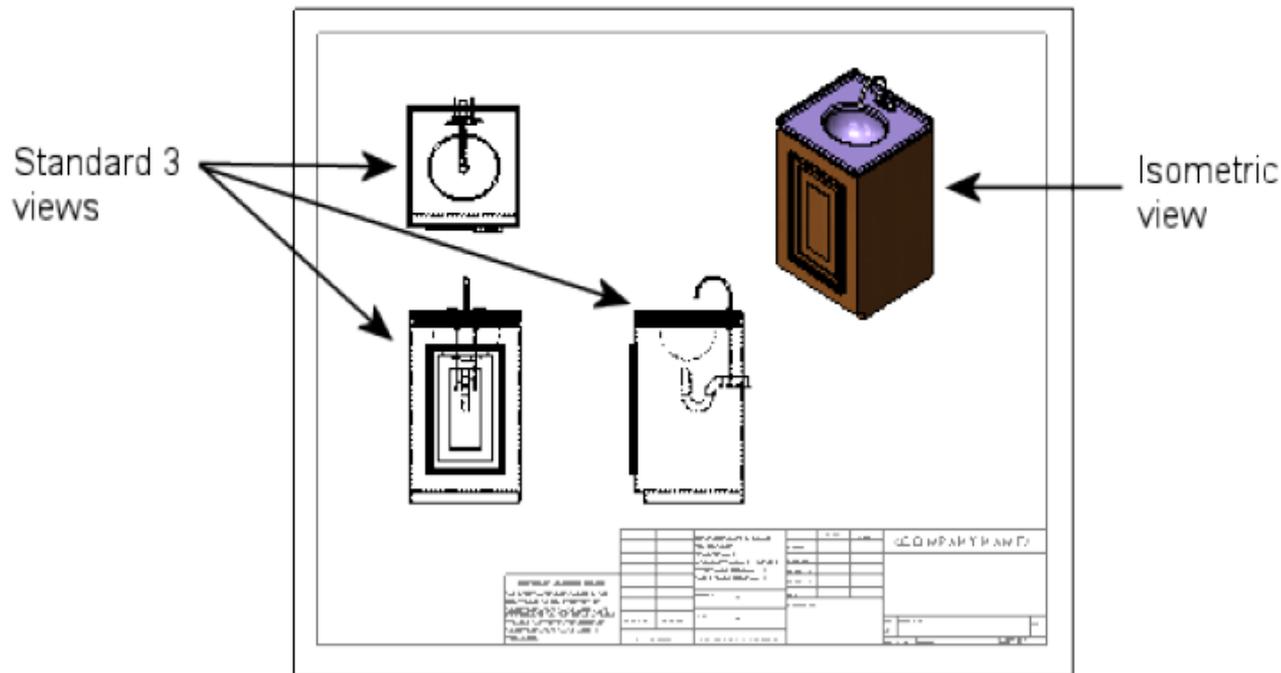
You integrate the parts in an assembly using **Mates**, such as **Concentric** and **Coincident**. Mates define the allowable direction of movement of the components. In the faucet assembly, the faucet base and handles have concentric and coincident mates.



Drawings

You create drawings from part or assembly models.

Drawings are available in multiple views such as standard 3 views and isometric views (3D). You can import the dimensions from the model document and add annotations such as datum target symbols.



Model Editing

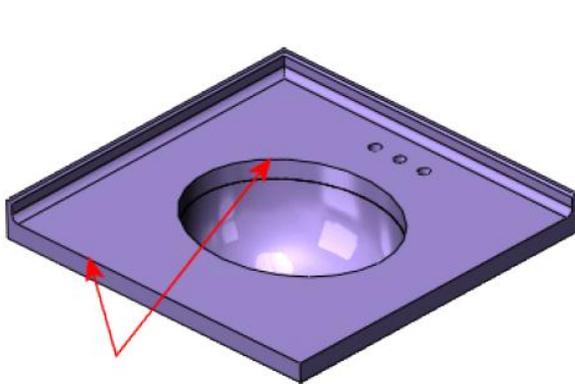
Use the SOLIDWORKS Feature Manager design tree and the Property Manager to edit sketches, drawings, parts, or assemblies. You can also edit features and sketches by selecting them directly from the graphics area. This visual approach eliminates the need to know the name of the feature. Editing capabilities include:

Edit sketch

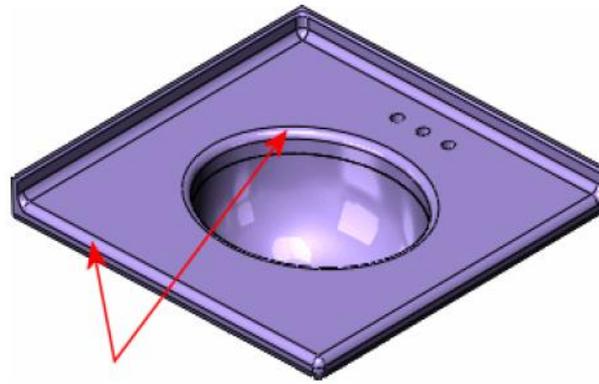
You can select a sketch in the FeatureManager design tree and edit it. For example, you can edit sketch entities, change dimensions, view or delete existing relations, add new relations between sketch entities, or change the size of dimension displays. You can also select the feature to edit directly from the graphics area.

Edit feature

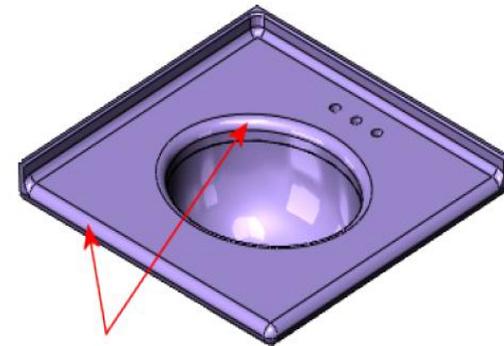
Once you create a feature, you can change most of its values. Use **Edit Feature** to display the appropriate PropertyManager. For example, if you apply a **Constant radius** fillet to an edge, you display the Fillet PropertyManager where you can change the radius. You can also edit dimensions by double-clicking the feature or sketch in the graphics area to show the dimensions and then change them in place.



No fillet feature



Fillet feature: 12mm applied



Fillet feature: 18mm applied