SolidWorks helps you move through the design cycle smarter. We live in a 3D world, so by designing in a 3D environment, your team can create real solutions faster, more accurately and more creatively.
2D design tools and SolidWorks have fundamentally different approaches. In 2D design tools, you design in a 2D environment. In SolidWorks, you design in a 3D environment, and you create 2D drawings based on the 3D model.

Types of Models
Computer-Aided Design software packages handle models in the following ways:

• 2D drawings
• Wireframe models

• Surface models (organic shapes)
• Solid models (mechanical parts and assemblies)

SolidWorks creates solid models, but it can also import, create, and manipulate surfaces, view models in wireframe mode, and generate 2D drawings from the 3D solid models. ScanTo3D tools, available in SolidWorks Premium, import mesh and point cloud data from which you can create surfaces and solid models.

Sketching versus Drawing
In SolidWorks, drawings are the 2D documents that you create from 3D part or assembly models. The tools that are considered drawing tools in 2D CAD programs are sketching tools in SolidWorks. When developing models in SolidWorks, you sketch geometric entities (such as rectangles and circles) as the basis for solid features (such as extrusions, revolves, and cuts). You can sketch entities approximately, then dimension the entities exactly.
The general procedure, from sketch through model to drawing, is as follows:

1. IN A PART DOCUMENT, OPEN A SKETCH AND SKETCH AN ENTITY, SUCH AS A RECTANGLE, APPROXIMATELY.
2. DIMENSION THE SKETCH EXACTLY.

3. EXTRUDE THE SKETCH TO FORM A 3D SOLID BASE FEATURE, WHICH BECOMES THE BASIS OF A PART.
4. OPEN A NEW DRAWING, INSERT THE PART AS A 2D STANDARD 3 VIEW, AND INSERT THE DIMENSIONS.

Feature-based Models

Just as an assembly consists of individual parts, a SolidWorks part consists of individual features.

The first feature you create in a part is the base. This feature is the basis on which you create the other features. The base feature can be an extrusion, a revolve, a sweep, a loft, thickening of a surface, or a sheet metal flange. However, most base features are extrusions. The following are some of the features you can use to make parts in SolidWorks.

- **Extrude** - Extrude creates a feature by extruding a 3D object from a 2D sketch, essentially adding the third dimension. An extrusion can be a base (in which case it always adds material), a boss (which adds material, often on another extrusion), or a cut (which removes material).
- **Revolve** - Revolve creates a feature that adds or removes material by revolving one or more sketch profiles around a centerline. The feature can be either a solid, a thin feature, or a surface.
- **Loft** - Loft creates a feature by making transitions between profiles. A loft can be a base, boss, cut, or surface.
- **Sweep** - Sweep creates a base, boss, cut, or surface by moving a profile (section) along a path.
- **Boundary** - Boundary creates very high quality, accurate features useful for creating complex shapes for the consumer product design, medical, aerospace, and mold markets. A boundary can be a base, boss, cut, or surface.
SolidWorks features are of two types: sketched and applied.

- **Sketched features** such as extrusions, revolves, sweeps, and lofts are based on sketch geometry.
- **Applied features** such as chamfers, fillets, and shells are applied directly to the model.

SolidWorks features are always added to the model, whether they add or remove material. You can modify features after creating them.

**Types of Files**

In SolidWorks, you can open any number of part, assembly, or drawing documents at the same time:

- **Part** (.sldprt)
- **Assembly** (.sldasm)
- **Drawing** (.slddrw)

SolidWorks gives the three basic file types their own extensions to facilitate finding and filtering files based on content. From an active document, you can open related files as follows:

- Open a drawing from its associated part or assembly document
- Open a part or assembly document from a drawing view
- Open a part from the component in its assembly document

Typically, you begin in a part document, creating a part. When you have several parts, you can assemble them in an assembly document. You can create drawings from both parts and assemblies.
Glass Box Visualization

In 2D CAD, you think of projecting the views of a 3D model onto the sides of a glass box to visualize the 2D drawings. If you have a 2D model, or are designing in 2D, think in the opposite direction. Visualize the 2D drawings folded up onto the sides of the glass box and projected into a 3D model.

The following example started as a 2D drawing and was converted into a 3D model.

Templates

SolidWorks provides templates for parts, assemblies, and a variety of drawing styles. You can create custom templates by opening existing templates (or any document file), setting options and inserting items (title blocks, base parts, and so on), then saving the documents as templates.

Template files have the following extensions:
- .prtdot (parts)
- .asmdot (assemblies)
- .drwdot (drawings)

The default template for the A size landscape drawing format includes standard formatting and text that you can edit in the drawing sheet format.
Parametric Dimensions

In SolidWorks, dimensions drive the model geometry; changing dimensions changes the shape of the model. You can relate dimensions to each other in equations.

1. OPEN A SKETCH, SKETCH A RECTANGLE, AND DIMENSION THE RECTANGLE.

2. MODIFY THE DIMENSIONS AS NEEDED WHILE CREATING THE SKETCH.

3. EXTRUDE A BLOCK BASE FEATURE.

4. COMPLETE THE FEATURE TO CLOSE THE SKETCH AND SHOW THE SOLID IN SHADED MODE.

5. TO MODIFY THE BLOCK, EDIT THE SKETCH, DOUBLE-CLICK A DIMENSION AND MODIFY THE VALUE.

6. EXIT THE SKETCH TO REBUILD THE SOLID WITH THE NEW DIMENSION.

Note: You can also use Instant3D to modify model geometry.
Design Intent

Design intent is how your model behaves when dimensions are modified.

An example of design intent is how you create and dimension a hole in a block. The hole can be a certain distance from a corner or edge, or it can be in the middle of the face, for example. If the size of the block or the hole changes, the part rebuilds correctly if the design intent has been considered in the definition.

SolidWorks captures the intent of a design, including relations, parameters, and model behavior. You can draw lines approximately, and later dimension them exactly. You can also change the sketch and feature dimensions at any time and rebuild the part.

In the following example, one hole is fixed, one is driven by an equation, and the other two are mirrored. As the size of the hinge changes, the holes remain properly spaced along the length and width.

Configurations

Configurations in SolidWorks allow you to create multiple variations of a part or assembly model within a single document. Configurations are a convenient way to develop and manage families of models with different dimensions, components, or other parameters.

You can create configurations manually, or you can use a design table to create multiple configurations simultaneously. Design tables provide a convenient way to create and manage configurations in a worksheet. You can use design tables in both part and assembly documents.
You can display design tables in drawings.

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**Exploded Views**

In SolidWorks, you can configure assemblies into exploded views, and you can include explode lines. When you insert assemblies into drawing views, you can specify that the exploded configurations be shown.
In Conclusion

Modeling in 3D helps you stay organized and in touch with the real world you're designing for. With SolidWorks, increased speed and accuracy free your design team to be more creative, so you can design products smarter, faster and better.