

# IMPLEMENTATION GUIDE: MODELING IN 3D WITH SOLIDWORKS

## Overview

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.

The screenshot displays the SolidWorks interface. On the left, a 3D model of a complex mechanical assembly is shown. Below the model is a BOM table with the following data:

ID	PART NUMBER	DESCRIPTION	QTY
1	RSM-92447	COVER CASING	1
2	RSM-90014	TAPER ROLLER BEARING	2
3	RSM-83734	SEAL	2
4	RSM-97486	INTERNAL GEAR	1
5	RSM-38311	KEY	1
6	RSM-83892	OUTPUT SHAFT	1
7	RSM-14578	BEARING RING	2
8	RSM-40113	OUTPUT SHAFT SEAL	2
9	RSM-46589	G-RING	1
10	RSM-98663	WASHER	1
11	RSM-98650	MECH. WASH.	1
12	RSM-94251	INTERNAL DISC/DRIVE SHAFT	1
13	RSM-84652	INTERNAL DISC/DRIVE SHAFT	1
14	RSM-40112	INPUT SHAFT SEAL	1
15	RSM-03323	INPUT SHAFT	1
16	RSM-64485	SKF 6205-1D C3 HC 1D 68	1
17	RSM-97177	WASHER	1
18	RSM-06418	CRANK	100
19	RSM-01225	TRUSS 5000-75-S075	1
20	RSM-97140	TRUSS 15000-68-S1488	1
21	RSM-04787	OUTPUT SHAFT COVER	1
22	RSM-97136	SS HASSET DISC/DRIVE SHAFT	1

On the right side of the interface, there is a legend for the BOM table:

- Missing Columns
- Extra Columns
- Missing Rows
- Extra Rows
- Failed Rows (On the basis of "PART NUMBER")

Below the legend is a table with the following data:

PART NUMBER	Column Name	BOM 1 Value	BOM 2 Value
RSM-20014	ITEM	2	3
RSM-83734	ITEM	3	4
RSM-97486	ITEM	4	5
RSM-38311	ITEM	5	6
RSM-83892	ITEM	6	7
RSM-14578	ITEM	7	8
RSM-40113	ITEM	8	9
RSM-46589	ITEM	9	10
RSM-98663	ITEM	10	11
RSM-98650	ITEM	11	12
RSM-94251	ITEM	12	13
RSM-84652	ITEM	13	14
RSM-40112	ITEM	14	15
RSM-03323	ITEM	15	16
RSM-64485	ITEM	16	17
RSM-93772	ITEM	17	18

At the bottom of the screenshot, an exploded view of a component is shown with a corresponding BOM table:

ID	PART NUMBER	DESCRIPTION	QTY
1	RSM-92447	COVER CASING	1
2	RSM-90014	TAPER ROLLER BEARING	2
3	RSM-83734	SEAL	2
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2D design tools and SolidWorks have fundamentally different approaches. With 2D design tools, you design in a 2D environment. With SolidWorks, you design in a 3D environment, and you create 2D drawings based on the 3D model.

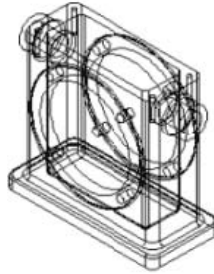
## DRAFTING

Computer-Aided Design (CAD) software packages handle models in the following ways:

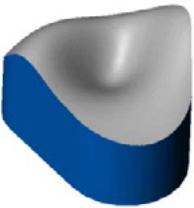
### 2D DESIGN



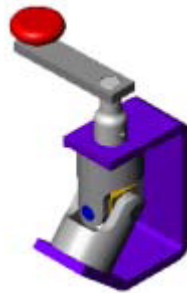
### WIREFRAME MODELS



### SURFACE MODELS (ORGANIC SHAPES)



### SOLID MODELS (MECHANICAL PARTS AND ASSEMBLIES)



*SolidWorks creates solid models. It can also import, create, and manipulate surfaces, view models in wireframe mode, and generate 2D drawings from the 3D solid models*

In SolidWorks, you can not only create solid models, but 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

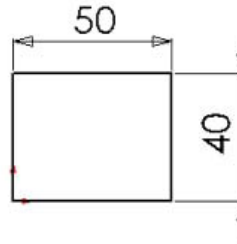
In SolidWorks, drawings are the 2D documents that you create from 3D part or assembly models. The tools that are considered drawing tools in 2D CAD programs are sketching tools in SolidWorks. When developing models in SolidWorks, you sketch geometric entities (such as rectangles and circles) as the basis 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:

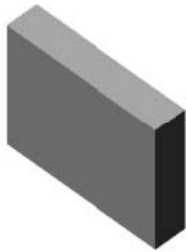
1. In a **part** document, **open** a sketch and sketch an entity, such as a **rectangle**, approximately.



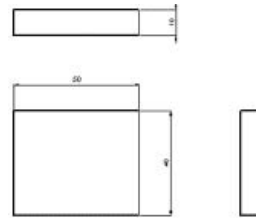
2. **Dimension** the sketch exactly.



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.



## PARTS VERSUS ASSEMBLIES

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.

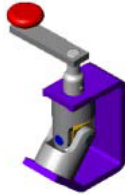
## TYPES OF FILES

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

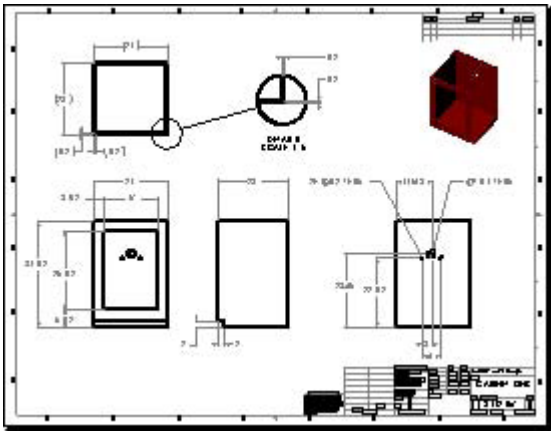
Part (.sldprt)



Assembly (.sldasm)



Drawing (.slddrw)



*Typically, you begin in a part document, creating a part. When you have several parts, you can assemble them in an assembly document*

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:

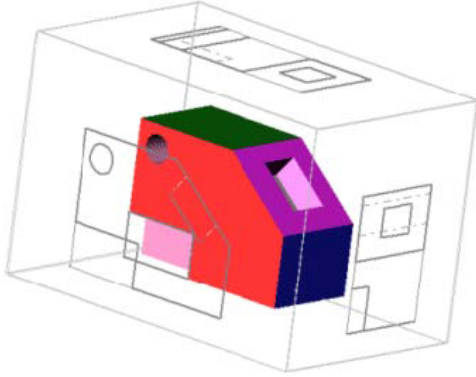
- Open a drawing from its associated part or assembly document
- Open a part or assembly document from a drawing view
- Open a part from the component in its assembly document

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.



*Visualize the 2D drawings folded up onto the sides of the glass box and projected 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)

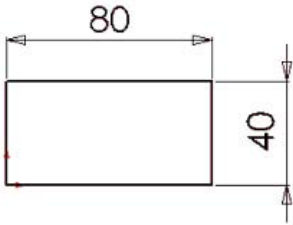
The default template for the A size landscape drawing format includes standard formatting and text that you can edit in the drawing sheet format.



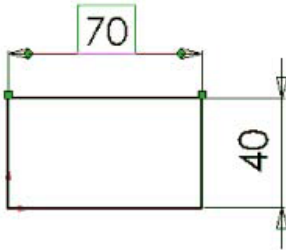
## PARAMETRIC DIMENSIONS

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

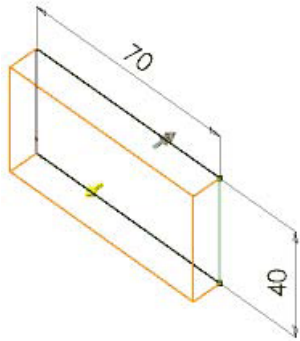
1. Open a sketch, sketch a rectangle, and dimension the rectangle.



2. Modify the dimensions as needed while creating the sketch.



3. Extrude a block base feature.

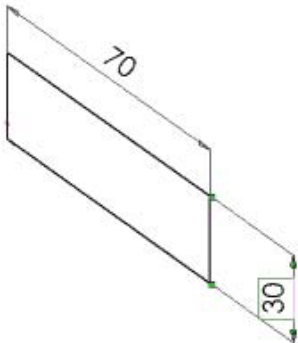


4. Complete the feature to close the sketch and show the solid in shaded mode.

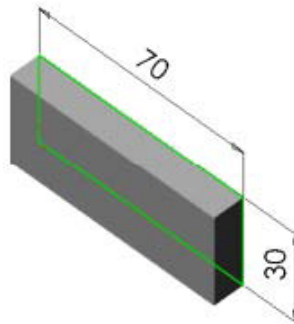


*In SolidWorks, dimensions drive the model geometry; changing dimensions changes the shape of the model*

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.

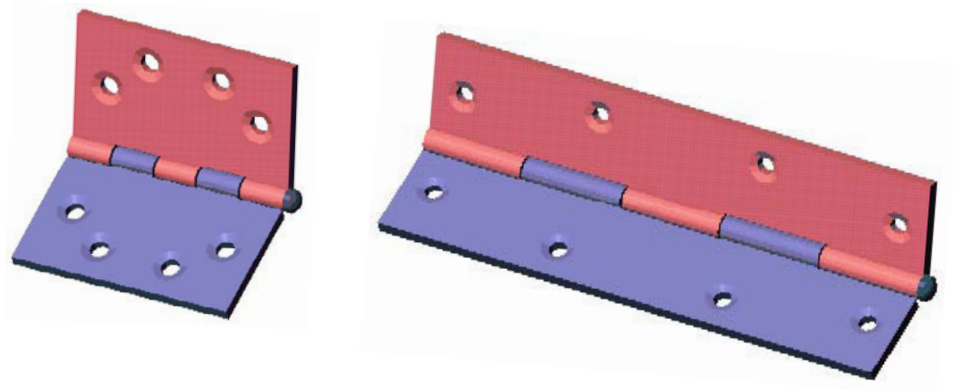
## DESIGN INTENT

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 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.

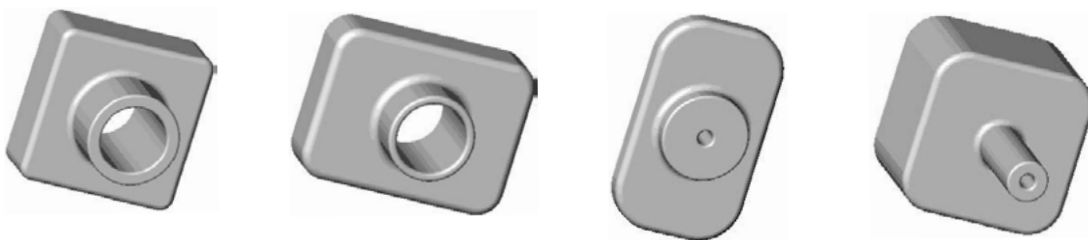


*Design intent is how your model behaves when dimensions are modified*

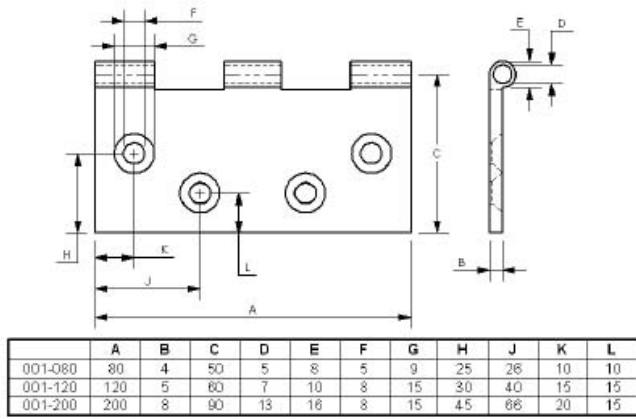
## CONFIGURATIONS

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.

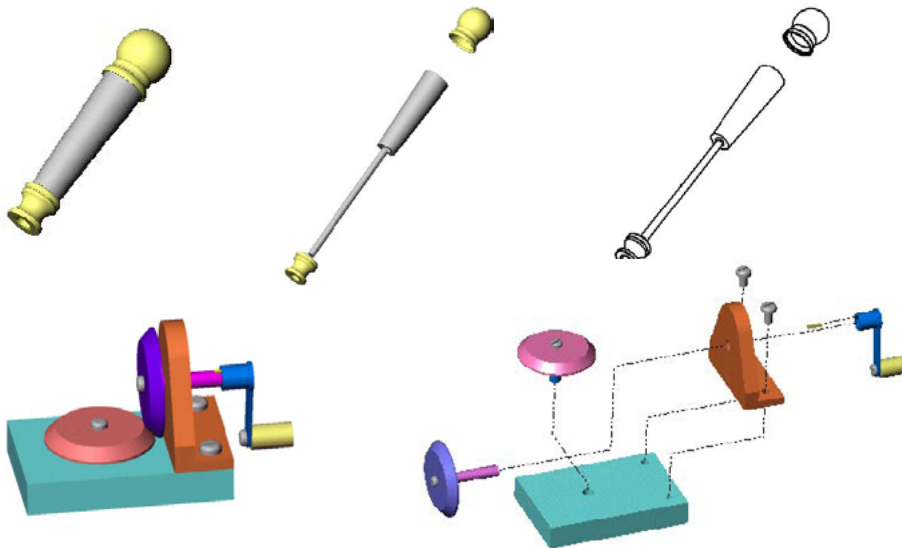


You can display design tables in drawings.



## 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.



## 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.

Additional ideas and help are available on the SolidWorks website at [www.solidworks.com](http://www.solidworks.com).

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