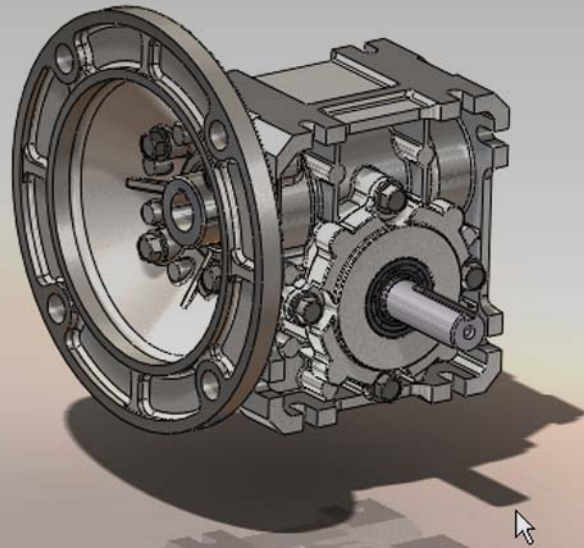


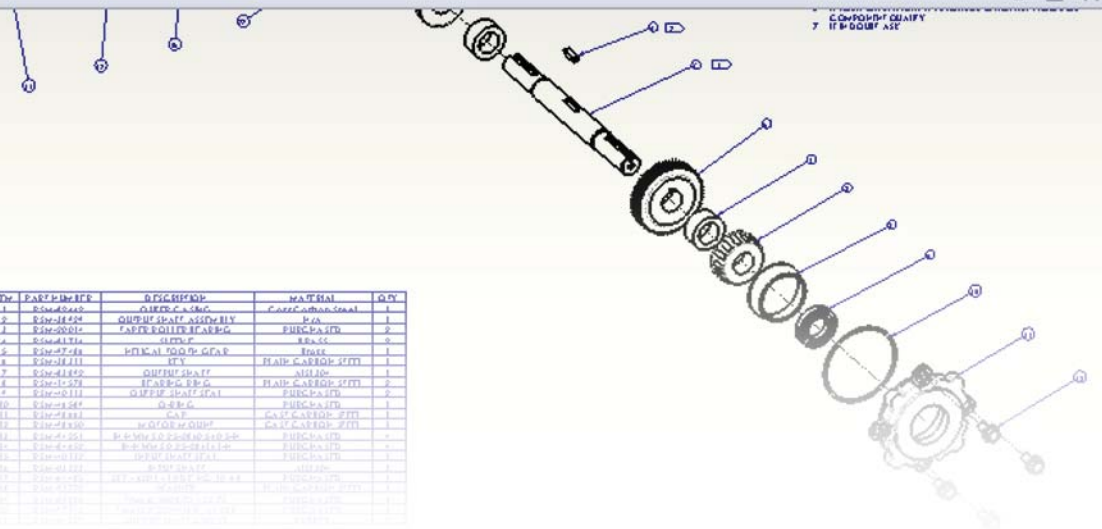
Implementation Guide: Modeling in 3D with SolidWorks®



Item	PART NUMBER	DESCRIPTION	QTY
1	RSM-92442	QUIET CASING	1
2	RSM-20014	INPUT ROLLER BEARING	2
3	RSM-83734	SHIM	2
4	RSM-97486	HELICAL INPUT 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	O-RING	1
10	RSM-98663	CAP	1
11	RSM-98650	MOTOR MOUNT	1
12	RSM-94251	N-HMM5 D25-28X10-S-N	▲
13	RSM-84652	N-HMM5 D25-28X10-N	▲
14	RSM-40112	INPUT SHAFT SEAL	1
15	RSM-03323	INPUT SHAFT	1
16	RSM-64485	SC1-6200-11 DE NC 10-68	1
17	RSM-93772	WASHER	1
18	RSM-06616	CRFASE	100
19	RSM-D1225	INVARC SDCB-7S-SD7S	1
20	RSM-97143	INVARC NSCDB-68-SI 688	1
21	RSM-04787	OUTPUT SHAFT COVER	1
22	RSM-71136	SS HAISCI D37S24XD37S-WX-N	1

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

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



Qty	PART NUMBER	DESCRIPTION	MATERIAL	QTY
1	RSM-98663	INPUT SHAFT SEAL	FLUOROPOLYMER	1
2	RSM-20014	INPUT ROLLER BEARING	STEEL	2
3	RSM-83734	SHIM	BRASS	2
4	RSM-97486	HELICAL INPUT GEAR	STEEL	1
5	RSM-38311	KEY	STEEL	1
6	RSM-83892	OUTPUT SHAFT	STEEL	1
7	RSM-14578	BEARING RING	STEEL	2
8	RSM-40113	OUTPUT SHAFT SEAL	FLUOROPOLYMER	2
9	RSM-46589	O-RING	FLUOROPOLYMER	1
10	RSM-98663	CAP	ALUMINUM	1
11	RSM-98650	MOTOR MOUNT	ALUMINUM	1
12	RSM-94251	N-HMM5 D25-28X10-S-N	STEEL	1
13	RSM-84652	N-HMM5 D25-28X10-N	STEEL	1
14	RSM-40112	INPUT SHAFT SEAL	FLUOROPOLYMER	1
15	RSM-03323	INPUT SHAFT	STEEL	1
16	RSM-64485	SC1-6200-11 DE NC 10-68	STEEL	1
17	RSM-93772	WASHER	STEEL	1
18	RSM-06616	CRFASE	STEEL	100
19	RSM-D1225	INVARC SDCB-7S-SD7S	INVAR	1
20	RSM-97143	INVARC NSCDB-68-SI 688	INVAR	1
21	RSM-04787	OUTPUT SHAFT COVER	ALUMINUM	1
22	RSM-71136	SS HAISCI D37S24XD37S-WX-N	STAINLESS STEEL	1

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.

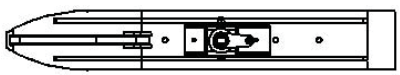


2D design tools and SolidWorks have fundamentally different approaches. In 2D design tools, you design in a 2D environment. In SolidWorks, you design in a 3D environment, and you create 2D drawings based on the 3D model.

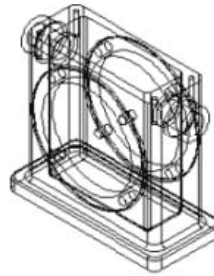
Types of Models

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

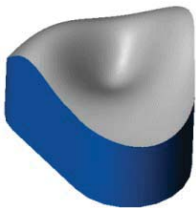
- 2D drawings



- 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

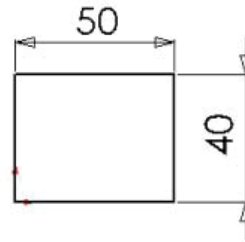
In SolidWorks, drawings are the 2D documents that you create from 3D part or assembly models. The tools that are considered drawing tools in 2D CAD programs are sketching tools in SolidWorks. When developing models in SolidWorks, you sketch geometric entities (such as rectangles and circles) as the basis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

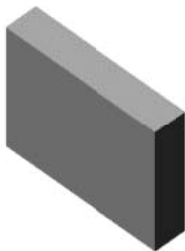
1. IN A **PART** DOCUMENT, **OPEN** A SKETCH AND SKETCH AN ENTITY, SUCH AS A **RECTANGLE**, APPROXIMATELY.



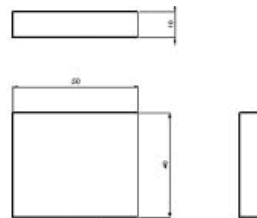
2. **DIMENSION** THE SKETCH EXACTLY.



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.

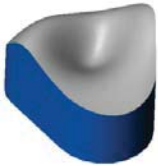
- **Sketched features** such as extrusions, revolves, sweeps, and lofts are based on sketch geometry.
- **Applied features** such as chamfers, fillets, and shells are applied directly to the model.

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:

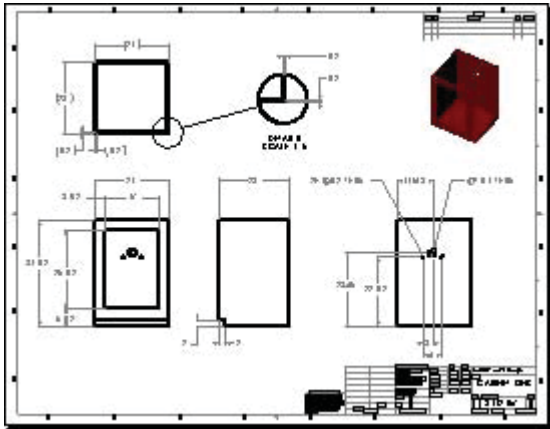
- Part (.sldprt)



- 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:

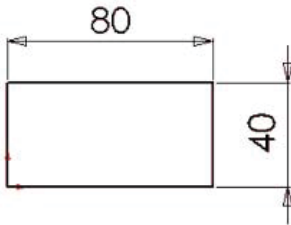
- Open a drawing from its associated part or assembly document
- Open a part or assembly document from a drawing view
- Open a part from the component in its assembly document

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.

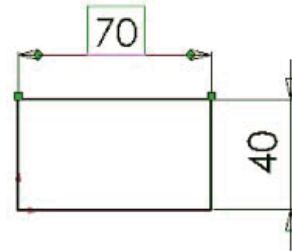
Parametric Dimensions

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

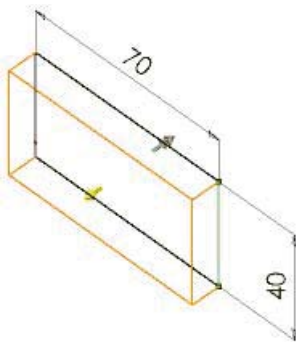
1. OPEN A SKETCH, SKETCH A RECTANGLE, AND DIMENSION THE RECTANGLE.



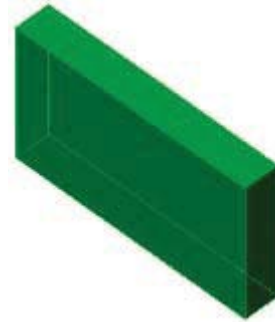
2. MODIFY THE DIMENSIONS AS NEEDED WHILE CREATING THE SKETCH.



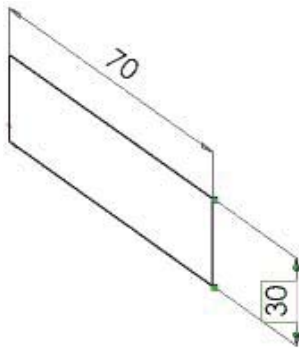
3. EXTRUDE A BLOCK BASE FEATURE.



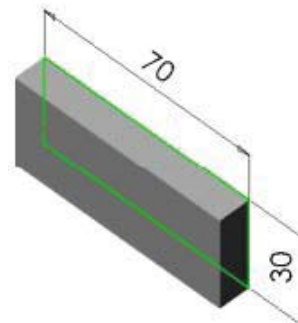
4. COMPLETE THE FEATURE TO CLOSE THE SKETCH AND SHOW THE SOLID IN SHADED MODE.



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.

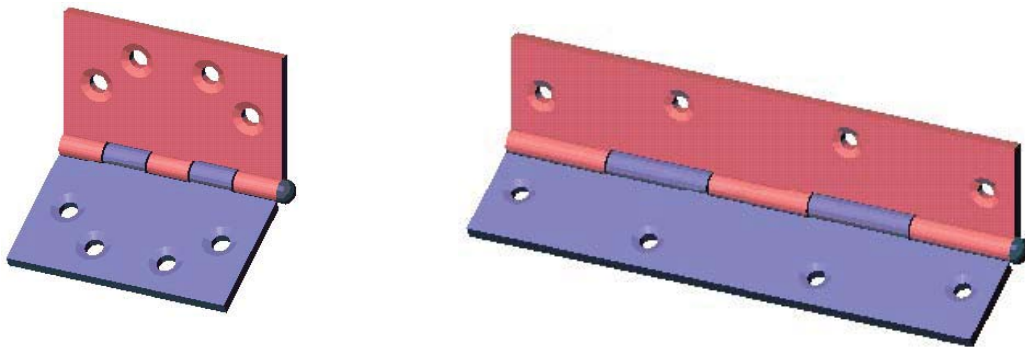
Design Intent

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.

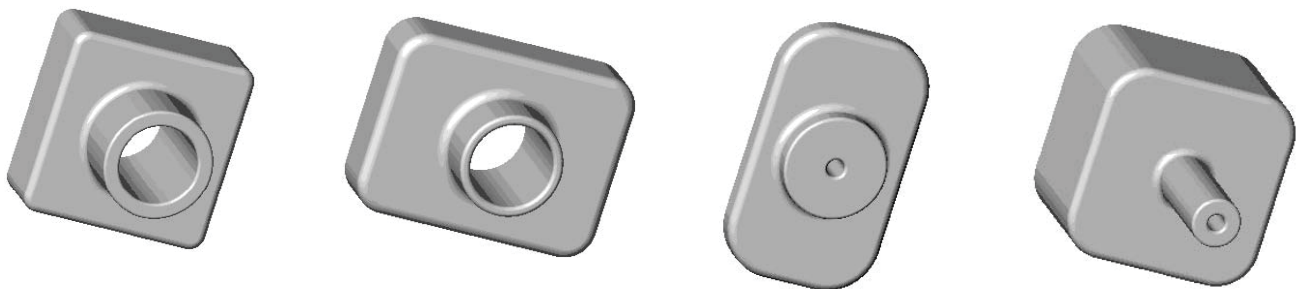
In the following example, one hole is fixed, one is driven by an equation, and the other two are mirrored. As the size of the hinge changes, the holes remain properly spaced along the length and width.



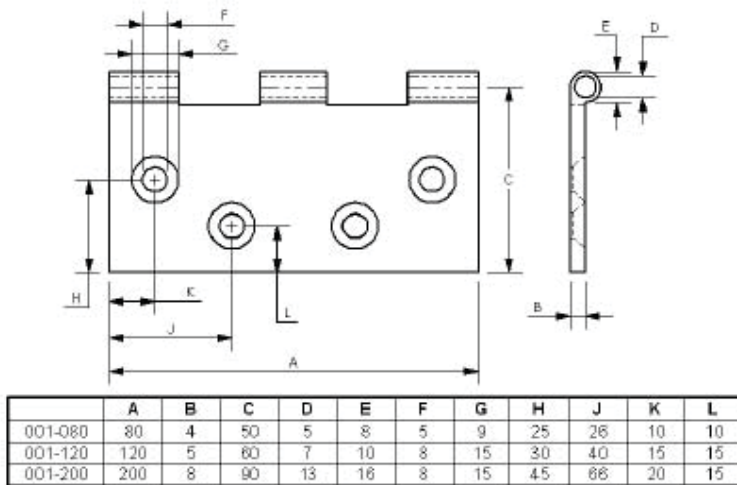
Configurations

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.

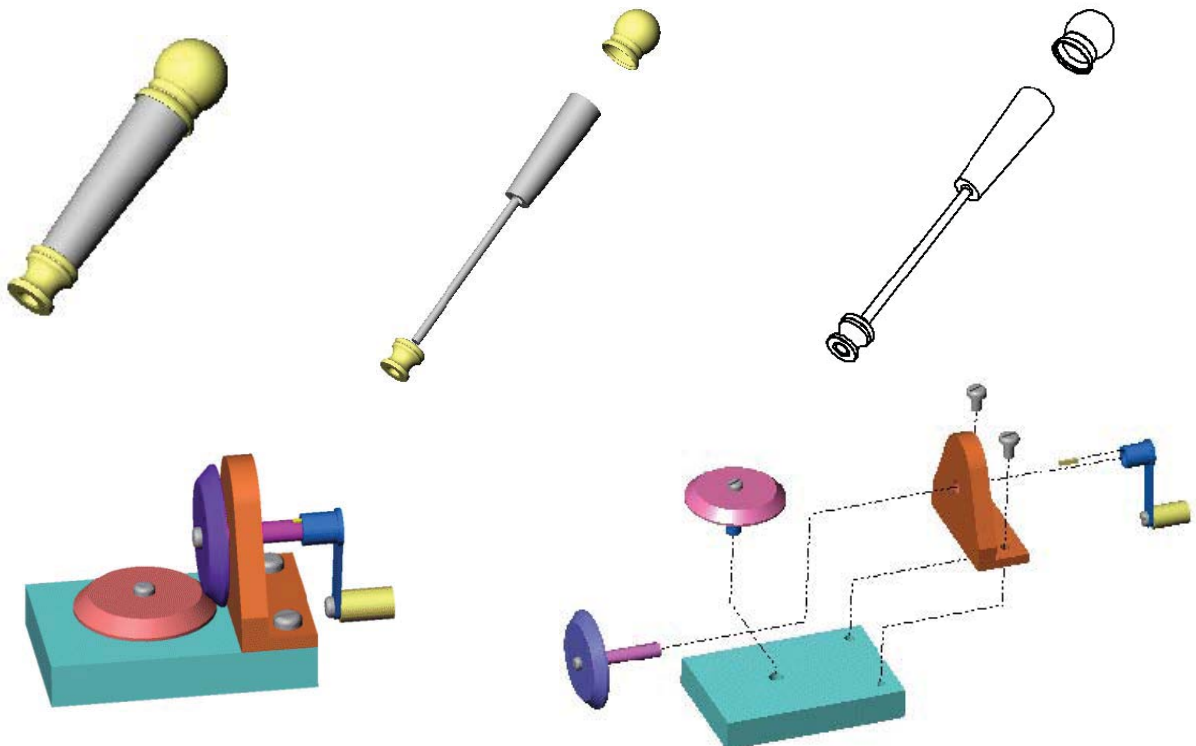


You can display design tables in drawings.



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.

Additional ideas and help are available on the SolidWorks web site at www.solidworks.com. The SolidWorks eNewsletter, press releases, and information on seminars, trade shows, and user groups are available at www.solidworks.com/pages/news/newsandevents.html.



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