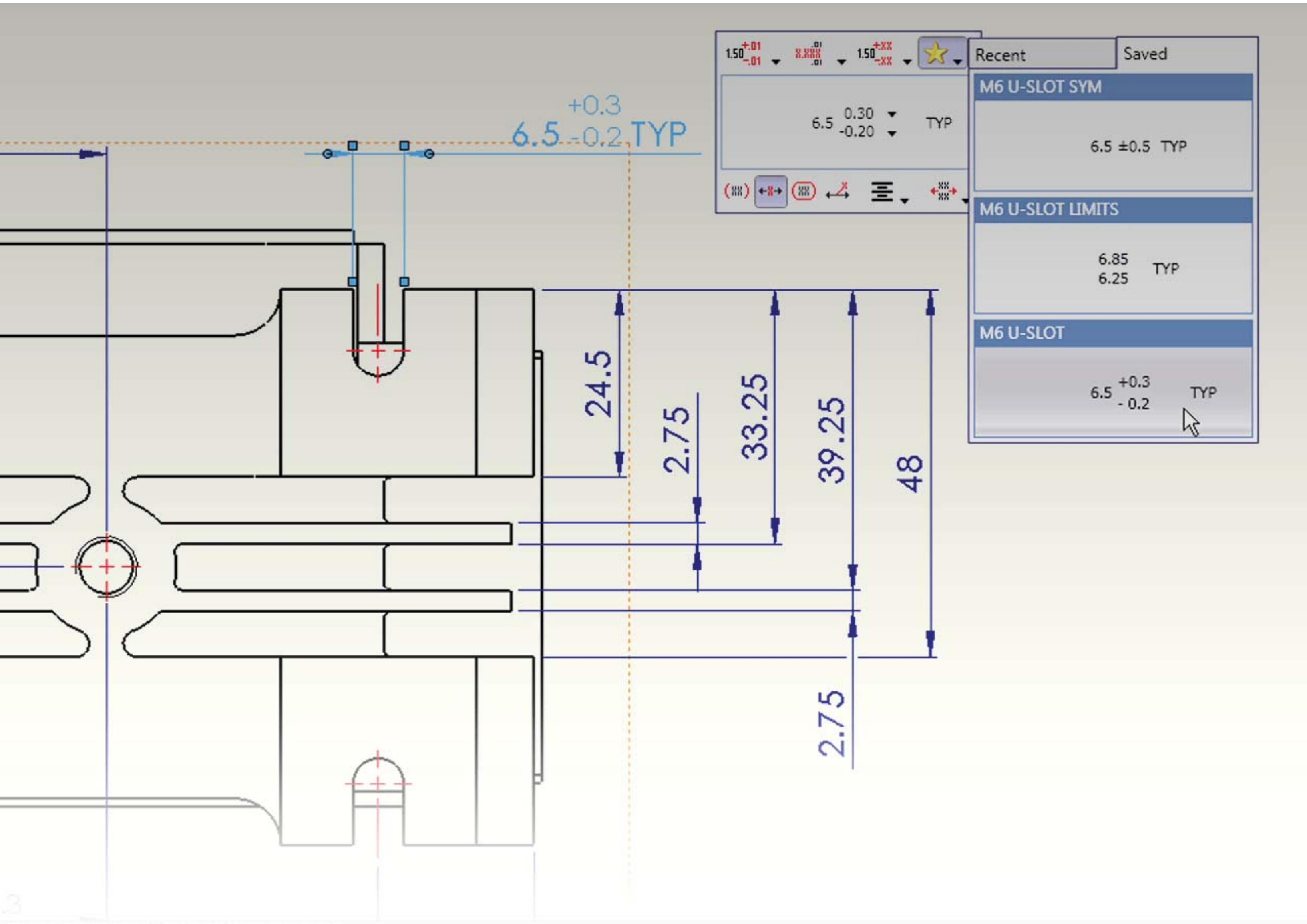
Implementation Guide:
Sketching with SolidWorks®



SolidWorks helps you move through the design cycle clearer. With intuitive sketching tools, your team can automatically dimension their sketches as they draw, for more accurate designs.

SolidWorks

Sketching in SolidWorks is the basis for creating features. Features are the basis for creating parts, which can be put together into assemblies. Sketch entities can also be added to **drawings**.

SolidWorks features contain intelligence so they can be edited. Design intent is an important consideration when creating SolidWorks models, so planning when sketching is important. The general procedure for sketching is to:

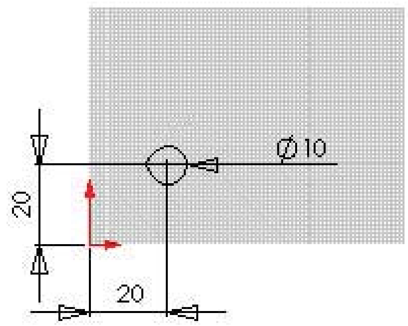
1. In a part document, select a sketch plane or a planar face (You can do this either before or after step 2.)
2. Enter the Sketch mode by doing one of the following:
   - Click **Sketch** on the Sketch toolbar.
   - Click a sketch tool (**Rectangle**, for example) on the Sketch toolbar.
   - Click **Extruded Boss/Base or Revolved Boss/Base** on the Features toolbar.
   - Right-click an existing sketch in the FeatureManager design tree and select **Edit Sketch**.
3. Create the sketch (sketch entities such as lines, rectangles, circles, splines, and so on).
4. Add dimensions and relations (you can sketch approximately, then dimension exactly).
5. Create the feature (which closes the sketch).

In general, it is better to use less complicated sketch geometry and more features. Simpler sketches are easier to create, dimension, maintain, modify, and understand. Models rebuild faster with simpler sketches.

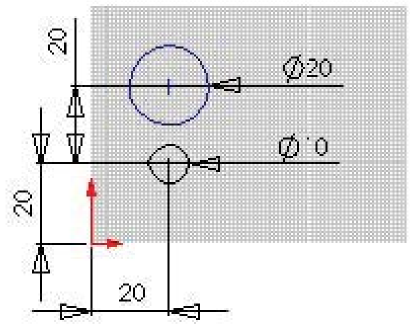
Sketch Dimensions

You can create features without adding dimensions to sketches. However, it is good practice to dimension sketches.

Dimension in accordance with the model's design intent; for example, you might want to dimension holes a certain distance from an edge, or else a certain distance from each other.

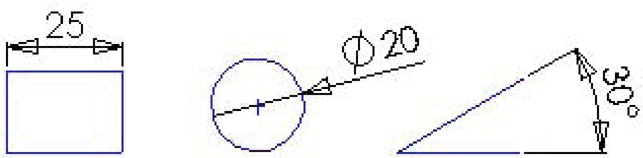


TO PLACE A HOLE A SPECIFIED DISTANCE FROM THE EDGES OF A BLOCK, DIMENSION THE DIAMETER OF THE CIRCLE AND DIMENSION THE DISTANCE BETWEEN ITS CENTER AND EACH EDGE OF THE BLOCK. CIRCLES ARE MEASURED FROM THE CENTER BY DEFAULT.



TO PLACE A HOLE A SPECIFIED DISTANCE FROM ANOTHER HOLE, DIMENSION THE DISTANCE BETWEEN THE CENTER OF THE HOLES. YOU CAN ALSO SPECIFY DIMENSIONS TO THE MINIMUM OR MAXIMUM POINT ON THE CIRCLE.

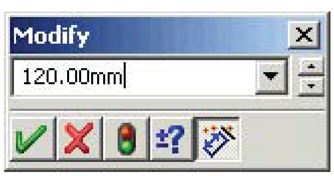
Most dimensions (linear, circular, or angular) can be inserted using a single tool, **Smart Dimension**  on the Dimensions/Relations toolbar.

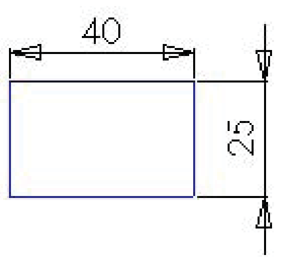


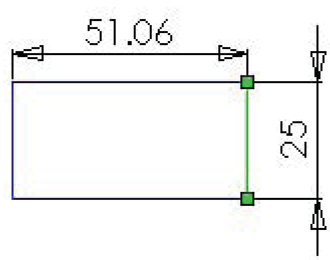
Additional dimension tools (**Baseline, Ordinate, Chamfer**) are available on the **Dimensions/Relations toolbar**.

You can dimension all entities in a sketch in one operation with **Fully Define Sketch**.

To change dimensions, double-click the dimension and edit the value in the **Modify** dialog box, or **drag** a sketch entity.







Snap

SolidWorks sketch entities can **snap** to points (endpoint, midpoints, intersections, and so on) of other sketch entities. With **Quick Snaps**, you can filter the types of sketch snaps that are available.

Additional snap functionality includes:

- **Grid** (displayed and snapped to)
- **Inferencing** (relations displayed as you sketch)
- **Relations** (added between sketch entities automatically through inferencing or manually)

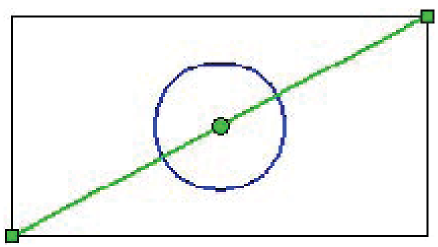
Sketch Relations

In SolidWorks, relations between sketch entities and model geometry are an important means of building in design intent. For example, you can draw two concentric circles. If you specify a concentric relation and then move one circle, the other circle moves with it, maintaining the relation.

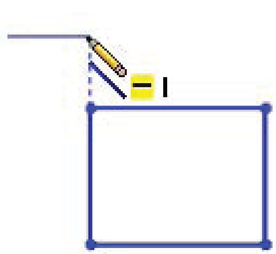
You can add relations in the following ways:

- **Automatically** by SolidWorks during sketching. The cursor changes to inform you of the relation it is **inferencing**.
- Manually after creating the sketch entities when you open entity PropertyManagers or the **Add Relations** PropertyManager. You can also **Display and Delete Relations**.

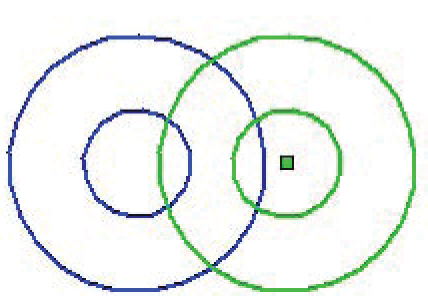
Equations create mathematical relations between model dimensions, but outside of sketches.



TO PLACE A HOLE IN THE CENTER OF THE BLOCK, SKETCH A CENTERLINE FROM CORNER TO CORNER, THEN SPECIFY A MIDPOINT RELATION BETWEEN THE CENTER OF THE CIRCLE AND THE CENTERLINE.



1. THE INFERENCING LINE SHOWS A VERTICAL RELATION BETWEEN THE ENDPOINTS OF THE TWO LINES.
2. THE IN THE POINTER DISPLAY INDICATES THAT THE LINE BEING SKETCHED IS HORIZONTAL. THE HORIZONTAL RELATION IS ADDED TO THE ENTITY PROPERTIES AUTOMATICALLY.



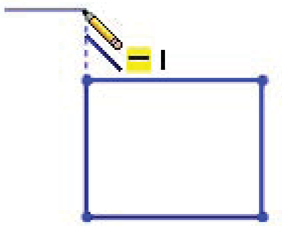
THE TWO CIRCLES ARE SPECIFIED TO BE CONCENTRIC. WHEN YOU MOVE ONE, THE OTHER MOVES WITH IT.

Inferencing

Inferencing displays relations by means of dotted inferencing lines, pointer display, and highlighted cues such as endpoints and midpoints.

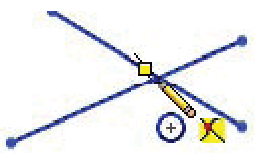
Inferencing lines

Inferencing lines appear as you sketch, displaying relations between the pointer and existing sketch entities (or modelgeometry).



Pointer display

The pointer display indicates when the pointer is over a geometric relation (an intersection, for example), what tool is active (line or circle), and dimensions (angle and radius of an arc). If the pointer displays a relation (such as  for a horizontal relation) and you click to accept the sketch entity while the relation is displayed, the relation is added automatically to the entity.



NOTE: You can turn off automatic relations. Click **Tools , Sketch Settings, Automatic Relations**.

Highlighted cues

Geometric relations such as endpoints, midpoints, and vertices **highlight** when the pointer approaches them, then change color when the pointer is poised to select them.

ON THE LEFT, A MIDPOINT HIGHLIGHTS AND THE POINTER SHOWS THAT A COINCIDENT RELATION  IS POSSIBLE AT ITS CURRENT POSITION. ON THE RIGHT, THE MIDPOINT HAS CHANGED COLOR AND THE POINTER SHOWS THAT IT RECOGNIZES THE MIDPOINT  .

Trim

You can **trim sketch entities**, including **infinite lines**, and **extend sketch entities** (lines, centerlines, and arcs) to meet other entities.

Trim Entities includes options:

- **Power trim**. Trim multiple adjacent sketch entities by dragging the pointer across the entities, or extend entities by selecting them and dragging the pointer.
- **Corner**. Trim or extend two sketch entities until they intersect at a virtual corner.
- **Trim away inside**. Trim open sketch entities inside two bounding entities.
- **Trim away outside**. Trim open sketch entities outside two bounding entities.
- **Trim to closest**. Trim or extend sketch entities to the closest intersection.

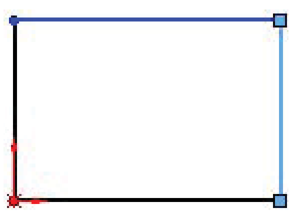
Sketch States

Sketches are generally in one of the following states:

- Under defined
- Fully defined
- Over defined

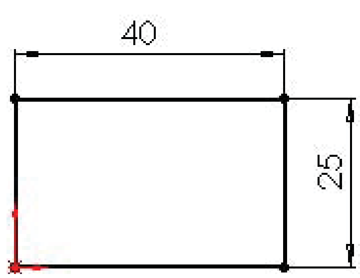
The sketch **status** appears in the window status bar. **Colors** indicate the state of individual sketch entities.

Under defined. As you begin a sketch, you can drag the entities to change their shape or position. In this rectangle, the black left and bottom lines are fixed to the origin, but you can drag the top and right lines. Blue indicates that the entity is not fixed, and light blue indicates that the entity is selected.



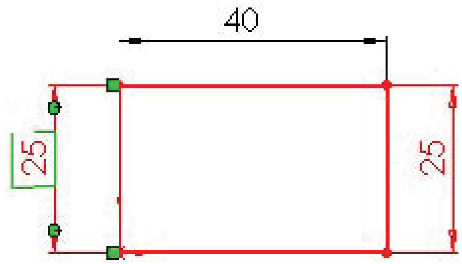
To add **relations** to a sketch, click **Add Relations** on the Dimensions/Relations toolbar.

Fully defined. Adding dimensions to the top and right fixes the sizes of all the sides of the rectangle because of the implied equal relations between top and bottom and the two sides. The rectangle itself is fixed to the origin. All the entities turn black, indicating that the rectangle is fully defined.



You can add relations (parallel, perpendicular, equal length, and so on) to a fully defined sketch. The sketch tolerates these logically redundant relations.

Over defined. Redundant dimensions over define a sketch. The red rectangle is over defined. When you insert dimensions, they are assumed to be driving dimensions. To have two dimensions driving the same geometry is invalid. A dialog box appears allowing you to designate the redundant dimension as driven.



You can view and delete relations. Click **Display/Delete Relations** on the Dimensions/Relations toolbar.

It is possible to create geometry that is unsolvable or invalid. The items that prevent the solution are displayed in pink (unsolvable) or yellow (invalid). Sketches with these types of geometry are labeled **No Solution Found** or **Invalid Solution Found**.

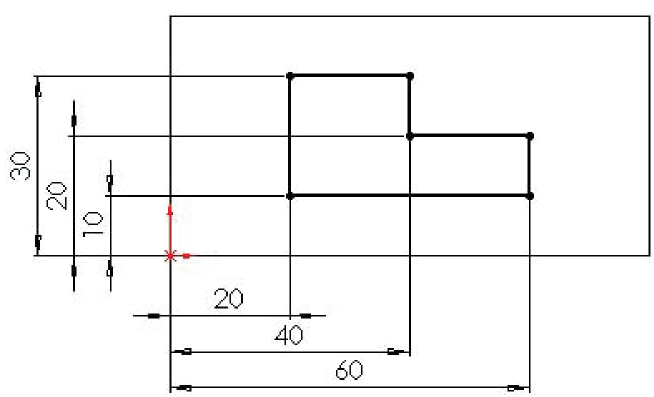
Dimensions and relations are two types of constraints. You define sketches with either type, or both.

Although you can create features using sketches that are not fully defined, it is a good idea to fully define sketches for production models. Sketches are parametric, and if they are fully defined, changes are predictable. However, sketches in **drawings**, although they follow the same conventions as sketches in parts, do not need to be fully defined since they are not the basis of features.

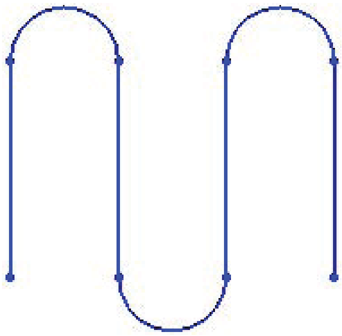
Automatic Sketch Operations

Automatic operations increase productivity in sketching. Automatic **relations** and **inferencing** also improve efficiency in sketching.

You can dimension all entities or selected entities in a sketch, including model edges, with the **Fully Define Sketch**  tool on the Dimensions/Relations toolbar.

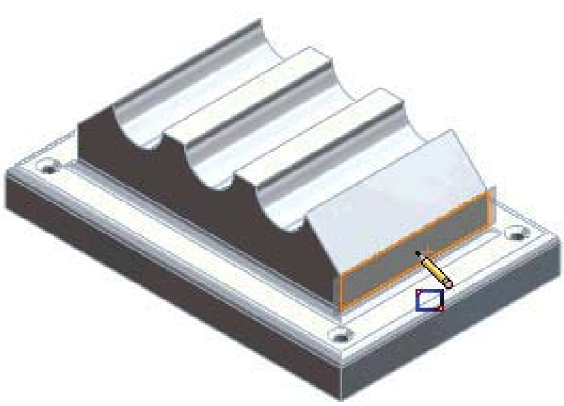


YOU CAN SOLVE OVER DEFINED SKETCHES USING SKETCHXPERT, WHICH CYCLES THROUGH POTENTIAL SOLUTIONS.



YOU CAN TRANSITION AUTOMATICALLY FROM LINE TO TANGENT ARC AND BACK, SO YOU CAN CREATE SKETCHES LIKE THIS WITHOUT CHANGING TOOLS.

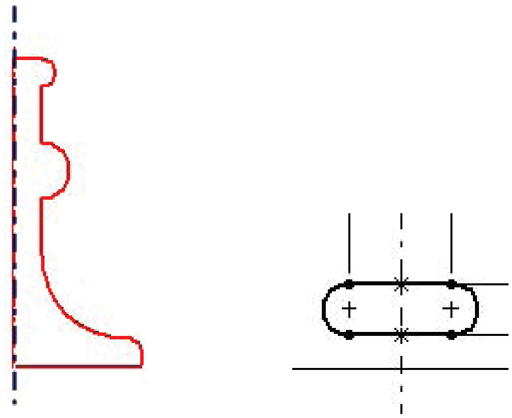
YOU CAN CONVERT RASTER DATA TO VECTOR DATA USING AUTO TRACE TOOLS.



YOU CAN HIGHLIGHT AND ACTIVATE PLANAR FACES OR PLANES TO QUICKLY CREATE SKETCHES USING RAPIDSKETCH.

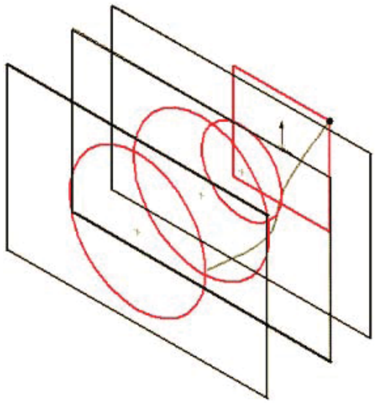
Construction Entities

In SolidWorks, any sketch entity can be specified for construction. Points and centerlines are always construction entities only.



USE A CENTERLINE AS THE AXIS ABOUT WHICH A SKETCH REVOLVES TO CREATE A BASE FEATURE, OR TO MIRROR SKETCH ENTITIES.

SolidWorks also has Reference Geometry (planes, axes, and coordinate systems) as a basis for creating features outside of sketches.



USE PLANES TO CONSTRUCT A SERIES OF SKETCHES AS THE BASIS OR A LOFT FEATURE.

With simple click and drag sketching from SolidWorks, designing is more accurate so you can design better products faster. 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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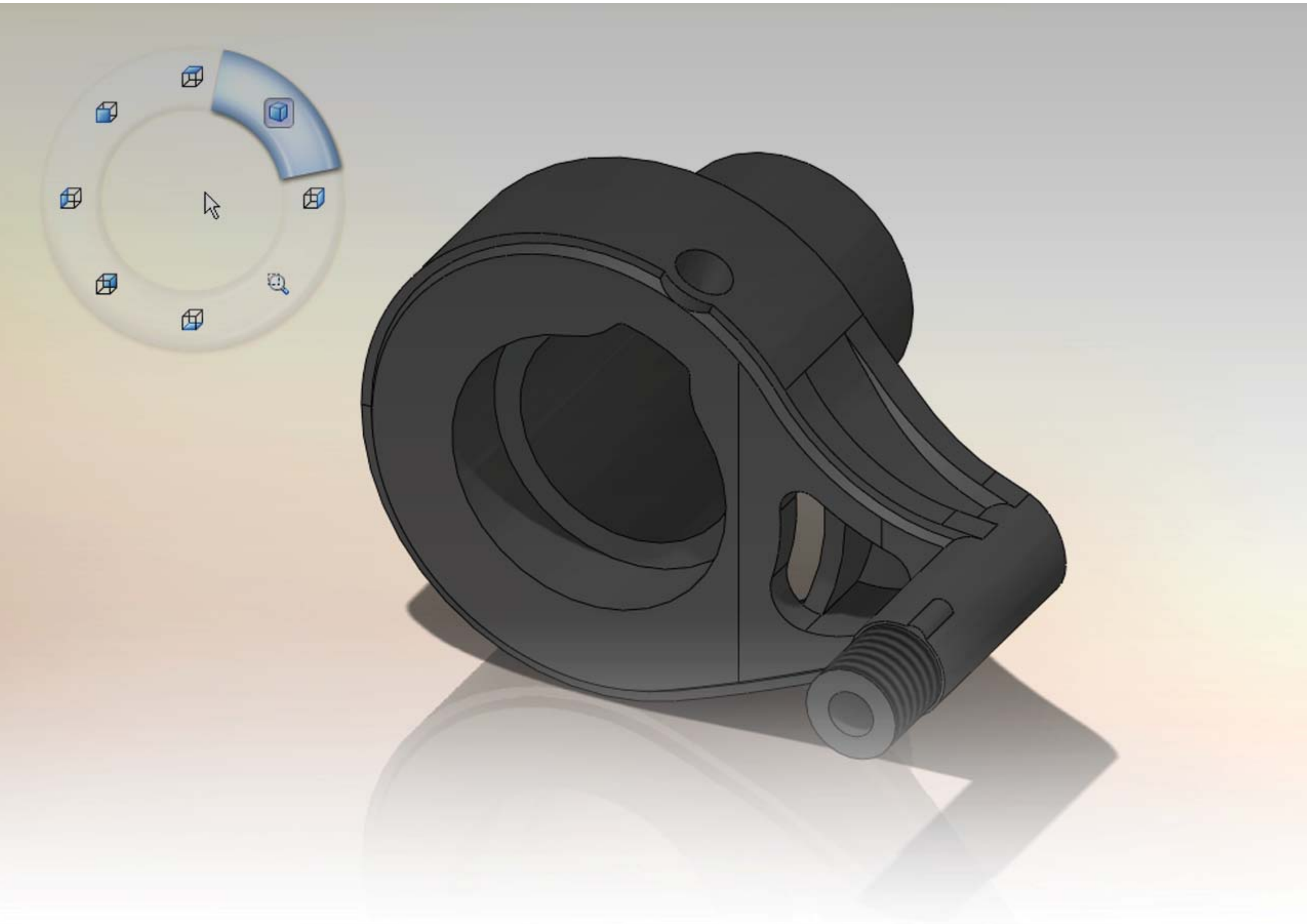
Implementation Guide:
Navigating the SolidWorks® User Interface



SolidWorks helps you move through the design cycle easier. With a customizable user interface, each team member team can create their own convenient and efficient SolidWorks environment.

Since most 2D CAD and SolidWorks are applications in the Microsoft® Windows environment, tool buttons, toolbars, and the general appearance of the windows look similar. However, many aspects of the environment differ.

Access to Tools

The most efficient method of working in SolidWorks is to use the tools on the toolbars and, when necessary, menus.

CommandManager

The **CommandManager** is context-sensitive. Its embedded toolbars change based on the document type.

This CommandManager appears in a part document. When you click a tab below the Command Manager, it updates to show that toolbar. For example, if you click the Sketches tab, the Sketch toolbar appears.

The CommandManager has two areas:

- **Tabs**. To switch toolbars, select the name of the area for which you want related toolbars.
- **Toolbar**. To activate tools, click them in this area.

In addition to the tooltip displayed with the tool icon, a description appears when you hold the pointer over the tool, giving you further information on how to use the tool.

You can customize the CommandManager in each type of document to display the toolbars you use most. You can drag the CommandManager to different locations anywhere on your desktop or dock it automatically at the top or on either side of the SolidWorks window.

The CommandManager is efficient, convenient, and customizable. Most of the tools you use are in one place.

Toolbars

All **toolbars** are available in the familiar Microsoft Windows style. You can show, hide, and **customize** them.

The **Heads-up View toolbar** is a transparent toolbar in each viewport that provides all the common tools necessary for manipulating the view.

Menu Bar

The **Menu Bar** contains various ways to access SolidWorks tools and options.

Shortcut Bars

Customizable shortcut bars let you create your own set of non-context commands for these modes:

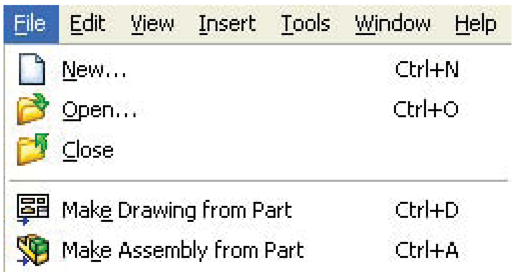
- Part
- Assembly
- Drawing
- Sketch

Keyboard Shortcuts

Shortcuts in SolidWorks are either **accelerator keys** or **keyboard shortcuts**.

Accelerator keys are available for every menu item and are indicated by underlined letters. They cannot be customized.

- To display the underlined letters on the main menu, press **Alt**.
- To access a menu, press **Alt** plus the underlined letter; for example **Alt+F** for the **File** menu.
- To execute a command, press the underlined letter; for example, **Alt+F**, then **C** to close the active document.



Keyboard shortcuts are key combinations such as those displayed at the right of the menu, which can be **customized**.

Undo and Redo

You can **undo** most recent changes. In sketches in part and assembly documents, you can also redo recent **undo** commands.

To undo your last action:

- Click **Undo**  on the Standard toolbar
- Click **Edit, Undo**
- Press **Ctrl+Z**

To redo your last Undo action:

- Click **Redo**  on the Standard toolbar
- Click **Edit, Redo**
- Press **Ctrl+Y**

The SolidWorks software keeps a list of available undo and redo actions, so you can choose from the list to undo or redo the selected action and all actions above it.

Repeat Last Command

You can repeat the last command, and also view and repeat any of the ten most **recent commands**.

To repeat the last command:

- Click **Edit, Repeat Last Command**.

To repeat a recent command:

1. Right-click in the graphics area and select **Recent Commands**.
2. Select a command from the list as your next command.

Screen Layout

When you open the SolidWorks application for the first time, the **Task Pane** appears and the Standard toolbar is available with tools such as **New, Open**, and **Save**.

When you open documents, additional tools become available. For all documents, the following appear:

- **Heads-up View toolbar**
- **Menu Bar**
- Panel with the **FeatureManager design tree**

In addition, in a part document, the following appear:

- **CommandManager** with the **Features** and **Sketch** toolbars
- **Triad** (for reference only)

In an assembly document, the following appear:

- **CommandManager** with the **Assemblies** and **Sketch** toolbars
- **Triad** (for reference only)

In a drawing document, the following appear:

- **CommandManager** with the **Drawings, Sketch**, and **Annotations** toolbars
- **Drawing sheet** with optional **sheet format** (selected when you open a new drawing)

To display additional toolbars, right-click an edge of the SolidWorks window and select a toolbar. The toolbar docks to an edge of the window. You can drag toolbars to any edge, or drag them into the graphics area, where they become floating palettes. Other modifications you can make to the layout include:

- Change the screen **background colors**.
- Open a **command line**.
- Set system and document options in **Tools, Options**.

When you change the screen layout and options, the changes apply to future SolidWorks sessions.

Some commands are executed immediately, some open dialog boxes, and many open a PropertyManager in the **Management Panel** at the left of the graphics area.

Task Pane

The **Task Pane** is a center for accessing resources and documents. It appears when you open the SolidWorks software, and it contains these tabs:

SolidWorks Resources	Commands for **Getting Started** and links to the SolidWorks **Community** and **Online Resources**.
Design Library	Reusable parts, assemblies, and other items, including **3D ContentCentral**, annotation favorites, and **Library Features**.
File Explorer	Duplicate of Windows Explorer on your computer, plus **Recent Documents** and **Open in SolidWorks**.
Search	Results of search operation. If you dissect files into Design **Clipart**, thumbnails of reusable geometry, such as sketches and features, appear on this tab. Drag the thumbnails onto the model to reuse geometry.
View Palette	Images of standard views, annotation views, section views, and flat patterns (sheet metal parts) to drag onto a drawing sheet.
Appearances	Provides a simplified way to display models in a photorealistic setting using a library of appearances and scenes. With PhotoWorks added in, the tab also contains a library of decals and lights.
Custom Properties	Enter custom and configuration-specific properties into SolidWorks files.
Document Recovery	If auto-recovery is enabled in **Tools, Options, System Options, Backup/Recover** and the system terminates unexpectedly, recovered files appear on this tab the next time you start the application.

The Task Pane can be in the following states:

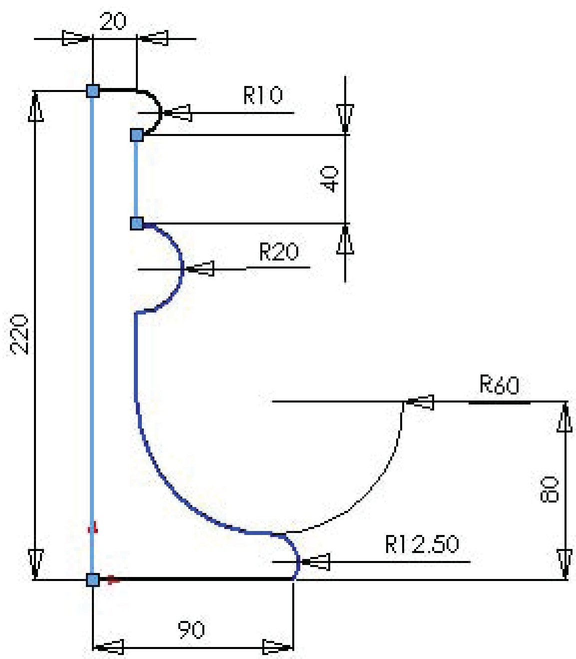
- Visible or hidden
- Expanded or collapsed
- Pinned or unpinned
- Docked or floating

You can drag documents from the **File Explorer** tab and the **Design Library** tab into the graphics area and from the graphics area or FeatureManager design tree into the **Design Library**.

Background Color

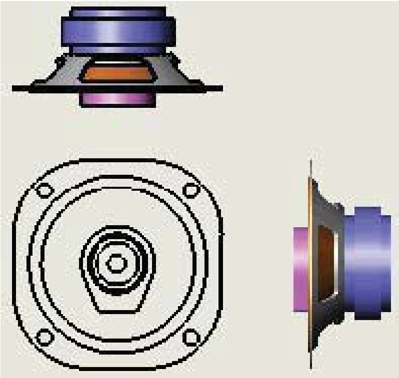
SolidWorks uses a blue gradient background in its graphics area. Although you can change the background color in SolidWorks, you will find that blue works best with shaded models and the various colors that indicate status.

> **NOTE**: You can also drag scenes onto models from the **Task Pane's Appearances** tab, under **Scenes**, to change the background color and model look. You can also click **Apply Scene** from the **Heads-up View** toolbar and select a scene.



IN A SKETCH, LIGHT BLUE INDICATES ENTITIES THAT ARE SELECTED. BLUE SHOWS ENTITIES THAT ARE NOT FULLY DEFINED. BLACK ENTITIES ARE FULLY DEFINED. THE SKETCH ORIGIN APPEARS IN RED. OTHER STATUS COLORS ARE YELLOW, PINK, AND GRAY.

COLORS IN A SHADED VIEW SHOW TO ADVANTAGE ON A BLUE GRADIENT BACKGROUND.



DRAWING SHEETS ARE THE COLOR OF MYLAR. YOU CAN DISPLAY DRAWING VIEWS IN VARIOUS SHADED AND LINE MODES.

To specify different colors, click **Tools, Options, System Options, Colors.** Some of the items for which you can specify color include:

- **Viewport Background**
- **Top Gradient Color**
- **Bottom Gradient Color**
- **Drawings, Paper Color**
- **Drawings, Background**
- **Grid Lines, Major**
- **Annotations, Imported**

Menus

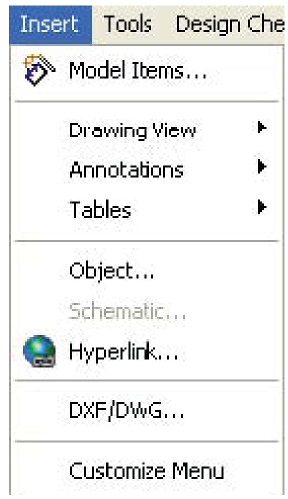
SolidWorks has a context-sensitive menu structure. The menu titles remain the same for all three types of documents, but the menu items change depending on which type of document is active. For example, the **Insert** menu includes features in part documents, mates in assembly documents, and drawing views in drawing documents.

You can access menu items through **keyboard shortcuts**, and you can **customize menus**.

PART

ASSEMBLY

DRAWING



Shortcut Menus

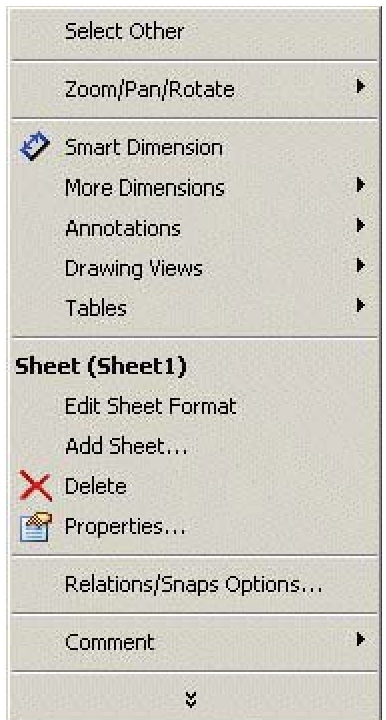
In SolidWorks, you activate context-sensitive (shortcut) menus when you click the right mouse button.

Shortcut menus are available in the graphics area, for drawing views or drawing sheets, and for items in the FeatureManager design tree, for example.

> **NOTE**: When you select items in the graphics area or FeatureManager design tree, **context toolbars** appear and provide access to frequently performed actions for that context.

IN THE GRAPHICS AREA OF A NEW PART DOCUMENT TREE

IN THE GRAPHICS AREA OF A NEW DRAWING



A FEATURE IN THE FEATUREMANAGER DESIGN

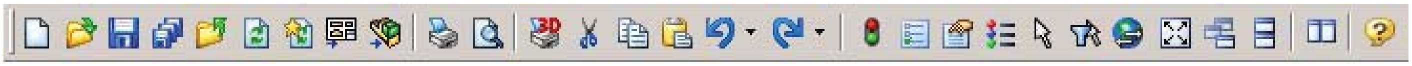
Toolbars

The SolidWorks toolbars are sensitive to the document type. The major toolbars that apply to each type of document appear when the appropriate type of document is opened. In addition, you can display any other toolbars.

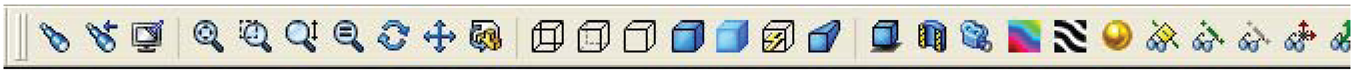
SolidWorks remembers the state of the toolbars from session to session. For example, if you make the Mold Tools toolbar visible in a part document, that toolbar is visible when you open a new part document.

You can customize SolidWorks toolbars by adding, deleting, and moving tools, and by arranging the toolbars in the SolidWorks window.

Standard toolbar



View toolbar



Annotation toolbar

Command Line

The SolidWorks **2D Emulator** is an optional add-in that simulates the 2D CAD command line. The commands available in the emulator, which are equivalent to SolidWorks sketching tools, include:

- Drawing entities (POINT, LINE, ARC, and so on)
- Other drawing tools (FILLET, CHAMFER, DIM, and so on)
- View tools (PAN, VIEW, ZOOM)
- Entity properties (COLOR, and so on)
- Information (LIST, and so on)
- Feature creation (EXTRUDE, REVOLVE)
- System tools (ALIGN, PLOT, and so on)

You can activate the 2D Command Line Emulator by clicking **Tools, Add-Ins** and selecting **SolidWorks 2D Emulator** from the list of add-ins. The command line appears at the bottom of the screen when you open a document. To turn off the emulator, click **Tools, Add-Ins**, and clear the **SolidWorks 2D Emulator** check box.

While in a SolidWorks document with the 2D Emulator active, you can display or hide the command line. Click **View, 2D Command Emulator.** A check mark beside the menu item indicates that the command line is displayed.

To access help for the 2D Command Line Emulator, click **Help, 2D Command Emulator Help**, or type **Help** in the command line.

Other ways you can customize the SolidWorks environment include:

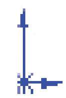
- Customize **keyboard shortcuts**
- Customize **shortcut bars**
- Customize **menus**
- Customize **tools** and **toolbars**
- Customize the **SolidWorks Resources** tab, toolbars, and menus based on **work flow**

- Record and customize **macros**
- Set **options**
- Customize **drafting standards**

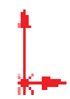
Coordinate Systems

SolidWorks uses a system of coordinate systems with origins. A part document contains an original origin. Whenever you select a plane or face and open a sketch, an origin is created in alignment with the plane or face. An origin can be used as an anchor for the sketch entities, and it helps orient perspective of the axes. A three-dimensional reference triad orients you to the X, Y, and Z directions in part and assembly documents.

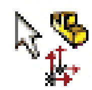
PART ORIGIN (ONE IN EACH PART DOCUMENT)



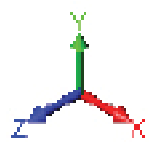
SKETCH ORIGIN (ONE FOR EACH NEW SKETCH)



INFERENCING TO AN ASSEMBLY ORIGIN (ISOMETRIC ORIENTATION)



REFERENCE TRIAD IN PART AND ASSEMBLY DOCUMENTS

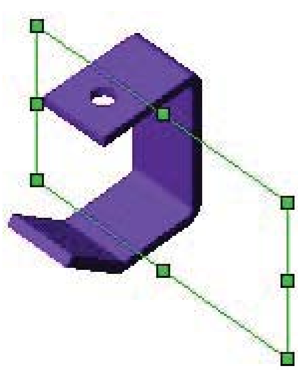


Planes

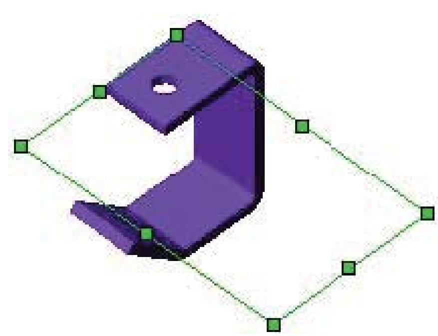
SolidWorks provides **Front, Top**, and **Right** planes as defaults. The **orientations** (**Front, Top, Right,** and so on) relate to these planes. Planes are used for sketching and for creating geometry for features.

You can create **reference planes** in addition to the default planes, and you can open sketches on planar **model faces**.

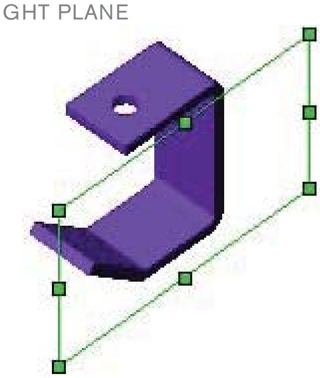
FRONT PLANE



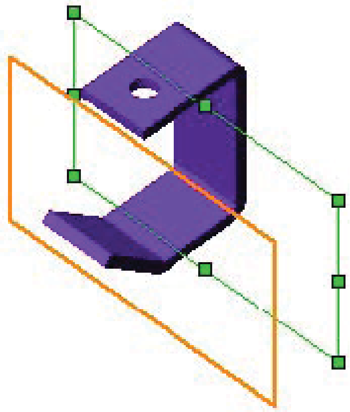
TOP PLANE



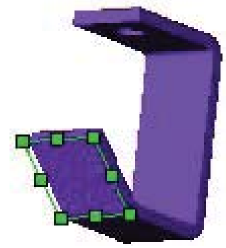
RIGHT PLANE



PLANE OFFSET

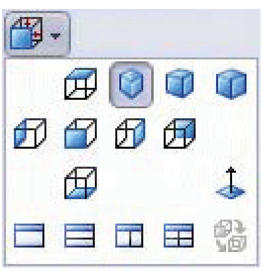


PLANE ON FACE



Orientation

The SolidWorks Standard Views toolbar and flyout toolbar contain **Front, Back, Top, Bottom, Right, Left, Isometric, Trimetric**, and **Dimetric** orientations. **Normal To** is normal (perpendicular) to the sketch plane or the selected plane. To access the Standard Views flyout toolbar, click **View Orientation** on the **Heads-up View toolbar**.



The **Orientation** dialog box contains the views on the Standard Views toolbar, plus user custom views. To access the **Orientation** dialog box, select **View Orientation** on the Standard Views toolbar, press the space bar, or right-click in the graphics area and select **View Orientation**. You can add custom views using the **New View** tool.

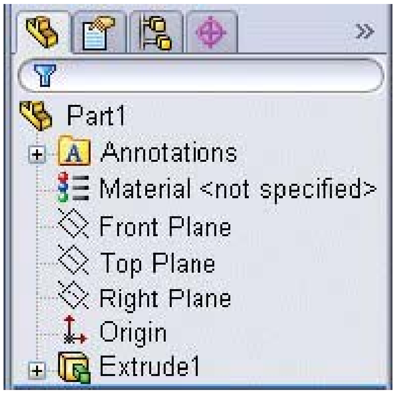
Management Panel

The left panel of the SolidWorks window manages part and assembly designs, drawing sheets, properties, configurations, and third party applications. The **CommandManager** provides access to the SolidWorks tools.

FeatureManager Design Tree

Names of features are displayed from top to bottom in the order created in the **FeatureManager design tree**, unless you reorder them. (Features can be considered as components of parts.)

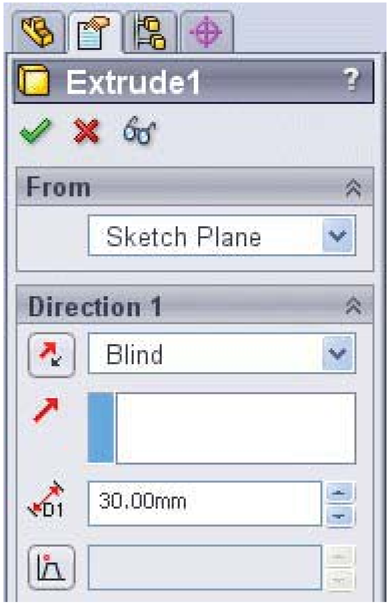
The FeatureManager design tree in assemblies displays components (parts or subassemblies and their features), a **Mates** folder, and assembly features.



The FeatureManager design tree in drawings contains an icon for each sheet. Under each sheet are icons for the sheet format and each view. Under each view are the parts and assemblies that belong to the view.

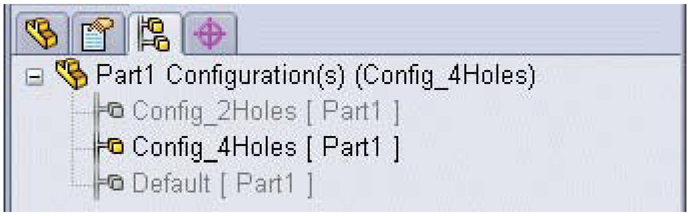
PropertyManager

Most sketch, feature, and drawing tools in SolidWorks open a **PropertyManager** in the left panel. The PropertyManager displays the properties of the entity or feature so you specify the properties without a dialog box covering the graphics area.



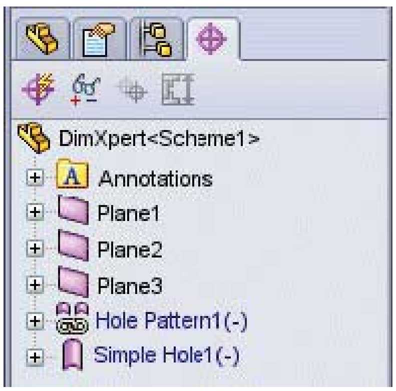
ConfigurationManager

The **ConfigurationManager** is a means to create, select, and view multiple configurations of parts and assemblies.



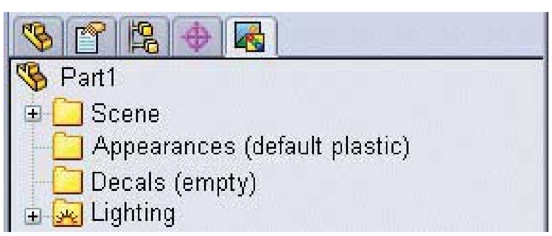
DimXpertManager

The DimXpertManager lists the tolerance features defined by **DimXpert for parts**. It also displays DimXpert tools that you use to insert dimensions and tolerances into parts. You can **import** these dimensions and tolerances into drawings.



Third Party Applications

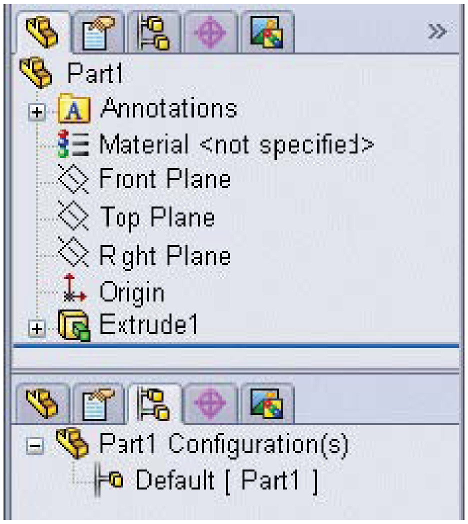
Third party programs are certified and integrated into the SolidWorks software. Such applications often include menus and left panel management tabs in the SolidWorks window.



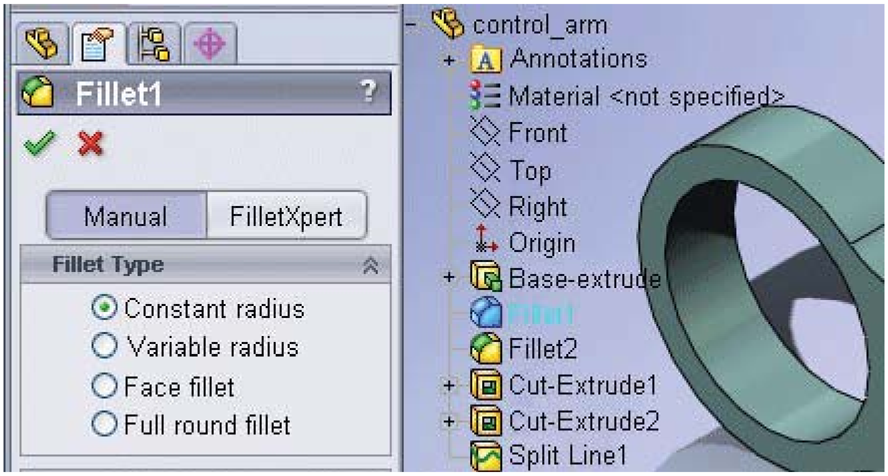
Manager Display

You can **switch** between the FeatureManager design tree, PropertyManager, ConfigurationManager, and third party managers by clicking the tabs at the top of the left panel in the SolidWorks window.

You can **split** the panel and display more than one manager or multiple copies of one manager.



When you are in a PropertyManager, you can click to view a **flyout** FeatureManager design tree simultaneously.



Selection Methods

In SolidWorks, you can select objects as follows:

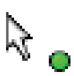
- Click **objects** in the graphics area
- Press **Ctrl** while clicking to **select** more than one object
- Drag the pointer from left to right to define a **box selection** or from right to left to define a **cross selection**
- Right-click one entity of a sketch object with multiple entities in a chain (such as a rectangle or polygon) and choose **Select Chain** from the shortcut menu
- Right-click one edge in a loop of edges in a part and choose **Select Loop** for operations such as feature fillet and chamfer
- Right-click two edges in a loop of edges in a part and choose Select Partial Loop to select a series of connecting edges
- Select features, components, planes, drawing views, and other items in the **FeatureManager design tree**

For many operations, you can select the objects either before or after selecting the tool you want to apply.

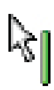
Feedback

To help you select, entities **highlight** as you pass the **pointer** over them, and the pointer changes to let you know what type of entity it senses.

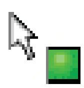
VERTEX



EDGE

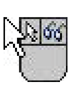


FACE



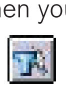
Select Other

You can right-click an object and choose **Select Other**  to step through all the items under the pointer. When you choose a face, the face is hidden so you can see inside the model.



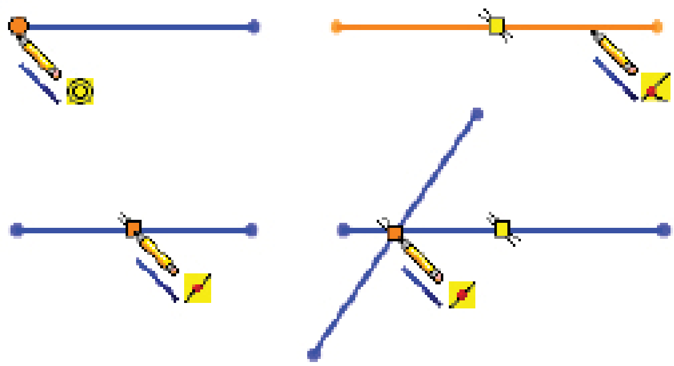
Selection Filter

You can set the **Selection Filter** to the kind of item that you want to select: faces, edges, vertices, surface bodies, reference geometry, sketch entities, dimensions, and various types of annotations. With the filter set, the kinds of items that you specify are identified when you pass the pointer over them.

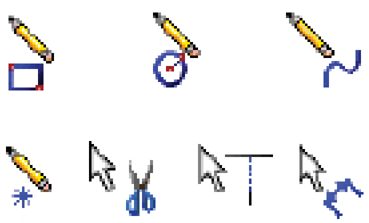
Click **Toggle Selection Filter Toolbar**  on the Standard toolbar to make the Selection Filter toolbar visible.

Selection Feedback

The pointer changes shape in SolidWorks to show the type of object it sees; for example, a vertex, an edge, or a face. In sketches, the pointer shows relations such as endpoints, midpoints, intersections, and types of entities such as lines, rectangles, and circles.



RELATIONS: **ENDPOINT, COINCIDENT, MIDPOINT, INTERSECTION**

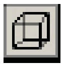


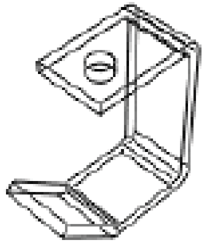
ENTITIES AND TOOLS: **RECTANGLE, CIRCLE, SPLINE, POINT, TRIM, EXTEND, DIMENSION**

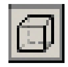
Display Functions

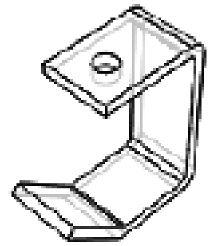
SolidWorks has familiar **zoom** and **pan** functions, plus additional display tools, on the View toolbar or the **Heads-up View toolbar**.

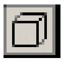
In addition to **Zoom to Selection**, **Zoom to Fit**, and **Rotate View**, SolidWorks has tools to display models in wire-frame, hidden lines visible, hidden lines removed, shaded, edges in shaded mode, and shadows in shaded mode. Models can be displayed in shaded mode in drawings as well as in part and assembly documents. **Section views** of the model (not **drawing section views**), perspective view, and shadows are also available on the View toolbar.

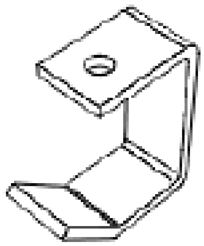
 **Wireframe**

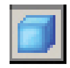


 **Hidden Lines Visible**

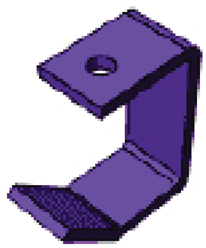


 **Hidden Lines Removed**

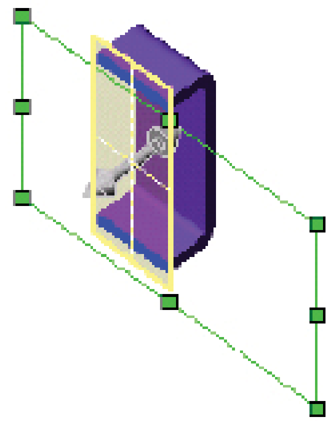


 **Shaded**

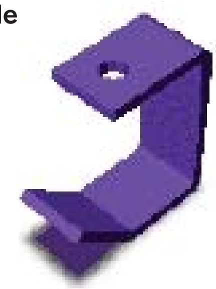
 **Shaded With Edges**



 **Section View**



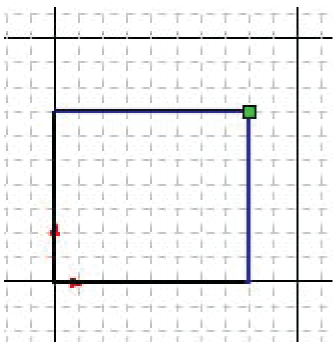
 **Shadows In Shaded Mode**



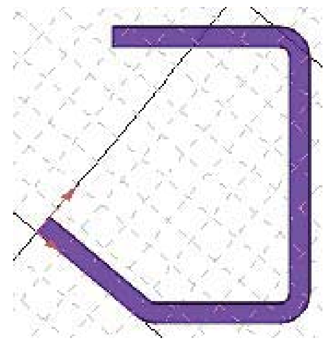
Grid and Snap

SolidWorks snaps to sketch geometry on the fly. For example, as the pointer approaches a line endpoint, the pointer changes to recognize the endpoint so you can choose to select it.

SolidWorks offers a display grid and snap grid while in sketch mode and in drawings. You can align the grid to a model edge and you can snap to an angle. The grid and snap capabilities are not often used in SolidWorks since dimensions and relations provide the required accuracy.



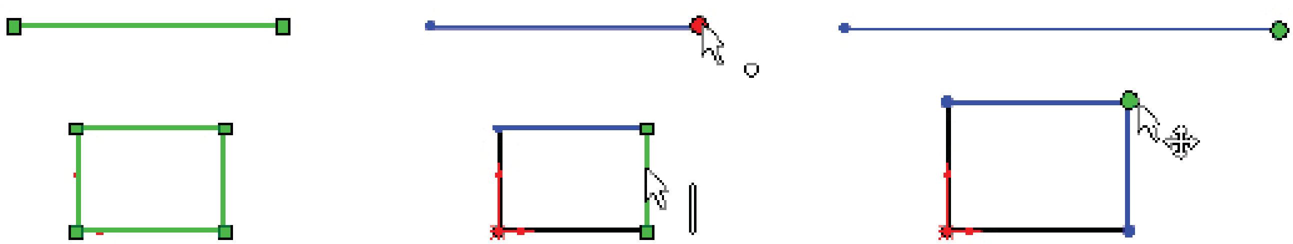
GRID WITH SNAP IN A SKETCH



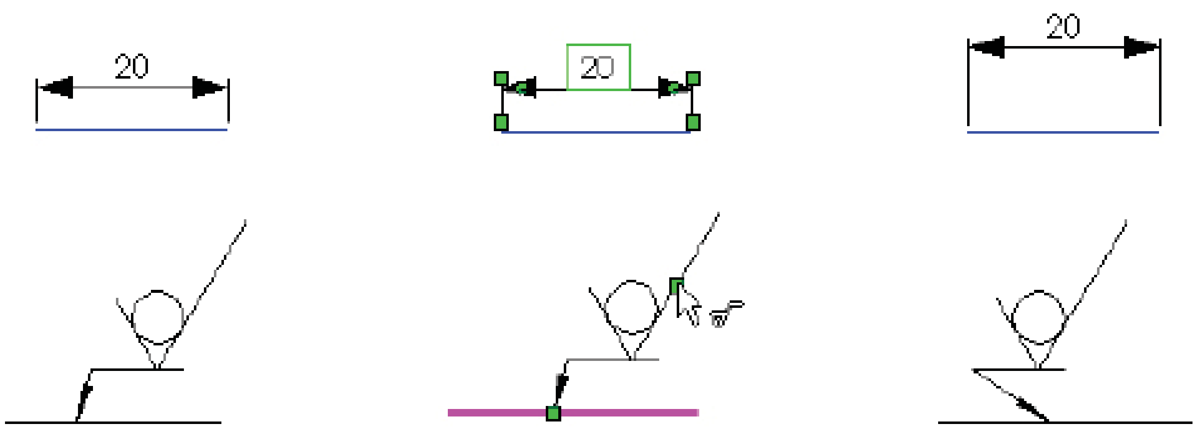
GRID ALIGNED TO A SPECIFIED MODEL EDGE

Dragging

In the SolidWorks software, you can move sketch entities by selecting them and dragging. You can also stretch sketch entities by dragging. For example, select a **line** and drag an endpoint, or select a side or vertex of a **rectangle** and drag to stretch the rectangle.



DRAG DIMENSIONS AND ANNOTATIONS TO POSITION THEM.



You can also drag drawing views.

To drag drawing views:

- Select a view (the pointer changes to ) and press **Alt** while dragging.

  \- or -

- Select the edge of a view (the pointer changes to ), then drag.

Many other items are available for dragging, including feature previews, components in assemblies, and so on.

Options

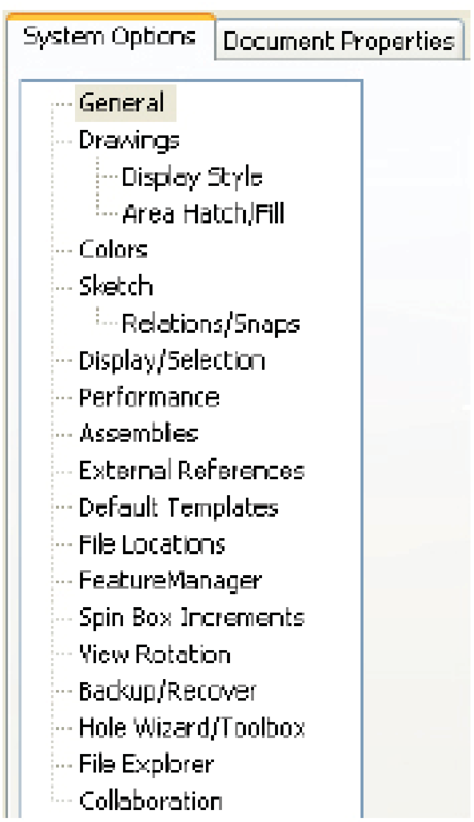
SolidWorks options are divided into the following categories:

- **System Options** apply to all documents, current and future.
- **Document Properties** apply only to the currently active document.

New documents get their document settings (such as **Units, Image Quality**, and so on) from the document properties of the template used in creating the document. The **Copy Settings Wizard** exports registry settings so that options can be copied to other computers. See the SolidWorks Import and Export dialog boxes for import and export options.

To access the **Options** dialog box, click **Options**  (Standard toolbar) or **Tools, Options**.

System Options



Document Properties

Help

Help in SolidWorks is context-sensitive and in HTML format. Help is accessed in many ways, including:

- **Help** buttons in all dialog boxes and PropertyManagers (or press **F1**)
- **Help** tool on the Standard toolbar for SolidWorks Help
- **Flyout menu** of Help options
- **Help** menu for SolidWorks or other Help (such as API, third-party software, and so on)

Glossary. The SolidWorks Help contains a glossary of terms. Click **Glossary** at the bottom of the table of contents.

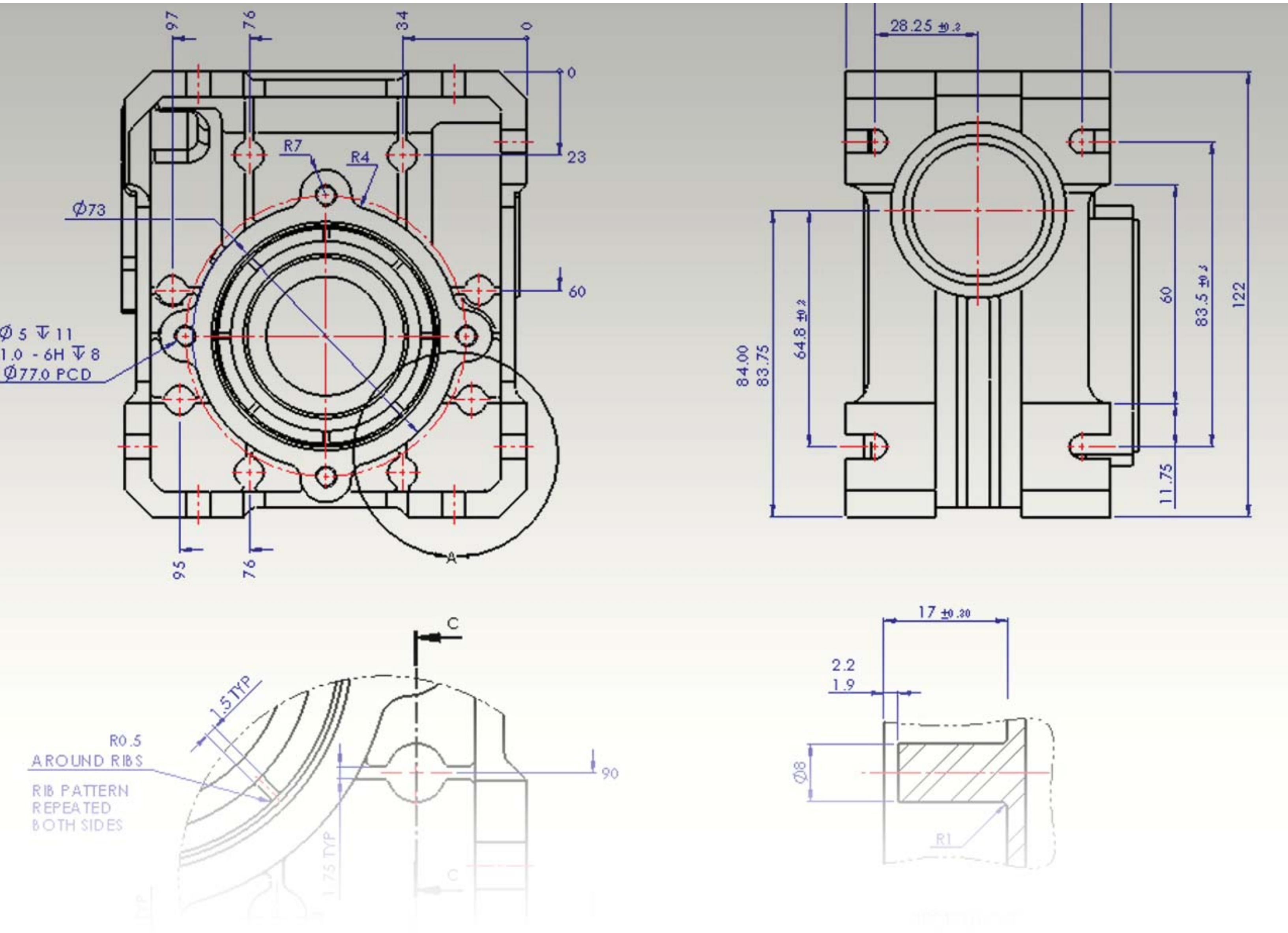
In addition to the Help facility, SolidWorks provides help in the following ways:

- What's New (on the **Help** menu) - new functionality added since the last major SolidWorks release
- **Interactive What's New** (click in new menus and new or changed PropertyManagers) - links to topics in the What's New book.
- **Quick Tips** (click on the status bar) - pop-up messages that give hints and options based on the current SolidWorks mode.
- Tooltips - information about tools on toolbars and in PropertyManagers and dialog boxes.
- Status bar information (at the bottom of the SolidWorks window) - pointer coordinates, sketch status, and brief descriptions of selected commands.
- **Tip of the Day** (at the bottom of the **SolidWorks Resources** tab in the Task Pane) - a new tip appears each time you start SolidWorks.
- **SolidWorks Tutorials** (on the **Help** menu and on the **SolidWorks Resources** tab in the Task Pane) - step-by-step lessons on features, parts, assemblies, drawings, and third-party applications.
- **SolidWorks Resources** tab in the Task Pane includes commands, links, and information. The **General Information** link provides access to Documentation Central. Documentation Central is the reference library for beginning and advanced users. Documentation Central features tutorials and simulations, an interactive environment for collaborative development, access to published manuals and updates, and late-breaking and experimental documentation.

Through customizable, interactive menus and smart pointers, your team can quickly and conveniently navigate the SolidWorks environment. With SolidWorks on your team, the user interface works for you so you can design products easier.

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.**

Implementation Guide:
Creating Drawings with SolidWorks®



SolidWorks helps you move through the design cycle smarter. With fully integrated drawing, your team can create drawings directly from 3D models, ensuring accuracy and preserving correspondences.

You can generate drawings in SolidWorks the same way you would generate them in 2D drafting and drawing systems. However, creating 3D models and generating drawings from the model have many advantages; for example:

- Designing models is faster than drawing lines.
- SolidWorks creates drawings from models, so the process is efficient.
- You can review models in 3D and check for correct geometry and design issues before generating drawings, so the drawings are more likely to be free of design errors.
- You can insert dimensions and annotations from model sketches and features into drawings automatically, so you do not have to create them manually in drawings.
- Parameters and relations of models are retained in drawings, so drawings reflect the design intent of the model.
- Changes in models or in drawings are reflected in their related documents, so making changes is easier and drawings are more accurate.

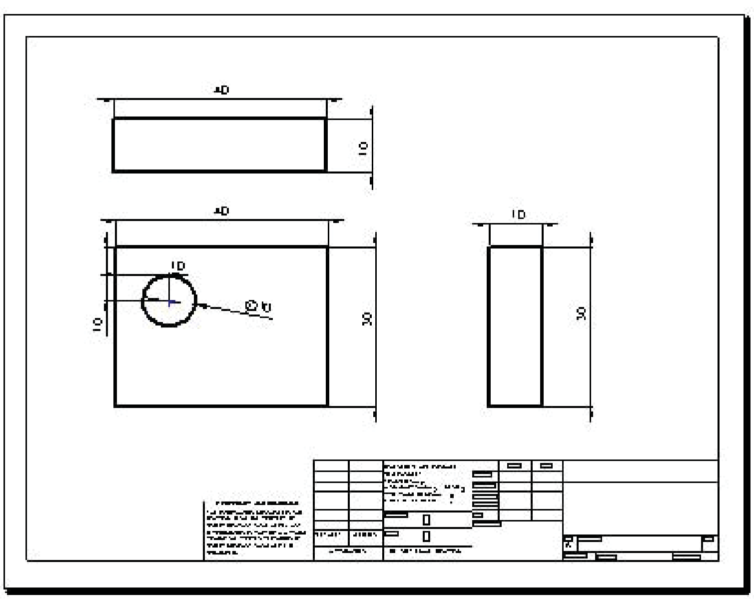
Creating Drawings

Drafting in SolidWorks

To draft a drawing in SolidWorks without creating a model:

1. Open a **New** drawing document. Choose a template.
2. Draw lines, rectangles, circles, and other entities with the tools on the Sketch toolbar.
3. Dimension the entities with the **Smart Dimension** tool on the Dimensions/Relations toolbar.
4. Add annotations **(Notes, Geometric Tolerance Symbols, Balloons**, and so on) with tools on the Annotation toolbar.

NOTE: See the next section for an alternative approach. See **Drafting** for further details on sketching in drawings.



Creating Drawings from Models

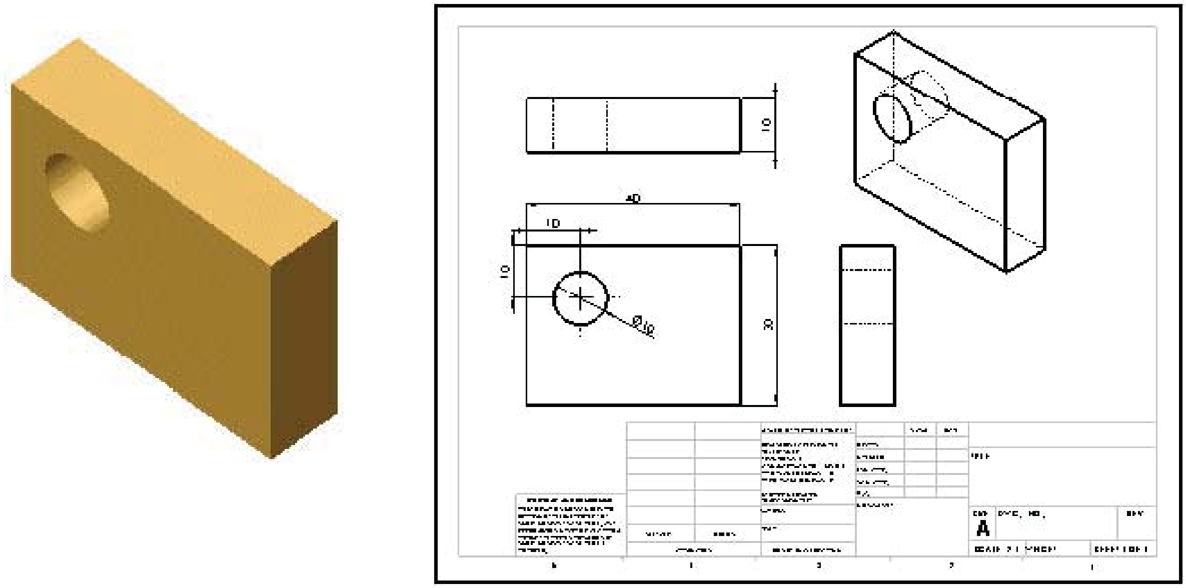
To generate drawings from part and assembly documents:

1. In a part or assembly document, click **Make Drawing from Part/Assembly** on the Standard toolbar and select a template in the **Sheet Format/Size** dialog box.

   The **View Palette** opens on the right side of the window.

2. Click to pin the View Palette.
3. Drag a view from the View Palette onto the drawing sheet.
4. In the **Drawing View** PropertyManager, set options such as orientation, display style, scale, etc. then click .
5. Repeat steps 3 and 4 to add views.

Note: You can have any drawing views of any models in a given drawing document.



MODEL PART DRAWING WITH SEVERAL VIEWS AND DIMENSIONS INSERTED

Drafting

You can **draft in 2D** in SolidWorks drawing documents using Sketch tools, Dimension tools, and Annotations as described in **Creating Drawings**. Concepts to consider include:

Sketch entities	In SolidWorks drawing documents, you can add **sketch entities** (lines, circles, rectangles, and so on) at any time. You can create your own line styles using **layers**, the **Line Format** tools, or **Line Style Options**.
Drawing views	You can add sketch entities and annotations to the drawing sheet or to **drawing views**. Drawing views allow you to **move** and **scale** all the items in the view in one operation. You can insert **empty views** onto drawing sheets to contain drafted entities.
Standards	The drafted elements follow the **standard** specified in **Tools, Options, Document Properties**, **Drafting Standard**. Such items as dimension arrows, tolerances, annotation display, and so on are generated based on the standard, but you can also edit the items manually (choose a different arrowhead style, for example).
Sheet formats	SolidWorks drawing **templates** contain drawing sheet **formats**. You can edit the formats and **save** them. You can also use a template without the format and create your own format, or **import** a block from your 2D CAD system (a **title block**, for example).
Grid	To display a grid, right-click and select **Display Grid**. Specify the grid spacing and snap control in **Tools, Options, Document Properties**, **Grid/Snap**.
Dimensions	**Dimensions** in SolidWorks control the geometry. The sketch entity or model element must agree with its dimension. You cannot sketch an entity at a certain size and display a dimension of a different size. However, you can scale entities in a drawing sheet or drawing view.
Relations	**Relations** (such as **Horizontal, Concentric, Tangent**) also control geometry. Some relations are **inferenced** as you sketch. You can **add**, display, and **delete** relations. To prevent **automatic** relations, press **Ctrl** as you sketch, or clear **Automatic relations** in **Tools, Options, System Options, Sketch**, **Relations/Snaps**.
Annotations	Most **annotations** work with sketch entities the same as they do with drawings derived from 3D models. Some exceptions are **hole callout** and **autoballoon**. Single **balloons** and **stacked balloons** appear with question marks, which you can replace with custom text. You can **import into drawings** the dimensions and tolerances you create with **DimXpert for parts**.

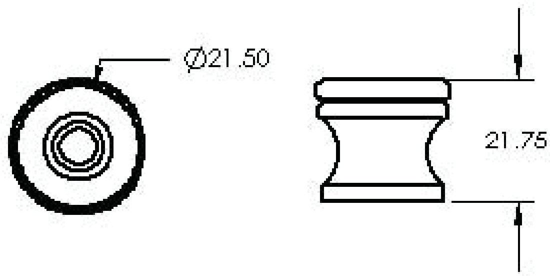
Standards

You can set up **styles** in SolidWorks to format dimensions, but it is not necessary to do so for dimensions and other annotations to follow a drawing standard.

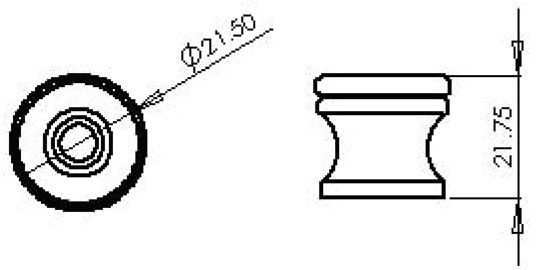
In SolidWorks, you set the standard for the current document in T**ools, Options, Document Properties**, **Drafting Standard**. The standard can be **ANSI, ISO, DIN, JIS, BSI, GOST**, or **GB**.

You can also set the standard in a drawing document **template.**

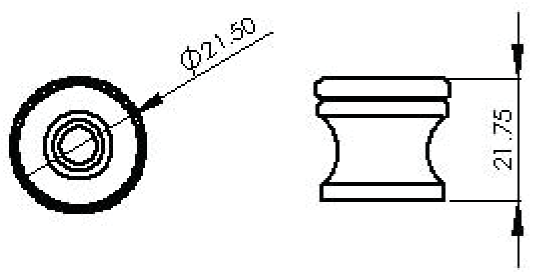
ANSI



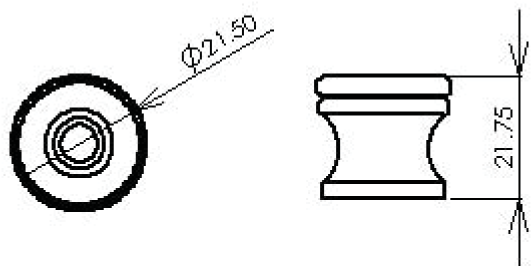
ISO



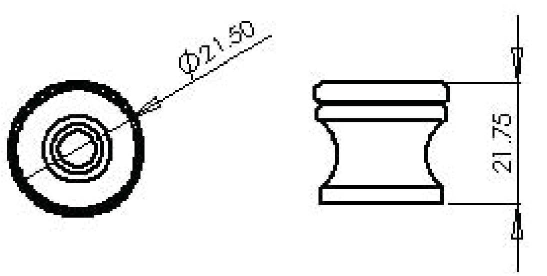
DIN



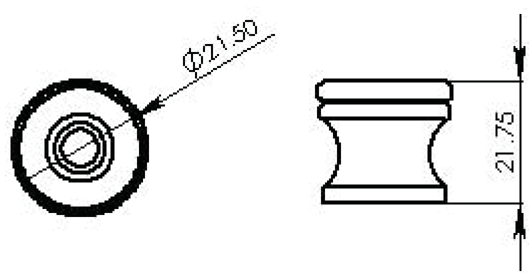
JIS



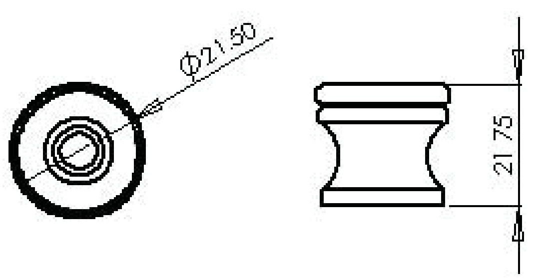
BSI



GOST



GB



Scaling

In SolidWorks, drawing views can be at any scale (2:1, 1:2, for example) in relation to the model.

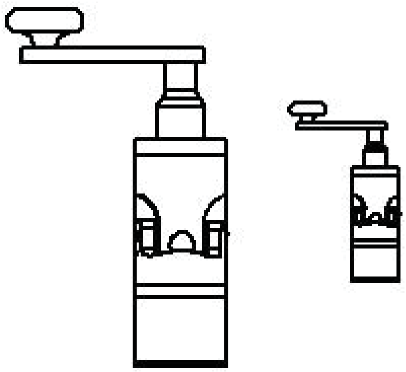
Drawing Sheets
You can set separate scales for each drawing sheet in the **Sheet Properties dialog box**. Right-click the drawing sheet outside any drawing views and select **Properties**. The scale of a drawing sheet appears in the status line at the bottom of the SolidWorks window.

Drawing Views
The scale of a drawing view is set in the PropertyManager when you select the view in the graphics area. A drawing view uses the scale of the drawing sheet unless:

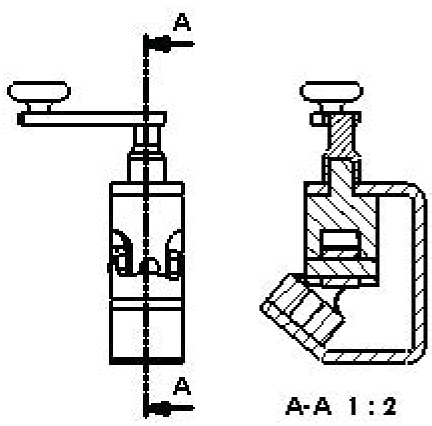
- You specify another scale, either when creating the view or any time afterwards.

  - or -

- The software needs to fit the view on the sheet with a certain scale.



When you create a child view (Section View, Detail View, and so on), the scale of the child view can be the same as the parent view, the same as the drawing sheet, or a custom scale. This section view has the same scale as its parent view.



Multiple Drawings

In SolidWorks, you can have multiple drawing sheets in a drawing document, which is like having a set of drawings all in the same file. The sheets can contain drawing views of any parts or assemblies. You can switch between sheets by selecting a named tab at the bottom of the SolidWorks window. You can also add and delete sheets using the shortcut menu.

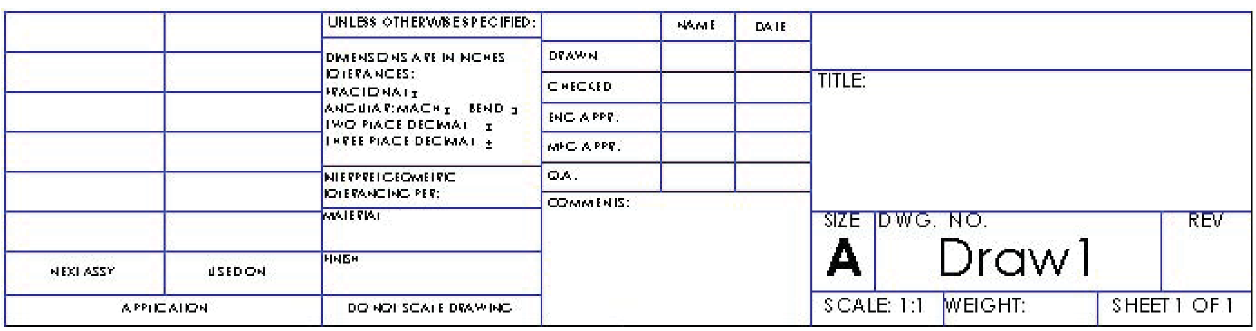
Title Blocks

When you start a new drawing in SolidWorks, you select a **template** with a specified paper size and drawing **sheet format.** The format can be standard, customized, or no format (specifying size only). When you define a title block, you can specify which template fields are editable and hotspot areas you can click to enter title block data.

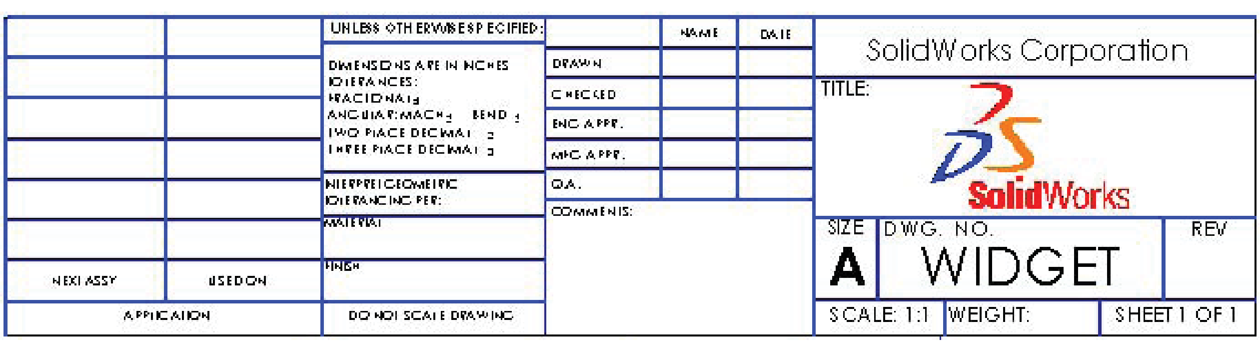
Standard formats contain title blocks. SolidWorks allows you to **edit** the sheet format. (You can also **save** sheet formats for use in future drawings.) You can add, move, format, and delete lines and text.

You can **link** note data to document properties such as file name, date, sheet number, and so on, or to custom properties that you define.

The title block in a default landscape sheet format contains the following lines and text:



In this example of editing the sheet format, a note with the company name is added, the note with the drawing name is edited, the lines are thickened, and a graphic is added.



Drawing Views

Drawing views are containers. Generally the contents are views of models. When you sketch in a drawing, or insert annotations or blocks, the entities belong to the active drawing **view** or drawing **sheet**. In SolidWorks you create drawing views as follows:

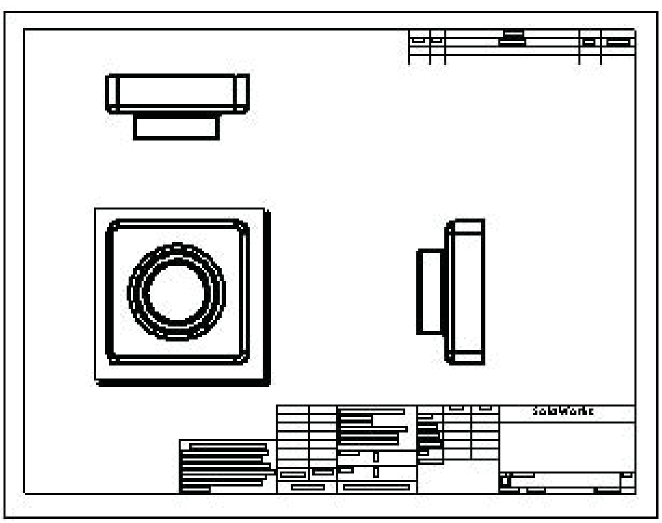
- **Standard views**, such as standard 3 views, various named model views (such as isometric), and relative views created automatically from the model.
- **Derived views** (projected, auxiliary, section, detail, broken, broken-out section, alternate position views) created in one or two steps from another view (such as drawing a profile for a detail view).
- **Empty views** (for sketch entities, notes, and so on) inserted with the menu item **Insert, Drawing View, Empty**.

Any changes in the model are automatically reflected in the drawing views.

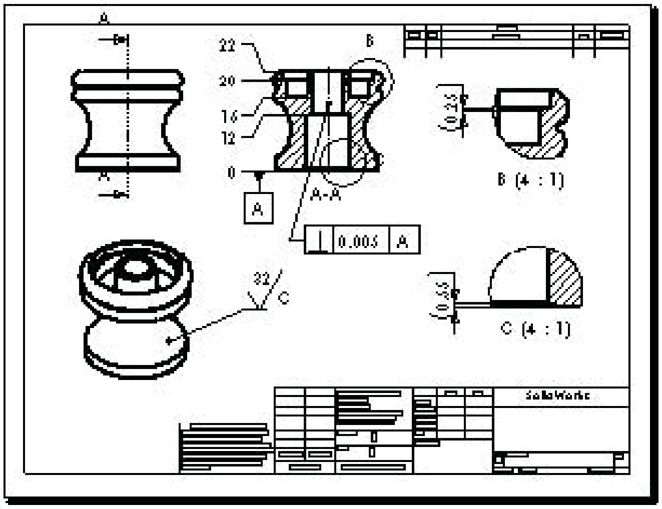
Aligning Views

Alignment between views in SolidWorks is automatic and adjustable. For example, standard 3 views are automatically aligned vertically and horizontally, while section, projected, and auxiliary views are aligned in the appropriate direction.

You can drag the views within the correct alignment. You can also break the alignment and drag views anywhere on the drawing sheet. You can rotate views and hide or show views.



STANDARD 3 VIEWS ARE ALIGNED AUTOMATICALLY. THE TOP VIEW IS CONSTRAINED HORIZONTALLY AND THE RIGHT VIEW IS CONSTRAINED VERTICALLY BY DEFAULT.



SECTION VIEWS ARE ALIGNED AUTOMATICALLY IN THE DIRECTION OF THE CUT. DETAIL VIEWS ARE NOT ALIGNED.

Dimensions in Drawings

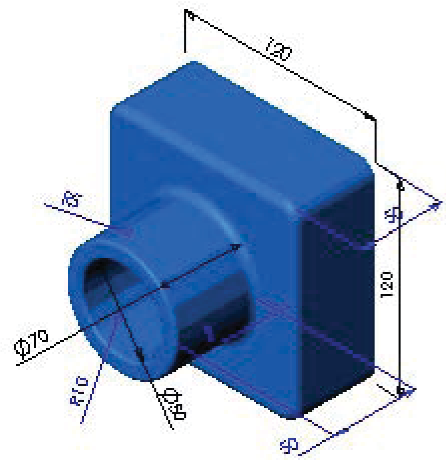
Usually you specify dimensions when you design a part, then insert the dimensions from the model into the drawing. Changing a dimension in one document changes it in any associated documents.

> **NOTE**: You can set an option during installation of SolidWorks that prevents changes in dimensions in drawings from affecting part or assembly models.

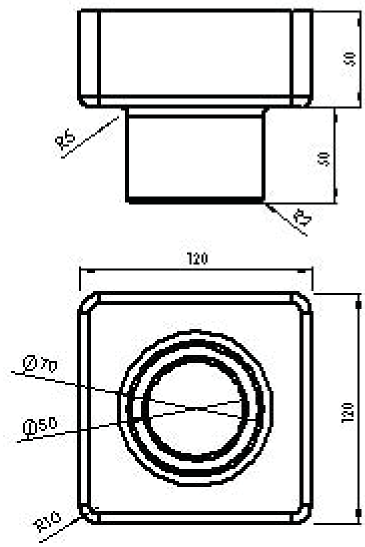
In SolidWorks, dimension formatting follows the standard that is set for the document in **Tools, Options, Document Properties**, **Drafting Standard** by default. You can change the document or template defaults for each type of dimension listed under **Tools, Options, Document Properties**, **Dimensions**. SolidWorks uses **styles** to save particular formatting.

Reference dimensions cannot be modified and do not change model geometry. However, when a model changes, reference dimensions update automatically. Model dimensions are linked to the model parametrically, using dimension names, and, when changed (in drawings or in model documents), modify the model.

When you insert dimensions in part and assembly documents, they are marked for drawings unless you specify otherwise. When you insert model dimensions with **Model Items**, **automatically** for a new drawing view, or with **Autodimension**, only the dimensions marked for drawings are inserted. When you insert an **annotation view** into a drawing, all annotations in the part or assembly are inserted in the drawing.



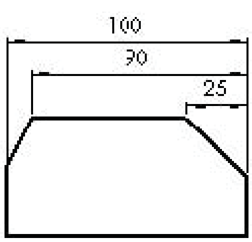
DIMENSIONS DEFINE THE GEOMETRY IN THE MODEL SKETCHES.



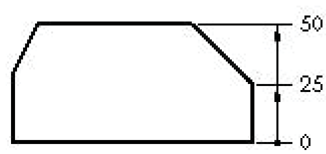
THE MODEL DIMENSIONS ARE TRANSFERRED INTO THE DRAWING USING INSERT, MODEL ITEMS.

Baseline dimensions, ordinate dimensions, chamfer dimensions, and hole callouts are available in drawings. Ordinate dimensions are also available in sketches.

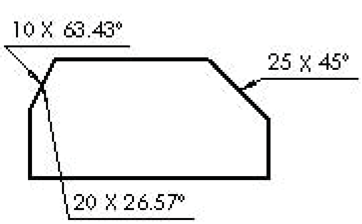
Baseline dimensions



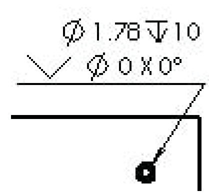
Ordinate dimensions



Chamfer dimensions



Hole callout

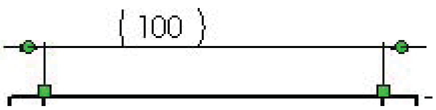


Dimension Formats

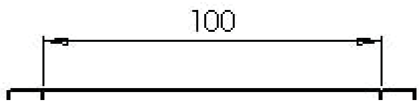
You can format dimensions individually or as a group in sketches and drawings. If you select a group of dimensions, only those properties the dimensions have in common are available for editing.

Editing in the Graphics Area
To position dimensions, select and drag them. To change the direction of the **arrows**, click the circular handles. Several display options, such as **Show Parentheses** and **Inspection Dimension**  , are available in the **Dimension Value PropertyManager.** In the following example, the arrows are flipped, the parentheses removed, and the dimension value centered.

Before



After



PropertyManagers
Select the dimension (or dimensions) and edit properties in these PropertyManagers:

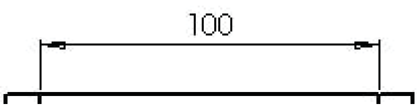
- **Dimension Value PropertyManager**
- **Dimension Leaders PropertyManager**
- **Dimension Other PropertyManager**

The PropertyManager properties include:

- **Dimension Style**
- **Tolerance/Precision**
- **Witness/Leader Display**
- **Dimension Text (including alignment and symbols)**
- **Primary Value**
- **Display Options**
- **Break Lines**
- **Layer**

In this example, the arrow style has been changed (from the default open arrows to solid arrows) and tolerance and text have been added in the **Dimension** PropertyManager.

Before



After

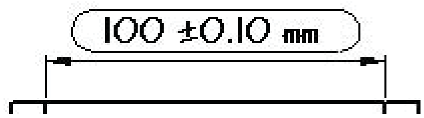
Other properties you can modify using the PropertyManagers include:

- **Value**
- **Name**
- **Units**
- **Precision**
- **Font**
- Various check boxes and buttons

In this example, you modified the font size and style, then added an inspection display.

Before

After

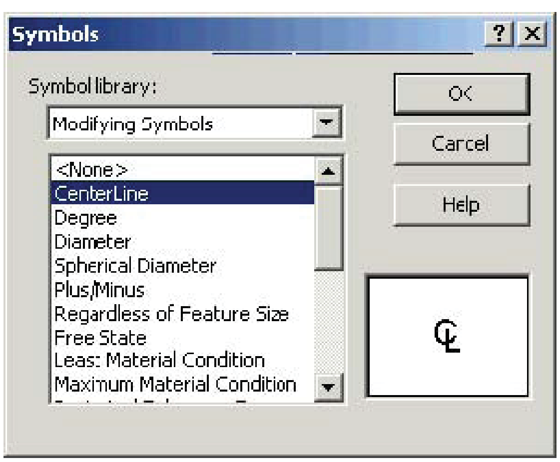


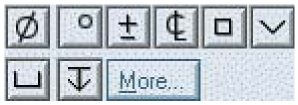
Dimension Styles

You can save any dimension property as part of a **Dimension Style**. You can also name favorites, apply them to multiple dimensions, update, and save them.

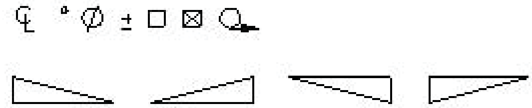
Symbols

SolidWorks has a library of symbols (such as degrees, depth, and so on). In the **Dimension Value** PropertyManager, click **More Symbols** under **Dimension Text** to access the library. Symbol libraries for various annotations, such as Notes, Geometric Tolerance Symbols, Surface Finish Symbols, Weld Symbols, and so on, are also available in PropertyManagers.



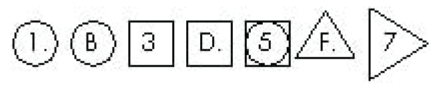


SYMBOL BUTTONS IN THE **DIMENSION VALUE PROPERTYMANAGER**



SOME SYMBOLS FROM THE **MODIFYING SYMBOLS** LIBRARY

COMPLETE **HOLE SYMBOLS** LIBRARY



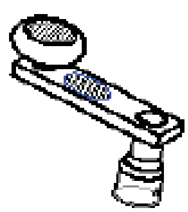
REPRESENTATIVE **FLAG** SYMBOLS

Annotations

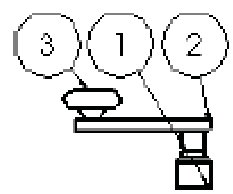
SolidWorks has many tools for specific annotations, as shown below. You can control many properties of the annotations in PropertyManagers and dialog boxes.

Some annotations, such as dowel pin symbols and area hatch, are available only in drawings. Many others, such as notes and weld symbols, can be added in model documents during the design phase and then inserted automatically from the model documents into the drawings.

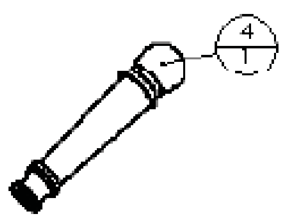
Area Hatch/Fill



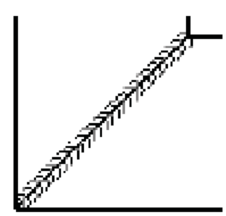
AutoBalloon



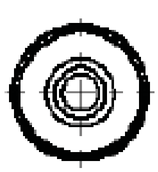
Balloon



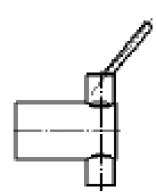
Caterpillar



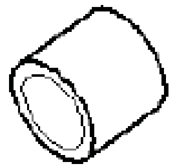
Center Mark



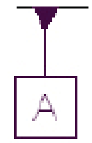
Centerline



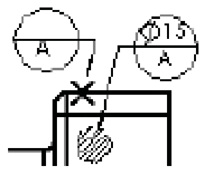
Cosmetic Thread



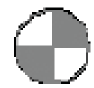
Datum Feature Symbol



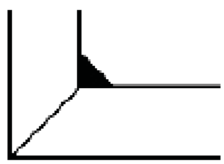
Datum Target Symbol



Dowel Pin Symbol

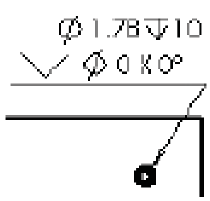


End Treatment

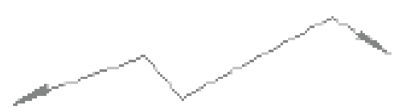


Geometric Tolerance Symbol

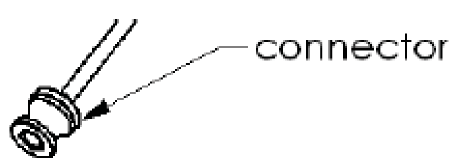
Hole Callout



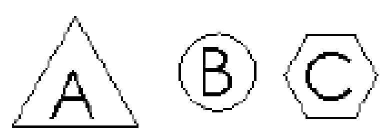
Multi-jog Leader



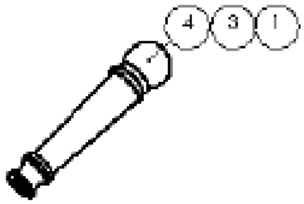
Note



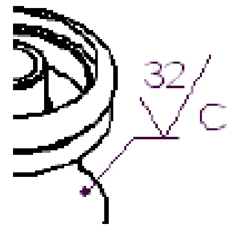
Revision Symbol



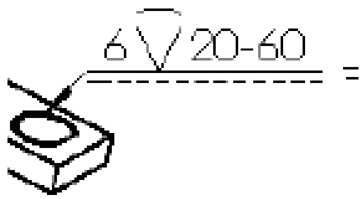
Stacked Balloon



Surface Finish Symbol



Weld Symbol



Automatic Drawing Operations

In addition to the autodimensioning and autotransitioning in **sketching**, automated operations in drawings increase productivity.

3D annotations. Annotation toolbar. Insert annotations into a part or assembly document. The 3D annotations are organized into annotation views that correspond to the model's orthographic views, such as front, bottom, etc. You can then use the annotation views in a drawing. The annotation views are converted into 2D drawing views; the annotations you inserted in the model are retained in the drawing.

Model Items. Annotation toolbar. Insert dimensions, annotations, and reference geometry from a part or assembly document into a drawing in one operation. You can specify all dimensions or only those marked for drawings.

AutoBalloon. Annotation toolbar. Add balloons to all components in a drawing view in one operation, choosing a layout and balloon style, size, and text.

Smart Dimension, Autodimension. Dimensions/Relations toolbar. Insert horizontal and vertical reference dimensions into drawing views as baseline, chain, or ordinate dimensions.

Center Marks. Annotation toolbar. Add center marks to all appropriate entities in a drawing view in one operation, choosing single, linear, or circular style, mark size, extended lines, font, angle, and named layer.

Centerlines. Annotation toolbar. Add centerlines to all appropriate entities in a drawing view in one operation.

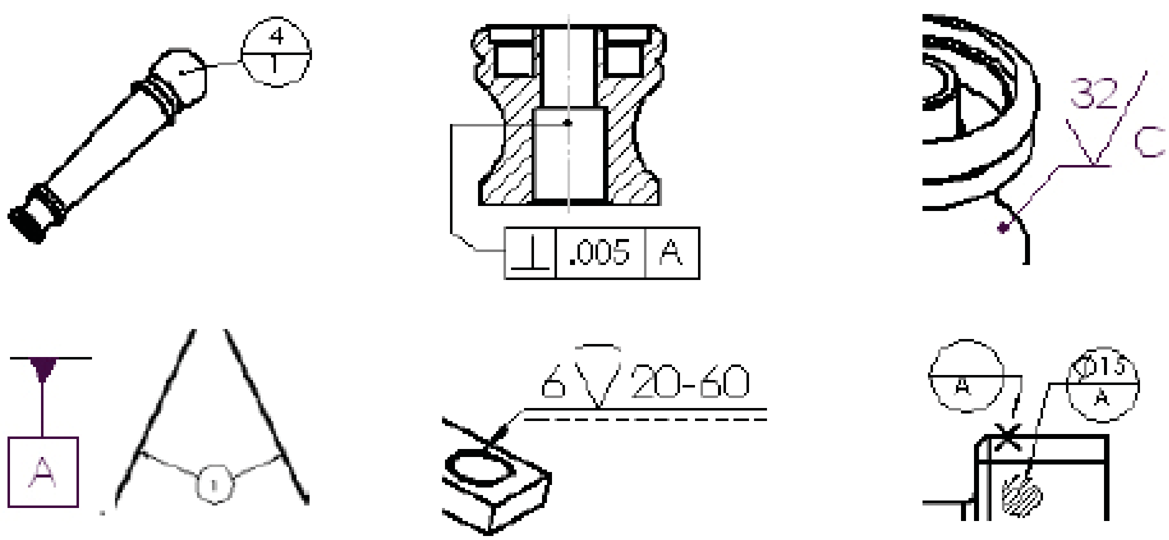
You can specify in **Tools, Options, Document Properties**, **Detailing** that the following items be inserted automatically into new drawing views:

- **Center Marks**
- **Centerlines**
- **Balloons**
- **Dimensions marked for drawings**

Leaders

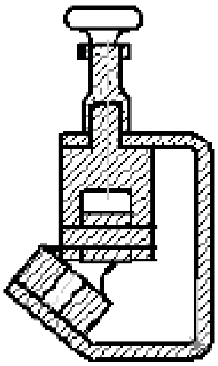
In SolidWorks, leaders are available with all annotations that use leaders. You can choose straight, bent, or multi-jog leaders. You can also create **multi-jog leaders** separately, and you can add **multiple leaders**.

When an annotation moves, the leader attached to the annotation moves with it. The leader also moves with any model to which it is attached.

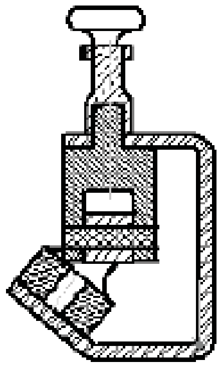


Crosshatching

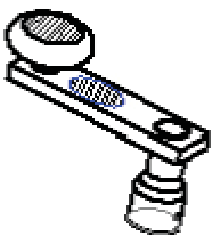
SolidWorks adds crosshatching to section views automatically. You can modify the crosshatch pattern manually. You can also add area hatching to faces or to closed sketch entities in drawings.



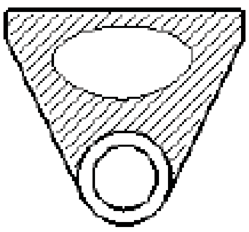
AUTOMATIC CROSSHATCHING IN A DRAWING SECTION VIEW



PROPERTIES (MATERIAL, SCALE, AND ANGLE) OF INDIVIDUAL CROSSHATCHED SECTIONS SPECIFIED MANUALLY



AREA HATCH ADDED TO A FACE AND A SKETCHED ELLIPSE



AREA HATCH REGION BOUNDED BY A COMBINATION OF MODEL EDGES AND SKETCH ENTITIES

Tables

The following types of **tables** are available on the Table toolbar in drawings:

General Table

Bill of Materials

Hole Table

Revision Table

Weldment Cut List

Design Table (Excel-based tables for managing configurations)

Table functionality includes:

- Standard or custom **templates**
- **Anchor** points
- Drag to move and resize
- Snap to elements in the sheet format
- Use context toolbars to **edit cells and table format**
- Add columns and rows
- Split or merge tables and cells
- Sort column contents
- Control color with **layers**

Each table has PropertyManagers for:

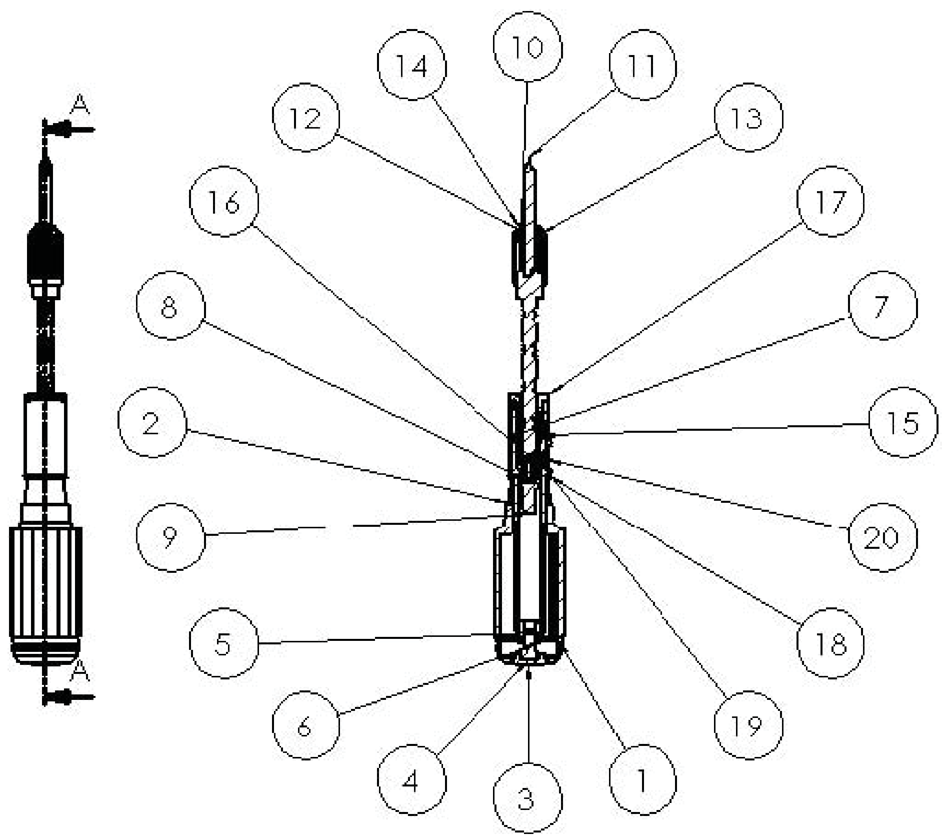
- Table Properties
- Table Format
- Column Properties
- Cell Properties
- Row Properties

Bill of Materials

SolidWorks automatically populates a **Bill of Materials** (BOM) with item numbers, quantities, part numbers, and custom properties in assembly drawings. You can anchor, move, edit, and split a BOM.

When you insert **balloons** into a drawing, the item numbers and quantities in the balloons correspond to the numbers in the Bill of Materials. If an assembly has more than one configuration, you can list quantities of components for all configurations or selected configurations.

You can create BOMs in assembly files and multibody part files. You can insert a BOM saved with an assembly into a referenced drawing. You do not need to create a drawing first.



ITEM NO.	PART NUMBER	DESCRIPTION	C0/QTY.	C1/QTY.	C2/QTY.
1	8112156	Handle	1	1	-
2	8112174	Handle-shoulder	1	1	-
3	8113199	End cap	1	1	-
4	9113155	Embedded bolt	1	1	1
5	8112992	Housing	1	1	1
6	8116170	Gearshaft screw	1	-	1
7	112-135	Crescent washer	1	1	1
8	112-139	Shaft screw	1	1	1
9	113-144	Shaft spring	1	1	1
10	8112188	Shaft	1	1	1
11	403-112	Phillips	1	-	-
12	8114175	Collar	1	1	1
13	Purchased	Push collar washer	2	2	2
14	Purchased	Clip	1	-	1
15	8112001-1	Turn gear	1	1	1
16	8112001-2	Turn gear	1	1	1
17	8115777	Switch cover nut	1	1	1
18	8115142	Switch casing	1	1	1
19	8117199	Turn selector plate	2	2	2
20	9581-12	Selector switch	1	1	1

Layers

In SolidWorks, you can specify the color, style, and thickness of lines in named layers. You can move objects into layers, and you can turn layers on and off. The layer list is built into many annotation and **dimension** dialog boxes.

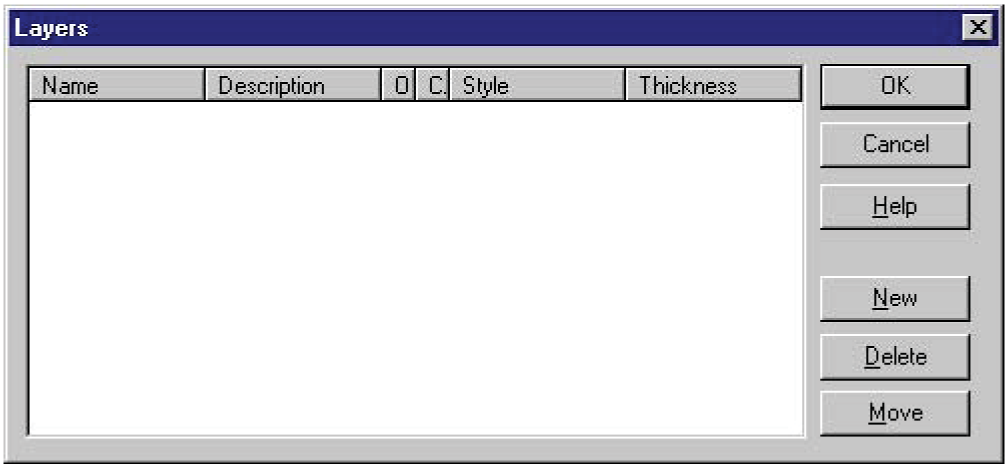
You can also format lines individually using the **Line Format** tools. You can specify document-level line thickness and style. Click **Tools, Options, Document Properties** and set **Line Font, Line Style**, and **Line Thickness**.

SolidWorks has multiple drawing sheets, and you can hide and show drawing views, assembly components, lines, and various other items without using layers.

In addition to creating layers in SolidWorks, you can **import** drawings with layers into SolidWorks. All 2D drawings layers are preserved in SolidWorks. When exporting from SolidWorks, you can map entity types to specific layers.

The Layer toolbar contains a list of layers in the drawing and the **Layer Properties** tool  .

Click the **Layer Properties**  tool to bring up the **Layers dialog box**. Create new layers and specify the **Color, Style**, and **Thickness** of lines in each layer.



Blocks

You can make, save, edit, and insert **blocks** for drawing items and **sketch entities** that you use often, such as standard notes, title blocks, label positions, and so on. You can attach blocks to geometry or to drawing views, and you can insert them into sheet formats.

Blocks can include the following items:

- Text (Notes)
- Dimensions
- Balloons
- Imported entities and text
- Area hatch

To create blocks, select items (from the list above) in the graphics area and click **Tools, Block, Make**.

You can save a sketch directly to a block file. Click **Save Sketch as Block**  (Blocks toolbar) or **Tools, Blocks, Save**.

When you insert blocks into drawings, you insert instances of the block definition, which you can modify as follows:

- Scale
- Rotate
- Add leaders
- Edit values of **attributes**

Additional functionality for blocks includes:

- Dynamically edit block definitions, including file definitions
  - Editing is in-place (no separate block editor window)
  - You can add or remove entities while editing
- Explode blocks in the graphics area
- Move, copy, and paste block instances
- Save blocks to file, or create and use in a drawing without saving to file
- Use part or drawing blocks interchangeably
- Change block base points
- Change leader attachment points and leader anchor points
- Reference external definitions, including existing blocks
- Snap to and infer from sketches to block points on a drawing sheet
- Add dimensions and constraints between sketch entities of two block instances
- Move block instances to and from layers
  - Once a block (instance) is moved to a layer, all entities inside the block take the layer properties.

Conclusion

Drawing in SoldWorks is just like drawing in 2D programs, but by integrating the drawing process with 3D modeling, you save time both creating and correcting your designs. With SolidWorks on your team, linked drawings and models assure consistency so you can design products smarter.

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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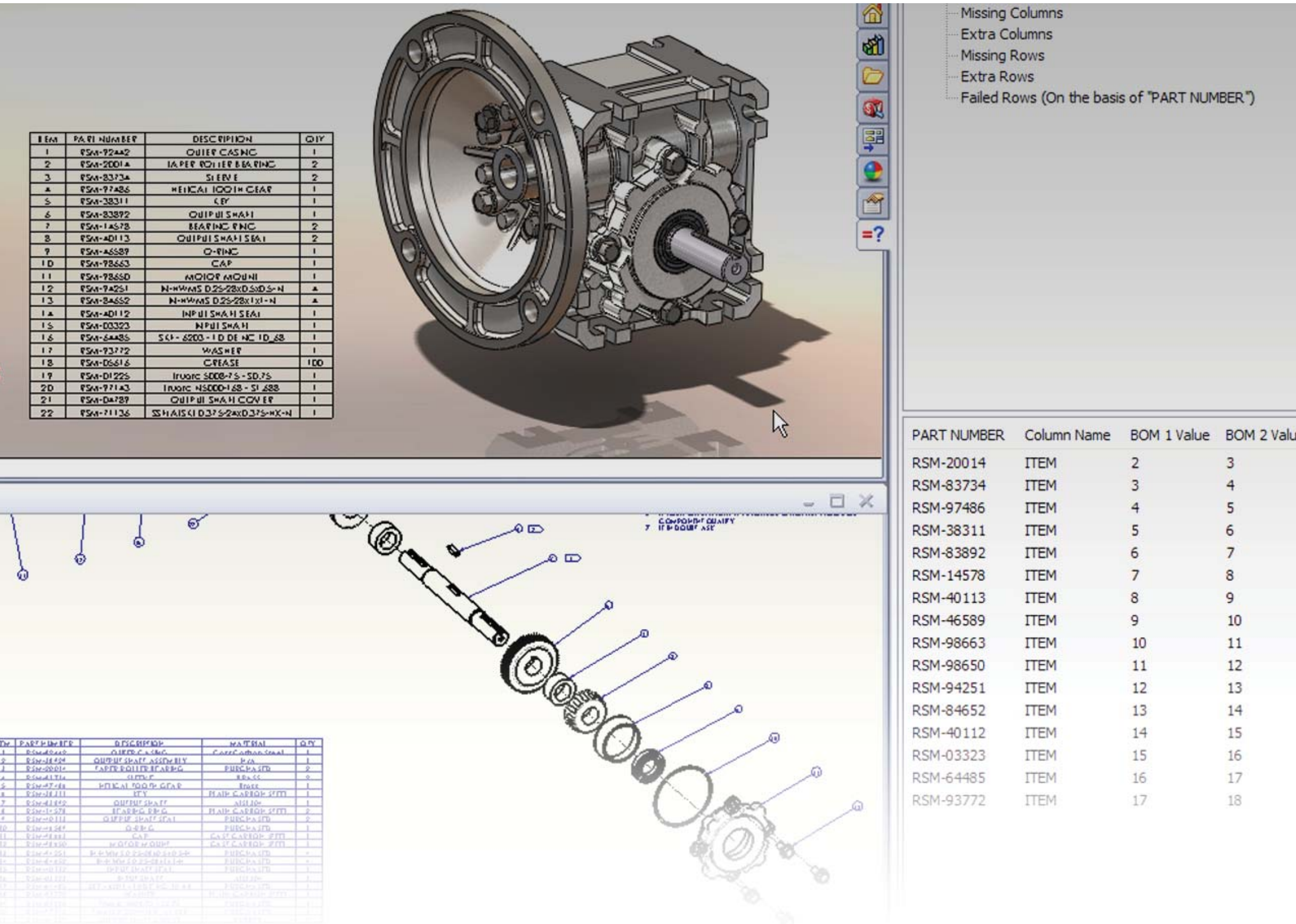
Implementation Guide:
Modeling in 3D with SolidWorks®



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.

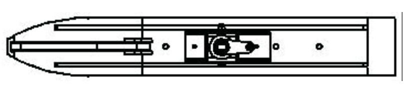
SolidWorks

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.

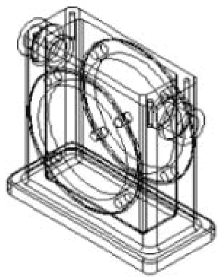
Types of Models

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

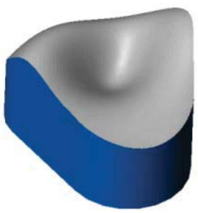
- 2D drawings



- 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.

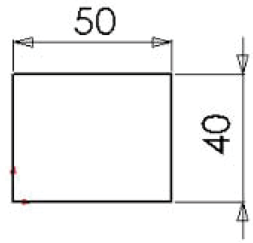
Sketching versus Drawing

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.

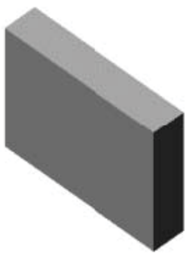
The general procedure, from sketch through model to drawing, is as follows:

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

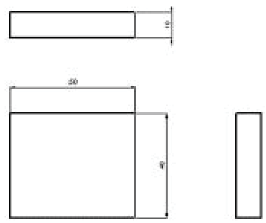
2. **DIMENSION** THE SKETCH EXACTLY.



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.

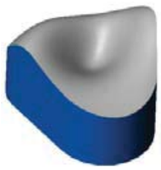
- **Sketched features** such as extrusions, revolves, sweeps, and lofts are based on sketch geometry.
- **Applied features** such as chamfers, fillets, and shells are applied directly to the model.

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

Types of Files

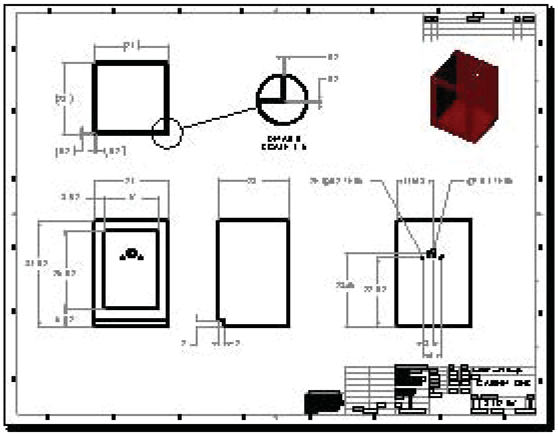
In SolidWorks, you can open any number of part, assembly, or drawing documents at the same time:

- 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:

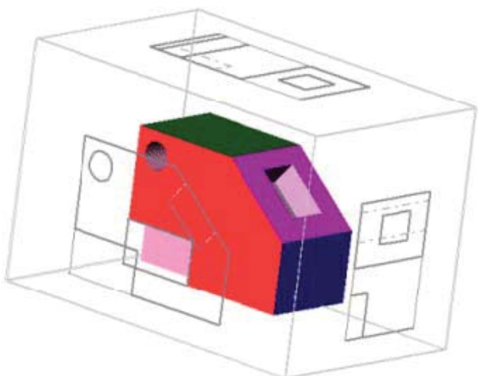
- Open a drawing from its associated part or assembly document
- Open a part or assembly document from a drawing view
- Open a part from the component in its assembly document

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.



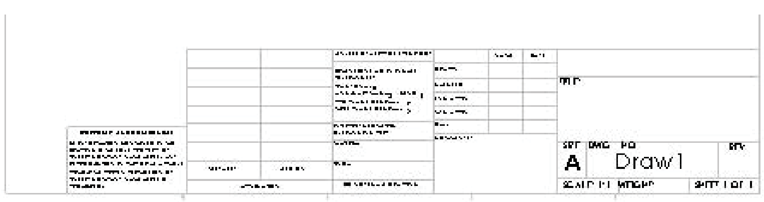
Templates

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)

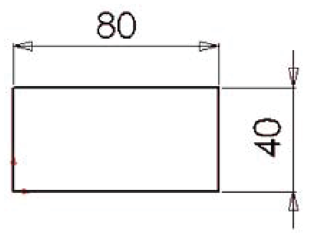
The default template for the A size landscape drawing format includes standard formatting and text that you can edit in the drawing sheet format.



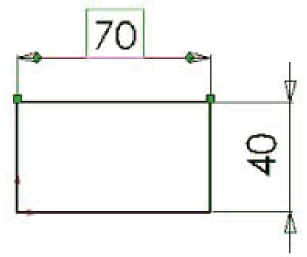
Parametric Dimensions

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

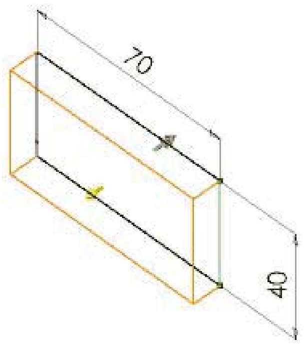
1. OPEN A SKETCH, SKETCH A RECTANGLE, AND DIMENSION THE RECTANGLE.



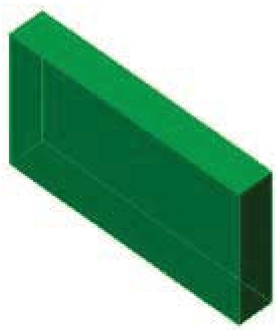
2. MODIFY THE DIMENSIONS AS NEEDED WHILE CREATING THE SKETCH.



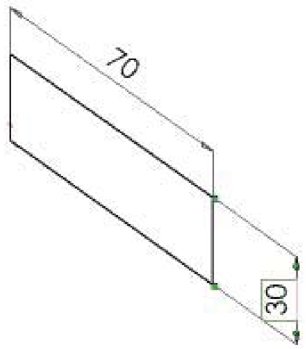
3. EXTRUDE A BLOCK BASE FEATURE.



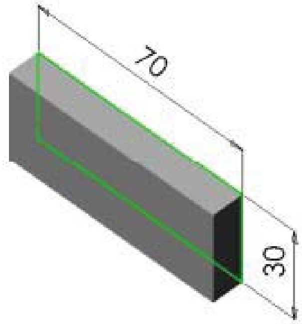
4. COMPLETE THE FEATURE TO CLOSE THE SKETCH AND SHOW THE SOLID IN SHADED MODE.



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.

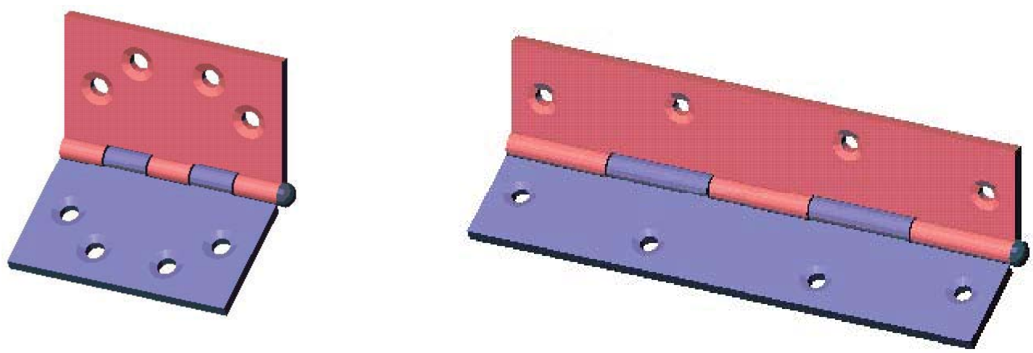
Design Intent

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.

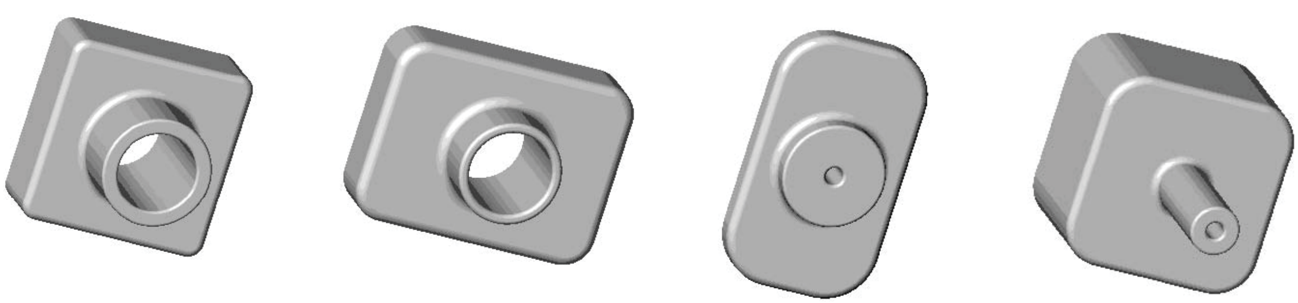
In the following example, one hole is fixed, one is driven by an equation, and the other two are mirrored. As the size of the hinge changes, the holes remain properly spaced along the length and width.



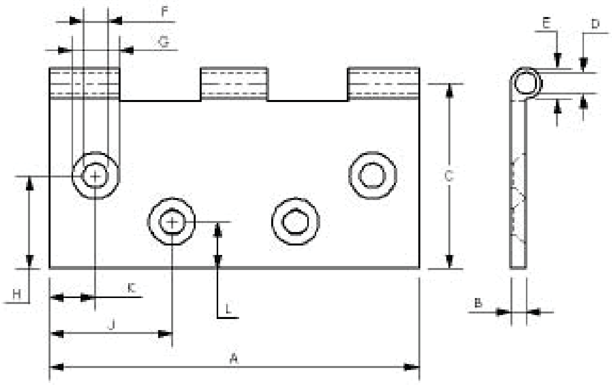
Configurations

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.



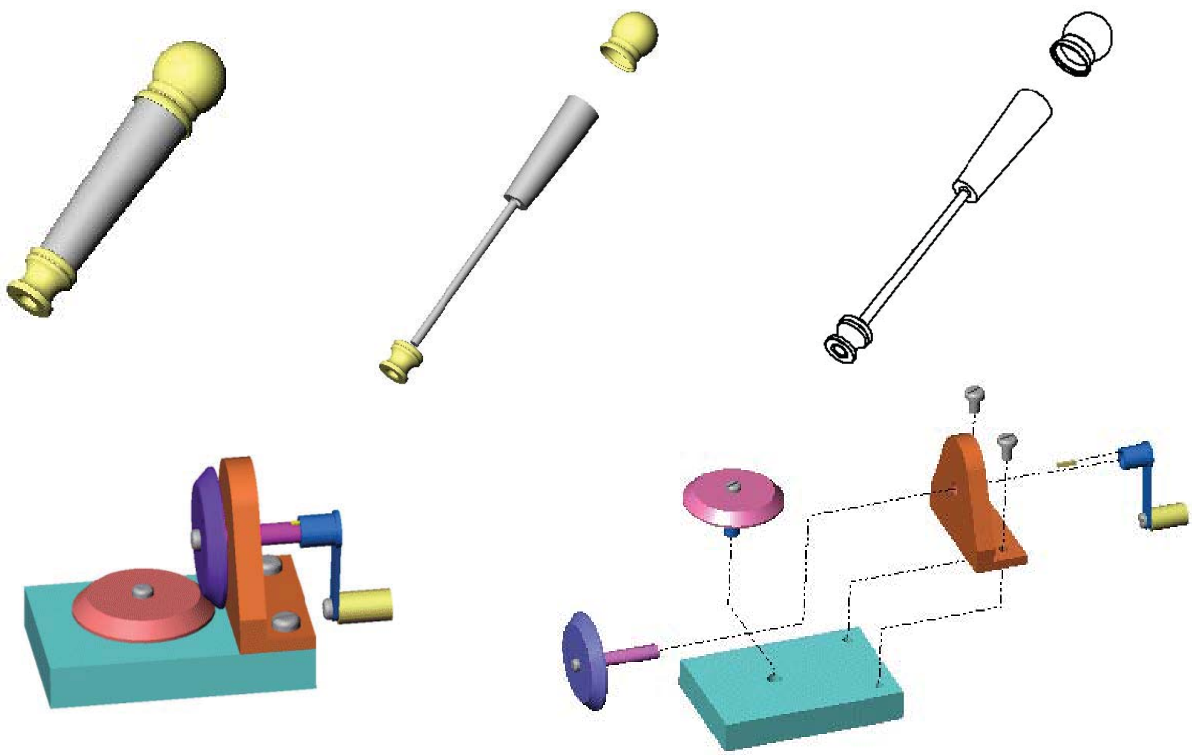
You can display design tables in drawings.



	A	B	C	D	E	F	G	H	J	K	L
001-080	80	4	50	5	8	5	9	25	26	10	10
001-120	120	5	60	7	10	8	15	30	40	15	15
001-200	200	8	90	13	16	8	15	45	66	20	15

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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