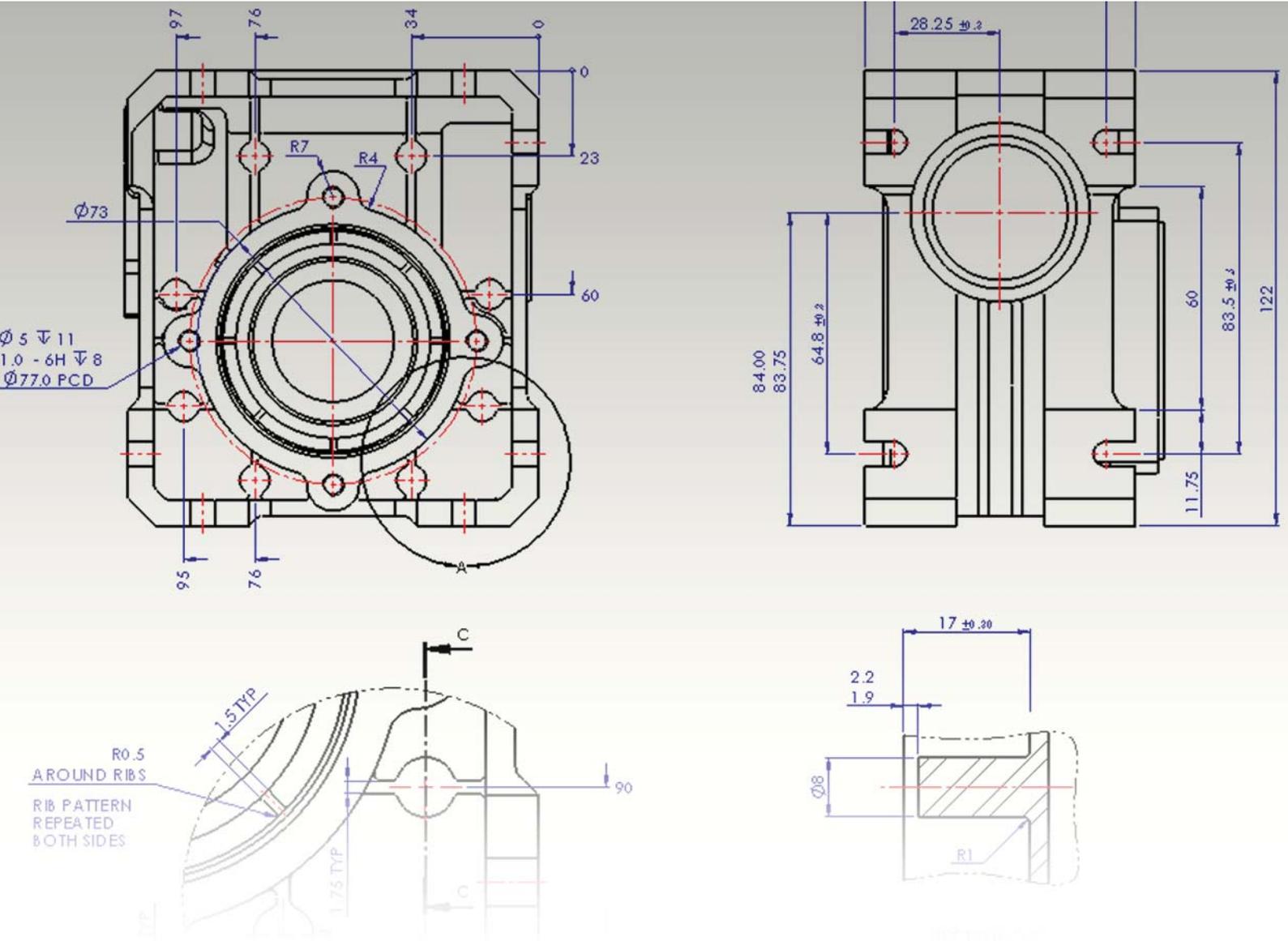# Implementation Guide:
Creating Drawings with SolidWorks®

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.

ƎS SolidWorks

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.
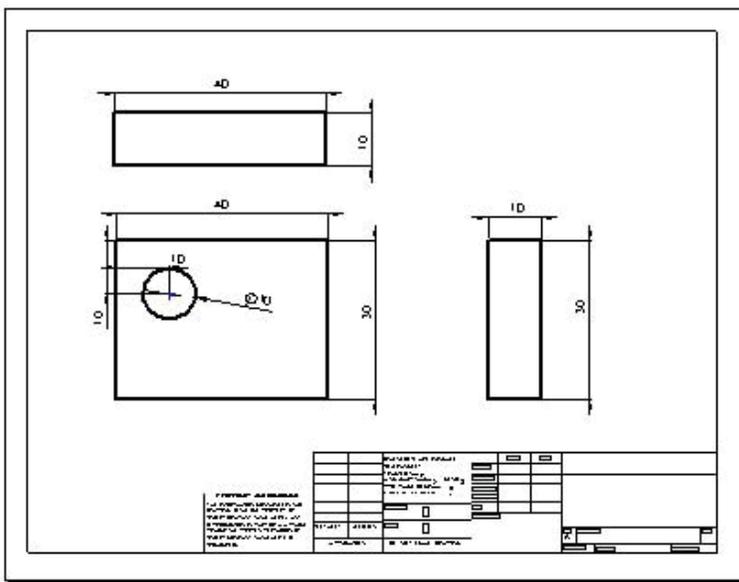
Creating Drawings

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** [icon] 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 [icon] 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 [icon].

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

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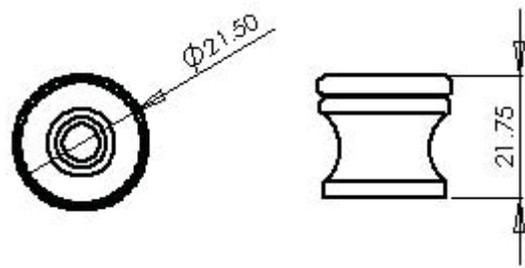
In SolidWorks, you set the standard for the current document in T**ools, Options, Document Properties**, **Drafting Standard**. The standard can be **ANSI, ISO, DIN, JIS, BSI, GOST**, or **GB**.

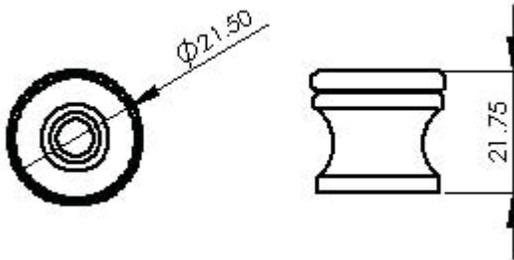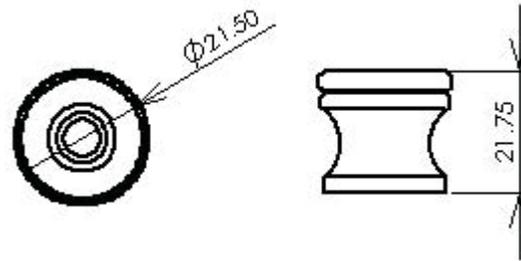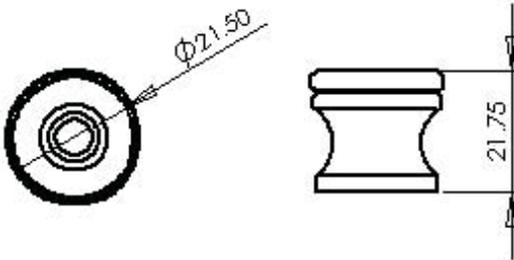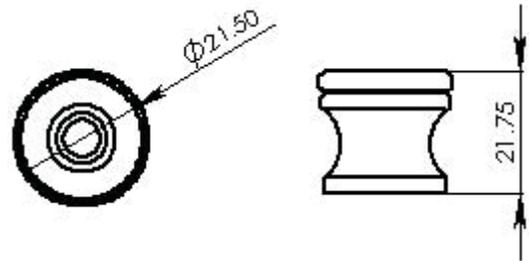You can also set the standard in a drawing document **template**.
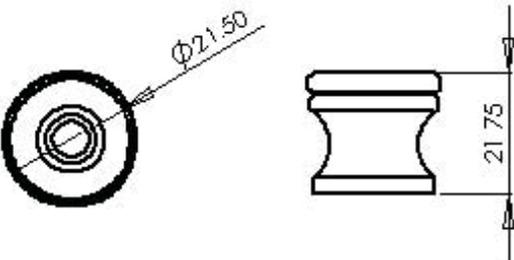
ANSI
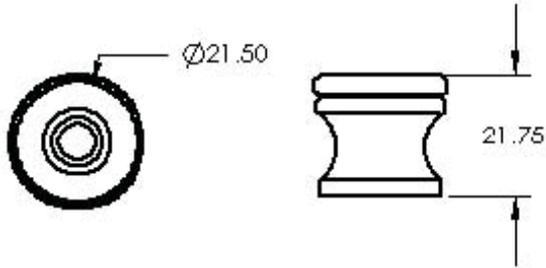
ISO

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.
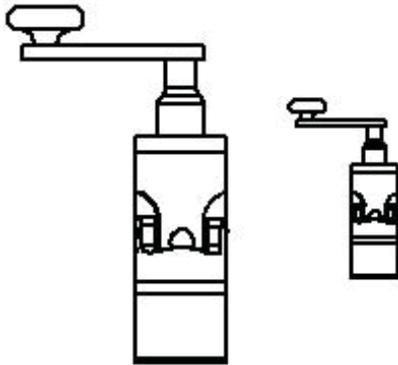
Drawing Sheets

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.
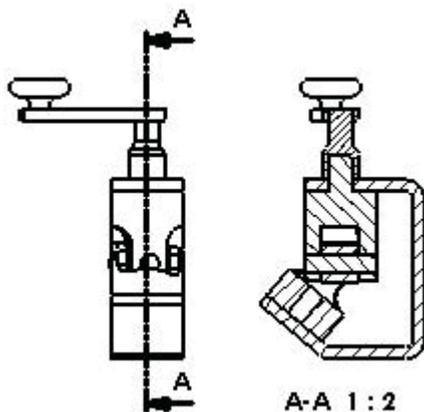
Drawing Views

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.

A-A 1:2

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.
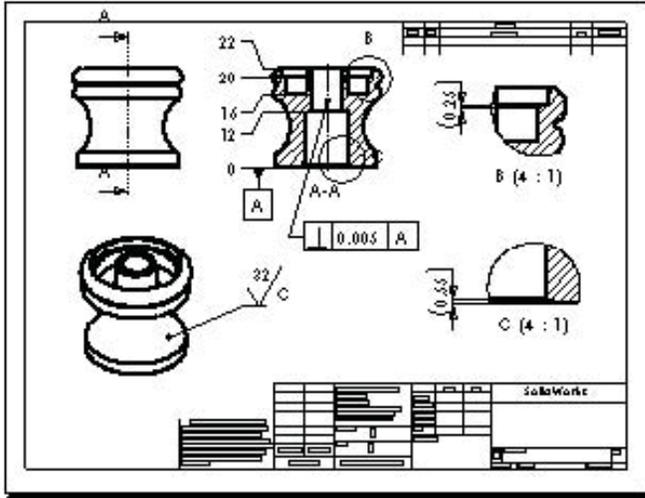
Aligning Views

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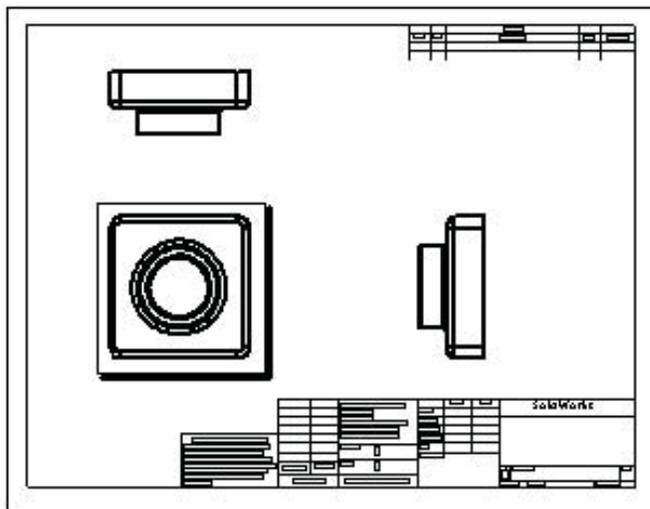
STANDARD 3 VIEWS ARE ALIGNED AUTOMATICALLY. THE TOP VIEW IS CONSTRAINED HORIZONTALLY AND THE RIGHT VIEW IS CONSTRAINED VERTICALLY BY DEFAULT.
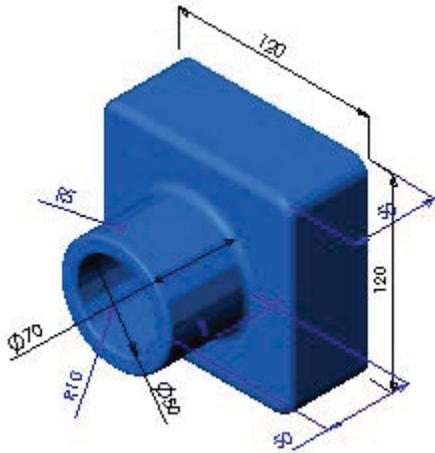
Dimensions in Drawings

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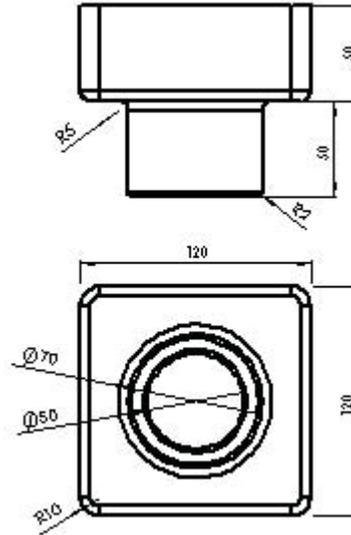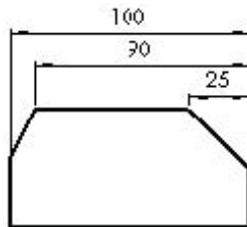
Baseline dimensions, ordinate dimensions, chamfer dimensions, and hole callouts are available in drawings. Ordinate dimensions are also available in sketches.
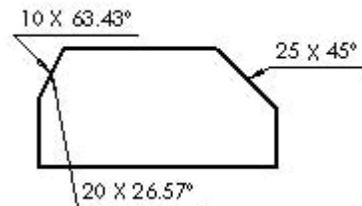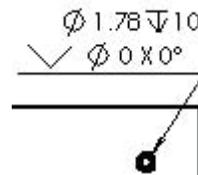
Baseline dimensions

Ordinate dimensions
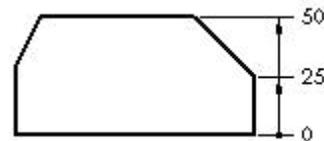
Chamfer dimensions


Hole callout

9

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.
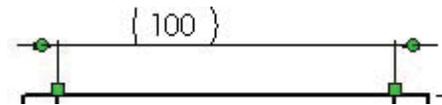
Before   After
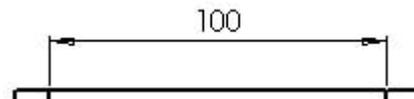
PropertyManagers

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

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

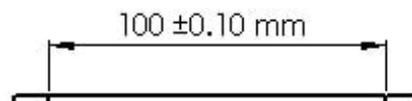The PropertyManager properties include:

- **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.
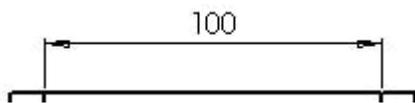
Before   After
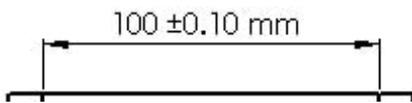
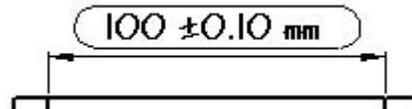Other properties you can modify using the PropertyManagers include:

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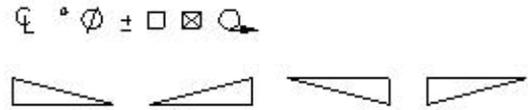
SolidWorks has a library of symbols (such as degrees, depth, and so on). In the **Dimension Value** PropertyManager, click **More Symbols** under **Dimension Text** to access the library. Symbol libraries for various annotations, such as Notes, Geometric Tolerance Symbols, Surface Finish Symbols, Weld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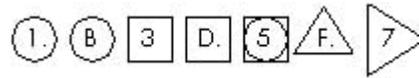
COMPLETE **HOLE SYMBOLS** LIBRARY

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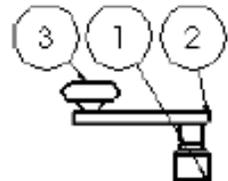
Some annotations, such as dowel pin symbols and area hatch, are available only in drawings. Many others, such as notes and weld symbols, can be added in model documents during the design phase and then inserted automatically from the model documents into the drawings.
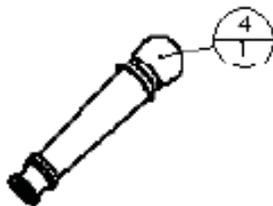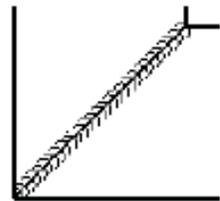
Area Hatch/Fill

AutoBalloon

Balloon

Caterpillar

Center Mark

Centerline
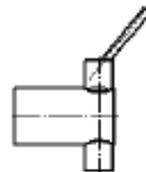
⎁	**Cosmetic Thread**	⎁	**Datum Feature Symbol**
	Datum Target Symbol		**Dowel Pin Symbol**
	End Treatment		**Geometric Tolerance Symbol**
	Hole Callout		**Multi-jog Leader**
A	**Note**		**Revision Symbol**
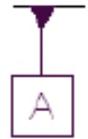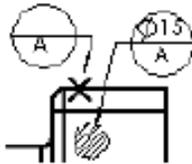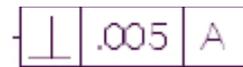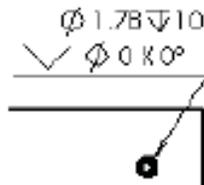
connector
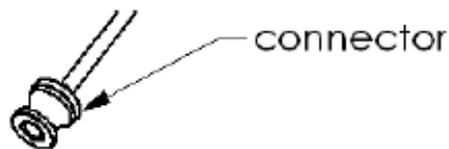
Stacked Balloon
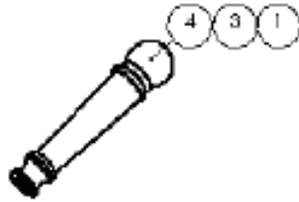
Surface Finish Symbol
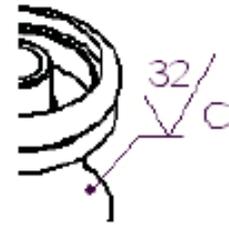
Weld Symbol
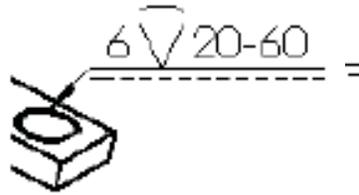
In addition to the autodimensioning and autotransitioning in **sketching**, automated operations in drawings increase productivity.

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.
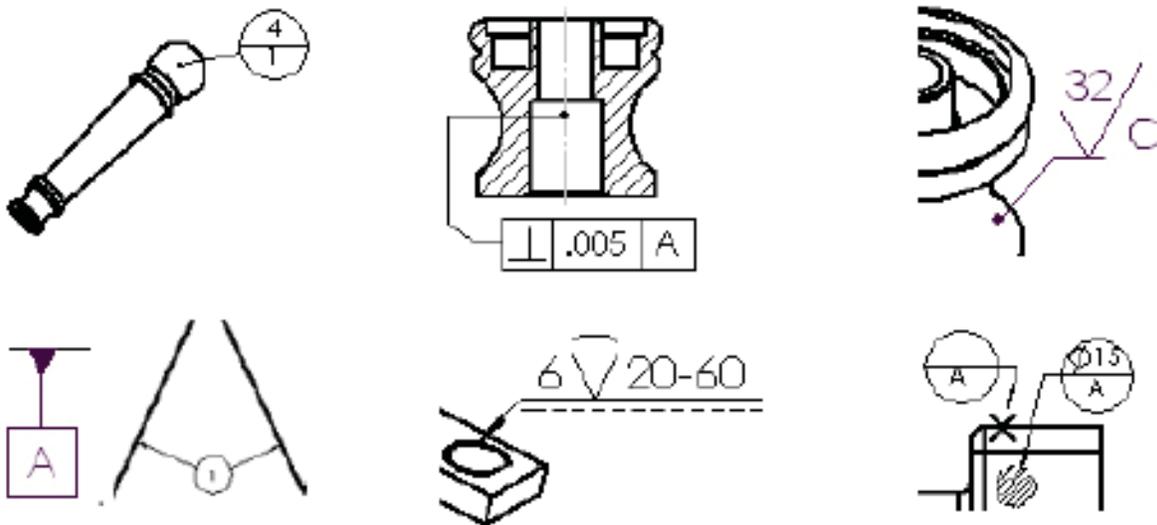
Crosshatching

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.
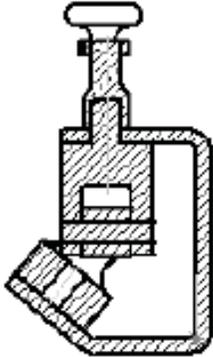
AUTOMATIC CROSSHATCHING IN A DRAWING
SECTION VIEW
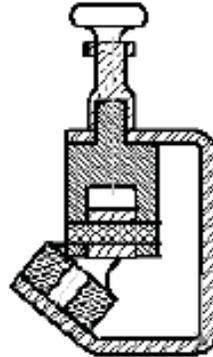
PROPERTIES (MATERIAL, SCALE, AND ANGLE)
OF INDIVIDUAL CROSSHATCHED SECTIONS
SPECIFIED MANUALLY

AREA HATCH ADDED TO A FACE AND
A SKETCHED ELLIPSE
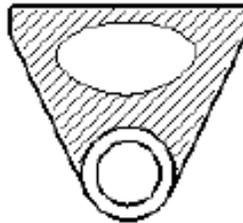
AREA HATCH REGION BOUNDED BY A
COMBINATION OF MODEL EDGES AND
SKETCH ENTITIES

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.
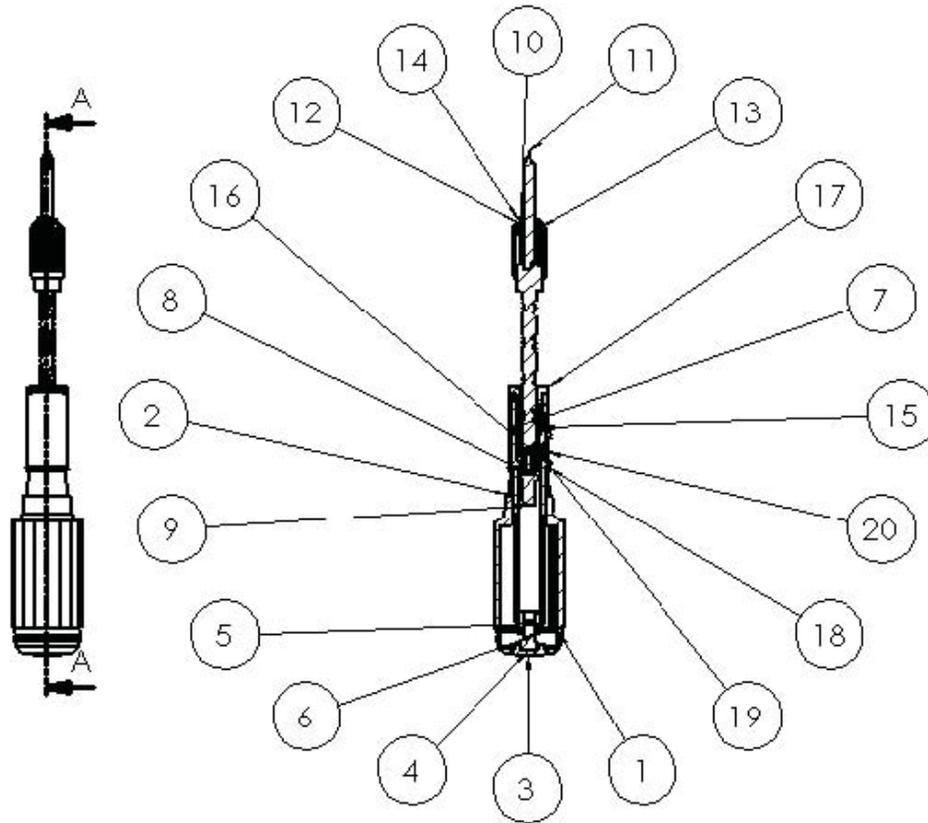
ITEM NO.	PART NUMBER	DESCRIPTION	C0/QTY.	C1/QTY.	C2/QTY.
1	8112156	Handle	1	1	-
2	8112174	Handle-shoulder	1	1	-
3	8113199	End cap	1	1	-
4	9113155	Embedded bolt	1	1	1
5	8112992	Housing	1	1	1
6	8116170	Gearshaft screw	1	-	1
7	112-135	Crescent washer	1	1	1
8	112-139	Shaft screw	1	1	1
9	113-144	Shaft spring	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.
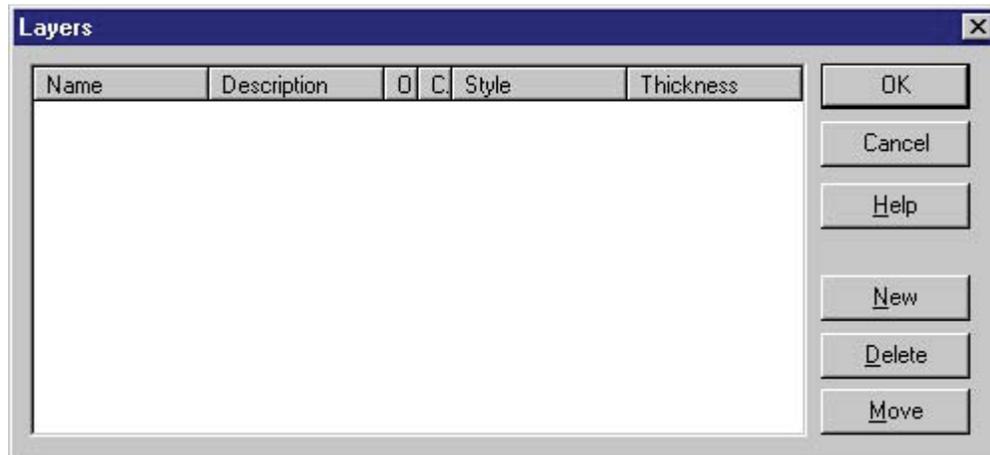
Blocks

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**.