



Drawing and Detailing with SolidWorks 2010

By Planchard & Planchard CSWP

Referencing the ASME Y14 Engineering Drawing and Related Documentation Practices

REVISIONS			
REV.	DESCRIPTION	DATE	APPROVED

3X $\varnothing 3.50 \nabla 52$
 $\square \varnothing 6.50 \nabla 3.50$

	TOLERANCES:	NAME	DATE	D&M ENGINEERING CYLINDER ASSEMBLY
	ONE PLACE DECIMAL ±.1	DRAWN	DCP	
	TWO PLACE DECIMAL ±.01	CHECKED		
	ANGULAR ±5°	ENG APPR.		
MATERIAL	FINISH	MFG APPR.		
INTERPRET DIM AND TOL PER ASME Y14.5-1994				SEE DWG. NO. 22105-991 REV. A



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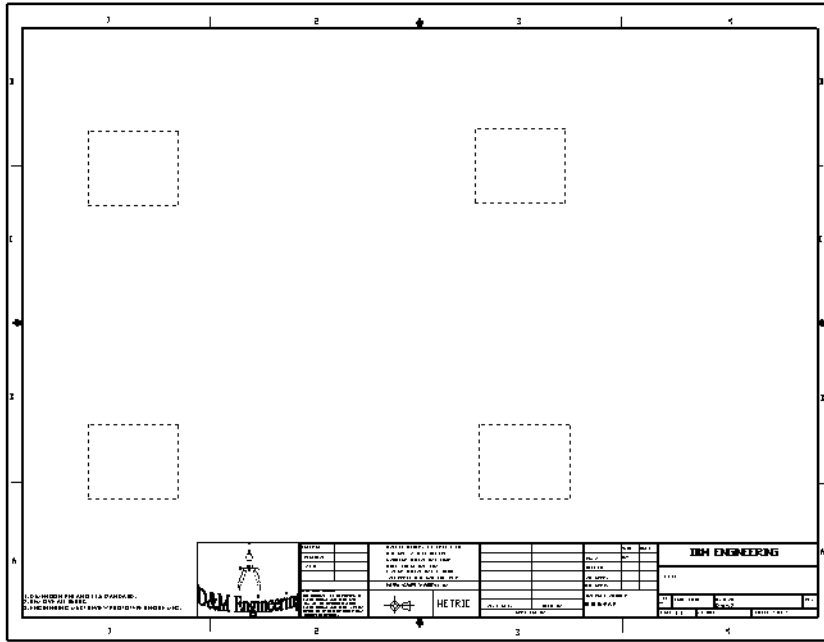
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Chapter 2

Drawing Templates and Sheet Formats



Below are the desired outcomes and usage competencies based on the completion of Chapter 2.

Desired Outcomes:	Usage Competencies:
<ul style="list-style-type: none"> • Two Drawing Templates: <ul style="list-style-type: none"> ○ C (ANSI) size Drawing template ○ A (ANSI) size Drawing template 	<ul style="list-style-type: none"> • Ability to apply Document Properties to reflect the ASME Y14 Engineering Drawing and Related Drawing Practices. • Understand System Options and Document Properties, which influence the drawing and Drawing template.
<ul style="list-style-type: none"> • One C size Sheet format. 	<ul style="list-style-type: none"> • Import an AutoCAD file as a Sheet format. Insert SolidWorks System Properties and Custom Properties.
<ul style="list-style-type: none"> • New file location for Drawing templates and Sheet format. 	<ul style="list-style-type: none"> • Ability to create new SolidWorks Document tabs.

Notes:



Chapter 2 - Drawing Templates and Sheet Formats

Chapter Objective


Create two Drawing templates. Create a C (ANSI) size Drawing template and an A (ANSI) size Drawing template. Create a C (ANSI) size Sheet format.

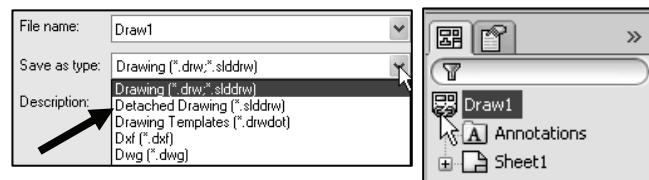
On the completion of this chapter, you will be able to:

- Establish a SolidWorks drawing document session.
- Distinguish between System Options and Document Properties as they relate to drawings and templates.
- Create a new SolidWorks File Location for a Drawing template.
- Set Reference Document Properties in a Drawing template.
- Create an empty C-size Drawing template. Propagate the settings to the drawing sizes.
- Import an AutoCAD drawing as a SolidWorks C-size Sheet format.
- Combine an empty Drawing template and Sheet format to create a C-ANSI-MM Drawing template.
- Develop Linked Notes to SolidWorks Properties and Custom Properties in the Sheet format.
- Insert a company logo with a relation in the Title block.
- Create an A-ANSI-MM Drawing template by combining information from the C-size Drawing template and A-size Sheet format.

Templates are part, drawing, and assembly documents that include user-defined parameters and are the basis for new documents. You can maintain many different document templates. For example, you can create:

- A Document template using millimeters and another document template using inches.
- A Document template using ANSI and another document template using ISO dimensioning standard.
- A Document template for a Detached drawing.

 A *Detached drawing* is design so you can open and work in drawing files without the model files being loaded into memory or even being present.



When opening a Detached drawing, SolidWorks checks all sheets in the drawing to be sure that they are synchronized with the model. If not, you are warned. To save a standard drawing to a Detached Drawing, click **File, Save As**, select the Save as type: **Detached Drawing (*.slddrw)**.

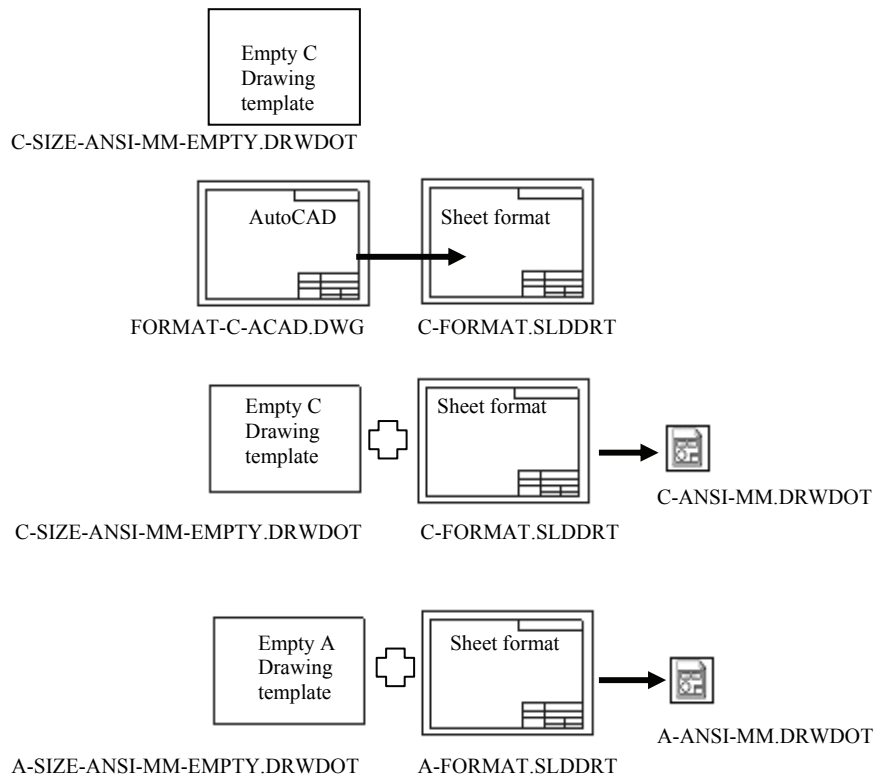
Chapter Overview

Your responsibilities as the designer include developing drawings that adhere to the ASME Y14 American National Standard for Engineering Drawing and Related Documentation Practices. The foundation for a SolidWorks drawing is the Drawing template. Drawing size, drawing standards, units and other properties are defined in the Drawing template.

Sheet formats contain the following: *Border, Title block, Revision block, Company name, logo, SolidWorks Properties and Custom Properties*. You are under time constraints to complete the project. Conserve drawing time. Create a custom Drawing template and Sheet format.

Perform the following tasks in this Chapter:

- Modify Document Properties and create an empty C (ANSI) size Drawing template.
- Import an AutoCAD drawing and save the drawing as a C-size Sheet format.
- Add System Properties and Custom Properties to the Sheet format.
- Create an A-ANSI-MM Drawing template.



- Combine the empty drawing template and imported the Sheet format to create the C-ANSI-MM Drawing template.
- Generate an empty A (ANSI) size Drawing template.
- Modify an existing SolidWorks A-size Sheet format.

Engineering Drawing and Related Documentation Practices

Drawing templates in this section are based on the American Society of Mechanical Engineers ASME Y14 American National Standard for Engineering Drawing and Related Documentation Practices.

These standards represent the drawing practices used by U.S. industry. The ASME Y14 practices supersede the American National Standards Institute ANSI standards.

The ASME Y14 Engineering Drawing and Related Documentation Practices are published by The American Society of Mechanical Engineers, New York, NY. References to the current ASME Y14 standards are used with permission.

ASME Y14 Standard Name:	American National Standard Engineering Drawing and Related Documentation:	Revision of the Standard:
ASME Y14.100M-1998	Engineering Drawing Practices	DOD-STD-100
ASME Y14.1-1995	Decimal Inch Drawing Sheet Size and Format	ANSI Y14.1
ASME Y14.1M-1995	Metric Drawing Sheet Size and Format	ANSI Y14.1M
ASME Y14.24M	Types and Applications of Engineering Drawings	ANSI Y14.24M
ASME Y14.2M(Reaffirmed 1998)	Line Conventions and Lettering	ANSI Y14.2M
ASME Y14.3M-1994	Multi-view and Sectional View Drawings	ANSI Y14.3
ASME Y14.41-2003	Digital Product Definition Data Practices	N/A
ASME Y14.5M –1994 (Reaffirmed 1999)	Dimensioning and Tolerancing	ANSI Y14.5-1982 (R1988)

This book presents a portion of the ASME Y14 American National Standard for Engineering Drawing and Related Documentation Practices. Information presented in Chapters 2 - 6 represents sample illustrations of drawings, various drawing views, and or dimension types.

The ASME Y14 Standards committee develops and maintains additional Drawing Standards. Members of these committees are from Industry, Department of Defense, and Academia.

Companies create their own drawing standards based on one or more of the following:

- ASME Y14
- ISO or other International drawing standards
- Older ANSI standards

- Military standards



There is also the “We’ve always done it this way” drawing standard or “Go ask the Drafting supervisor” drawing standard.

File Management

File management organizes parts, assemblies and drawings. Why do you need file management? A large assembly or drawing can contain hundreds or even thousands of parts.

Parts and assemblies are distributed between team members to save time. Design changes occur frequently in the development process. How do you manage and control changes? Answer: Through file management. File management is a very important tool in the development process.

Utilize file folders to organize projects, vendor parts and assemblies, templates and various libraries.

Folders exist on the local hard drive, example C:\. Folders also exist on a network drive, example Z:\. The letters C:\ and Z:\ are used as examples for a local drive and a network drive respectfully. The files and folders required to complete the projects in this book are located on the enclosed CD.

Activity: File Management

Create a new folder in Windows to download files.

- 1) Click **Start** from the Windows Taskbar.
- 2) Click **My Documents** in Windows.
- 3) Click **File, New, Folder** from the Main menu.

Enter the new folder name.

- 4) Enter **DRAWING-W-SOLIDWORKS-2010**.



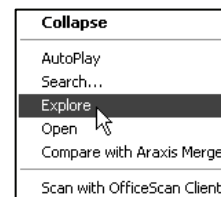
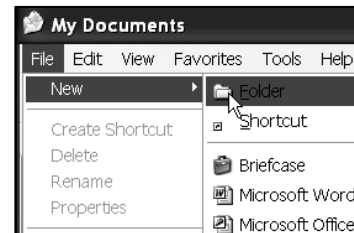
Select the Microsoft Windows commands either from the Main menu, toolbar icons or with the right mouse button. Windows XP is used in this section.

Return to the DRAWING-W-SOLIDWORKS-2010 folder.

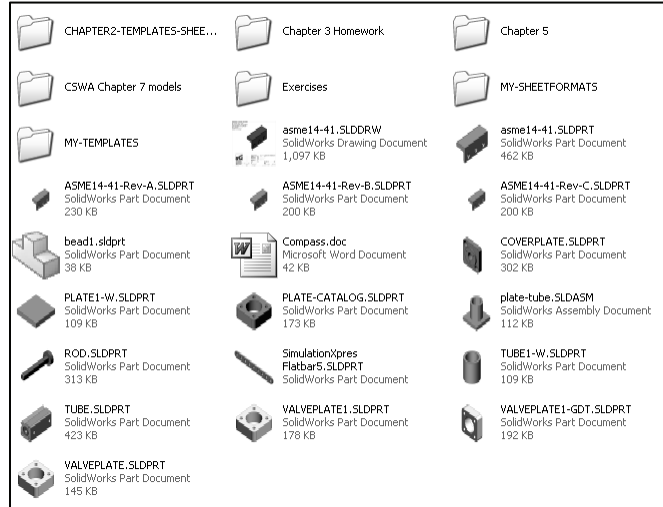
- 5) Click the **DRAWING-W-SOLIDWORKS-2010** folder.


Copy the files from the enclosed CD in the book to the new folder.

- 6) **Insert** the enclosed CD. Select your **CD drive**.
- 7) Right-click **Explore**. View the available files and folders.
- 8) **Copy** the files and folders to the DRAWING-W-SOLIDWORKS-2010 folder. The DRAWING-W-SOLIDWORKS-2010 folder is the working folder for this book.




Store chapter Drawing templates in the MY-TEMPLATES file folder. Store Chapter Sheet formats in the MY-SHEETFORMATS folder.



 Drawing templates and Sheet formats that are created in this chapter, are located in the CHAPTER 2-TEMPLATES-SHEETFORMATS folder on the CD. Check for proper path location on your system for created Sheet formats and Drawing templates.

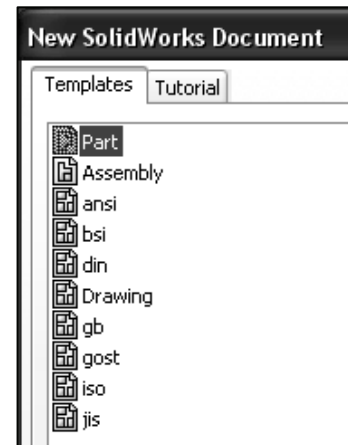
Default Drawing Template, Sheet Format and Sheet Size

The foundation of a SolidWorks drawing is the Drawing template. Drawing sheet size, drawing standards, company information, manufacturing and or assembly requirements: units, layers, line styles and other properties are defined in the Drawing template.

 SolidWorks 2010 provides additional Drawing Templates for various Overall drafting standards.

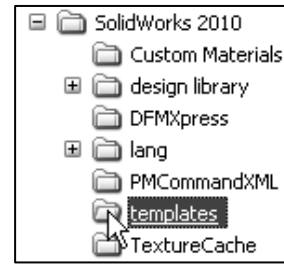
A Sheet format is incorporated into the Drawing template. The Sheet format contains the following items:

- Sheet border
- Title block
- Revision block information
- Company name and or logo information
- Custom Properties
- SolidWorks Properties
- And more



SolidWorks starts with a default Drawing template, (*.drwdot).

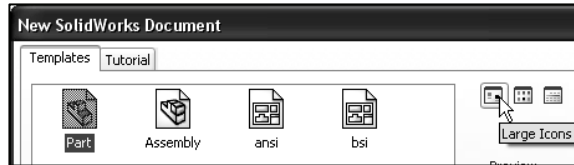
The default Drawing template is located in the C:\Documents and Settings\All Users\Application Data\SolidWorks\SolidWorks 2010\templates folder on a non-network system. SolidWorks is the name of the installation folder.



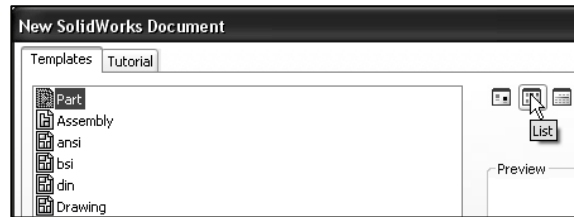
New SolidWorks Document

The Templates folder corresponds to the Templates tab displayed in the New SolidWorks Document dialog box.

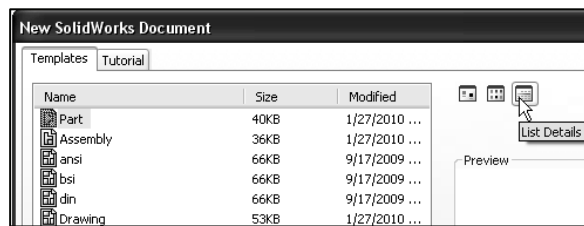
The Large Icons option displays the full name, and a large document icon.



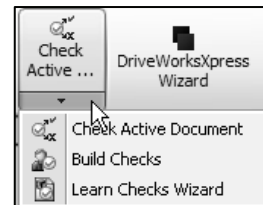
The List option displays the document icons in a list format.



The List Details option displays the document name, size and last modified date.

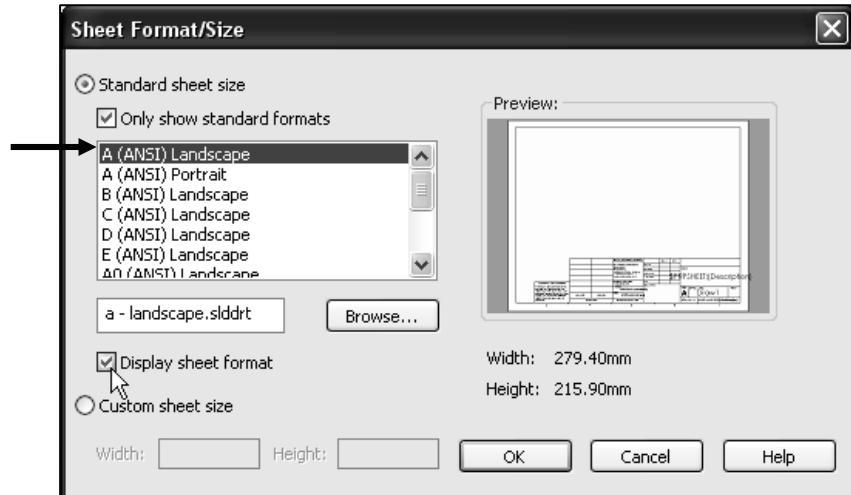



SolidWorks Design Checker is a set of tools used to check parts, assemblies and especially drawing files against a checklist of company specified standards. The **Build Checks** tool is used to create a checklist and the **Check Active Document** tool to apply it to a SolidWorks document. The checker generates a list of failed checks and detailed information about each. The results can be sent to a report.

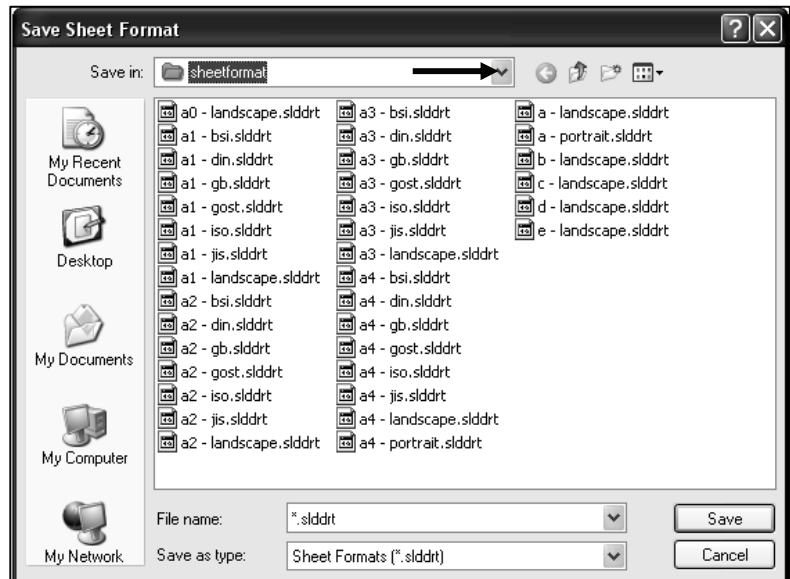


Sheet Format/Size

The Sheet Format/Size dialog box defines the Sheet format and the paper size. The U.S. default Standard Sheet Format is A (ANSI) Landscape. The Display sheet format box toggles the Sheet format display on/off.



 The Standard Sheet formats are located in the C:\Documents and Settings\All Users\Application Data\SolidWorks\SolidWorks 2010\lang\english\sheetformat in a non-network system.

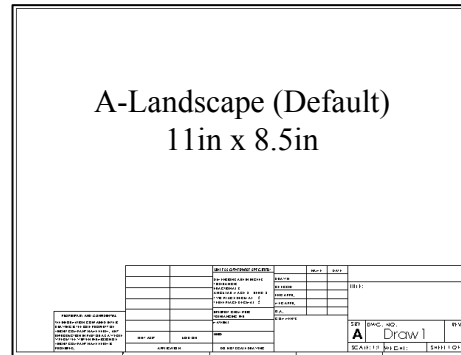


ASME Y14.1 Drawing Sheet Size and Format

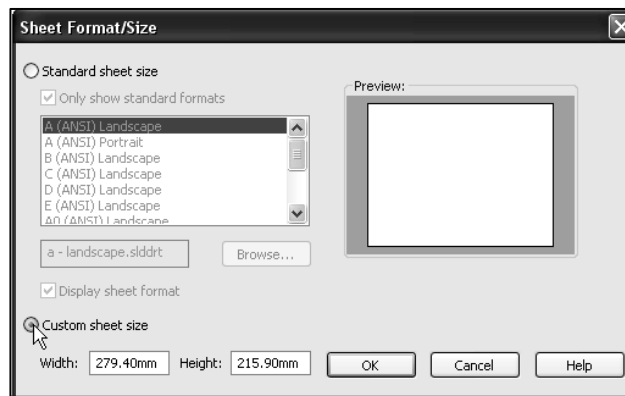
There are two ASME standards that define sheet size and format. They are:

1. ASME Y14.1-1995 Decimal Inch Drawing Sheet Size and Format
2. ASME Y14.1M-1995 Metric Drawing Sheet size

Drawing size refers to the physical paper size used to create the drawing. The most common paper size in the U.S. is the A-size: (8.5in. x 11in.).



The most common paper size internationally is the A4 size: (210mm x 297mm). The ASME Y14.1-1995 and ASME Y14.1M-1995 standards contain both a horizontal and vertical format for A and A4 size respectively. The corresponding SolidWorks Sheet format is Landscape for horizontal and Portrait for vertical.




SolidWorks predefines U.S. drawing sizes A through E. Drawing sizes: F, G, H, J, & K utilize the Custom sheet size option. Enter values for width and height. SolidWorks predefines metric drawing sizes A4 through A0. Metric roll paper sizes utilize the Custom sheet size option.

The ASME Y14.1-1995 Decimal Inch Drawing and ASME Y14.1M-1995 Metric Sheet size standard are as follows:

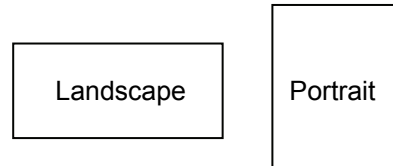
Drawing Size: "Physical Paper"	Size in inches:	
	Vertical	Horizontal
A horizontal (landscape)	8.5	11.0
A vertical (portrait)	11.0	8.5
B	11.0	17.0
C	17.0	22.0
D	22.0	34.0
E	34.0	44.0
F	28.0	40.0
G, H, J and K apply to roll sizes, User Defined		

Drawing Size: "Physical Paper" Metric	Size in Millimeters:	
	Vertical	Horizontal
A0	841	1189
A1	594	841
A2	420	594
A3	297	420
A4 horizontal (landscape)	210	297
A4 vertical (portrait)	297	210

 Use caution when sending electronic drawings between U.S. and International colleagues. Drawing paper sizes will vary. Example: An A-size (11 in. x 8.5 in.) drawing (280 mm x 216 mm) does not fit a A4 metric drawing (297 mm x 210 mm). Use a larger paper size or scale the drawing using the printer setup options.

Start a new session of SolidWorks. Create a new drawing with the default Drawing template. Utilize C ANSI Size paper with no Sheet format displayed.

The sheet border defines the C-size drawing: 22 in. x 17 in., (558.80 mm x 431.80 mm). A new Graphics window displays the C ANSI Landscape Drawing, named Draw1.

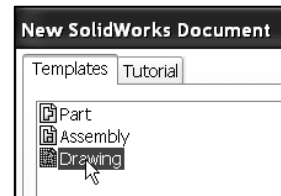



Landscape indicates that the larger dimension is along the horizontal. A-Portrait and A4-Portrait indicate that the larger dimension is along the vertical.

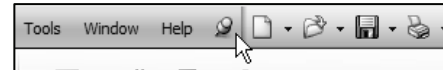
Activity: Default Drawing Template

Start a SolidWorks session.


- 9) Click **Start, All Programs** from the Windows Main menu.
- 10) Click the **SolidWorks 2010** folder.
- 11) Click the **SolidWorks 2010** application. The SolidWorks Graphics window is displayed.



 Pin the Menu bar toolbar and the Menu bar menu.

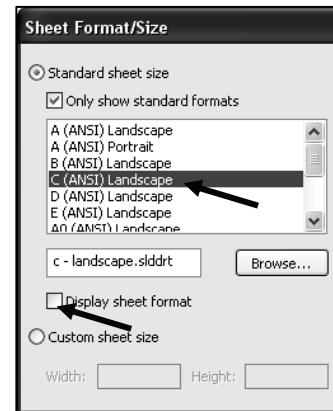


Select the default Drawing template.


- 12) Click **New**  from the Menu bar toolbar.
- 13) Double-click **Drawing** from the Templates tab.


Create a C-Landscape sheet.

- 14) Select **C (ANSI) Landscape** from the Standard sheet size drop-down menu.
- 15) **Uncheck** the Display sheet format box.
- 16) Click **OK** from the Sheet Format/Size dialog box.



Exit the Model View PropertyManager.

- 17) Click **Cancel**  from the Model View PropertyManager. The FeatureManager is displayed with Draw1 as the default drawing name.

 A goal of this book is to expose various SolidWorks design tools and features. The most direct way, may not be always shown.



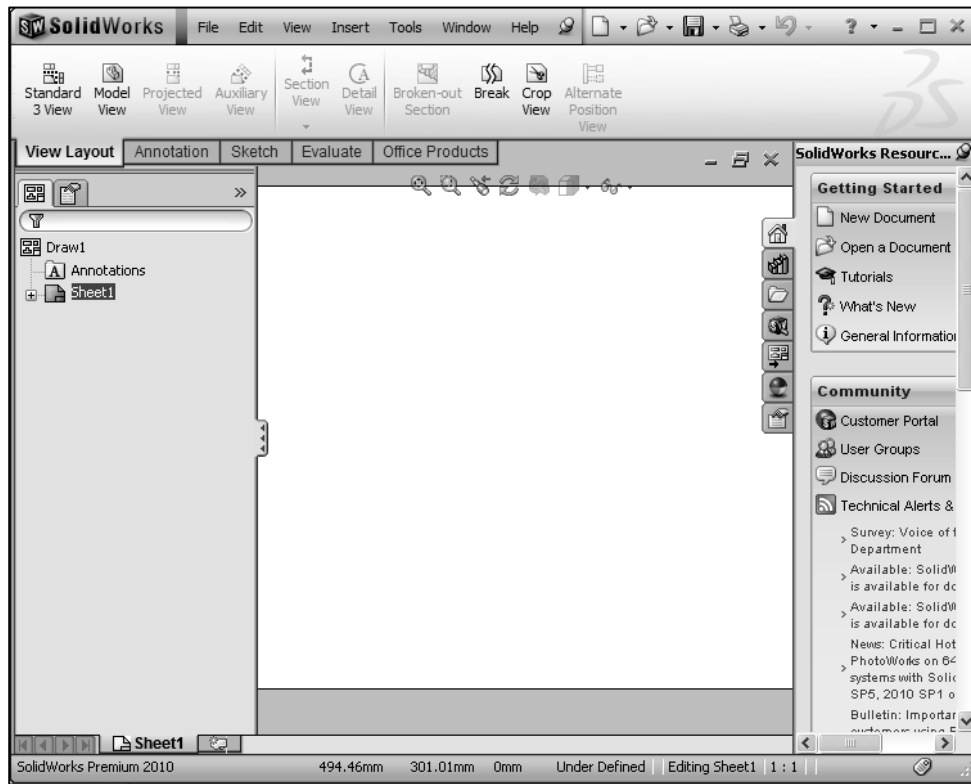
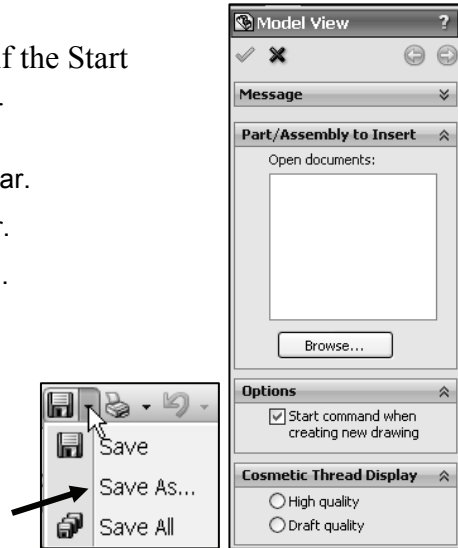
The Model View PropertyManager is displayed if the Start command when creating new drawing box is checked.

Save Draw1.

18) Click **Save As** from the Consolidated Menu bar toolbar.

19) Select the **DRAWING-W-SOLIDWORKS-2010** folder.

20) Click **Save**. The Draw1 FeatureManager is displayed.



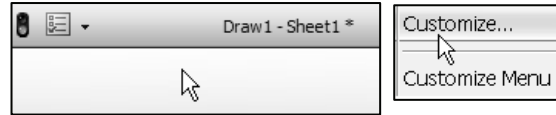
If you do not see the above illustration, right-click inside the Sheet boundary, click Edit Sheet Format. Click Edit Sheet. SolidWorks is presently working on this reported bug.

Activity: Display the Line Format Toolbar

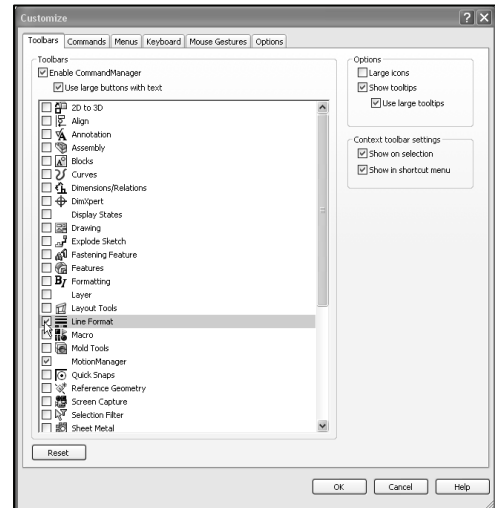
Review the CommandManager options and display the Line Format toolbar.

21) Right-click in the **grey area** of the CommandManager as illustrated.

22) Click **Customize**. The Customize dialog box is displayed. The Toolbars tab is displayed by default.




23) Check the **Line Format** box. The Line Format toolbar is displayed. Explore the tabs and your options to customize your options.



24) Click and drag the **Line Format toolbar** off the Graphics window.

Close the Customize dialog box.

25) Click **OK**.







 By default, the Show tooltips option is selected. Apply the Customize dialog box to set short cut keys from the keyboard.



Line Format Toolbar




The Line Format toolbar controls the following options: *Layer*

Properties , *Line Color* , *Line Thickness* , *Line Style* , *Hide Show edges*  and *Color Display Mode* .

Utilize the Line Format toolbar when creating a Drawing template. Select the tools and menu options that are displayed in bold icons and black text.

The tools and menu options that are displayed in gray are called grayed-out. The gray icon or text cannot be selected. Additional information is required for these options.

 You can also display the Line Format toolbar by clicking **View, Toolbars, Line Format** from the Menu bar menu.

Activity: Create a Keyboard Shortcut

Customize the Keyboard.

- 26) Click **Tools, Customize** from the Menu bar menu. The Customize dialog box is displayed.

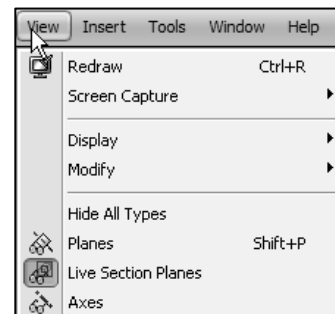
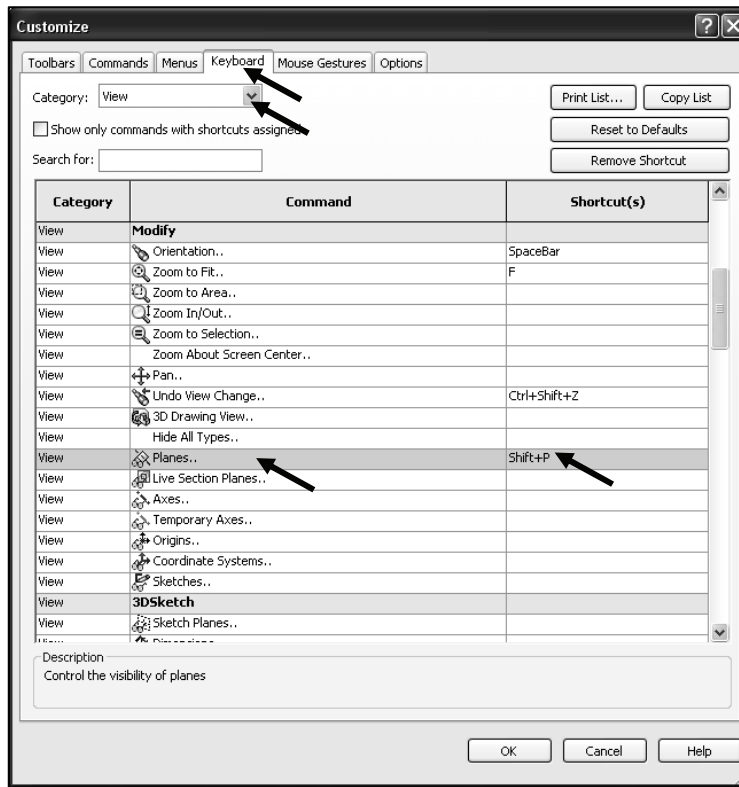


Note: There are numerous ways to access commands and menus in SolidWorks.

- 27) Click the **Keyboard** tab.
- 28) Select **View** for Categories.
- 29) Select **Planes** for Commands.
- 30) Click a **position** inside the Shortcut(s) box.
- 31) Enter **P** for new shortcut key. Note: Shift+P is displayed in the Shortcut(s) box.
- 32) Click **OK** from the Customize dialog box.
- 33) **Save** the drawing. Draw1 is the default name.

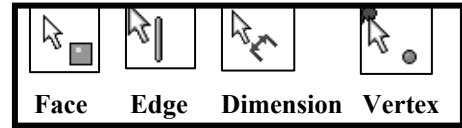




The short cut key P is displayed next to the Planes option in the View menu. Create additional short cut keys as an exercise.




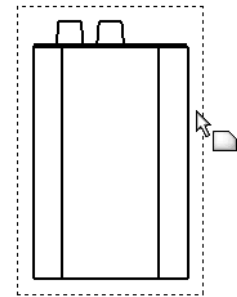
Cursor Feedback


SolidWorks provides system feedback by attaching a symbol to the mouse pointer cursor. The system feedback symbol indicates what you are selecting or what the system is expecting you to select. As you move the mouse pointer across your model, system feedback is provided to you in the form of symbols, riding next to the cursor. The cursor has an important role in the SolidWorks User Interface.





The mouse pointer provides feedback in both the Drawing Sheet  and Drawing View  modes.


The mouse pointer displays the Drawing Sheet  icon when the Sheet properties and commands are executed.




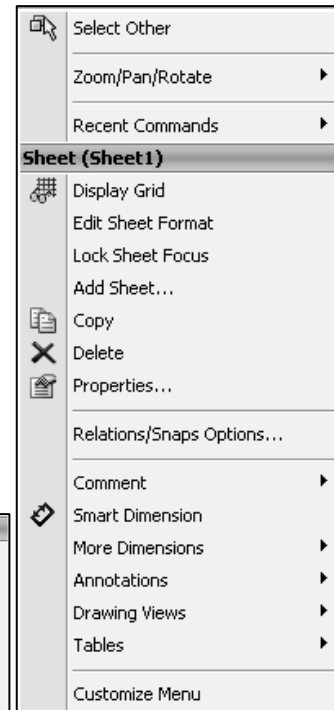
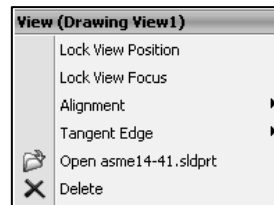
The mouse pointer displays the Drawing View  icon when the View properties and commands are executed.


 View the mouse pointer for feedback to select Sheet, View, Component and Edge properties in the Drawing.

Sheet Properties display properties of the selected sheet. Right-click in the sheet boundary .

View Properties display properties of the selected view. Right-click on the view boundary .

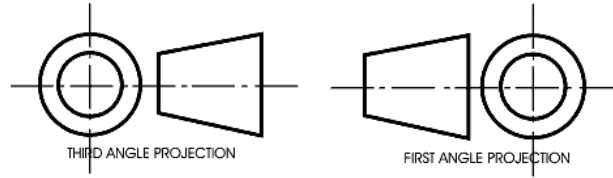
 Use the Lock View Position command to prevent a view from being moved by dragging.



 Use the Lock View Focus command when you need a view to remain active as you work within it. This allows you to add sketch entities to a view, even when the mouse pointer is close to another view. You can be sure that the items you are adding belong to the view you want.

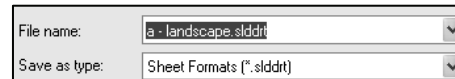
Sheet Properties

Sheet Properties display properties of the selected sheet. Sheet Properties define the following: *Name of the Sheet, Sheet Scale, Type of Projection (First angle or Third angle), Sheet Format, Sheet Size, View label and Datum label.*



The Sheet format and Sheet size are set in the default Drawing template. Review the Sheet Properties. The Standard sheet size option is grayed out.

The Sheet format file extension is *.slddrt. The Sheet format option is grayed out. The C-size paper, width, and height dimensions are listed under the Custom sheet size option.



By default, sheet formats are save in the <SolidWorks install directory>\data folder. However formats can be save in any location. The data folder is overwritten when a new version of SolidWorks is loaded. The files will be lost if they are stored here.

Activity: Sheet Properties

Display the Sheet Properties.

34) Right-click inside the **Sheet boundary**.

35) Click **Properties**. The Sheet Properties dialog box is displayed. Default Name of the sheet is Sheet1. Default Scale is 1:1.

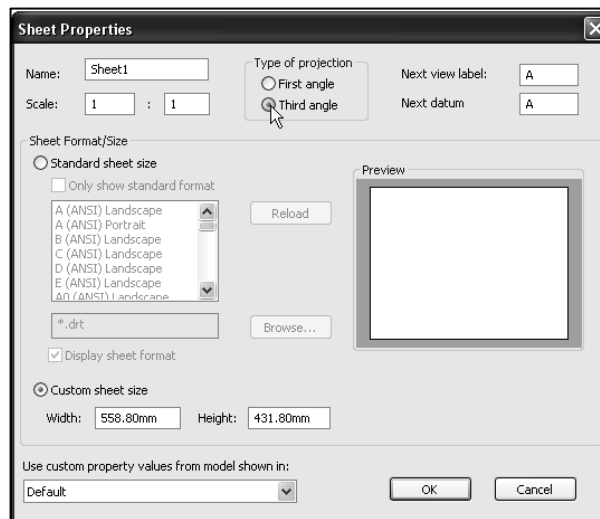
36) Set Type of projection. Click the **Third angle** box.



First or Third Angle projection was set at the initial installation of the SolidWorks software.

Exit the Sheet Properties dialog box.

37) Click **OK** from the Sheet Properties dialog box.



The Sheet name is Sheet1. The FeatureManager and Sheet tab display the Sheet name. The Sheet Scale is 1:1. The Preview box contains no Sheet format. Custom sheet size is 22in x 17in (558.80mm x 431.80mm).

Glass Box and Six Principal Views

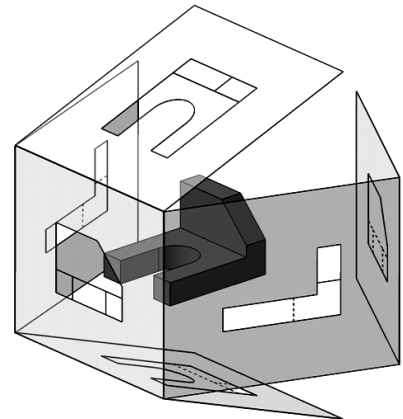
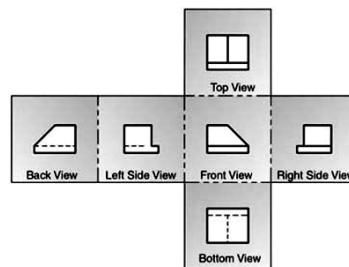
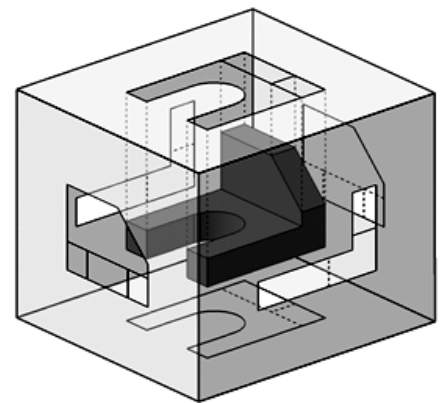
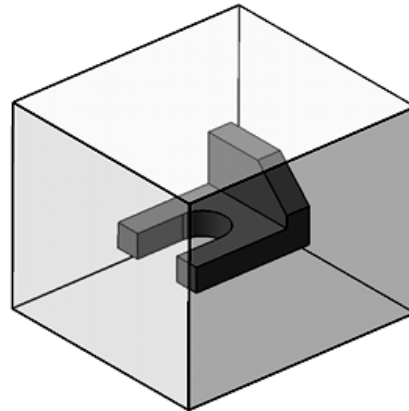
The Glass box method is a traditional method of placing an object in an *imaginary glass box* to view the six principle views.

Imagine that the object you are going to draw is placed inside a glass box, so that the large flat surfaces of the object are parallel to the walls of the box.

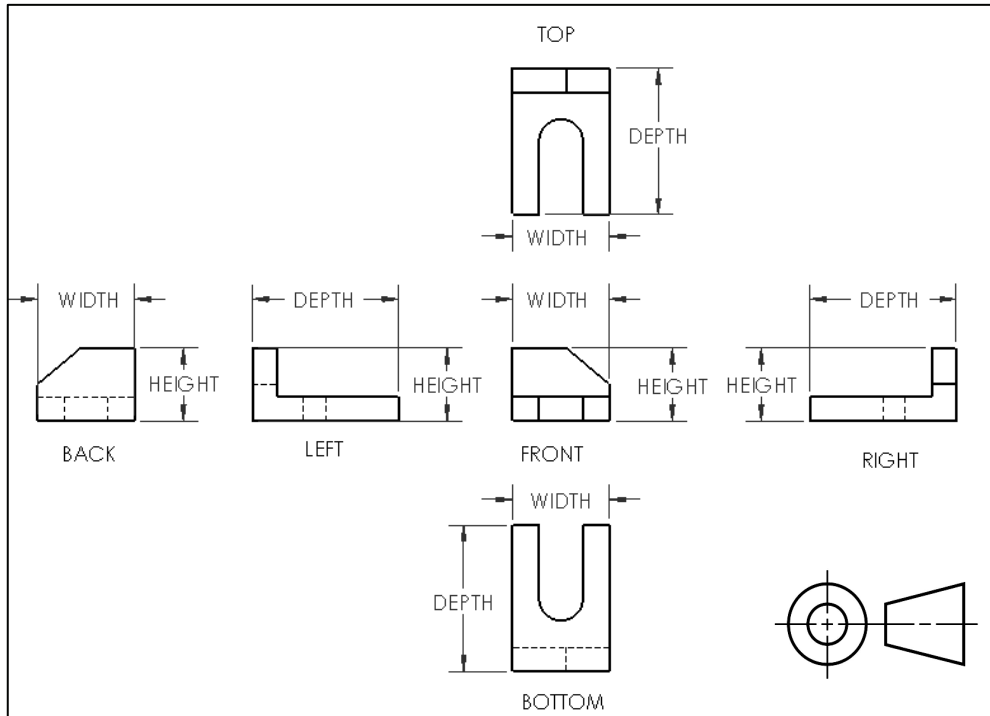
From each point on the object, imagine a ray, or projector perpendicular to the wall of the box forming the view of the object on that wall or projection plane.

Then *unfold the sides* of the imaginary glass box to create the orthographic projection of the object.

There are two different types of Angle Projection: First and Third Angle Projection. First Angle Projection is used in Europe and Asia. Third Angle Projection is used in the United States.



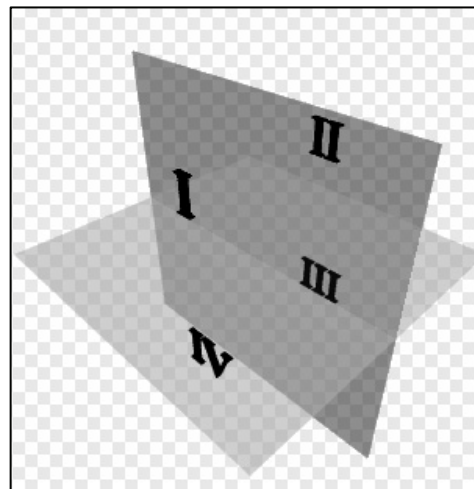
Third Angle Projection is used in the book. Imagine that the walls of the box are hinged and unfold the views outward around the Front view. This will provide you with the six principle views.



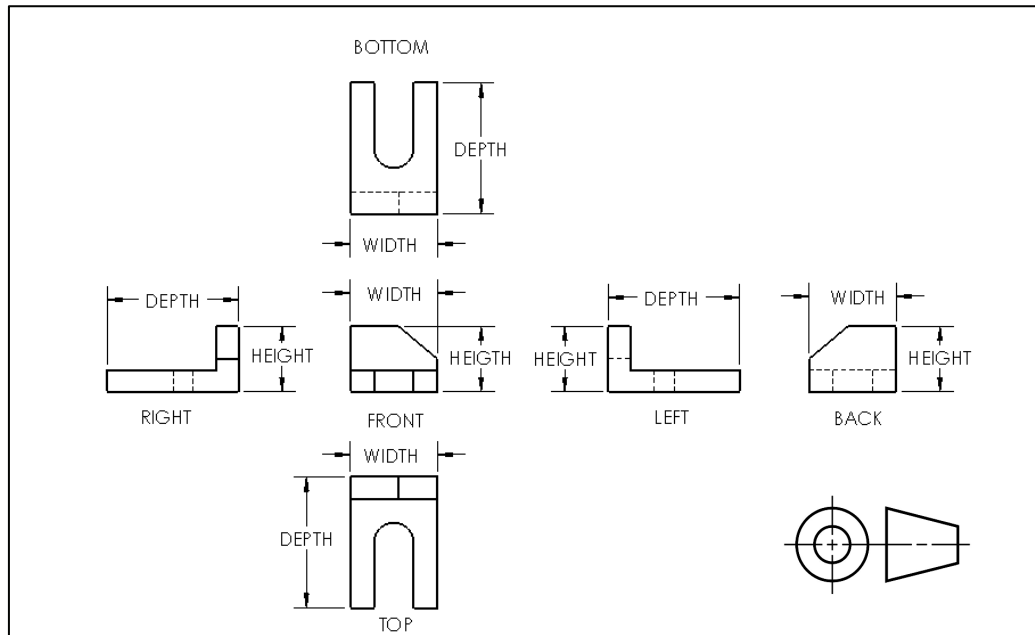
 SolidWorks uses BACK view vs. REAR view.

Modern orthographic projection is derived from Gaspard Monge's descriptive geometry. Monge defined a reference system of two viewing planes, horizontal H ("ground") and vertical V ("backdrop"). These two planes intersect to partition 3D space into four quadrants. In Third-Angle projection, the object is conceptually located in quadrant III.

Both First-angle and Third-angle projections result in the same six views; the difference between them is the arrangement of these views around the box.

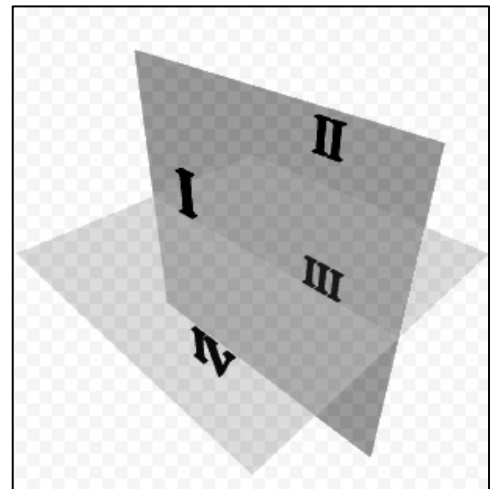


Below is an illustration of First Angle Projection.



In First Angle projection, the object is conceptually located in quadrant I, i.e. it floats above and before the viewing planes, the planes are opaque, and each view is pushed through the object onto the plane furthest from it.

💡 SolidWorks uses BACK view vs. REAR view.



Display Styles / Modes

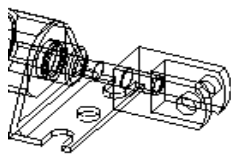
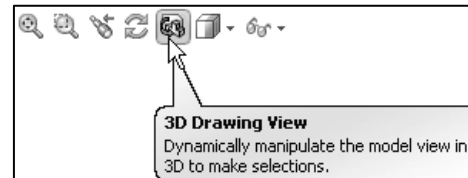
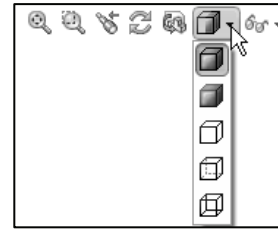
Display modes for a Drawing view are similar to a part except with the addition of the 3D Drawing view tool. This tool provides the ability to rotate the model in an existing view.

Wireframe and Shaded Display modes provide the best Graphic performance. Mechanical details require Hidden Lines Visible display and Hidden Lines Removed display. Select Shaded/Hidden Lines Removed to display Auxiliary Views to avoid confusion.

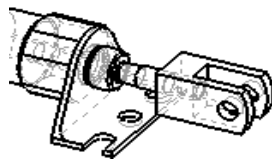
Tangent Edges Visible provides clarity for the start of a Fillet edge. Tangent Edges Removed provides the best graphic performance.

💡 ANSI standards prefers no Tangent Edges display, however individual company standards may display Tangents Edges for clarity.

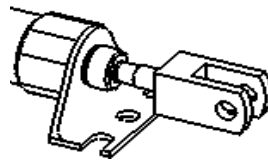
💡 Tangent Edges are displayed for educational purposes.



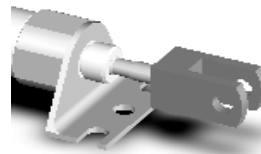
Wireframe



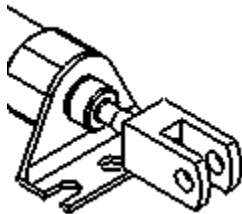
Hidden Lines Visible



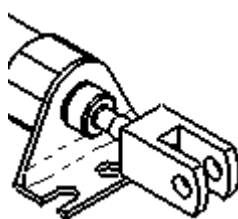
Hidden Lines Removed



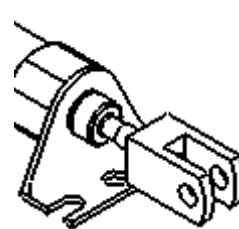
Shaded



Tangent Edges Visible




Tangent Edges With Font

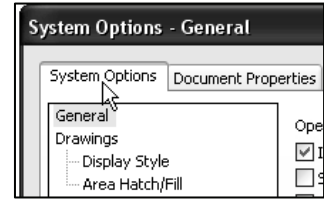
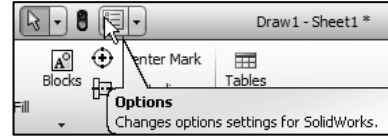


Tangent Edges Removed

System Options


System Options are stored in the registry of the computer. System Options are not part of the document. Changes to the System Options affect all current and future documents. There are hundreds of Systems Options. Review a few of the options in this section.

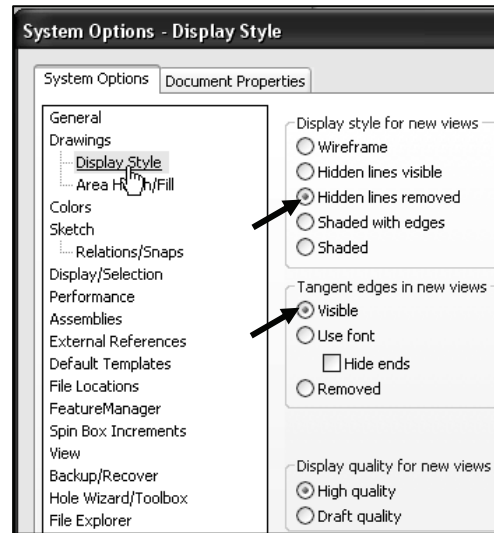
Click the Options  icon from the Menu bar toolbar to activate the System Options dialog box.




Activity: System Options - Display Style

Set the default display style.

- 38) Click **Options**  from the Menu bar toolbar. The System Options - General dialog box is displayed.
- 39) Click the **Display Style** folder from the System Options tab.
- 40) Check the **Hidden lines removed** box for the Display style for new views.
- 41) Check the **Visible** box for the default Tangent edges in the new views. Note: High quality is the default option for display quality.



 There are two new system properties for drawing documents in SolidWorks 2010: SW-View Name and SW-View Scale. These properties are similar to the system properties SW-Sheet Name and SW-Sheet Scale.

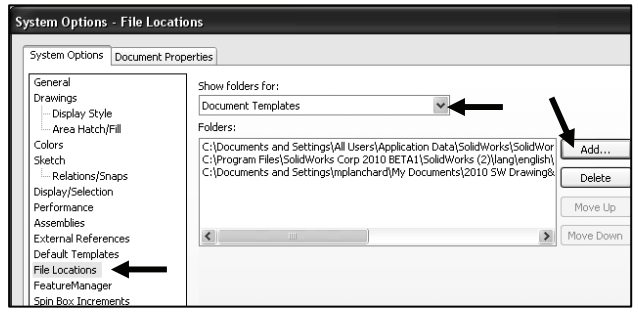
File Locations

System Options, File Locations, Document Templates option determines the path to locate a Custom Drawing template. Add the MY-TEMPLATES folder to the File Locations. The folder listed in the Document Templates option determines the tabs displayed in the New SolidWorks Document dialog box.

Activity: System Options-File Locations


Set file locations for Drawing Templates.

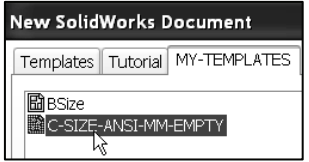
- 42) Click the **File Locations** folder from the System Options tab.
- 43) Select **Document Templates** from the Show folders for drop-down menu.
- 44) Click the **Add** button.
- 45) Click **Browse**.
- 46) Select the **DRAWING-W-SOLIDWORKS-2010MY-TEMPLATES** folder.
- 47) Click **OK** from the Browse For Folder box.
- 48) Click **Yes**.
- 49) Click **OK** from the System Options dialog box.
- 50) Click **Yes**.



Save Draw1.

- 51) **Save** Draw1.

 The MY-TEMPLATES tab is displayed in the New SolidWorks Drawing dialog box. The MY-TEMPLATES tab is *not displayed if the folder is empty*. The System Option, File Locations list determines the order of the tabs. Save the Drawing Templates to the MY-TEMPLATES folder.



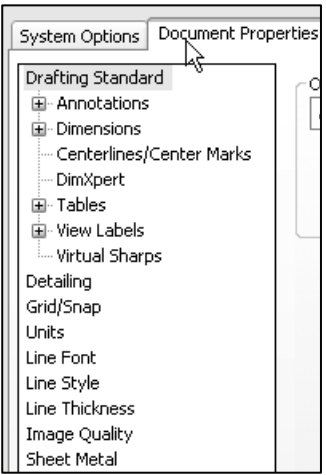
Document Properties

Document Properties apply to the current document. Set the following: *Drafting Standard, Grid/Snap, Units, Line Fonts, and Image Quality* in Document Properties.

When the current document is saved as a template, the current parameters are stored with the template. New documents that utilize the same template contain the stored parameters.

Conserve drawing time. Set the Document Properties in the Drawing template. Document Properties options contain hundreds of parameters. Examples are addressed in this section. Explore other parameters through SolidWorks Help Topics.


There are numerous text styles and sizes available in SolidWorks. Companies develop drawing format standards and use specific text height for Metric and English drawings.

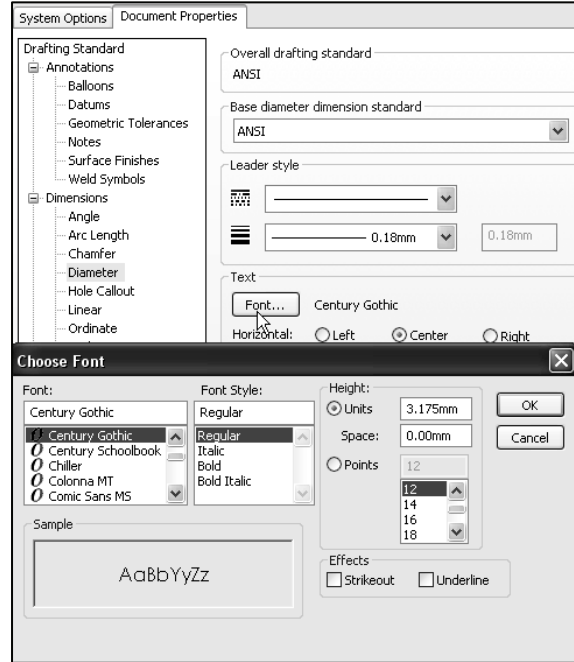


The ASME Y14.2M-1992(R1998) standard lists the following: *lettering, arrowhead, line conventions* and *lettering conventions* for engineering drawings and related documentation practices.

Font

Century Gothic is the default SolidWorks font.

 Create an assessment page to test that your Printer/Plotter drivers support the default SolidWorks font.



Minimum Drawing Letter Height based on ASME Y14.2.				
Annotation	Inch drawing sizes: A, B, C Metric drawing sizes: A2, A3, A4		Inch drawing sizes: D, E Metric drawing sizes: A0, A1	
	Inch	Millimeter	Inch	Millimeter
Drawing Title, Drawing Size, Cage Code, Drawing Number and Revision letter positioned inside the Title block.	.12in	3mm	.24in	6mm
Section views, Zone letter and numerals.	.24in	6mm	.24in	6mm
Drawing block headings in Title block.	.10in	2.5mm	.10in	2.5mm
All other characters inside the Sheet boundary. Corresponds to the SW Dimension and Note font.	.12in.	3mm	.12in	3mm.

Arrowheads

Control arrowheads through the Drafting Standard, Dimensions option in an active drawing document.

Utilize a solid filled arrowhead with a 3:1 ratio. The arrowhead width is proportionate to the line thickness. The Dimension line thickness is 0.3mm.

The Dimension arrow is based on the Dimension line. SolidWorks defines arrow size with three options:

- *Height*
- *Width*
- *Length*

Height corresponds to the arrow width. Width corresponds to the arrow tail length. Length corresponds to the distance from the tip of the arrow to the end of the tail.

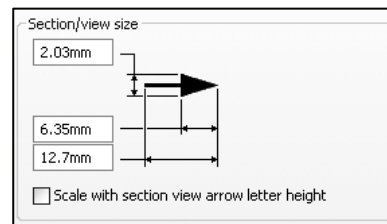
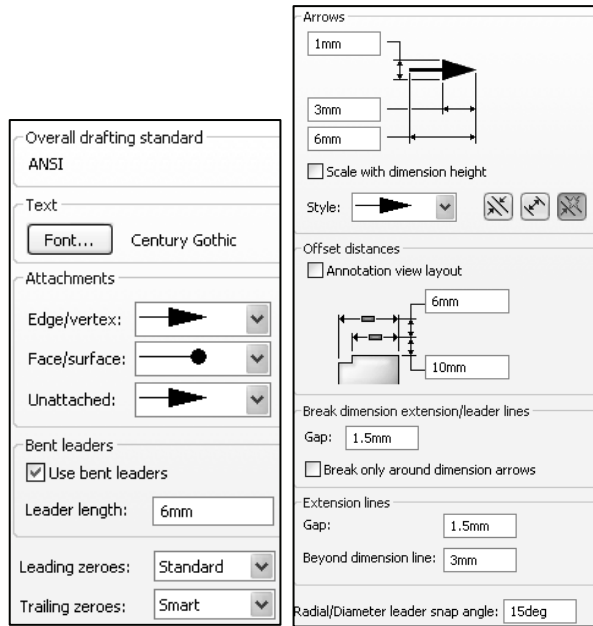
The Section line thickness, (Drafting Standard, View Labels, Section) is 0.6mm. The Section arrow is based on the Section line. The Section arrow length is 6mm. The Section arrow width is 2mm.



The illustration displays the default mm values.



The illustrations are based on SolidWorks SP1.0. The illustrations may vary slightly per your SolidWorks release.



Line Widths

The ASME Y14.2M-1992 (R1998) standard recommends two line widths with a 2:1 ratio. The minimum width of a thin line is 0.3mm. The minimum width of a thick, “normal” line is 0.6mm.



A single width line is acceptable on CAD drawings. Two line widths are used in this Chapter: Thin: 0.3mm and Normal: 0.6mm.

Apply Line Styles in the Line Font Document Properties. Line Font determines the appearance of a line in the Graphics window. SolidWorks styles utilized in this chapter are as follows:

SolidWorks Line Style	Thin: (0.3mm)	Normal: (0.6mm)
Solid		
Dashed		
Phantom		
Chain		
Center		
Stitch		
Thin/Thick Chain		

Various printers/plotters provide variable Line Weight settings. Example: Thin (0.3mm), Normal (0.6mm), and Thick (0.6mm).

Refer to the printer/plotter owner’s manual for Line weight setting. Utilize the Document Property, Line Style option to create, save, load, and delete line styles.

Line styles:

Name	Appearance	
Solid		
Dashed		
Phantom		
Chain		
Center		
Stitch		
Thin/Thick Chain		

Line length and spacing values:

Formatting key:
 A = Normal Line
 B = Bold segments on ends of lines
 Positive value indicates line segment
 Negative value indicates gap between segments

Example:

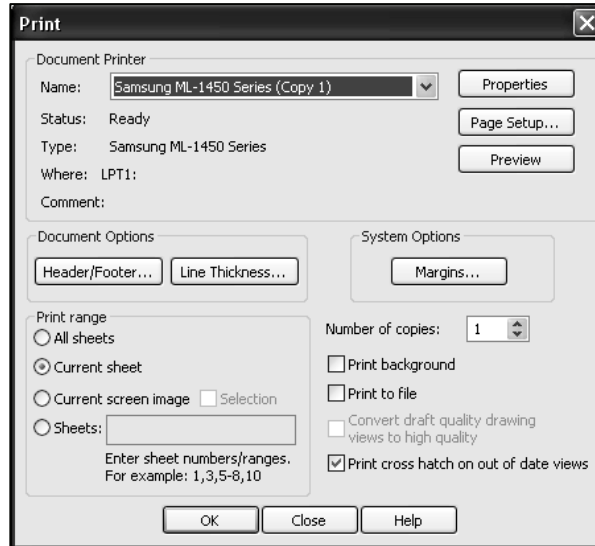
Definition:	Resulting line style:
A,1,-1	
A,1,-1,.5,-.5	
B,1,-1	

Scale large drawing sheets with the Resolution and Scale option located in the File, Page Setup Menu bar menu. Use the Scale to fit option to resize the drawing sheet to the physical paper size.

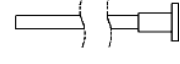
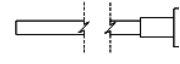
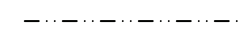
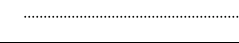
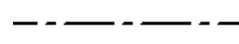
Use Scale to resize the drawing sheet by a percentage to the physical paper size.

Line Font


The ASME Y14.2M-1992(R1998) standard addresses the type and style of lines used in engineering drawings. Combine different Line Styles and use drawing layers to achieve the following types of ASME lines:

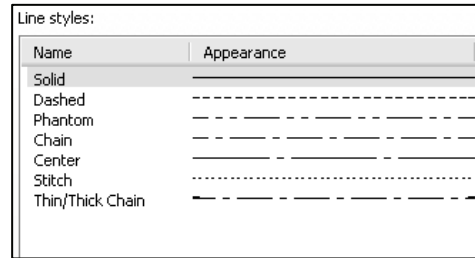


ASME Y14.2-1992(R1998) TYPE of LINE & example:	SolidWorks Line Font Type of Edge:	Style:	Thickness:
Visible line displays the visible edges or contours of a part.	Visible Edge 	Solid	Thick "Normal"
Hidden line displays the hidden edges or contours of a part.	Hidden Edge 	Dashed	Thin
Section lining displays the cut surface of a part assembly in a section view.	Crosshatch 	Solid	Thin Different Hatch patterns relate to different materials
Center line displays the axes of center planes of symmetrical parts/features.	Construction Curves 	Center	Thin
Symmetry line displays an axis of symmetry for a partial view.			Sketch Thin Center Line and Thick Visible lines on drawing layer.
Dimension lines/Extension lines/Leader lines combine to dimension drawings.	Dimensions 	Solid	Thin
Cutting plane line or Viewing plane line display the location of a cutting plane for sectional views and the viewing position for removed views.	Section Line View Arrows 	Phantom Solid	Thick Thick, "Normal"

Break line displays an incomplete view. Short Breaks Long Breaks		Curved	Broken view Use Curved for Short Breaks Use Small Zig Zag for Long Breaks
		Small Zig Zag	
Phantom line displays alternative position of moving parts.			Sketch Thin Phantom Line on drawing layer
Stitch line displays a sewing or stitching process.			Sketch Thin Stitch Line on drawing layer
Chain line displays a surface that requires more consideration or the location of a projected tolerance zone.			Sketch Thick Chain Line on drawing layer

The following default lines are defined in SolidWorks: *Solid, Dashed, Phantom, Chain, Center, Stitch, Thin/Thick Chain.*

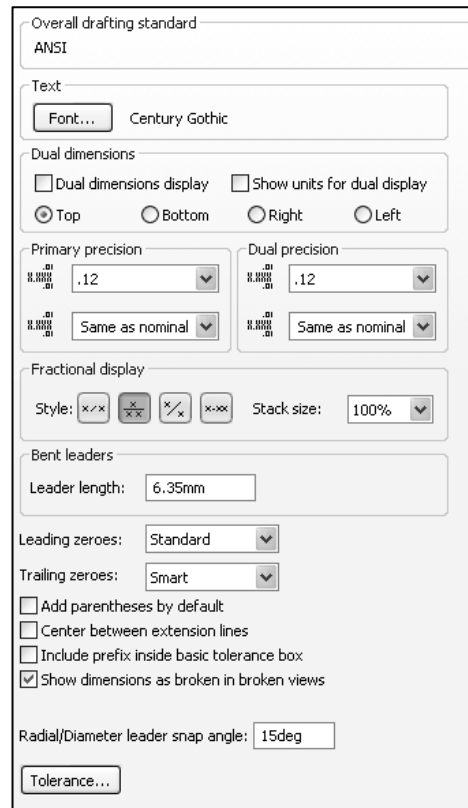
 Define these line types on a separate drawing layer.



Document Properties-Dimensions

Control the Dimensions options through Document Properties. The Drafting Standard, Dimensions determines the display on the drawing. Millimeter dimensioning and decimal inch dimensioning are the two key types of units specified on engineering drawings.

There are other dimension types specified for commercial commodities such as pipe sizes and lumber sizes. Develop separate Drawing templates for decimal inch units.



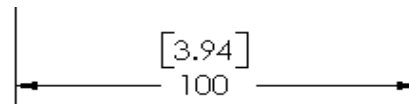


ASME Y14.2-1992(R1998) and the ASME Y14.2M Line Conventions and Lettering standard define text height, arrows and line styles for inch and metric values. Review the Detailing Document Properties options function before entering their values.

Drafting (Dimensioning) Standard

The Drafting standard options are:

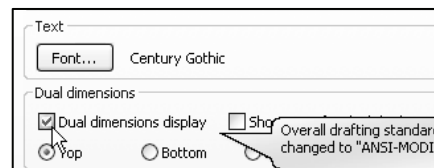
- **ANSI:** American National Standards Institute
- **ISO:** International Standards Organization
- **DIN:** Deutsche Institute für Normung (German)
- **JIS:** Japanese Industry Standard
- **BSI:** British Standards Institution
- **GOST:** Gosndarstuennye State Standard (Russian)
- **GB:** Guo Biao (Chinese)



Dual dimensions Display Option

The Dual dimensions display check box shows dimensions in two types of units on the drawing.

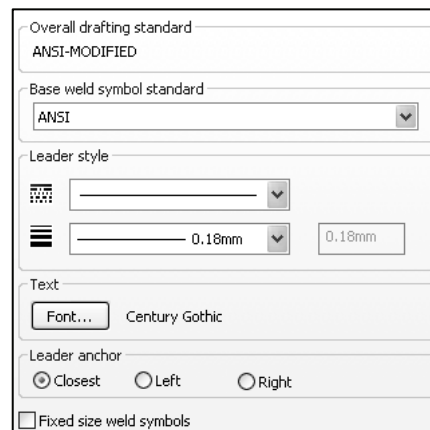
Select Dual dimensions display. Select the On top option. Select Dual Dimension Length units. The primary units display is 100mm. The secondary units display is [3.94]in.



Type	Unit	Decimals
Basic Units		
Length	millimeters	.12
Dual Dimension Length	inches	.12
Angle	centimeters	.12
Mass/Section Properties	meters	
	microinches	

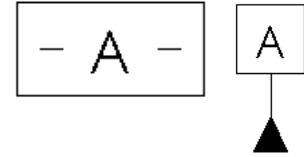
Fixed Size Weld Symbols Option

The Fixed size weld symbols checkbox displays the size of the weld symbol. Scale the symbols according to the dimension font size.



Display Datums per 1982 Option

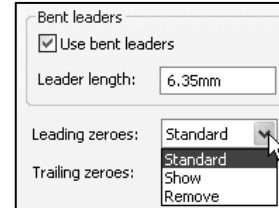
The Display datums per 1982 checkbox displays the ANSI Y14.5M-1982 datums. Use the ASME Y14.5M-1994(R1999) datums in this text.



Leading Zeroes and Trailing Zeroes Option

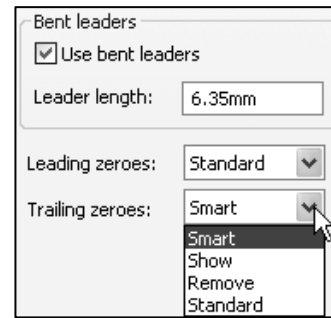
The Leading zeroes list box contains three options:

- **Standard:** Active by default. Zeros are displayed based on the dimensioning standard.
- **Show:** Zeros before decimal points are displayed.
- **Remove:** Zeros are not displayed.



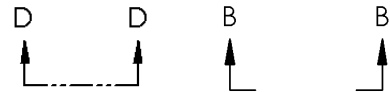
The Trailing zeroes list box contains four options:


- **Smart:** Active by default. Trailing zeros are trimmed for whole metric values. (Conforms to ANSI and ISO standards.)
- **Show:** Dimensions have trailing zeros up to the number of decimal places specified in Tools, Options, Document Properties, Units section.
- **Remove:** All trailing zeros are removed.
- **Standard:** Trims trailing zeroes to the ASME Y14.5M-1994 standard.



Alternative Section Display Option

The ASME Y14.2M-1992(R1998) standard supports two display styles. The default section line displays a continuous Phantom line type (D-D).

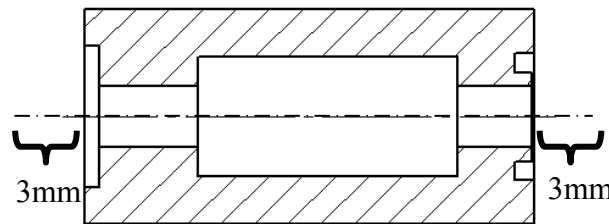


 Check the Alternate section display checkbox to allow for a gap in the section line (B-B).

Centerline Extension and Center Marks Option

The Centerline extension value controls the extension length beyond the section geometry.

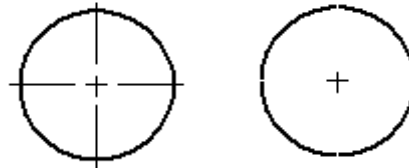
Centerlines are created as font lines and arcs in the drawing views.



Centerlines should be added to the drawing prior to the addition of dimensions and annotations. You can resize them or modify their appearance. Resize their appearance by dragging the control points on each side of the centerline.

Center marks specify the default center mark size used with arcs and circles. Center marks are displayed with or without Center mark lines.

The Center mark lines extend pass the circumference of the selected circle. Select the Center mark size based on the drawing size and scale.



The Center mark command creates a center mark, or a center point on selected circular edges. Selecting a circle creates a center mark. Selecting an arc creates a center point.

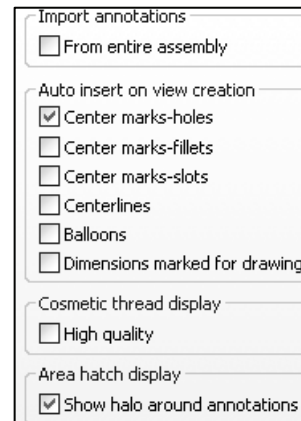
Center Marks should be added to the drawing prior to the addition of dimensions and annotations. You can resize them or modify their appearance.



Center Marks and Centerlines are annotations used to mark circle centers and describe the geometry size on the drawing.

Auto Insert on View Creation Option

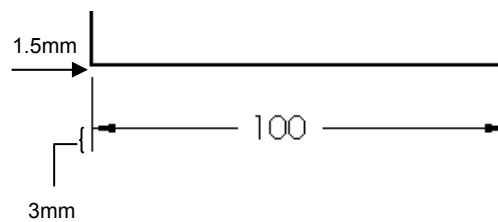
Auto insert on view creation locates Center marks on the appropriate entities when a new view is inserted into a drawing. By default Center marks-holes, Center marks-fillets, Center marks-slots, Centerlines, Balloons, and Dimensions marked for drawing options are not checked.



Save detailing time. Uncheck the Center marks option when parts contain multiple size holes and holes positioned at angles. Insert all dimensions and then insert the Center marks tool from the Annotation toolbar.

Extension Lines Option

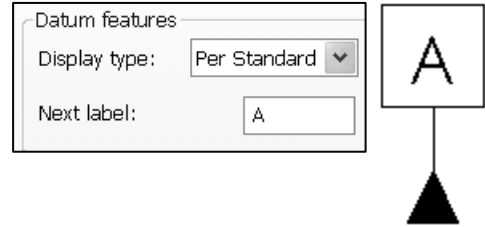
The ASME Y14.2M-1992(R1998) and ASME Y14.5M-1994(R1999) standard defines extension line length and gap. A visible gap exists between the extension line and the visible line. The extension line extends 3mm - 4mm past the dimension line.



The values 1.5mm and 3mm are a guide. Base the gap and extension line on the drawing size and scale.

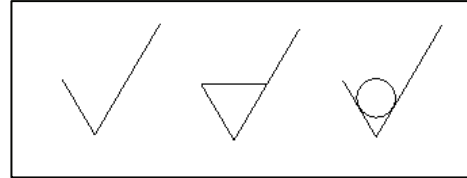
Datum Feature Option

The Next label specifies the subsequent upper case letter used for the Datum feature symbol. The default value is A. Successive labels are in alphabetical order. The Datum Display type Per Standard option displays a filled triangular symbol on the Datum feature.



Surface Finish Symbols

For the ISO standard, Surface finish symbols display per the 2002 standard.



Break Line Option

The Break line gap specifies the size of the gap between the Broken view break lines.



Automatic Update on BOM Option

The Automatic Update on BOM option updates the Bill of Material in a drawing if related model custom properties change.

ITEM NO.	QTY.	PART NO.	MATERIAL
1	1	10-0408	ALUMINUM
2	1	10-0409	STEEL

Set the values in SolidWorks to meet the ASME standard.

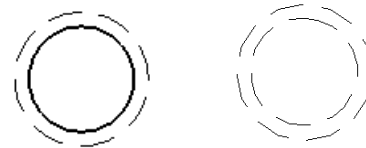


Set units before entering values for Detailing options. Units for the Default Templates are determined from initial SolidWorks installation options.

Cosmetic Thread Display Option

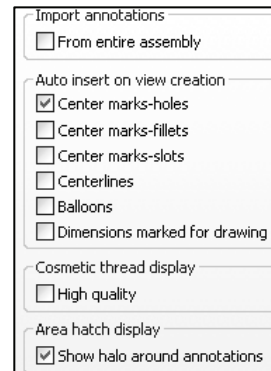
The High quality option displays Cosmetic threads visible or hidden in a selected drawing view. A blind hole Cosmetic thread is visible in the Front view and hidden in the Back view.

The Cosmetic thread feature is used to describe the attributes of a specific hole without having to add real threads to the model. It represents the minor diameter of a tread on a boss or the major diameter of a thread on a hole and can include a hole callout.




Visible

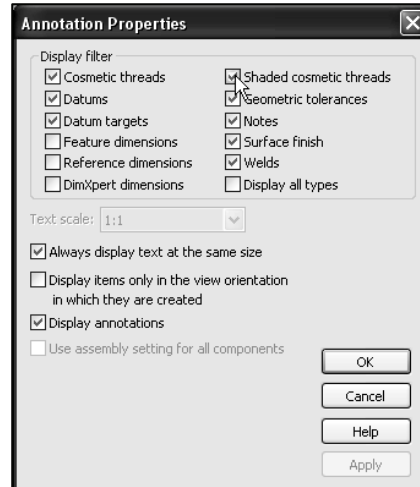
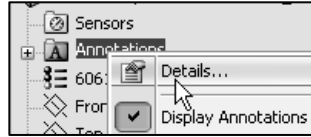
Hidden





In SolidWorks 2010, the Cosmetic thread can be applied at the part or drawing level. Either way, a cosmetic thread differs from other annotations in that it is an absorbed feature of the item to which it is attached. For example, the cosmetic thread on a hole is in the FeatureManager design tree as Thread1 under the Hole feature, along with the sketches used to create the hole as illustrated.

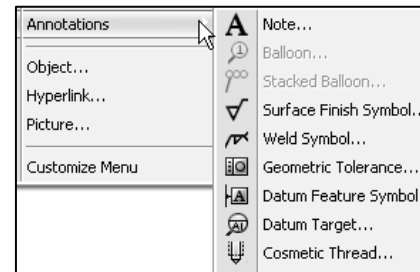


 To display a Cosmetic thread, right-click the Annotations folder from the FeatureManager and click Details. The Annotation Properties dialog box is displayed. Check the Cosmetic threads box and the Shaded cosmetic threads box. Click OK. View the cosmetic thread.




 To insert a Cosmetic thread, click Insert, Annotations, Cosmetic thread from the Menu bar menu.

 Review the Document Properties before inserting views, dimensions and annotations into your drawing. Modify the Document Properties that correspond to the part. For example, check/uncheck Dual dimension display if required for manufacturing. Uncheck the Auto insert on view creation, Center mark option when a part contains multiple size hole, rotated at different angles.



Activity: Document Properties-Detailing

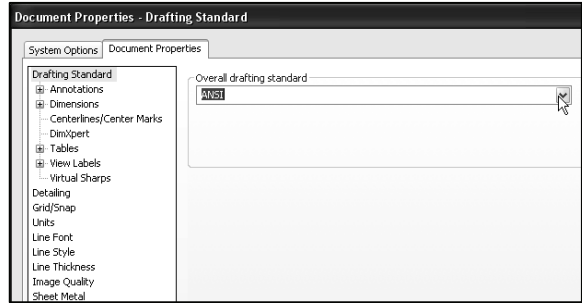
Set Detailing options.

52) Click **Options** , **Document Properties** tab from the Menu bar toolbar.

53) Select **ANSI** from the Overall drafting standard drop-down menu. Various options are available depending on the selected standard.

54) Click the **Dimensions** folder.

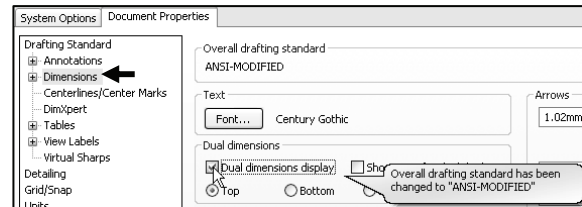
55) Check the **Dual dimension display** box.



Modify the Dimension Extension line value.

56) Enter **1.5mm** for Extension lines Gap.

57) Enter **3mm** for Extension lines Beyond dimension line.

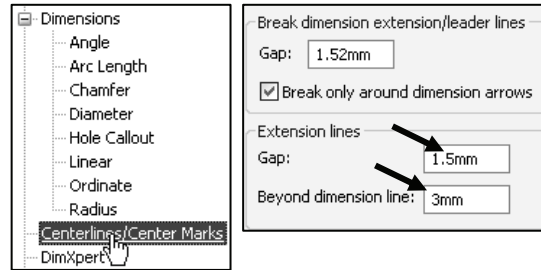


Modify the Centerline / Center Marks.

58) Click the **Centerlines/Center Marks** folder under Dimensions.

59) Enter **3mm** for the Centerline extension.

60) Enter **0.5mm** for the Center marks Size.

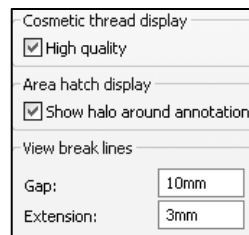
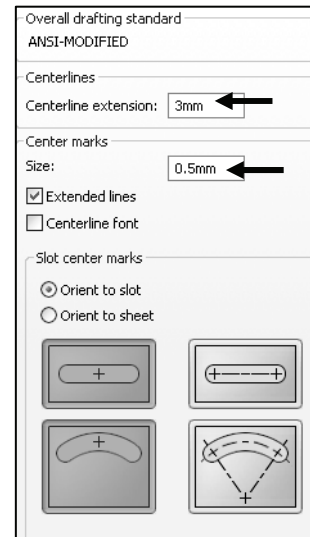
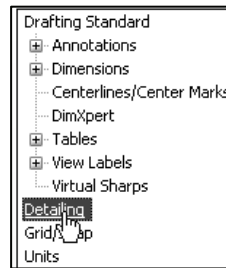


Modify the Break line gap.

61) Click the **Detailing** folder under Drafting Standard.

62) Enter **10mm** for the View Break lines Gap.

63) Enter **3mm** for the View break lines Extension.



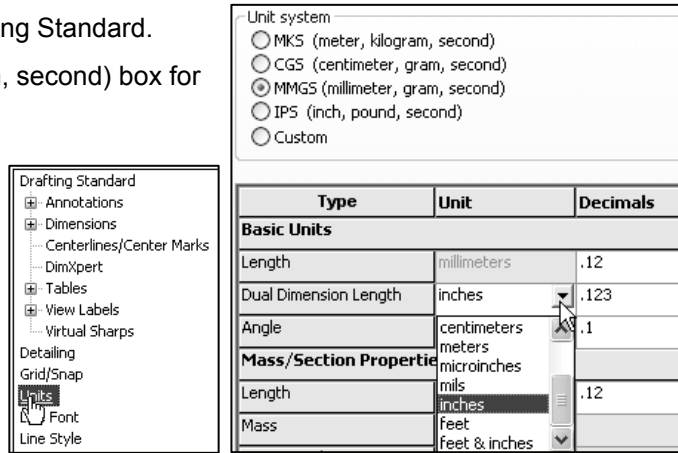


There is no set value for the Break line gap. Increase the value to accommodate a revolved section.



Set units.

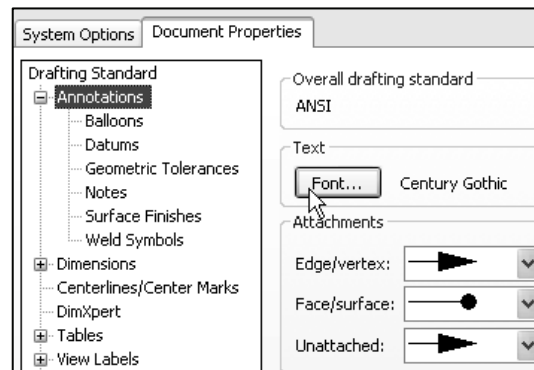
- 64) Click the **Units** folder under Drafting Standard.
- 65) Click the **MMGS** (millimeter, gram, second) box for Unit system.
- 66) Select **.12** from the drop-down menu for decimal places for Length units millimeters.
- 67) Select **inches** from the drop-down menu for Dual Dimension Length.
- 68) Select **.123** from the drop-down menu for inch Decimal places.
- 69) Select **.1** from the drop-down menu for Decimal places for Angular units.



- 70) Click **OK** from the Document Properties - Units dialog box.
- 71) Click **Save** .

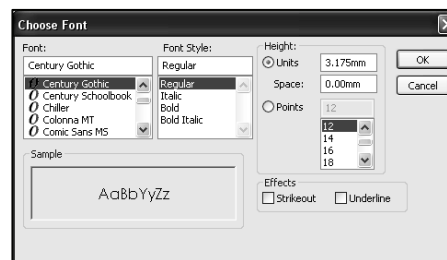
Document Properties, Annotations Font

The Annotations font controls the text height in the Drawing template for the following Annotation types: *Balloons*, *Datums*, *Geometric Tolerances*, *Notes*, *Surface Finishes* and *Weld Symbols*.



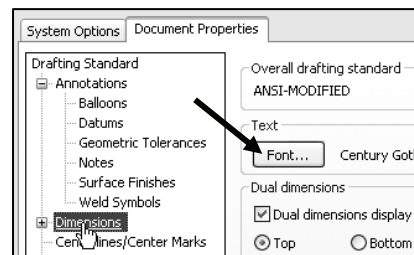
Notes Font

The Notes font option under the Annotations Font specifies the font type and size for notes and view labels.



Dimensions Font

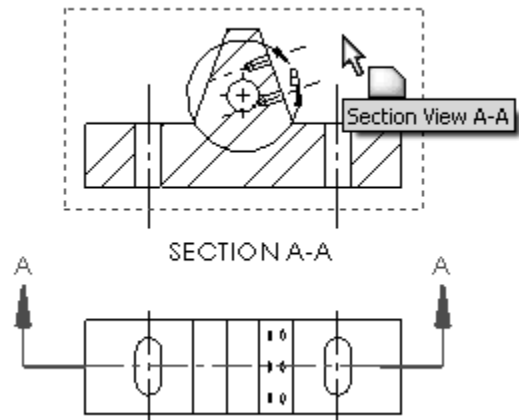
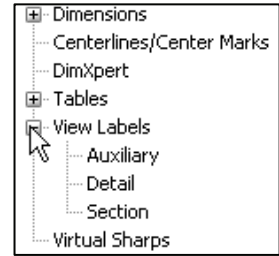
The Dimensions font option specifies the font type and size for the dimension text.



Auxiliary View / Auxiliary View Label Font

The Auxiliary View and the Auxiliary View Label fonts specify the font type and size used for the letter labels on the auxiliary arrow and the auxiliary view label text.

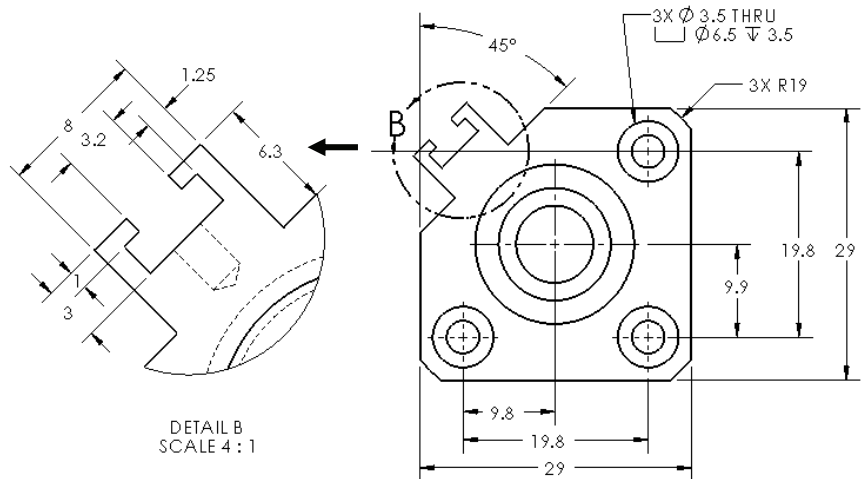
An Auxiliary View in general is similar to a Projected view, but it is unfolded normal to a reference edge in an existing view. The reference edge can be an edge of a part, a silhouette edge, an axis, or a sketched line. If you sketch a line, activate the drawing view first.



Detail View / Detail View Label Font

The Detail View and the Detail View Label fonts specify the font type and size used for the letter labels on the detail circle and the text below the detail view.

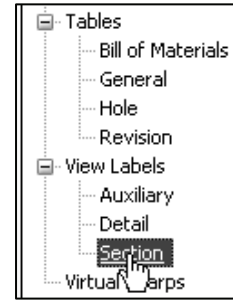
A Detail View in general is used to create a new drawing view which is an enlarged portion of an existing view. The enlarged portion is enclosed using sketch geometry, usually a circle or other closed contour like a spline.




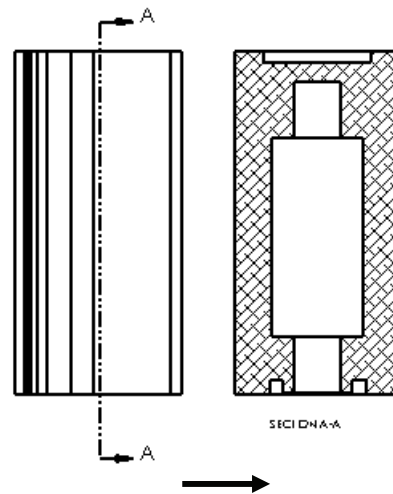
Section View / Section View Label Font

The Section View and the Section View Label fonts specify the font type and size used for the letter labels on the section lines and the text below the Section view.

A Section view in general is used to create a new drawing view that is defined by cutting an existing view with a section line (Sketch).



 Some drawing views require or allow sketching within the view, rather than just over the view. These views, most notably sections and details require that the view be active before sketching so that the sketch geometry will be associated with the view.




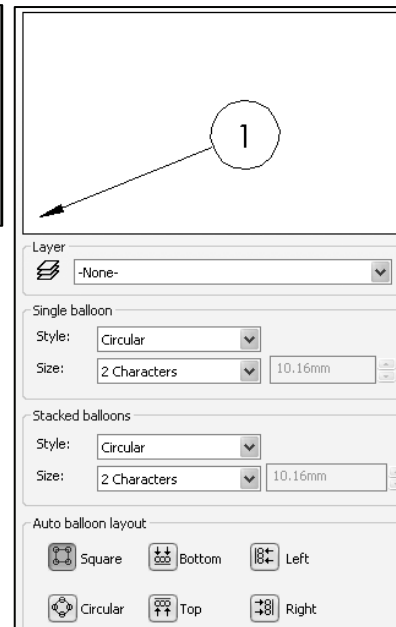
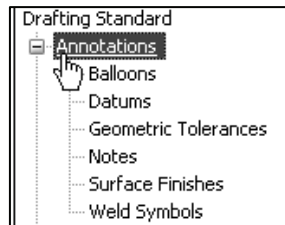
Annotations Arrow Font

The Annotations font specifies the font type and size used for general annotations.

Balloons, Datums, Geometric Tolerances, Notes, Surface Finished and Weld symbols Font

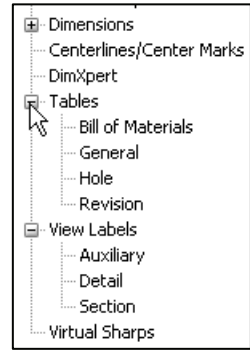
The Balloon, Datum, Geometric Tolerance, Note, Surface Finish and Weld symbol folders specifies the font type and size.

 ANSI standard states that a Leader line of a balloon is displayed as an arrow that points to an edge or a dot which points to a face.




Tables Font

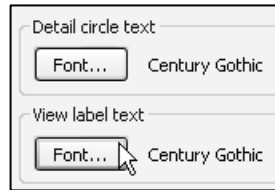
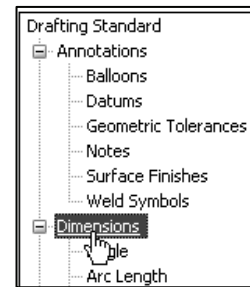
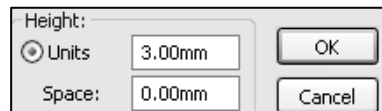
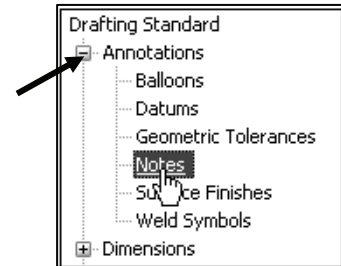
The Tables font varies from company to company. Tables font controls the *Bill of Materials*, *General*, *Hole* and *Revision*.



Activity: Document Properties-Annotations Font

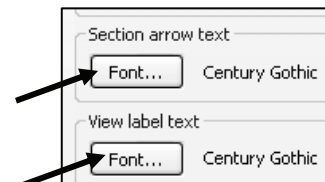
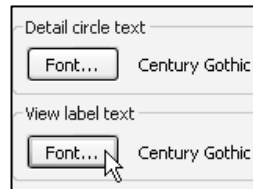
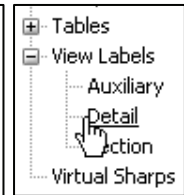
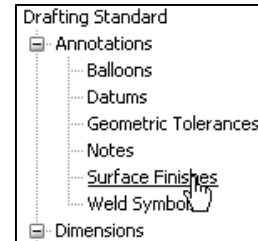
Set the Font.

- 72) Click **Options** , **Document Properties** tab from the Menu bar toolbar.
- 73) Expand the **Annotations** folder.
- 74) Click the **Notes** folder.
- 75) Click the **Font** button. The Choose Font dialog box is displayed.
- 76) Enter **3mm** for text height.
- 77) Click **OK** from the Choose Font dialog box.
- 78) Repeat the above procedure to set Font text height for **Dimensions**, (**View Labels Detail**, **View Label text**), **Surface Finishes**, **Weld Symbols**, **Tables**, and **Balloons** font.



Set the Section and View Arrow font.

- 79) Click the **Detail** folder under the View Labels folder.
- 80) Click the **Font** button for View label text.
- 81) Enter **6mm** for text height.
- 82) Click **OK** from the Choose Font dialog box.
- 83) Repeat for the above procedure for the **Section/Section View label text**, and the **Section/Section arrow text**.



84) Click **OK** from the Document Properties dialog box.

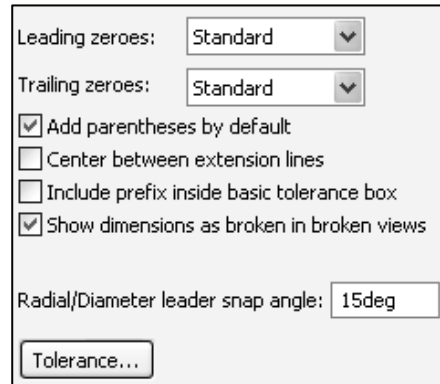
85) Click **Save** .



Companies vary the size of their default font. ASME Y14.2 lists the annotation values as minimum letter heights.

Document Properties, Dimensions Options

The Document Properties, Drafting Standard, Dimensions options determine the display of dimensions. The Dimensions option determines the display and position of the text and extension lines.



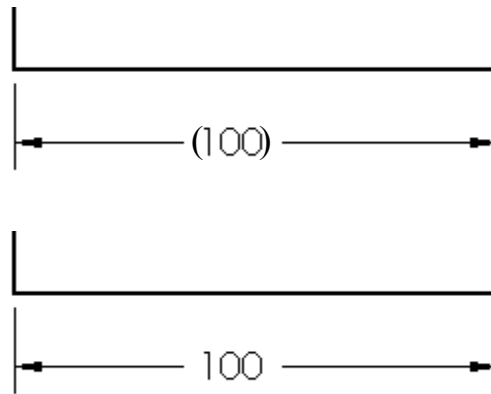
Reference dimensions require parentheses. Symmetric feature dimensions in the part require a redefined dimensioning scheme in the drawing.



Uncheck the Add parentheses by default to conserve design time.



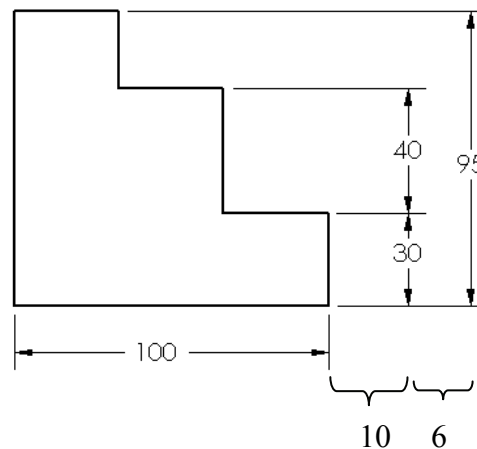
Add Parenthesis to a dimension in the drawing. Right-click on the dimension text. Click Properties. Check the Display with parentheses box.



Offset Distances Option


The ASME Y14.5M-1994(R1999) standard sets guidelines for dimension spacing. The space between the first dimension line and the part profile is 10mm or greater.


The space between subsequent parallel dimension lines is 6mm or greater. Spacing differs depending on drawing size and scale. Set the From last dimension option to 6mm. Set the From model option to 10mm.



Arrows Option

The Arrows option controls the display of the Arrowheads. The ASME Y14.2M-1992(R1998) standard recommends a solid filled arrow head.

 Arrowheads are drawn between extension lines if space is available. If space is limited, see the preferred arrowhead and dimension location order as illustrated.

 Double-click the dimension, to access the control points for the arrowheads in SolidWorks.

Break Dimension/Extension Option

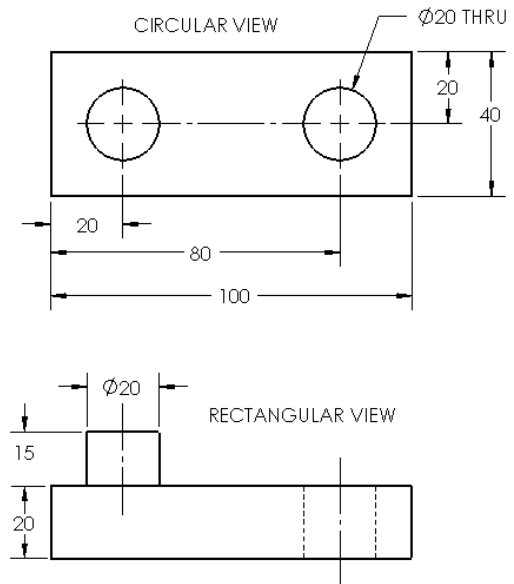
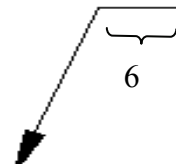
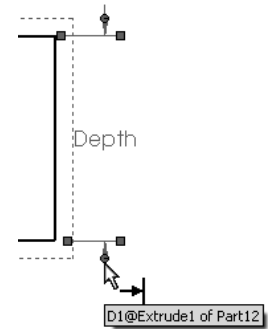
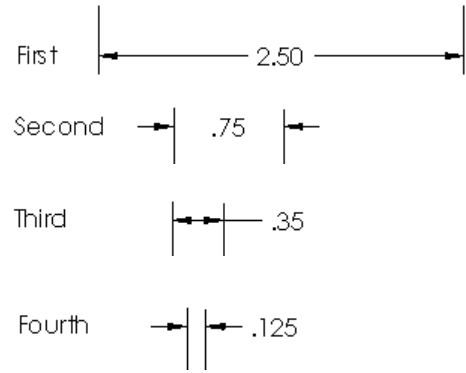
The ASME Y14.5M-1994(R1999) standard states do not cross dimension lines. Break the extension line when the dimension line crosses close to an arrowhead.

Drag the extension line above the arrowhead. Sketch a new line collinear with the extension line below the arrowhead.

Uncheck the Break around dimension arrows only option. Control individual breaks in the drawing for this chapter.


Bent Leader Length Option

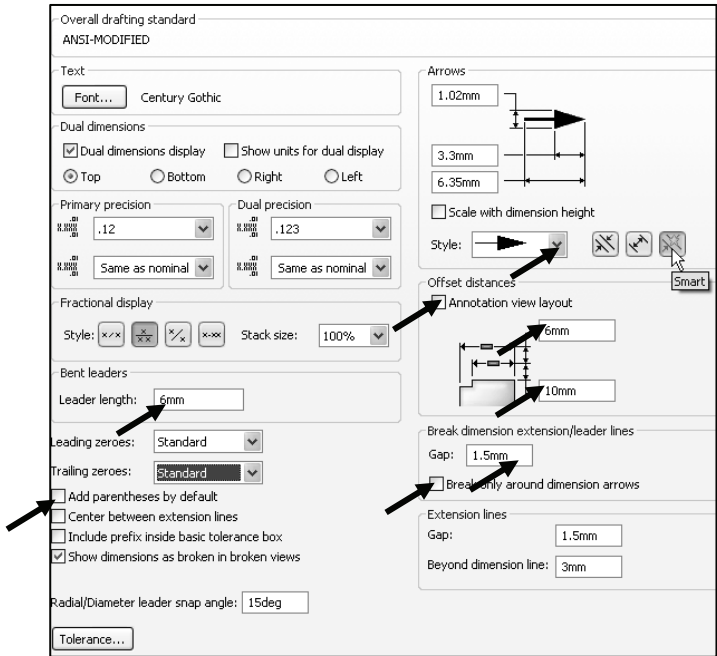
Create ASME leader lines with a small horizontal segment. This is called the Bent Leader length.



Activity: Document Properties-Dimensions

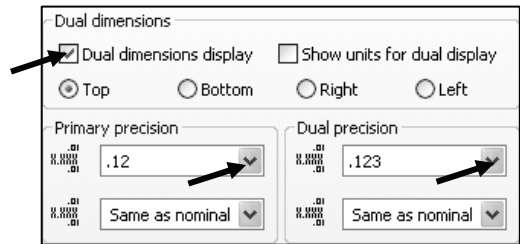
Set the Dimensions options.

- 86) Click **Options** , **Document Properties** tab from the Menu bar toolbar.
- 87) Click the **Dimensions** folder.
- 88) Uncheck the **Add parentheses by default** box.
- 89) Uncheck the **Annotation view layout** box.
- 90) Set the Offset distances to **6mm** and **10mm** as illustrated.
- 91) Set the Arrow Style to **Solid**.
- 92) Check the **Smart** box.
- 93) Enter **1.5mm** for the Gap in the Break dimension extension lines box.
- 94) Uncheck the **Break around dimension arrows only** box.
- 95) Enter **6mm** for the Bent leader length (ASME only).



Set the Dimension Precision.

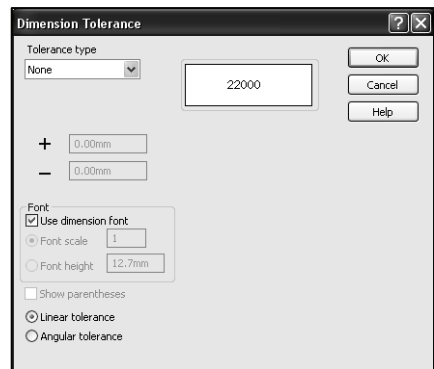
- 96) The primary units are millimeters. Select **.12** for two place decimal precision for Primary dimension.
- 97) Select **.123** for three place decimal precision for Dual precision.
- 98) Click **OK** from the Document Properties dialog box.



Save the drawing.

- 99) Click **Save** .

The Dimension Precision Value and Tolerance entries depend on drawing units and manufacturing requirements. The Tolerance button displays the Dimension Tolerance options. The Tolerance type is None by default. Control Tolerance type on individual dimensions.



Document Properties-Notes and Balloons Option

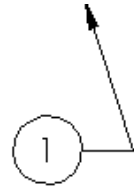
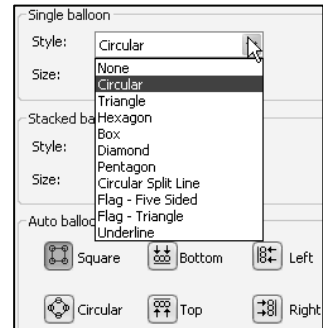
Note text positioned on the drawing, outside the Title block use the same font type and height size as the Dimension font. The exceptions to the rule are:



- ASME Y14.100M-1998 Engineering Drawing Practices extended symbols.
- Use Upper case letters for all Notes unless lower case is required. Example: HCl – Hardness Critical Item requires a lower case “l”.

Modify Notes Border Style to create boxes, circles, triangles and other shapes around the text. The Default Border style is set to None. Modify the border height. Use the Size option.

Balloon callouts label components in an assembly and relate them to the item numbers in the Bill of Materials. The default Balloon style is Circular.



Activity: Document Properties - Notes and Balloons

Set the Notes options.

100) Click **Options** , **Document Properties** tab from the Menu bar toolbar.

101) Expand the **Annotations** folder.

102) Click the **Notes** folder.

103) Check **Bent** for Leader display.

104) Uncheck the **Use document leader length** box.

105) Enter **6mm** for the Leader length.

Set the drawing Balloon Properties.

106) Click the **Balloons** folder.

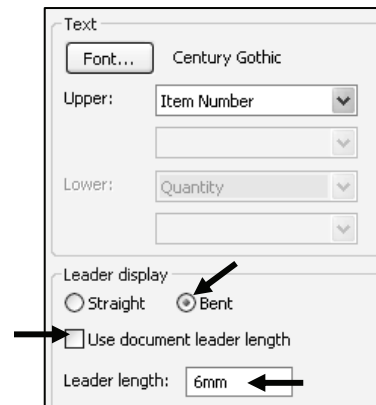
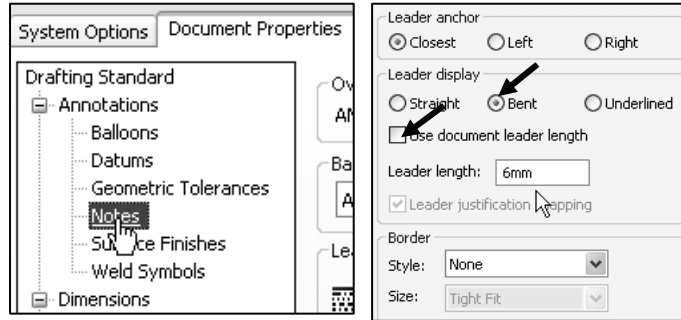
107) Click the **Bent** box.

108) Uncheck the **Use document leader length** box.

109) Enter **6mm** for Leader length.

110) Click **OK** from the Document Properties dialog box.

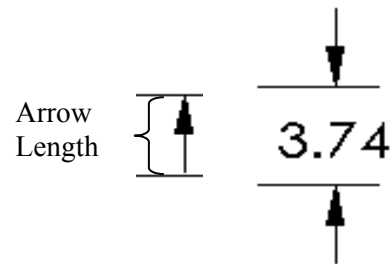
111) **Save** the drawing.



Document Properties - Arrows

Set Arrows Properties according to the ASME Y14.2M-1992(R1998) standard with a 3:1 ratio: Width to Height.

The Length value is the overall length of the arrow from the tip of the arrowhead to the end of the arrow tail. The Length is displayed when the dimension text is flipped to the inside. A Solid filled arrowhead is the preferred arrow type for dimension lines.



Activity: Document Properties - Arrows

Set the Dimension Arrow Properties.

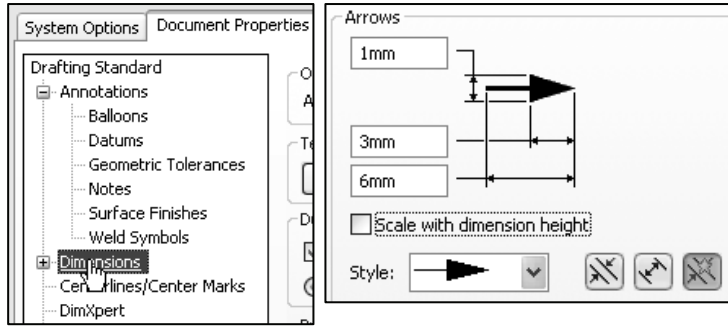
112) Click **Options** , **Document Properties** tab from the Menu bar toolbar.

113) Click the **Dimensions** folder.

114) Enter **1** for the arrow Height in the Size text box.

115) Enter **3** for the arrow Width.

116) Enter **6** for the arrow Length.



Set the Section View Arrow Properties.

117) Expand the **View Labels** folder.

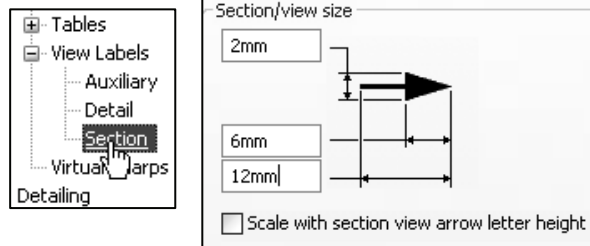
118) Click the **Section** folder.

Set the arrow style.

119) Under the Section/View size, enter **2** for Height.

120) Enter **6** for Width.

121) Enter **12** for Length.



Document Properties - Line Font

The Line Font determines the style and thickness for a particular type of edge in a drawing. Modify the type of edge, style and thickness to reflect the ASME Y14.2M-1992(R1998) standard. The ASME Y14.2M-1992(R1998) standard defines two line weights: 0.3mm and 0.6mm.

Thin Thickness is 0.3mm. Thick (Normal) Thickness is 0.6mm. Review line weights as defined in the File, Page Setup or in File, Print, System Options for your particular printer/plotter. Control the line weight display in the Graphics window.

Activity: Document Properties - Line Font

Set the Line Font Properties.

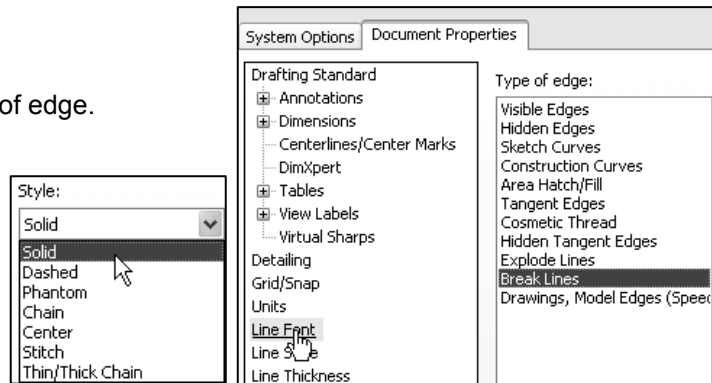
122) Click the **Line Font** folder.

123) Click **Break Lines** for the Type of edge.

124) Select **Solid** for Style.

Create a Custom Line Thickness.
125) Select **Custom Size** for Thickness.

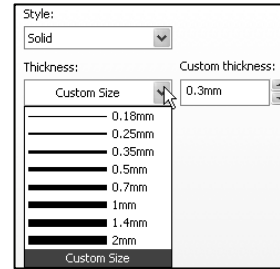
126) Enter **0.33mm** for Custom thickness.



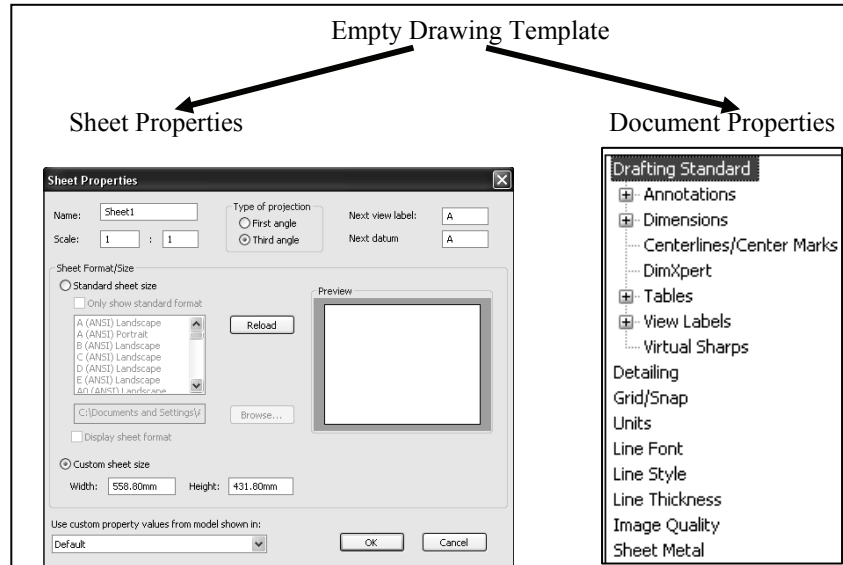
127) Click **OK** from the Document Properties - Line Font dialog box.

Save the drawing.

128) Click **Save**.



Draw1 is the current drawing. Utilize Draw1 to create a Drawing template. The empty Drawing template contains no geometry. The empty Drawing template contains the Document Properties and the Sheet Properties.



Predefined and Projected Views

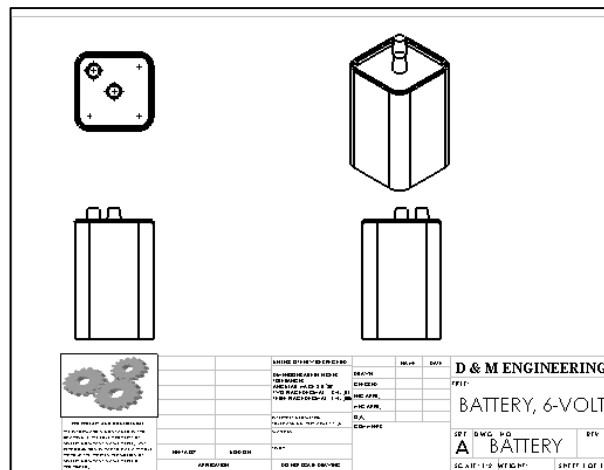
In Orthographic Projection - the six principle views are *Top, Front, Right, Back, Bottom* and *Left*. Drawings commonly display the Top, Front, Right, and an Isometric view. You can define a view in a drawing sheet and then populate the view. You can save a drawing document with Predefined views as a document template.

Insert the Top, Front, Right, and Isometric views into the drawing template. Utilize the Predefined command to create the Front and Isometric view. Utilize the Projected view command to create the Right and Top view.

The Drawing template contains a Sheet format. Leave space when positioning views.




Save Predefined views with the drawing template. Save the drawing template in the next section, before you insert a part into the Predefined views.




Activity: Insert Predefined and Projected Views

Insert a Front Predefined view.

129) Click **Insert, Drawing View, Predefined**  from the Menu bar menu.

130) Click the **lower left corner** of the drawing. The Drawing View1 PropertyManager is displayed.

 *Front view is the default view in the Orientation dialog box.

131) Click **Hidden Lines Removed** from the Display Style box.

132) Click **OK**  from the Drawing View1 PropertyManager.

Insert a Top Projected view.


133) Click the **View Layout** tab from the CommandManager.

134) Click **Projected view**  from the View Layout toolbar. The Projected View PropertyManager is displayed.

135) Check the **Use parent style** box to display Hidden Lines Removed.

136) Click a **position** directly above the Front view.

Insert the Right Projected view.


137) Click **Projected View**  from the View Layout toolbar.

138) Click inside the **Front** view.

139) Click a **position** directly to the right of the Front view.


Insert an Isometric Predefined view.


140) Click inside the **Front** view.

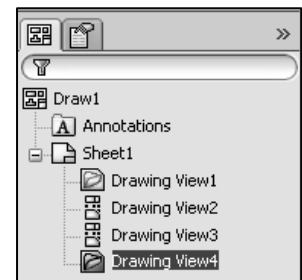
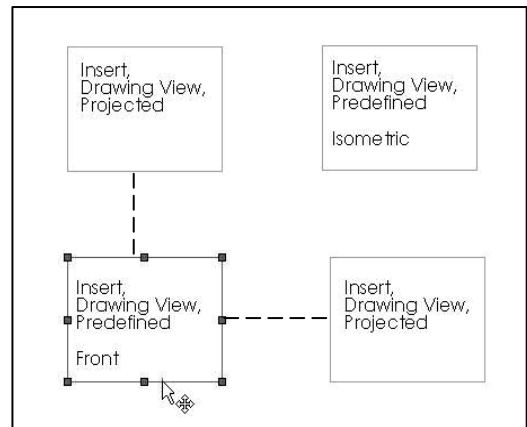
141) Click **Insert, Drawing View, Predefined**  from the Menu bar menu. The Drawing View PropertyManager is displayed.

142) Click a **position** in the upper right corner of the sheet as illustrated.

143) Click ***Isometric** from the Orientation box.

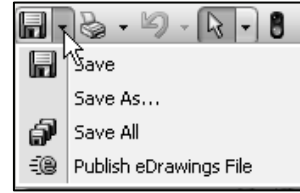
144) Click **OK**  from the Drawing View4 PropertyManager.

145) Click **Save** . View the drawing FeatureManager. Note the View icons for the Predefined and Projected views.



Save As

The Save As option provides the ability to save documents with various file types. The current document is a drawing named Draw1.slddrw. Save the document as a Drawing template (*.drwdot).



Select the Drawing Templates (*.drwdot) option for Save as type before you browse to the MY-TEMPLATES folder. SolidWorks selects the SolidWorks\data/templates folder by default when you select Drawing Templates (*.drwdot).

Test the Drawing template located in the MY-TEMPLATES folder. Create a new drawing document.

Activity: Save As and Test Drawing Template

Save the empty Drawing Template.

146) Click **Save As** from the Menu bar menu.

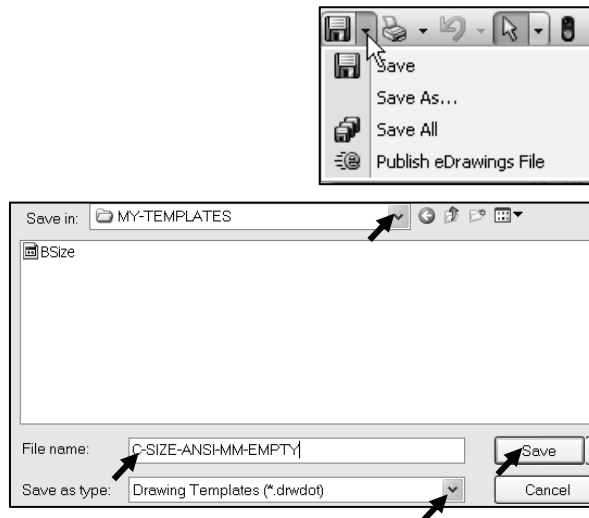
147) Select **Drawing Templates (*.drwdot)** from the Save as type.

148) **Browse** and select the **DRAWING-W-SOLIDWORKS-2010\ MY-TEMPLATES** for the Save in file folder.

149) Enter **C-SIZE-ANSI-MM-EMPTY** for the File name. The file extension for the template is .drwdot.

150) Click **Save** from the Save As dialog box.

151) Click **Windows, Close All** from the Menu bar toolbar.

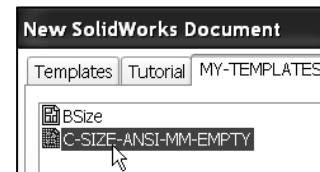


Create a new drawing.

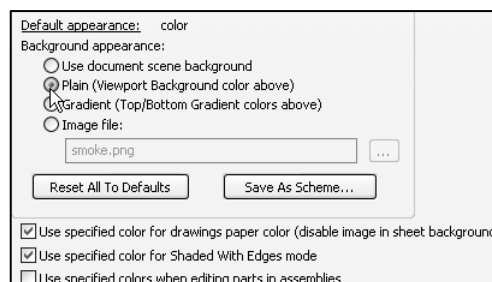
152) Click **New** from the Menu bar toolbar

153) Select **MY-TEMPLATES** tab from the New SolidWorks Document dialog box.

154) Double-click **C-SIZE-ANSI-MM-EMPTY**.




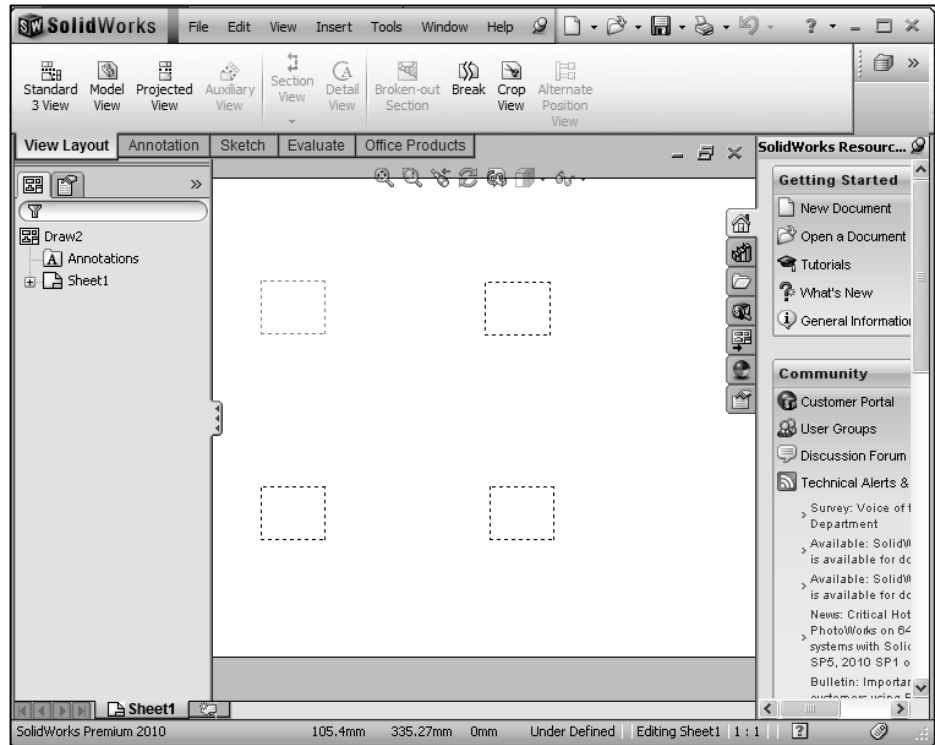
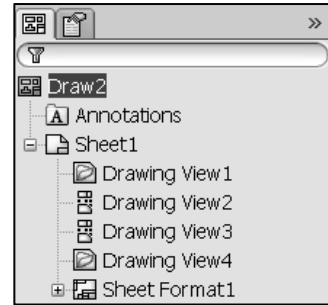
For improved drawing visibility, the default Drawing Sheet background color is modified to white.



The Sheet Format/Size box displays C (ANSI) Landscape.

155) If required, click **C (ANSI) Landscape**. Click **OK**.

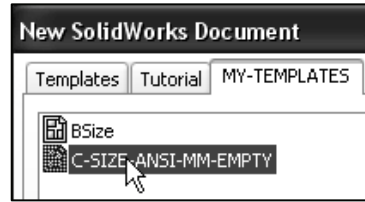
156) Click **Cancel**  from the Model View PropertyManager. Draw2 is the current drawing document. Note the drawing view icons in the FeatureManager.



Close all documents.

157) Click **Windows, Close All** from the Menu bar menu.

You created a C (ANSI) size drawing with no Sheet format when you selected the C-SIZE-ANSI-MM-EMPTY template from the New SolidWorks Document box. The Drawing template controls sheet size and Document Properties. The Sheet format controls the Title block, company logo, and Custom Properties.



Conserve design time. Utilize the C-SIZE-ANSI-MM-EMPTY template to create empty templates for A and B size drawings. Modify the Sheet Properties size option and utilize the Save As options for the drawing template.

More Information

Additional details on Sheet Properties, System Options, and Document Properties is available in SolidWorks Help Topics. Keywords: sheet properties, paper (size), drawings (display modes, edge and display), options (annotations, balloon, detailing, dimensions, file locations, font, note, and units).



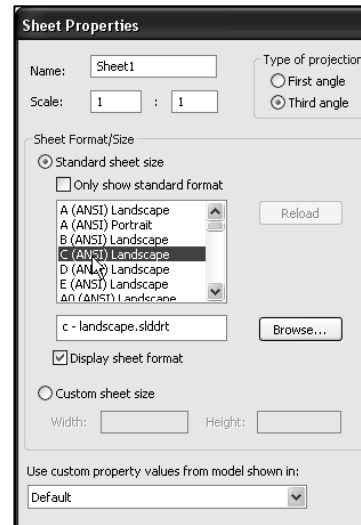
Review

The Sheet Properties option displayed: Sheet name, scale, size, Type of projection, and more. You selected C (ANSI) Landscape size paper with no Sheet format.

You reviewed the System Options Drawings and File Locations. The Drawings Display Style option controlled the display mode and tangent edges of the view.

The File Locations option created the MY-TEMPLATES folder tab in the New SolidWorks Document dialog box.

Document Properties are stored in the current document. You utilized the Detailing (Dimensions, Notes, Balloons, Arrows, and Annotations Font), Line Font, and Units options in the Drawing template. There are hundreds of System Options and Document Properties.



Sheet Format

Customize drawing Sheet formats to create and match your company drawing standards.

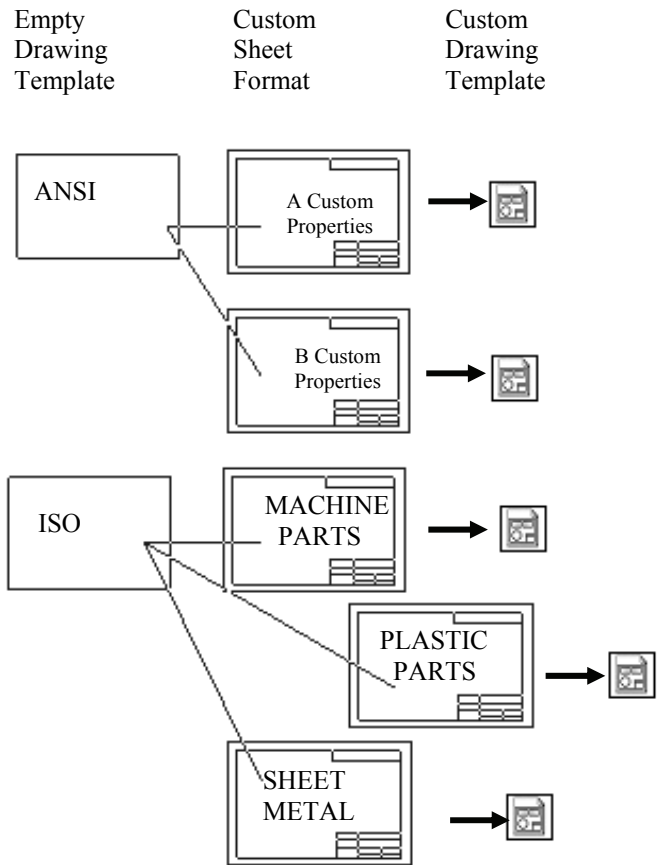
A customer requests a new product. The engineer designs the product in one location, the company produces the product in a second location and the field engineer supports the customer in a third location.

The ASME Y14.24M standard describes various types of drawings. Example: The Engineering department produces detail and assembly drawings. The drawings for machined, plastic and sheet metal parts contain specific tolerances and notes used in fabrication.

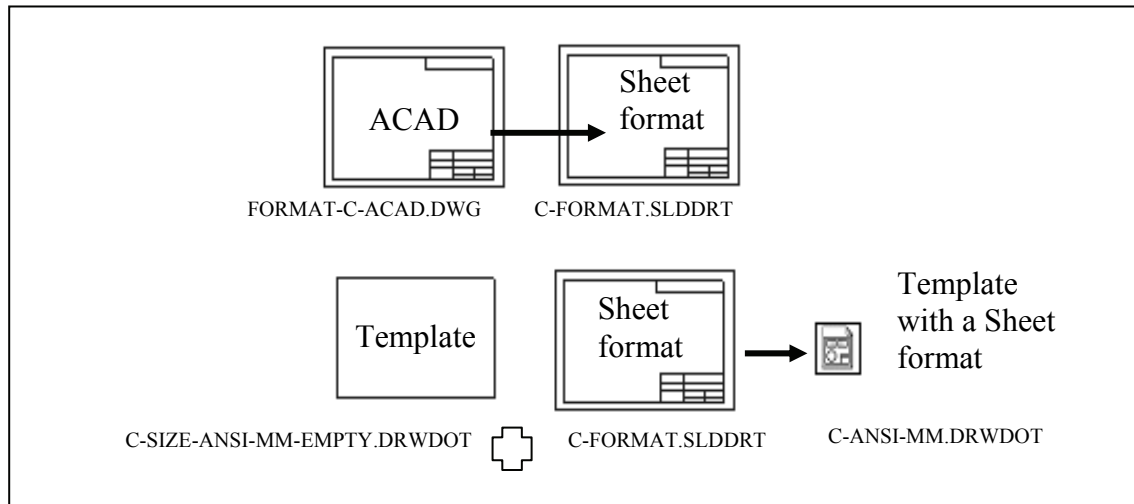
Manufacturing adds vendor item drawings with tables and notes. Field Service requires installation drawings that are provided to the customer.

Create Sheet formats to support various standards and drawing types. There are numerous ways to create a custom Sheet format:

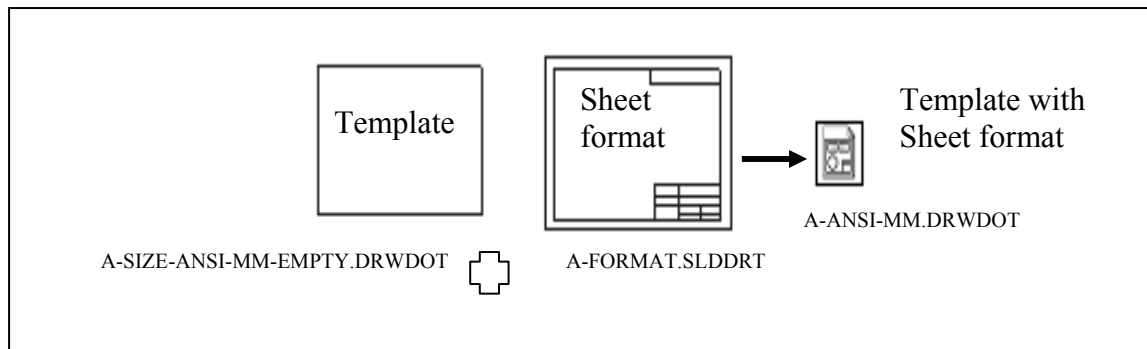
- Open a “.dwg” file created with another CAD application. Save the “.dwg” file as a Sheet format.
- Right-click in the Graphics window. Select Edit Sheet Format. Create drawing borders, Title block, notes, and zone locations for each drawing size. Save each drawing format.
- Right-click Properties in the Graphics window. Select Properties. Check the Display Sheet Format option from the Sheet format drop-down menu. Browse to select an existing Sheet format.
- Add an OLE supported Sheet format such as a bitmap file of the Title block and notes. Use the Insert, Object command or Insert, Picture command.



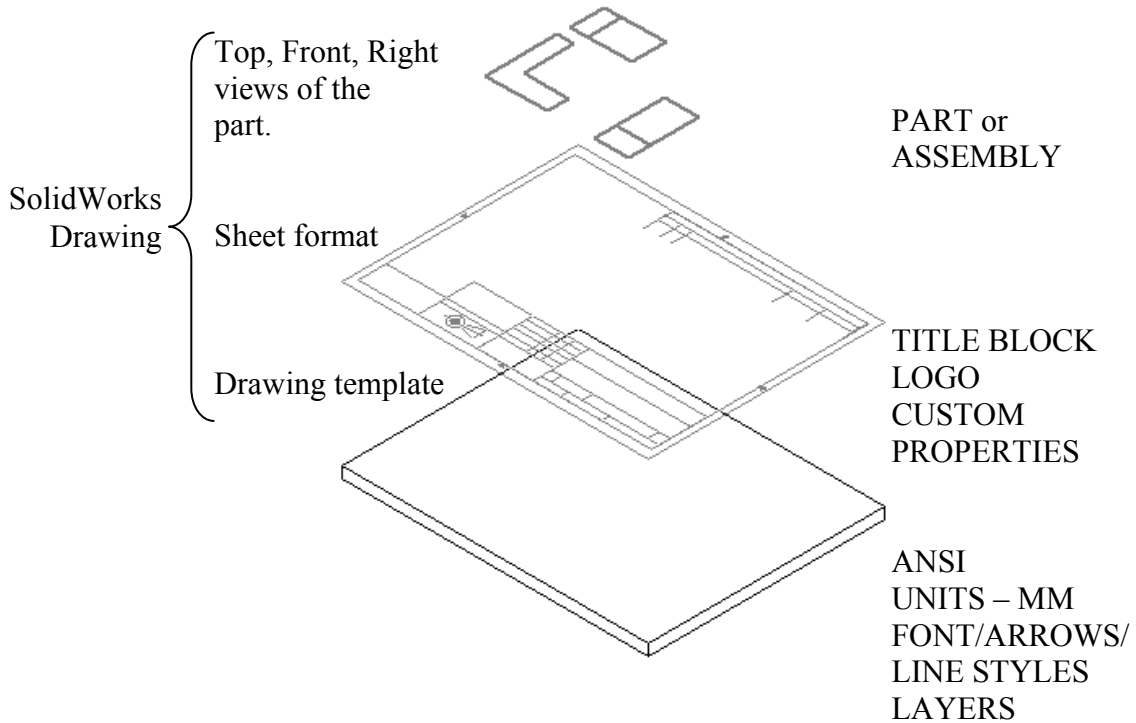
- Utilize an existing AutoCAD drawing to create a SolidWorks Sheet format.
- Open the AutoCAD drawing as the Sheet format. Save the C-FORMAT.sldprt
- Sheet format. Add the Sheet format C-FORMAT.sldprt to the empty C (ANSI) size Drawing template. Create a new Drawing template named C-ANSI-MM.drwdot.



- Add an A (ANSI) size Sheet format, A-FORMAT.sldprt to an empty A (ANSI) size Drawing template. Create an A-ANSI-MM.drwdot Drawing template.



Insert views from the part or assembly into the SolidWorks drawing.

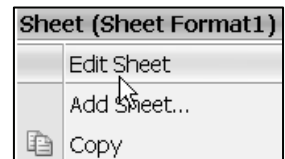
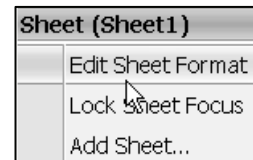


Data imported from other CAD systems for a Sheet format may require editing in SolidWorks. Delete extraneous lines in the imported Sheet format. The drawing sheet contains two modes:

- *Edit Sheet Format*
- *Edit Sheet*

Utilize the *Edit Sheet Format* command to add or modify notes and Title block information. Edit in the *Edit Sheet Format* mode for lines and text created in the AutoCAD Title block.

Utilize the *Edit Sheet* command to insert views and dimensions. The sheet boundary and major title block headings are displayed with a THICK line style. Modify the drawing layer THICKNESS.



Activity: Sheet Format, Import From AutoCAD

Open an AutoCAD drawing: FORMAT-C-ACAD.dwg.

158) Click **Open** from the Menu bar toolbar.

159) Select **DWG (*.dwg)** from the Files of type drop-down menu.

160) Double-click **FORMAT-C-ACAD** from the DRAWING-W-SOLIDWORKS-2010\MY-SHEETFORMATS folder. A DXF / DWG Import dialog box is displayed.

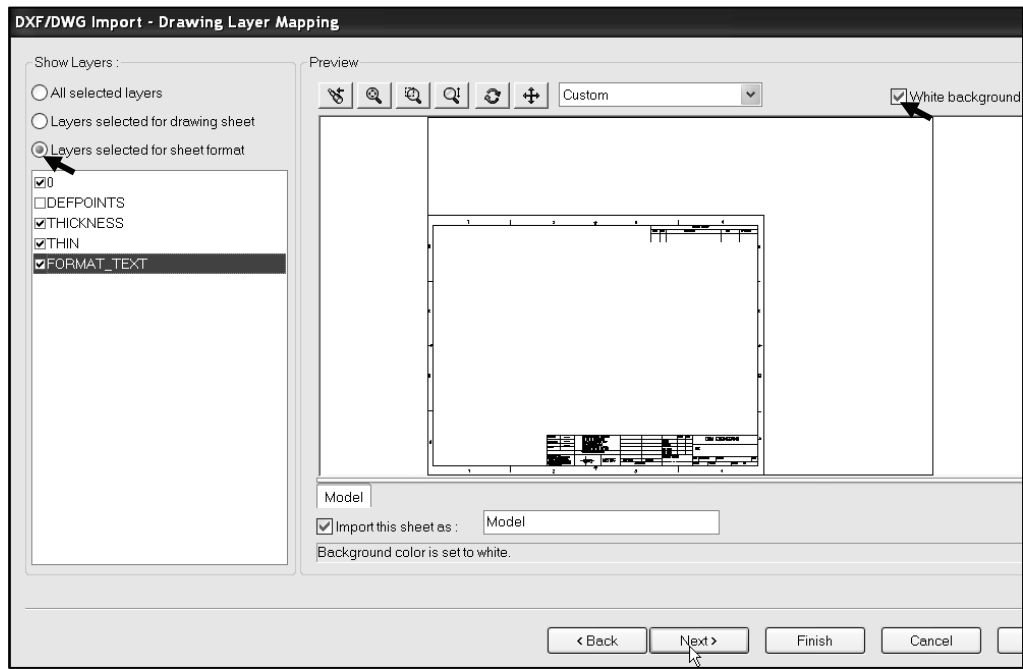
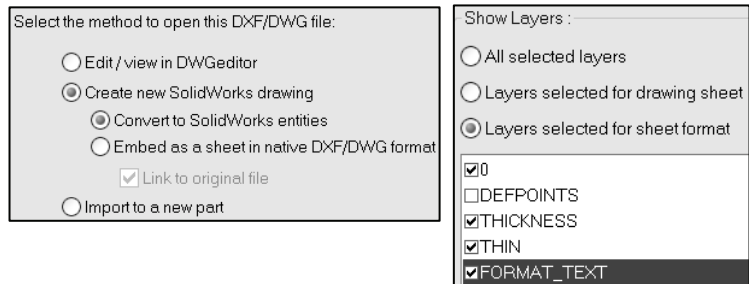
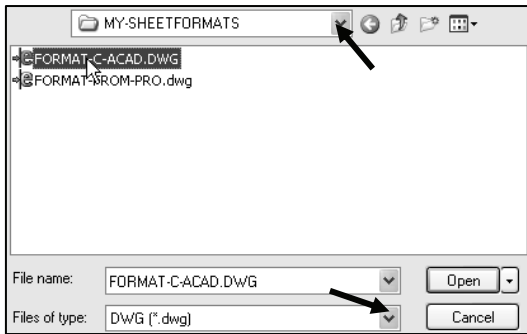
161) Accept the default settings. Click **Next>**.

162) Click **Layers selected for sheet format**.

163) Check **0**, **THICKNESS**, **THIN** and **FORMAT_TEXT** layers.

164) Check the **White background** box.

165) Click **Next>**.

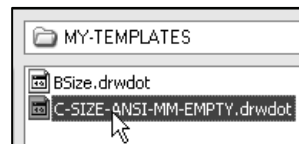


166) Select **Millimeters** for Data units.

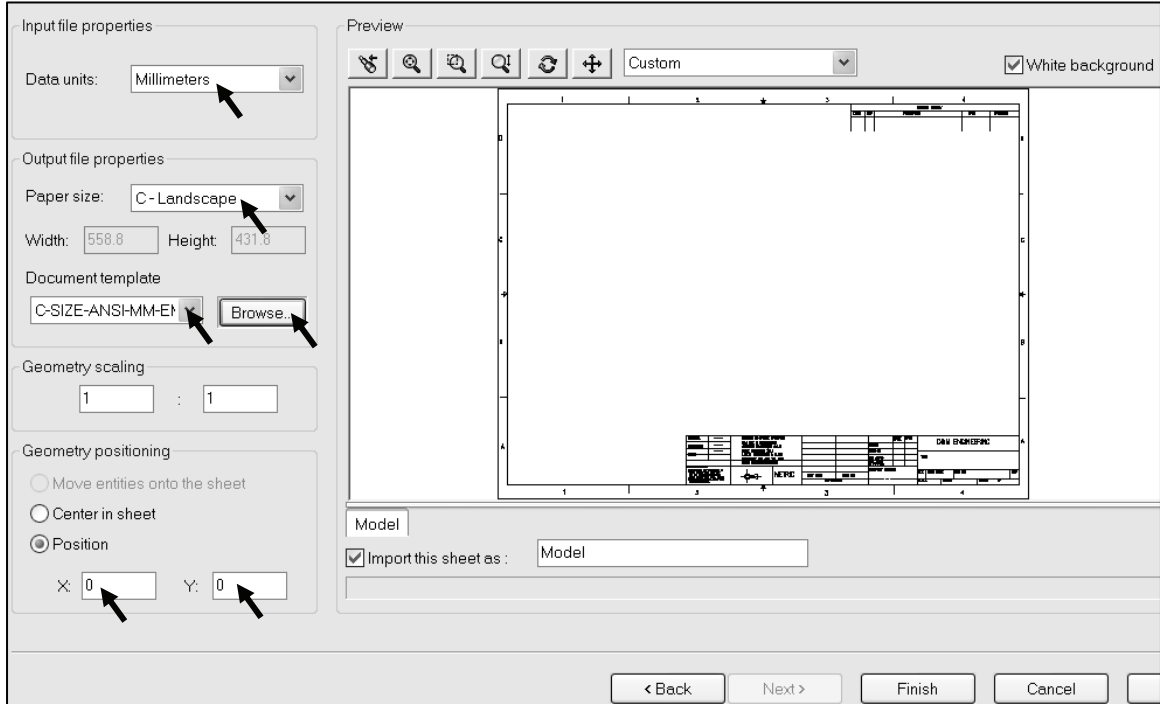
167) Select **C-Landscape** for Paper size.

168) Click the **Browse** button.

169) Select the **MY-TEMPLATES** folder.




- 170) Double-click **C-SIZE-ANSI-MM-EMPTY** for Drawing Template.
- 171) Enter **0** for the X position.
- 172) Enter **0** for the Y position.
- 173) Click **Finish**.



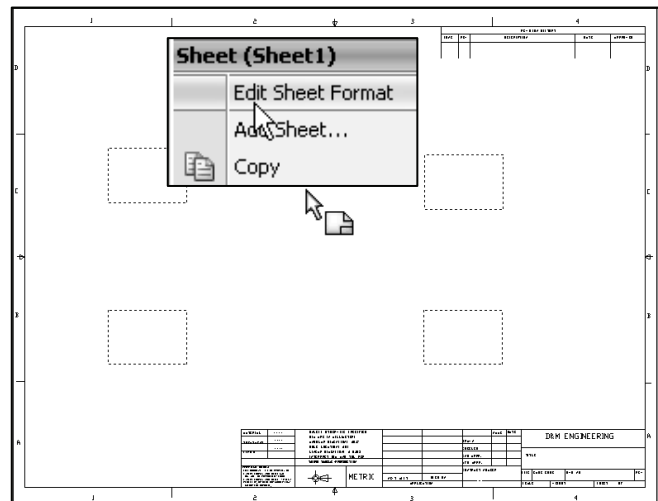
Save the Drawing.

- 174) Click **Save As** from the Menu bar toolbar.
- 175) Select the **DRAWING-W-SOLIDWORKS-2010** folder.
- 176) Enter **Draw3** for File name.
- 177) Click **Save**.

 Draw# is the temporary drawing document utilized to create the Sheet format.

Edit the Title block.

- 178) Right-click in the **Graphics window**.
- 179) Click **Edit Sheet Format**.



Delete the Title block lines.

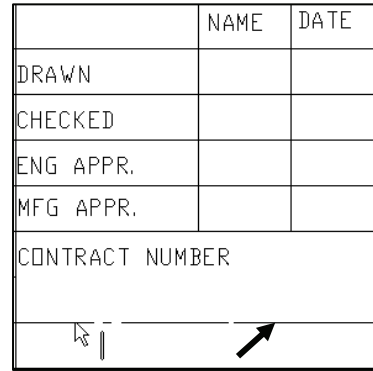
180) Zoom in on the Title block.

181) Click the first horizontal line below the CONTRACT NUMBER.

182) Right-click Delete.

183) Click the second horizontal line below the CONTRACT NUMBER.

184) Right-click Delete.



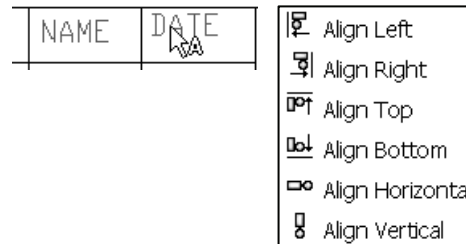
Align the NAME and DATE text.

185) Click and drag the NAME text.

186) Click and drag the DATE text.



You can use the Ctrl key, right-click Align and select the Align option.



Display the Layer toolbar.

187) Right-click a position in the gray area, to the right of the Help menu.

188) Check Layer. The Layer toolbar is displayed.

Modify Thick Layer properties.

189) Click the Layer Properties icon from the Layer toolbar.



Rename the AutoCAD layer

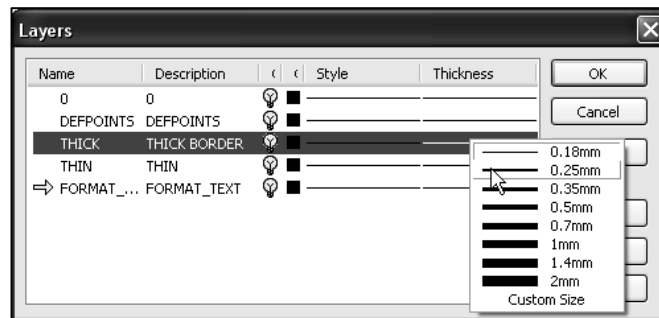
190) Rename Name from THICKNESS to THICK.

191) Rename Description from THICKNESS to THICK BORDER.

192) Click the line Thickness in the THICK layer.

193) Select the second line.

194) Click OK from the Layers dialog box.



The border and Title block display the Thick line. The left line in the Title block is on the Thin layer. Modify the line layer from the Thin layer to the Thick layer.

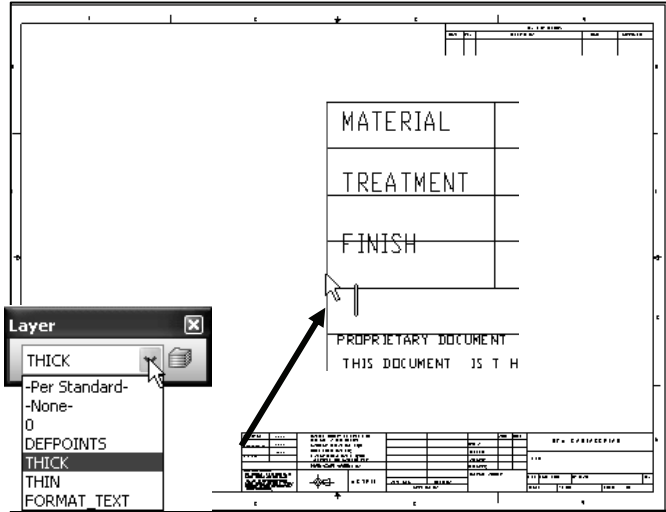
Modify a Line layer.

195) Click on the **left line** as illustrated.

196) Click **THICK** layer from the Options box.

197) Click **OK** ✓ from the Line Properties PropertyManager.

💡 Align the MATERIAL, TREATMENT and FINISH text as an exercise. You will need to retype the MATERIAL, TREATMENT and FINISH text before you align them.



Save the Sheet.

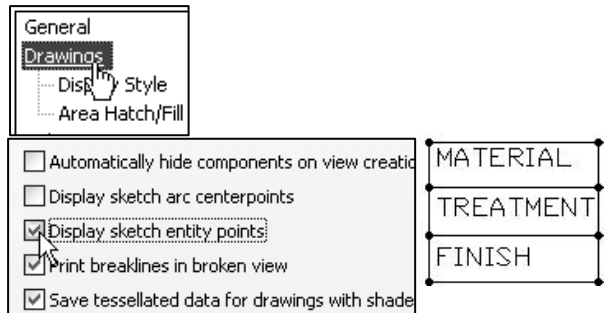
198) Click **Save**.

The C-FORMAT requires additional information and editing in the Title block. The Title block created from AutoCAD only contains text headings such as: Drawing Number, Revision, and Drawn by. Each heading is located in a different box in the Title block.

Insert additional Notes into the Title block in the Edit Sheet Format Mode. The Notes in the Sheet Format are linked to Properties. Properties are variables shared between parts, assemblies, and drawing documents.

💡 View Line segments clearly. The System Options, Drawings, Display sketch entity points option displays the endpoints of the line segments. Check this option before editing the lines in the Title block.

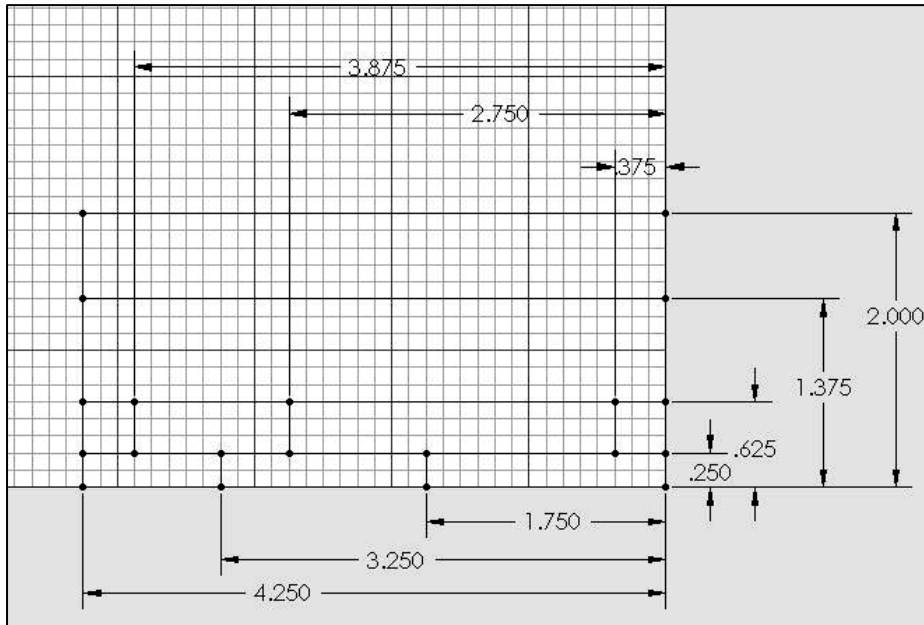
💡 Utilize the Sketch tools to create and edit Title block lines. Utilize dimensions and geometric relations to create Title block lines for A, B, C, D, and E sheet formats according to the ASME Y14.1 Decimal Inch Drawing Sheet Size and Format and ASME Y14.1M Metric Drawing Size and Format.





Utilize the Document Property, Grid/Snap for quick sketching. The ASME Y14.1 Title block is based on 0.125 increments. Set the Document Properties, Grid/Snap to 0.125 (English). The following dimensions below are recommended for A, B, C, and G sizes.

Detailing
Grid/Snap
Units
Line Font
Line Style



Title Block Notes and Properties

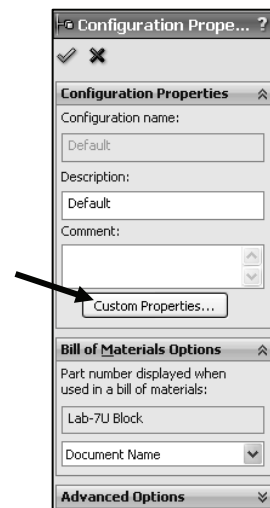
The Title block contains vital part and assembly information. Each company creates a unique version of a Title block. The imported AutoCAD sheet format contains heading names in each area of the Title block such as: TITLE, DWG NO., and SCALE.

Utilize SolidWorks System Properties and User defined Custom Properties to link Notes in the Sheet format to the drawing, part, and assembly.

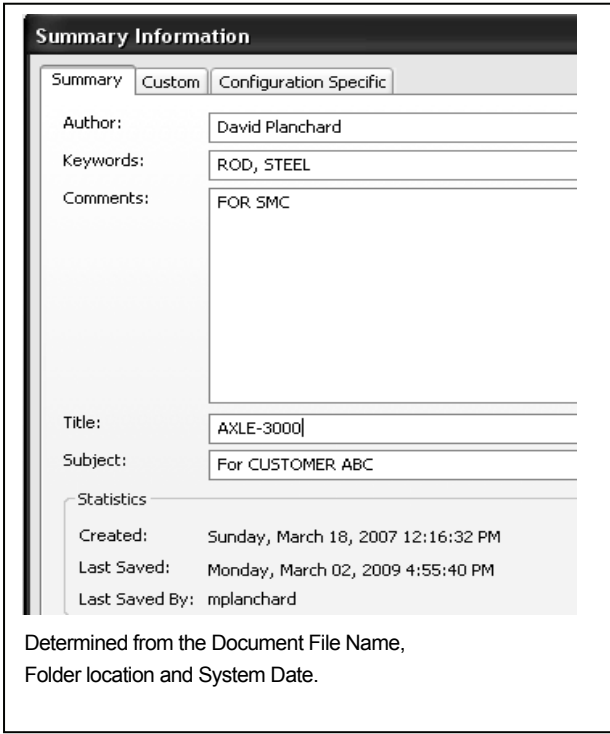
System Properties

System Properties extract values from the current drawing. System Properties are determined from the SolidWorks documents. Insert System Properties as linked Notes in the Sheet Format.

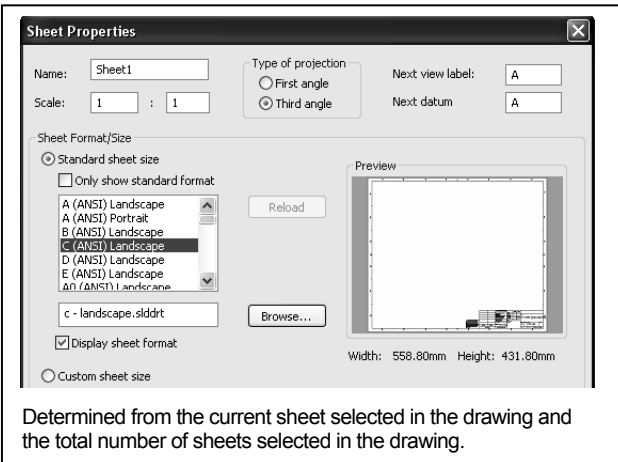
System Properties begin with the prefix SW. There are two categories of Properties: System Properties and Drawing Specific System Properties.



Set System Properties in the File, Properties, Summary Information dialog box as follows:

System Properties:	
SW-Author	
SW-Keywords	
SW-Comments	
SW-Title	
SW-Subject	
SW-Created Date	
SW-Last Saved Date	
SW-Last Saved By	
SW-File Name	
SW-Folder Name	
SW-Long Date	
SW-Short Date	

Set Drawing Specific System Properties: SW-Sheet Name, SW-Sheet Scale SW-Sheet Format Size and SW-Template Size in the Sheet Properties dialog box.

Drawing Specific:	
SW-Sheet Name	
SW-Sheet Scale	
SW-Sheet Format Size	
SW-Template Size	
SW-Current Sheet	
SW-Total Sheet	

User Defined Properties

There are two types of User defined Properties: Custom Properties and Configuration Specific Properties. Custom Properties link all of the configurations of a part or an assembly. Configuration Specific Properties link only a single configuration of a part or an assembly.

Assign User defined Property values to named variables in the document. The default variables are listed in the text file `\SolidWorks 2010\lang\english, properties.txt`. Create your own User defined Property named variables.



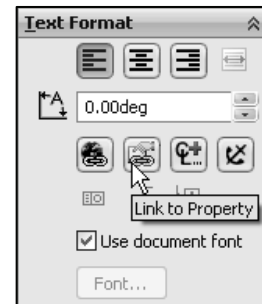
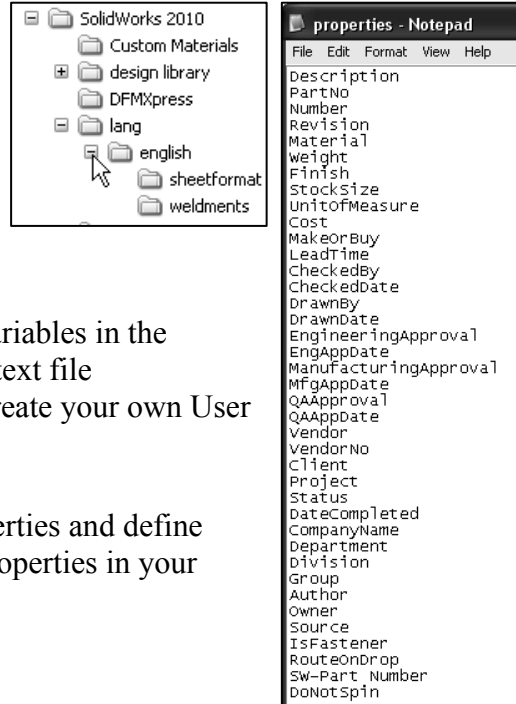
Conserve design time. Utilize System Properties and define Custom Properties and Configuration Specific Properties in your sheet formats.

Linked Notes

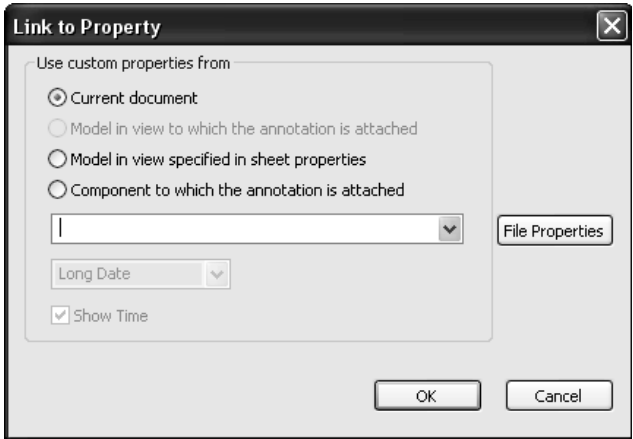
Insert Notes into the Title block. Link the Notes to SolidWorks Properties and Custom Properties.

Review your company's Engineering documentation practices to determine the Notes displayed in the Title block.

In the next activity, DWG NO. is linked to the SW-File Name System Property. Revision is linked to the Revision Custom Property in the part or assembly.



Linked Notes begin with the four different prefixes listed below:

Prefix:	Evaluated from:	
\$PRP:	Current document.	
\$PRPVIEW:	Model in the drawing view to which the Note is attached.	
\$PRPSHEET:	Model in view specified in Sheet Properties. For Sheet Format notes, the first view listed in the FeatureManager is used.	
\$PRPMODEL:	Component to which annotation is attached.	Utilize \$PRP and \$PRPSHEET in the Sheet Format.

Linked Notes that reference Custom Properties in the drawing utilize the prefix: \$PRP: Enter double quotes to define the property name: Example: \$PRP:"CompanyName".

Linked Sheet Format Notes that reference Custom Properties in the part utilize the prefix: \$PRPSHEET. Linked Sheet Format Notes are displayed blank in the Edit Sheet mode. Linked Sheet Format Notes are displayed with their property Name in the Edit Sheet Format mode. Example: \$PRPSHEET{Material}.

Insert the following Linked Notes:

System Properties Linked to fields in the default Sheet Format. Prefix: \$PRP	Custom Properties of drawings linked to fields in the default Sheet Formats. Prefix: \$PRP	Custom Properties copied from the default SW Sheet Format to a Custom Sheet Format. Prefix: \$PRP		Custom Properties of parts and assemblies linked to the fields in default Sheet Formats. Prefix:\$PRPSHEET
SW-File Name (in DWG. NO. field)	CompanyName	DrawnBy	DrawnDate	Description (in TITLE field):
SW-Sheet Scale	CONTRACT NUMBER	CheckedBy	CheckedDate	Weight
SW-Current Sheet		EngineeringApproval	EngAppDate	Material, Finish and TREATMENT
SW-Total Sheets		ManufacturingApproval	MfgAppDate	Revision

User-defined Custom Property Names CONTRACT NUMBER and TREATMENT are displayed in capital letters for clarity. Utilize Large and small letters for Custom Property Names. Create a new layer for the Title block notes. The large yellow arrow in the Name column indicates the current layer.

Activity: Title Block and SW-File Name

Insert the Title block TEXT layer.

199) Click the **Layer Properties** icon.

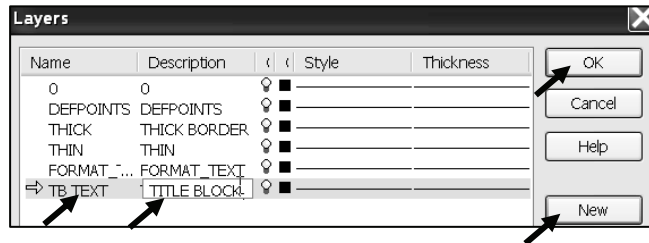


200) Click the **New** button.

201) Enter **TB TEXT** for Name.

202) Enter **TITLE BLOCK TEXT** for Description.

203) Click **OK** from the Layers box.



Create a Linked Note for the DWG NO System Property.

204) Click **Note** from the Annotation toolbar.



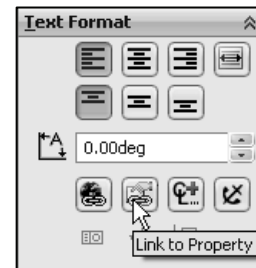
205) Click a **point** below the DWG NO. text.

206) Click **Link to Property** from the Text Format box.

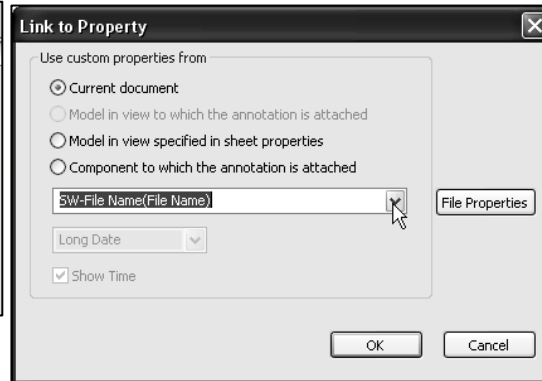
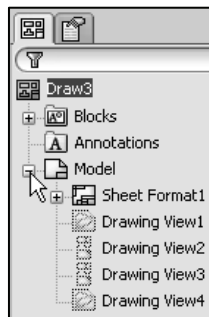
207) Select **SW-File Name** from the drop-down menu.

208) Click **OK** from the Link to Property dialog box.

209) Click **OK** from the Note PropertyManager.



210) Position the mouse pointer on the **Draw3** text. The variable name **\$PRP:"SW-File Name"** is displayed.




Save the Drawing.

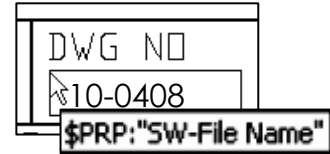
211) **Rebuild** the drawing.

212) Click **Save**.

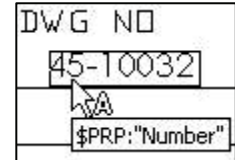
SIZE	CAGE CODE	DWG NO Draw3	REV
SCALE	WEIGHT	SHEET OF	\$PRP:"SW-File Name(File Name)"

 Draw3 is the current file name. The default Draw number varies depending on the number of drawings opened in a SolidWorks session.

The \$PRP:“SW-File Name(File Name)” property updates to contain the part or assembly filename. Example: Insert the part 10-0408 into a Drawing template. The filename 10-0408 is linked to the SW-FileName property and is displayed in the DWG NO. box.

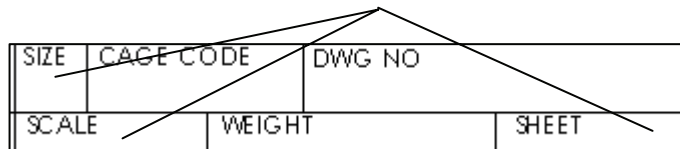


What action do you take to control the DWG NO. by a separate property not linked to the part filename? Answer: Create a Note linked to the Custom Property \$PRP: "Number" in the Sheet format. Enter the value 45-10032 for the Number Custom Property in the drawing document.



Size, Sheet and Scale Properties

Additional Linked Notes are required in the Title block. Create the SIZE, SHEET and SCALE text with Linked Properties. Position the text below the headings.



The Sheet Scale value changes to reflect the sheet scale properties in the drawing. The Sheet box combines two System Properties: SW-Current Sheet and SW-Total Sheets. The Current Sheet value and Total Sheets value change as additional sheets are added to the drawing.

Activity: Size, Sheet and Scale Properties

Create a Linked Property to the SIZE text.

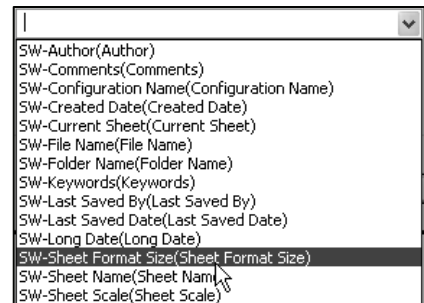
213) Click **Note**  from the Annotation toolbar.

214) Click a **point** below the SIZE text.

215) Click **Link to Property**  from the Text Format box.

216) Select **SW-Sheet Format Size** from the drop-down menu.

217) Click **OK** from the Link to Property dialog box.



Modify the font size and style of the SIZE text.



218) Double-click **C** in the SIZE box

219) Enter **3** for font height.

220) Click **Bold** for style.

221) Click **OK** ✓ from the Note PropertyManager.

Create a Linked Property to SCALE.

222) Click **Note A** from the Annotation toolbar.

223) Click a **point** to the right of the SCALE text.

224) Click **Link to Property**  from the Text Format box.

225) Select **SW-Sheet Scale** from the drop-down menu.

226) Click **OK**. 1:1 is displayed.

227) Click **OK** ✓ from the Note PropertyManager.

Delete text in the Title box.

228) Click the **OF** text in the lower right corner of the title box.

229) Press the **Delete** key.

Combine Link Properties for the SHEET text.

230) Double-click the **SHEET** text.

231) Position the **cursor** at the end of the text.

232) Press the **space bar**.

233) Click **Link to Property**  from the Text Format box.

234) Select **SW-Current Sheet** from the drop-down menu.

235) Click **OK**.

236) Press the **space bar**.

237) Enter the text **OF**.

238) Press the **space bar**.

239) Click **Link to Property**  from the Text Format box.

240) Select **SW-Total Sheets** from the drop-down menu.

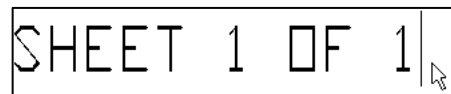
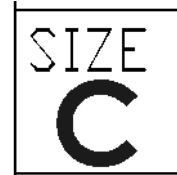
241) Click **OK**.

242) Click **OK**.

243) Click **OK** ✓ from the Note PropertyManager.

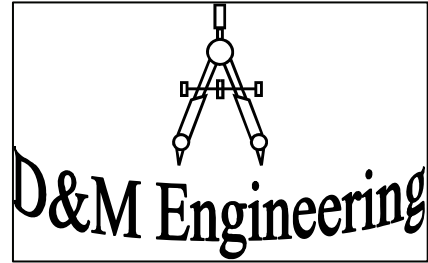
Save the Drawing.

244) Click **Save**.



Custom Property and Logo Picture

Utilize D&M ENGINEERING or your own value for CompanyName in the next step. The CompanyName Property is controlled through a Custom Property in the Sheet format.



Activity: Custom Property and Logo Picture

Delete the current Company Name Note text.

245) Right-click on the **D&M ENGINEERING** text in the drawing.



246) Click **Edit Text in Window**.

247) Delete **D&M ENGINEERING**.



Insert the CompanyName Property.

248) Enter **\$PRP: "CompanyName"** in the Note text box.

249) Click **OK**.

250) Click **Link to Property**  from the Text Format dialog box.

251) Click the **File Properties** button from the Link to Property box.

252) Click the **Custom** tab from the Summary Information box.

253) Click inside the **Property Name** box.

254) Select **CompanyName**.

255) Click inside the **Value / Text Expression** box.

256) Enter **D&M ENGINEERING** for CompanyName.

257) Click inside the **Evaluated Value** box.

258) Click **OK** from the Summary Information box.

259) Click **OK** from the dialog box.

260) Click **OK**  from the Note PropertyManager. View the results.

Summary		Custom		
Delete		BOM Quantity: - none -		Edit List
	Property Name	Type	Value / Text Expression	Evaluated Value
1	CompanyName	Text	D&M ENGINEERING	D&M ENGINEERING
2				



The Title block displays the CompanyName Linked Note. The current document stores the CompanyName Property. Select Custom Properties through the Link to Property drop-down menu.

Modify the font of D&M ENGINEERING.

261) Double-click the **D&M ENGINEERING** text.

262) Click **Bold**.

263) Click **OK** from the Note Property Manager.



Position the mouse pointer over the Linked Note to display the Custom Property value. Utilize Ctrl-A to select all the text in the Note text box.

A company logo is normally located in the Title block. Create a company logo by inserting a picture file or a file as an OLE object into the Title block. Example: The file COMPASS.doc is located in the MY-SHEETFORMATS folder. Utilize any picture file, scanned image, or bitmap.



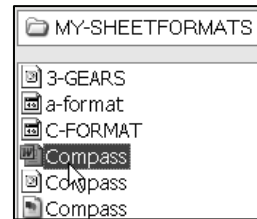
Insert a picture file for the Sheet Logo.

264) Click **Insert, Object** from the Menu bar menu. Note: You can insert a picture as a logo. Click **Create from File**.

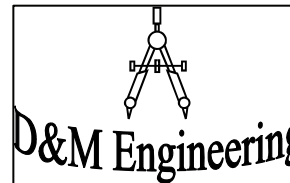
265) Click **Browse**.

266) Double-click **MY-SHEETFORMATS\Compass.doc**.

267) Right-click **OK**. The picture file is displayed on the Sheet.



268) Size the **picture** by dragging the picture handles in Sheet1 as illustrated.

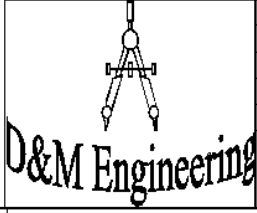
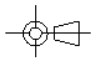


Save the Drawing.

269) Click **Save** .



If needed, you can add relations in the sheet format so that any modifications can be easily applied. For instance if you wanted to modify the format for different sheet sizes, or needed to extend a text area

	MATERIAL	----	UNLESS OTHERWISE SPECIFIED
	TREATMENT	----	DIM ARE IN MILLIMETERS
	FINISH	----	ANGULAR DIMENSIONS ±0.3°
			HOLE LOCATIONS ±0.1
			LINEAR DIMENSIONS ± 0.125
			INTERPRET DIM AND TOL PER
			ASME Y14.5M - 1994
			THRU ANGLE PROJECTION
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			METRIC


to accommodate a large amount of information in the Title block, then it would be easier to achieve if there were defined sketch entities.

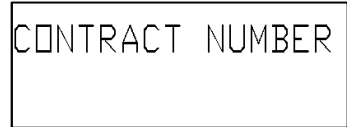
User Defined Custom Property

Your company has a policy that a contract number must be contained in the Title block for all associated drawings in the chapter. The contract number is not a predefined SolidWorks Custom Property. Create a user defined Custom Property named CONTRACT NUMBER. Add it to the drawing Title block. The Custom Property is contained in the sheet format.

Activity: User Defined Custom Property

Create a User defined Custom Property.

270) Click **Note**  from the Annotation toolbar.



271) Click a **point** in the upper left hand corner below the CONTRACT NUMBER text.

272) Click **Link to Property**  from the Text Format box.

273) Click the **File Properties** button.

274) Click the **Custom** tab.

275) Click inside the **Property Name** spin box.

276) Enter **CONTRACT NUMBER** for Name.

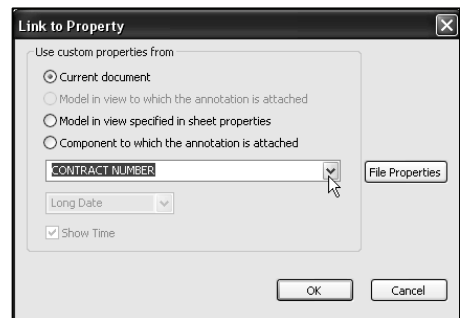
277) Click inside the **Value / Text Expression** box.

278) Enter **101045-PAP** for Value. Click inside the **Evaluated Value** box.

279) Click **OK** from the Summary Information box.

	Property Name	Type	Value / Text Expression	Evaluated Value
1	CompanyName	Text	D&M ENGINEERING	D&M ENGINEERING
2	CONTRACT NUMBER	Text	101045-PAP	101045-PAP

280) Select **CONTRACT NUMBER** in the Property Name text box. Click **OK** from the Link to Property box. View the results

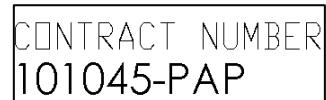


281) Click **OK**  from the Note PropertyManager.

Fit the drawing to the Graphics window.

282) Press the **f** key.

283) Click **Save**.




Copy/Paste Custom Properties

Conserve design time. Share information from Templates and Sheet Formats. Copy DrawnBy, DrawnDate, CheckedBy, CheckedDate, EngineeringApproval, EngAppDate, ManufacturingApproval and MfgAppDate from a default SolidWorks C-Sheet format to the Custom C-Format.

Activity: Copy/Paste Custom Properties

Open the default SolidWorks C-size Drawing template.


284) Click **New**  from the Menu bar toolbar.


285) Select the **Templates** tab.

286) Double-click **Drawing**.

287) Select **C (ANSI) Landscape** for the Sheet format.

288) Check **Display sheet format**. Click **OK**.

	NAME	DATE
DRAWN		
CHECKED		
ENG APPR.		
MFG APPR.		

289) If needed, click **Cancel**  from the Model View PropertyManager.

290) **Zoom in** on the NAME and DATE area in the Title block.

	NAME	DATE	
DRAWN	ERROR!: DrawnBy	ERROR!: DrawnDate	TITLE
CHECKED	ERROR!: CheckedBy	ERROR!: CheckedDate	
ENG APPR.	ERROR!: EngineeringApproval	ERROR!: EngAppDate	
MFG APPR.	ERROR!: ManufacturingApproval	ERROR!: MfgAppDate	

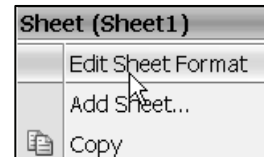
Display the Linked text.

291) Click **View, Annotation Link Errors** from the Menu bar menu. An Error indicates the value for the Custom Property is empty.

Edit the Sheet format.

292) Right-click in the **sheet boundary**.

293) Click **Edit Sheet Format**.



Copy the drawing Custom Properties.

294) Hold the **Ctrl** key down.

295) Select the **text** in the columns under Name and Date. Do not select the QA text row.

296) Release the **Ctrl** key.

297) Press **Ctrl + C**.

298) **Return** to the active custom C-Sheet format drawing.

299) Click a **position** between the NAME and DATE column and the CHECKED and ENG APPR. row.

	NAME	DATE
DRAWN	ERROR!: DrawnBy	ERROR!: DrawnDate
CHECKED	ERROR!: CheckedBy	ERROR!: CheckedDate
ENG APPR.	ERROR!: EngineeringApproval	ERROR!: MfgAppDate
MFG APPR.	ERROR!: ManufacturingApproval	ERROR!: EngAppDate
CONTRACT NUMBER 101045-PAP		

Paste the information.

300) Press **Ctrl + V**.

301) Drag the **text** to center in the NAME column and DATE column.

302) Position the **mouse pointer** on the DrawnBy text. The Custom Property \$PRP:"DrawnBy" is displayed.

Hide the Linked text.

303) Click **View**; uncheck **Annotation Link Errors** from the Menu bar menu.

Insert Custom Property DrawnBy.

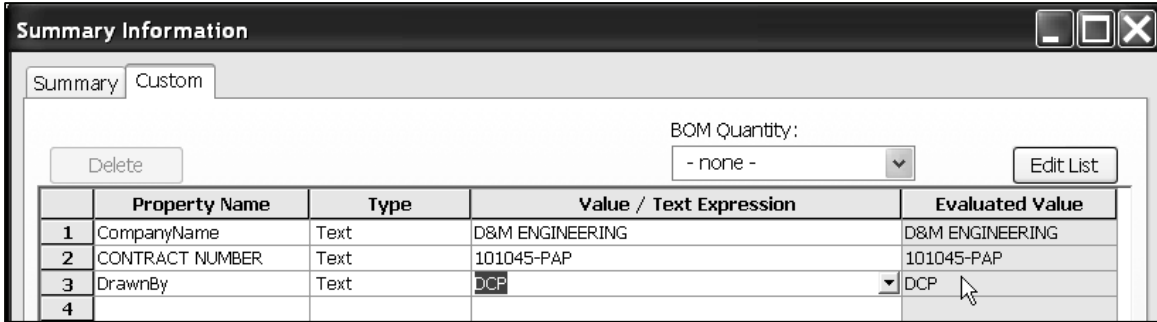
304) Click **File, Properties** from the Menu bar menu.

305) Click the **Custom** tab.

306) Select **DrawnBy** for Property Name.

307) Enter your name, example **DCP**, in the Value / Text Expression box.

308) Click inside the **Evaluated Value** box.



309) Click **OK**.

310) Click **Save** .



In this example, you saved the DrawnBy Custom Property with the Sheet format. The DrawnBy Custom Property may also be left blank in the Sheet format and entered by the designer in the drawing.

	NAME	DATE
DRAWN	DCP	
CHECKED		

Custom Properties in Parts and Assemblies

Define Custom Properties in parts and assemblies through the ConfigurationManager, Properties option. Insert Custom Properties from a part or assembly into the drawing. Create Description, Weight, Material, and Revision Custom Properties as Linked Notes in the sheet format. Enter values for these Custom Properties in the part or assembly.


Activity: Custom Properties in Parts and Assemblies

Insert the Description Property.

311) Click **Note A** from the Annotation toolbar.

312) Click a **position** to the right of the TITLE.


313) Enter **\$PRPSHEET:"Description"**.

314) Click **OK**  from the Note PropertyManager.




The Note displays \$PRPSHEET:{Description}. Enter the Description value in the part or assembly Custom Properties. The value is linked to the TITLE box Note.

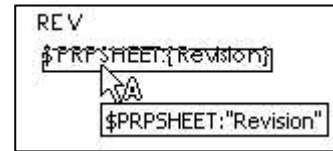
Insert the Revision Property.

315) Click **Note**  from the Annotation toolbar.

316) Click a **position** below the REV text.

317) Enter **\$PRPSHEET:"Revision"**.

318) Click **OK**  from the Note PropertyManager. The Note displays \$PRPSHEET:{Revision}.



Enter the Revision value in the part or assembly Custom Properties. Edit the WEIGHT text and append the text \$PRPSHEET:"WEIGHT".

Insert the Weight Property.


319) Right-click the **WEIGHT** text on Sheet1.

320) Click **Edit Text in Window**.

321) Delete the **WEIGHT** text.

322) Enter **\$PRPSHEET:"Weight"**.

323) Click **OK** from the Edit Text Window dialog box.

324) Click **OK**  from the Note PropertyManager.



Insert the Material Property.

325) Delete the ----- to the right of the MATERIAL box.

326) Click **Note**  from the Annotation toolbar.

327) Enter **\$PRPSHEET:"Material"** to the right of the MATERIAL box.

328) Click **OK**  from the Note PropertyManager.

329) Repeat for TREATMENT.

330) Enter **\$PRPSHEET:"Treatment"** to the right of the TREATMENT box.

331) Repeat for FINISH.

332) Enter **\$PRPSHEET:"Finish"** to the right of the FINISH box.

333) Click **Save** .



MATERIAL	\$PRPSHEET:{Material}	UNLES DIM A PROU HOLE LINEA
TREATMENT	\$PRPSHEET:{Treatment}	
FINISH	\$PRPSHEET:{Finish}	

Description, Revision, Weight, Material, and Finish are predefined Custom Properties. Assign values in the part and assembly. The TREATMENT Custom Property is not defined. Create the TREATMENT Custom Property Name and value in the part through the ConfigurationManager, Custom Properties, or a Design Table.

General Notes

General notes are annotations that describe additional information on a drawing. Conserve drawing time. Place common general notes in the Sheet format. The Engineering department stores general notes in a Notepad file, GENERALNOTES.TXT. General notes are usually located in a corner of a drawing.

Activity: General Notes

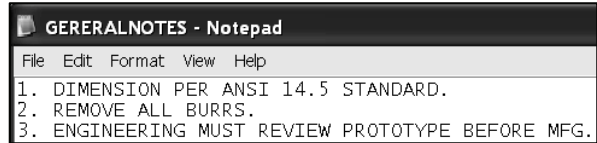
334) Minimize the **SolidWorks window**. Do not close.

Create general notes from a text file.

335) Double-click on the Notepad file, **MY-SHEETFORMATS\GENERALNOTES.TXT**.



336) Click **Ctrl + A** to select the text in the Notepad file.



337) Click **Ctrl + C** to copy the text into the windows clipboard.

338) Return to the open document in SolidWorks. Click the **Alt + tab**.

339) Click **Note A** from the Annotation toolbar.

340) Click a **point** in the lower left hand corner of the Sheet.



341) Click **inside** the Note text box.

Paste the three lines of text.

342) Click **Ctrl + V**.

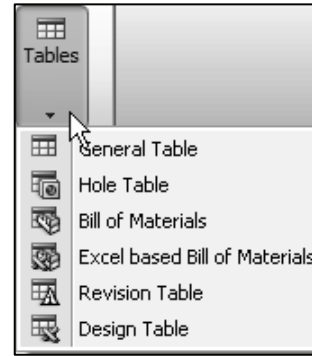
343) Click **OK** ✓ from the Note PropertyManager.

344) Click **Save** .

Tables

Various general SolidWorks tables are available: *General Table, Hole Table, Bill of Materials, Excel based Bill of Materials, Weldment Cut list, Revision Table and Design Table*. Each table contains an Anchor point. An Anchor point locates the Table position in the Sheet format. Access to the Anchor point is through the Table entry in the FeatureManager.

The Revision Table documents the history of a drawing. Locate the Revision Table Anchor point in the upper right corner of the sheet format. Address other tables in future projects.



Activity: Revision Table-Anchor Point

Delete the current Revision Table created in the AutoCad format.

345) Zoom in on the upper right corner of the Sheet Format.

346) Window-select the Revision Table.

347) Right-click Delete to remove all imported table lines and text.

REVISION HISTORY				
ZONE	REV	DESCRIPTION	DATE	APPROVED

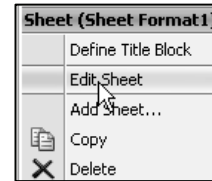
Return to the drawing sheet.

348) Right-click in the Graphics window.

349) Right-click Edit Sheet.

Fit the drawing to the Graphics window.

350) Press the f key.



Set the default layer.

351) Click None from the Layer text box.

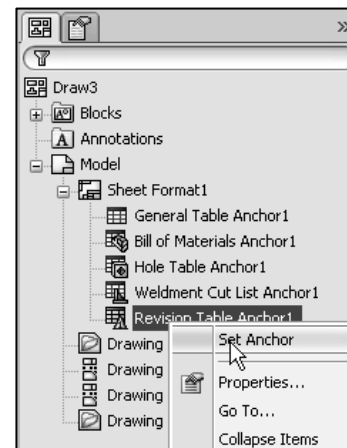


Set the Revision Table anchor point.

352) Expand Sheet Format1 in the Drawing FeatureManager.

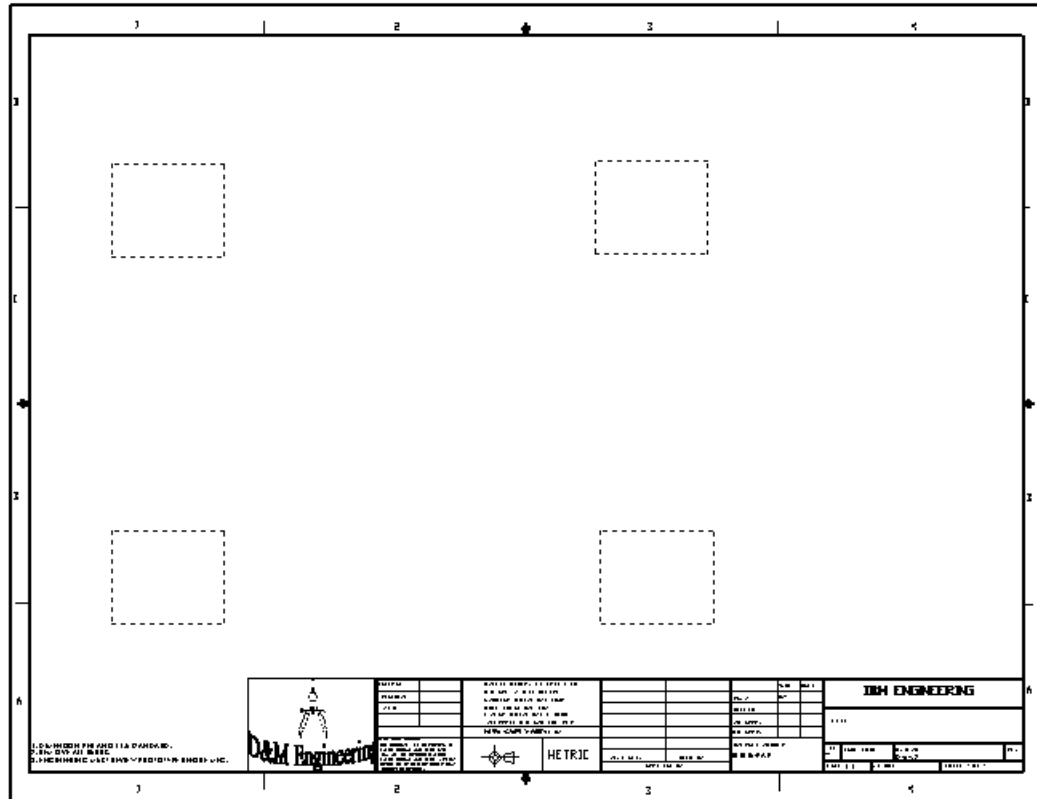
353) Right-click Revision Table Anchor1.

354) Click Set Anchor.



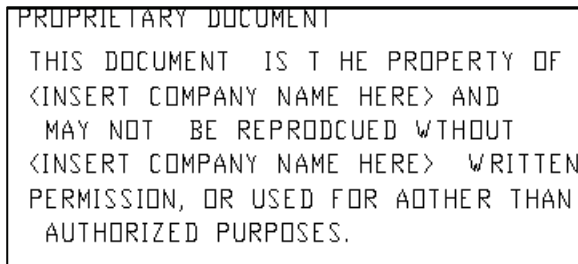
355) Click a **position** in the upper right corner of the Title block. You are in the Edit Sheet Format mode.

356) Click **Save** .



Two additional areas of the Title block require editing. Address this action as an exercise. The AutoCAD format utilized blocks in the original Proprietary Document statement.

The paragraph was imported. Letters are missing. Each line is a separate block.



Imported from Autocad, in block format.

Delete the old note. Retype the note in SolidWorks. Modify the text font to Century Gothic.

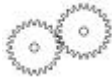
The Tolerance block is located in the Title block. The Tolerance block provides information to the manufacturer on the minimum and maximum variation for each dimension on the drawing. If a specific tolerance or note is provided on the drawing, the specific tolerance or note will override the information in the Tolerance block.

The design requirements and the manufacturing process determine the general tolerance values. The original Tolerance block lists values for inch parts. The Sheet format is developed for a metric part. Modify the LINEAR DIMENSIONS tolerance to +/- 0.2mm.

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Recreated in SolidWorks in paragraph format.

UNLESS OTHERWISE SPECIFIED
 DIM ARE IN MILLIMETERS
 ANGULAR DIMENSIONS ±0.3°
 HOLE LOCATIONS ±0.1
 LINEAR DIMENSIONS ± 0.2|
 INTERPRET DIM AND TOL PER
 ASME Y14.5M - 1994



Review

The Sheet format contains System Properties, Custom Properties and General Notes. SW-File Name, SW-Sheet Scale, SW-Current Sheet and SW-Total Sheets were Notes in the Sheet format linked to System Properties.

CompanyName, CONTRACT NUMBER, DrawnBy, and DrawnDate were Notes in the sheet format linked to the Drawing Custom Properties. DrawnBy and DrawingDate were copied from an existing default Sheet format.

Description, Revision, Material, Weight, Finish and TREATMENT were Notes in the sheet format linked to Custom Properties in the part and assembly.

You inserted a file for a company logo and General Notes from a text file. You utilized a table anchor point to position future Revision Tables in the Title block.



Create Sheet formats for different parts types. Example: sheet metal parts, plastic parts and high precision machined parts. Create Sheet formats for each category of the parts that are manufactured with unique sets of Notes and Custom Properties.



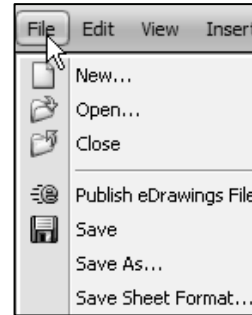
The illustrations are based on SolidWorks SP1.0. The illustrations may vary slightly per your SolidWorks release.

Review the Engineering Drawing Practices in your company as they relate to Custom Properties and Sheet formats. Create a table. List the following:

- Identify the required Sheet formats.
- Identify the required SolidWorks Properties to control the design process.
- Identify the required Custom Properties to control the design process.
- Determine the required values for each Property.
- Determine the correct location to define the Property: part, assembly, or drawing.

Save Sheet Format and Drawing Template

The Sheet format (*.slddrt) and Drawing template (*.drwdot) utilize two different commands to save the current drawing document (.drw). Utilize the File, Save Sheet Format option to create the Sheet format. Store Sheet formats in your MY-SHEETFORMATS folder.



Utilize the Save As command and select the Drawing template option to create the Drawing template. Combine the C-FORMAT Sheet format with the empty Drawing template. The C-FORMAT Sheet format is contained in every sheet of the drawing in the C-ANSI-MM Drawing template.

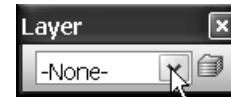
Save the Sheet format and Drawing templates in the Edit Sheet mode. Insert Views into the drawing in Edit Sheet mode. Views can't be displayed in the Edit Sheet Format mode. Set the layer option to None. The current layer is saved in the Drawing template.

Create a new drawing to test the Sheet format and the Drawing template. The Add Sheet option inserts a second sheet into the current drawing.

Activity: Save Sheet Format and Save Drawing Template

Set the Layer.

357) If needed, click **None** from the Layers toolbar.



Save the Sheet format.

358) Click **File**, **Save Sheet Format** from the Menu bar menu.

359) Enter **C-FORMAT** in the MY-SHEETFORMATS folder.

360) Click **Save**.



Close all documents.

361) Click **Windows**, **Close All** from the Menu bar menu.

Combine the C-SIZE-ANSI-MM-EMPTY template with the C-FORMAT Sheet format.

362) Click **New**  from the Menu bar menu.

363) Double-click **C-SIZE-ANSI-MM-EMPTY** from the MY-TEMPLETES folder.



364) Right-click in the **Graphics window**.

365) Click **Properties**.

366) Click the **Standard Sheet size** box.

367) Click **Browse**.

368) Double-click **C-FORMAT** from the MY-SHEETFORMATS folder.



369) Check **Display sheet format**.

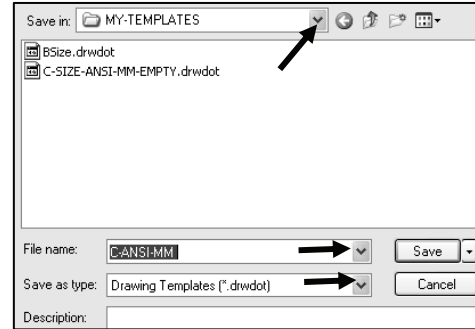
370) Click **OK** from the Sheet Properties dialog box.

Save the Drawing Template.

371) Click **Save As** from the Menu bar toolbar.

372) Select **Drawing Template (*.dwdot)** for Save as type.

373) Select **MY-TEMPLATES** for Save in folder.



374) Enter **C-ANSI-MM** for File name.

375) Click **Save**.

Close all documents.

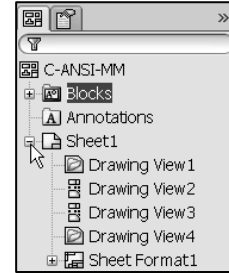
376) Click **Windows, Close All** from the Menu bar menu.

Verify the template.

377) Click **New** from the Menu bar toolbar.

378) Click the **MY-TEMPLATES** tab.

379) Double-click the **C-ANSI-MM** template. The C-ANSI-MM Drawing template is displayed with the Sheet format.



Add Sheet2

380) Right-click in **Sheet1**.

381) Click **Add Sheet**.

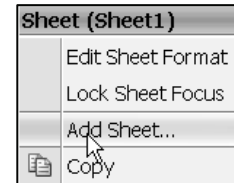


382) Click **No**. Sheet2 is displayed.

Save the drawing.

383) Click **Save As** from the Menu bar toolbar.


384) Select the **DRAWING-W-SOLIDWORKS-2010** folder.

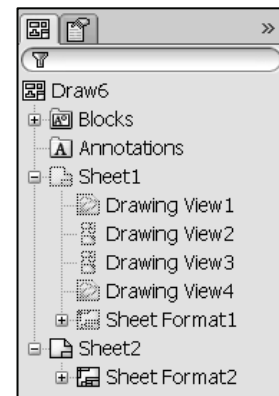



385) Enter **Draw6** for Filename.

Close all files.

386) Click **Windows, Close All** from the Menu bar menu.

 Drawing sheets are ordered as they are created. The names are displayed in the FeatureManager design tree and as Excel-style tabs at the bottom of the Graphics window. Activate a sheet by right-clicking in FeatureManager design tree and clicking Activate or click the tab name.



 *When adding a sheet, if the system can't locate the Sheet format, add the Sheet format in File Location under System Options or Browse to the correct Sheet format location.*



Reorder sheets by using the standard drag and drop technique with the tabs.



Rename a sheet by right-clicking the drawing tab, click Rename.


A (ANSI) Size Drawing Template

Create an A (ANSI) size Drawing template and an A (ANSI) size Sheet format. Text size for an A (ANSI) size drawing is the same as a C (ANSI) size drawing. Utilize the empty C (ANSI) size Drawing template to copy the Document Properties.

Create an A-ANSI-MM Drawing template. Add an A (ANSI) size Sheet format. SolidWorks copies the Document Properties in the C (ANSI) size Drawing template to the A-size Drawing template. The MY-SHEETFORMATS folder contains a predefined Sheet format named, A-FORMAT. The A-FORMAT contains geometry, text, and dimensions. The current layer is set to None. The Drawing template controls the units.

Activity: A (ANSI) Drawing Template

Create a new A (ANSI) Drawing template.

387) Click **New**  from the Menu bar toolbar.

388) Double-click **C-SIZE-ANSI-MM-EMPTY**.

389) Select **A (ANSI) Landscape** for Standard sheet size.

390) Uncheck **Display Sheet Format**.

391) Click **OK**.

392) Click and drag the **pre-determined views** into the Sheet boundary.

Fit the template to the Graphics window.

393) Press the **f** key.

Save the A-size Drawing Template.

394) Click **Save As** from the Menu bar toolbar.

395) Select **Drawing Templates** for Save as type.

396) **Browse** to the MY-TEMPLATES file folder.

397) Enter **A-SIZE-ANSI-MM-EMPTY** for File name.

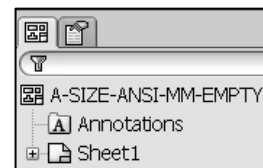
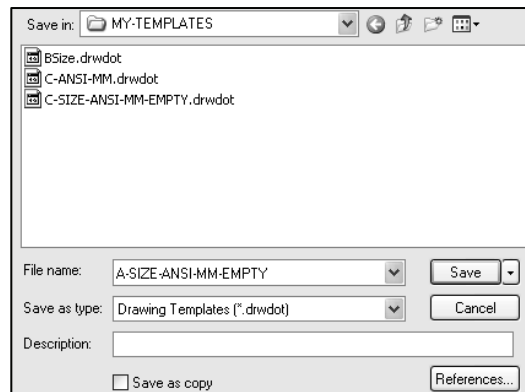
398) Click **Save**.

Load the Custom A-size Sheet format.

399) Right-click in the **Graphics window**.

400) Click **Properties**.

401) Click **Standard sheet size** for the Sheet format.



402) Click **Browse**.

403) Double-click **a-format** in the MY-SHEETFORMATS folder.

404) Check **Display sheet format**.

405) Click **OK**. Note: The current layer is set to None.

Save the new Drawing Template.

406) Click **Save As** from the Menu bar toolbar.

407) Select **Drawing Templates (*.drwdot)** for Save as type.

408) Select the **MY-TEMPLATES** file folder.

409) Enter **A-ANSI-MM**.

410) Click **Save**.

Close all documents.

411) Click **Windows, Close All** from the Menu bar toolbar.

Verify the template.

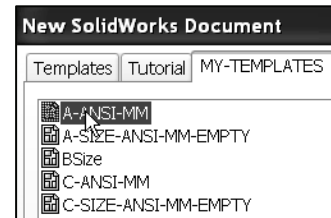
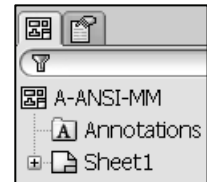
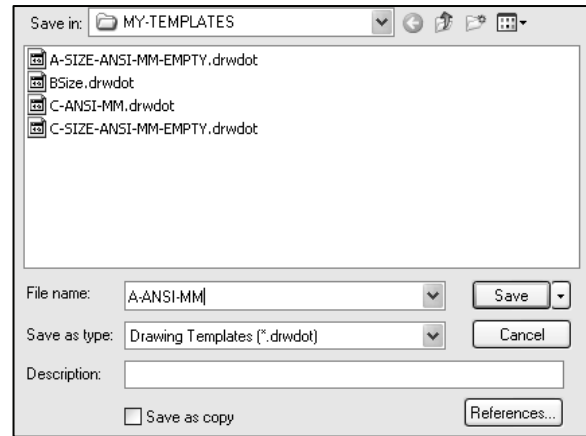
412) Click **New**  from the Menu bar toolbar.

413) Click the **MY-TEMPLATES** tab.

414) Double-click the **A-ANSI-MM** template. The new drawing is displayed in the Graphics window.

Close all documents.

415) Click **Windows, Close All** from the Menu bar menu.



Chapter Summary

In this chapter, you created a Custom C-size and A-size Drawing template and Sheet format. The Drawing template and Sheet format contained global drawing and detailing standards.

You obtained and applied drawing properties that reflect the ASME Y14 Engineering Drawing and Related Drawing Practices. You performed the task of importing an AutoCAD drawing to create and modify a Custom Sheet format.

The Sheet format utilized System Properties and User defined Custom Properties through Linked Notes. The A-ANSI-MM and C-ANSI-MM Drawing templates and A-FORMAT and C-FORMAT Sheet formats are use in the next Chapter.

Review additional topics in the chapter exercise. Example: Create drawing templates for inch Document Properties. Import a Pro\ENGINEER Sheet format into SolidWorks.

Chapter Terminology

ANSI: American National Standards Institute.

ASME: American Society of Mechanical Engineers. ASME is the publisher of the Y14 Engineering Drawing and Related Documentation Practices. ASME Y14.5M-1994 is a revision of ANSI Y14.5-1982.

CommandManager: The CommandManager is a Context-sensitive toolbar that dynamically updates based on the toolbar you want to access. By default, it has toolbars embedded in it based on the document type. When you click a tab below the Command Manager, it updates to display that toolbar. For example, if you click the **Sketches** tab, the Sketch toolbar is displayed.

Coordinate System: SolidWorks uses a coordinate system 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.

Cursor Feedback: The system feedback symbol indicates what you are selecting or what the system is expecting you to select. As you move the mouse pointer across your model, system feedback is provided.

DimXpertManager: The DimXpertManager lists the tolerance features defined by DimXpert for a part. It also displays DimXpert tools that you use to insert dimensions and tolerances into a part. You can import these dimensions and tolerances into drawings. DimXpert is not associative.

Drawing: A 2D representation of a 3D part or assembly. The extension for a SolidWorks drawing file name is .SLDDRW. Drawing refers to the SolidWorks module used to insert, add, and modify views in an engineering drawing.

Drawing Template: A document that is the foundation of a new drawing. The drawing template contains document properties and user-defined parameters such as sheet format. The extension for the drawing template filename is .DRWDOT.

Drawing Sheet: A page in a drawing document.

FeatureManager: The FeatureManager design tree located on the left side of the SolidWorks window provides an outline view of the active part, assembly, or drawing. This makes it easy to see how the model or assembly was constructed or to examine the various sheets and views in a drawing. The FeatureManager and the Graphics window are dynamically linked. You can select features, sketches, drawing views, and construction geometry in either pane.

Heads-up View toolbar: A transparent toolbar located at the top of the Graphic window.

Hidden Lines Removed (HLR): A view mode. All edges of the model that are not visible from the current view angle are removed from the display.

Hidden Lines Visible (HLV): A view mode. All edges of the model that are not visible from the current view angle are shown gray or dashed.

Import: The ability to open files from other software applications into a SolidWorks document. The A-size sheet format was created as an AutoCAD file and imported into SolidWorks.

Layers: Simplifies a drawing by combining dimensions, annotations, geometry and components. Properties such as: display, line style, and thickness are assigned to a named layer.

Menus: Menus provide access to the commands that the SolidWorks software offers. Menus are Context-sensitive and can be customized through a dialog box.

Model: 3D solid geometry in a part or assembly document. If a part or assembly document contains multiple configurations, each configuration is a separate model.

Mouse Buttons: The left, middle, and right mouse buttons have distinct meanings in SolidWorks. Use the middle mouse button to rotate and Zoom in/out on the part or assembly document.

OLE (Object Linking and Embedding): A Windows file format. A company logo or EXCEL spreadsheet placed inside a SolidWorks document are examples of OLE files.

Origin: The model origin is displayed in blue and represents the (0,0,0) coordinate of the model. When a sketch is active, a sketch origin is displayed in red and represents the (0,0,0) coordinate of the sketch. Dimensions and relations can be added to the model origin, but not to a sketch origin.

Part: A 3D object that consist of one or more features. A part inserted into an assembly is called a component. Insert part views, feature dimensions and annotations into 2D drawing. The extension for a SolidWorks part filename is .SLDPRT.

Plane: To create a sketch, choose a plane. Planes are flat and infinite. Planes are represented on the screen with visible edges.

Properties: Variables shared between documents through linked notes.

Sheet: A page in a drawing document.

Sheet Format: A document that contains the following: page size and orientation, standard text, borders, logos, and Title block information. Customize the Sheet format to save time. The extension for the Sheet format filename is .SLDDRT.

Sheet Properties: Sheet Properties display properties of the selected sheet. Sheet Properties define the following: Name of the Sheet, Sheet Scale, Type of Projection (First angle or Third angle), Sheet Format, Sheet Size, View label, and Datum label.

System Feedback: Feedback is provided by a symbol attached to the cursor arrow indicating your selection. As the cursor floats across the model, feedback is provided in the form of symbols riding next to the cursor.

System Options: System Options are stored in the registry of the computer. System Options are not part of the document. Changes to the System Options affect all current and future documents. There are hundreds of Systems Options.

Templates: Templates are part, drawing, and assembly documents that include user-defined parameters and are the basis for new documents.

Toolbars: The toolbar menus provide shortcuts enabling you to access the most frequently used commands. Toolbars are Context-sensitive and can be customized through a dialog box.

Questions

1. Name the drawing options defined in the Drawing template.
2. Name five drawing items that are contained in the Sheet format.
3. Identify the paper dimensions required for an A (ANSI) Landscape size horizontal drawing.
4. Identify the paper dimensions required for an A4 horizontal drawing.
5. Name the Size option you select in order to define a custom paper width and height.
6. Identify the primary type of projection utilized in a drawing in the United States.
7. Describe the steps to display and modify the Properties in a drawing sheet.
8. Identify the location of the stored System Options.
9. Name five Display Modes for drawing views.
10. True or False. SolidWorks Line Font Types define all ASME Y14.2 type and style of lines.
11. Identify all Dimensioning Standards Options supported by SolidWorks.
12. Identify 10 drawing items that are contained in a Title block.
13. SolidWorks Properties are saved with the _____ format.
14. The Drawing template ends with the SolidWorks file extension _____.
15. A Sheet format ends with the SolidWorks file extension _____.
16. An AutoCAD drawing ends with the file extension _____.
17. Describe the procedure to insert a picture into the Sheet format.
18. True or False. Custom Properties are defined only in the Drawing template.

Exercises

Notes for Exercise 2.1 through Exercise 2.3:

Create Drawing templates for both inch and metric units. ASME Y14.5M has different rules for English and Metric unit decimal display.

English decimal display: If a dimension value is less than 1in, no leading zero is displayed before the decimal point. See Table 1 for details.

Metric decimal display: If a dimension value is less than 1mm, a leading zero is displayed before the decimal point. See Table 1 for details.

Specify General Tolerances in the Title block. Specific tolerances are applied to an individual dimension.

Select ANSI for the SolidWorks Dimensioning Standard. Select inch or metric for Drawing units.

Table 1 Tolerance Display for INCH and METRIC DIMENSIONS (ASME Y14.5M)		
Display	Inch	Metric
Dimensions less than 1	.5	0.5
Unilateral Tolerance	1.417 ^{+0.005} _{-.000}	36 ⁰ _{-0.5}
Bilateral Tolerance	1.417 ^{+0.010} _{-.020}	36 ^{+0.25} _{-0.50}
Limit Tolerance	.571 .463	14.50 11.50

Exercise 2.1:

Create an A-size ANSI Drawing template using inch units. Use an A-FORMAT Sheet format. Create a C-size ANSI Drawing template using inch units. Use a C-FORMAT Sheet format.

The minimum ASME Y14.2M letter height for the Title block is displayed in Table 2. Create three new Layers named:

- DETAILS
- HIDE DIMS
- CNST DIMS (Construction Dimensions)

Create new Layers to display the CHAIN, PHANTOM, and STITCH lines.

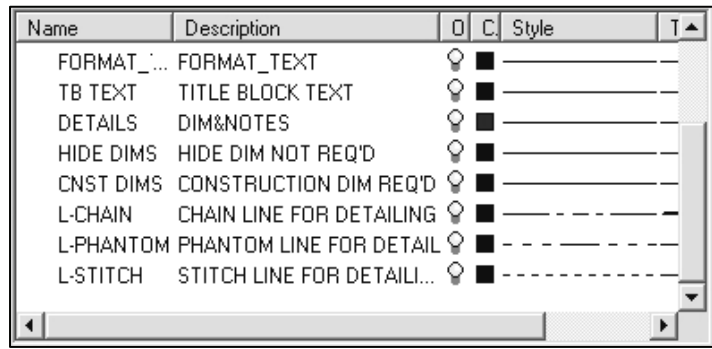


TABLE 2 Minimum Letter Height for TITLE BLOCK (ASME Y14.2M)	
Title Block Text	Letter Height (inches) for A, B, C Drawing Size
Drawing Title, Drawing Size, Cage Code, Drawing Number, Revision Letter	.12
Section and view letters	.24
Drawing block letters	.10
All other characters	.10

Exercise 2.2:

Create an A4(horizontal) ISO Drawing template. Use Document Properties to set the ISO dimension standard and millimeter units.

Exercise 2.3:

Modify the SolidWorks drawing template A4-ISO. Edit Sheet Format to include a new Sheet Metal & Weldment Tolerances box on the left hand side of the Sheet format, Figure EX2.3.

Display sketched end points to create new lines for the Tolerance box. Click Options, System Options, Sketch from the Menu bar toolbar. Check Display entity points. The endpoints are displayed for the Sketched lines.

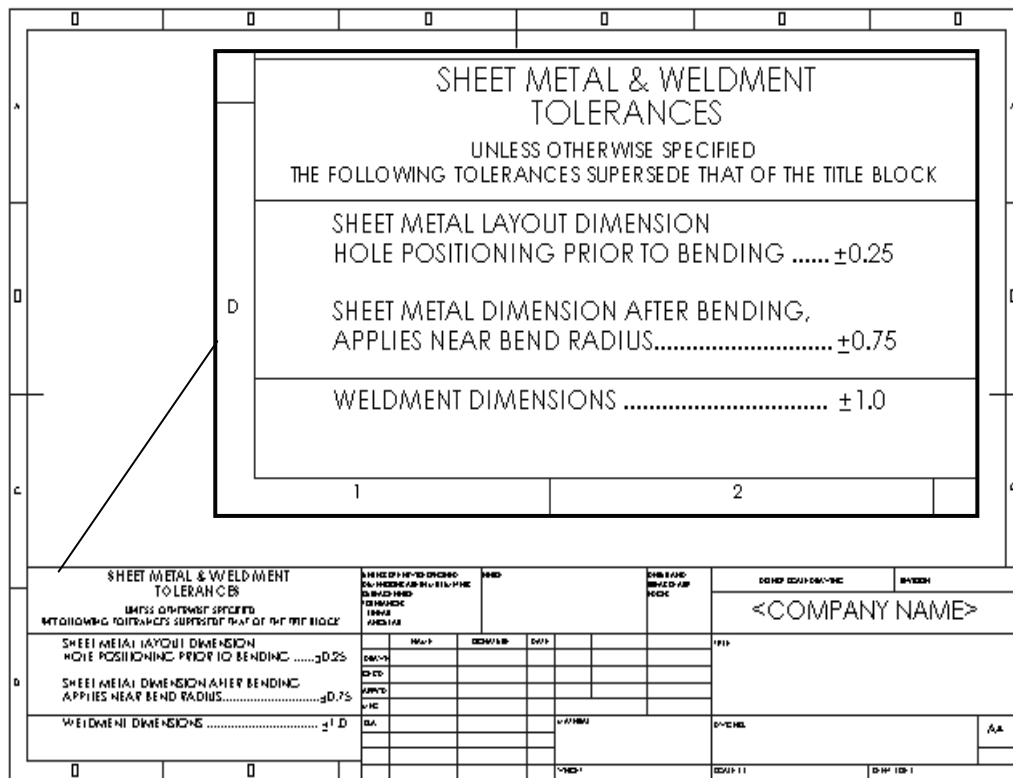


Figure EX2.3

SHEET METAL & WELDMENT TOLERANCES box

Courtesy of Ismeca, USA Inc. Vista, CA.

Exercise 2.4:

You are not required to have Pro/E to perform the following exercise. Your company uses SolidWorks and Pro/ENGINEER to manufacture sheet metal parts, Figure EX1.4. Import the empty A-size drawing format, FORMAT-A-PRO-E.DWG located in the DRAWING-W-SOLIDWORKS-2010 file folder. The document was exported from Pro/E as a DWG file. Save the Pro/E drawing format as a SolidWorks Sheet format.

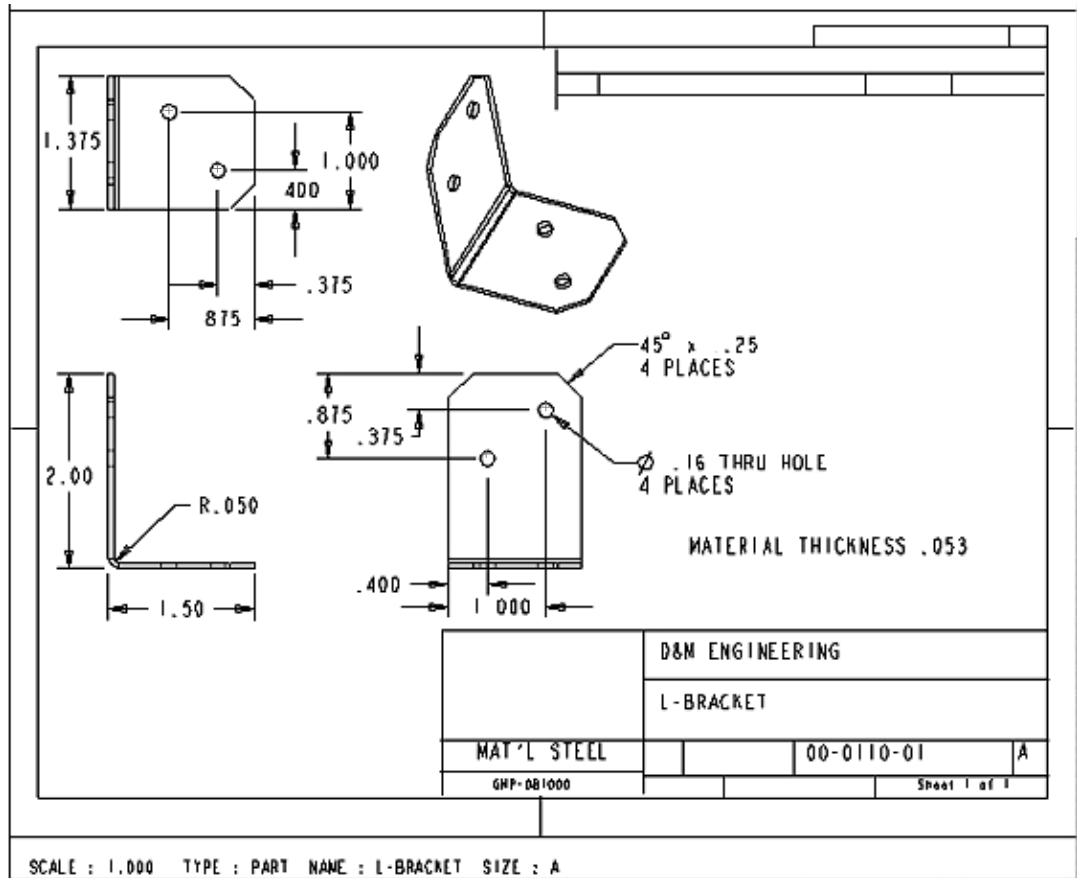


Figure EX2.4
Sheet Metal Strong Tie Reinforcing Bracket,
Courtesy of Simpson Strong Tie Corporation, CA, USA.

Exercise 2.5:

You are required to have AutoCAD to perform the following exercise. Your company uses SolidWorks and AutoCAD. Open an A-size Drawing template from AutoCAD. Review the Dimension Variables (DIMVARS) in AutoCAD. Record the DIMSTATUS for the following variables:

AutoCAD:	Function:
DIMTXSTY	Dimensioning Text Style
DIMASZ	Arrow size
DIMCEN	Center Mark size
DIMDEC	Decimal Places
DIMTDEC	Tolerance Decimal Places
DIMTXT	Text Height
DIMDLI	Space between dimension lines for Baseline dimensioning

Identify the corresponding values in SolidWorks Document Properties to contain the AutoCAD dimension variables. Utilize Help, Moving from AutoCAD to SolidWorks. Use CommandMap, Draw Toolbar, and Dimension Toolbar in this exercise.

Define Favorite dimension style settings for a particular dimension. Apply Favorite dimension styles to other dimensions on the drawing, part and assembly documents.



Early AutoCAD drawing formats contain fonts not supported in a Windows environment. These fonts imported into SolidWorks will be misaligned in the Sheet Format. Modify older AutoCAD formats to a True Type Font in SolidWorks.

For additional information on the transition between 2D AutoCAD and 3D SolidWorks, Use the Draw Toolbar option in SolidWorks help.

