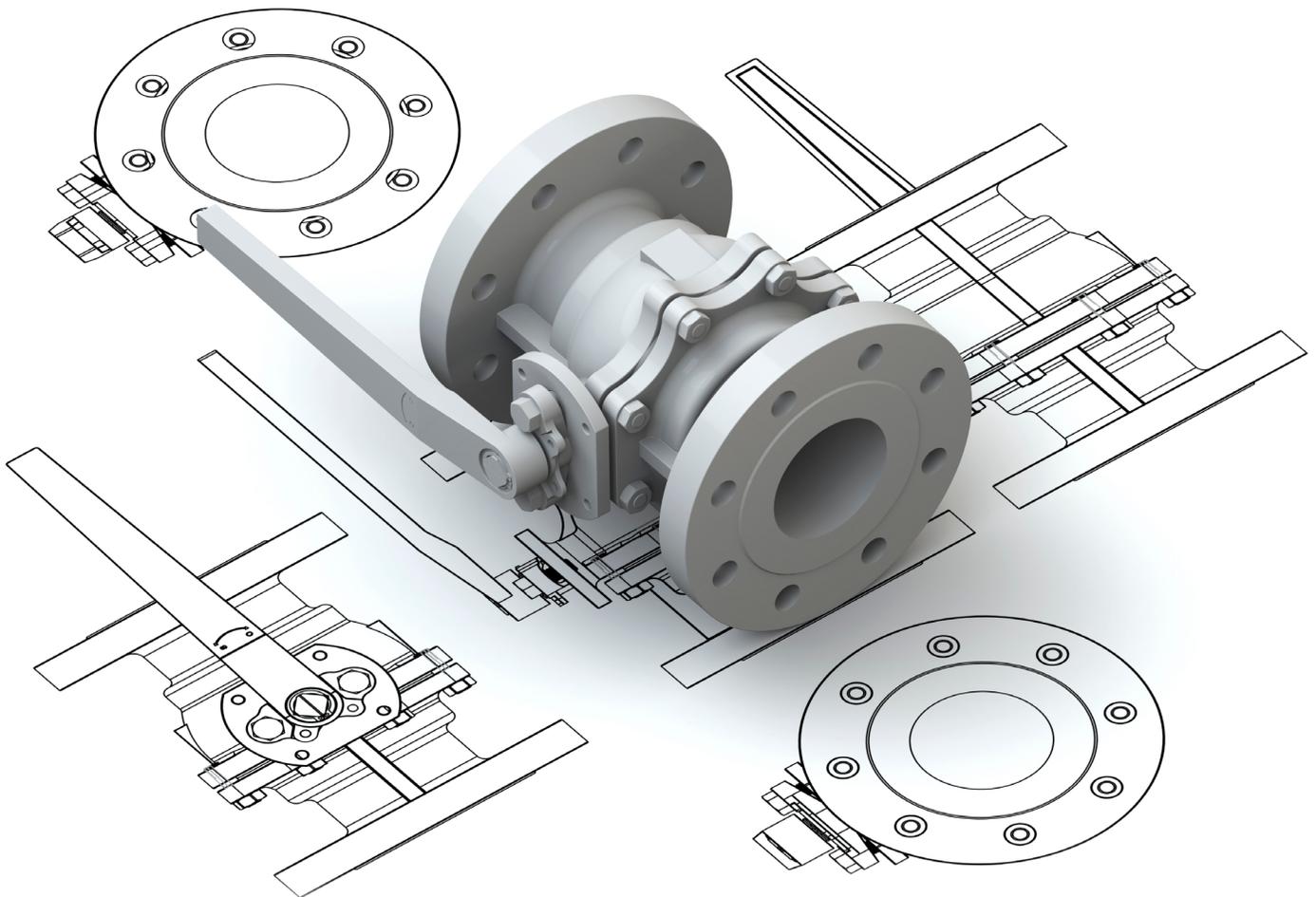


**Includes**  
Preparation for the  
CSWPA-DT certification

# Drawing and Detailing with **SOLIDWORKS** 2022

Referencing the ASME Y14.5 Engineering  
Drawing and Related Documentation Practices



David C. Planchard, CSWP,  
SolidWorks Accredited Educator



Better Textbooks. Lower Prices.  
[www.SDCpublications.com](http://www.SDCpublications.com)

Visit the following websites to learn more about this book:



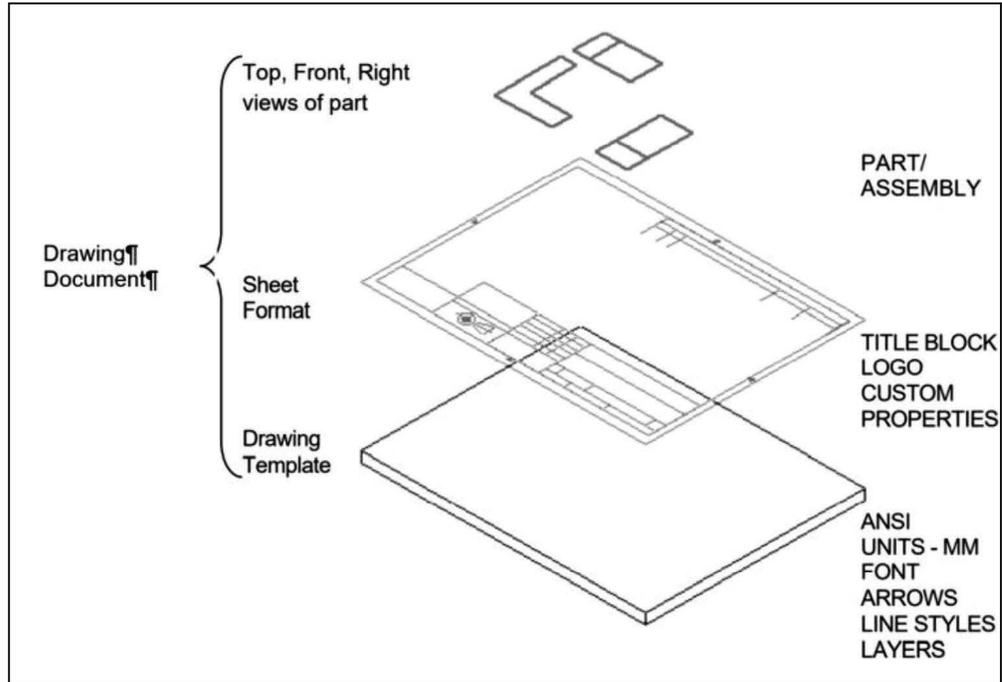
[amazon.com](https://www.amazon.com)

[Google books](https://books.google.com)

[BARNES & NOBLE](https://www.barnesandnoble.com)

# Chapter 5

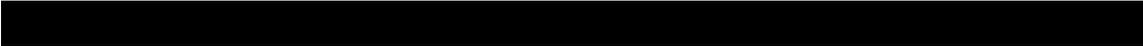
## Structure of a Drawing Document



Below are the desired outcomes and usage competencies based on the completion of Chapter 5.

Desired Outcomes:	Usage Competencies:
<ul style="list-style-type: none"> <li>• Drawing Templates:                             <ul style="list-style-type: none"> <li>○ C-(ANSI)-MM Landscape.</li> <li>○ A-(ANSI)-MM Landscape.</li> <li>○ C-(ANSI)-MM Landscape with Pre-defined views.</li> </ul> </li> </ul>	<ul style="list-style-type: none"> <li>• Understand the structure of a drawing document.</li> <li>• Work with System Options and Document Properties, which influence the drawing, Drawing template, and Sheet format.</li> </ul>
<ul style="list-style-type: none"> <li>• C-CUSTOM.</li> <li>• A-CUSTOM.</li> </ul>	<ul style="list-style-type: none"> <li>• Insert SOLIDWORKS System Properties and Custom Properties.</li> </ul>
<ul style="list-style-type: none"> <li>• New file location for drawing templates and Sheet format.</li> </ul>	<ul style="list-style-type: none"> <li>• Design a Drawing template without a Sheet format file.</li> <li>• Define the file location for custom document templates.</li> </ul>

Notes:



## Chapter 5 - Structure of a Drawing Document

### Chapter Objective

The purpose of this chapter is to provide an understanding of how SOLIDWORKS Drawing documents and templates are created and used. Create an awareness on the structure of a Drawing document. Provide a general knowledge of the ASME Y14 2009 Engineering Drawing and Related Documentation Practices. On the completion of this chapter, you will be able to:

- Identify elements which construct a Drawing document.
- Successfully develop a Drawing document.
- 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 a C-size (ANSI) Landscape Drawing Template and Sheet format.
- Create a A-size (ANSI) Landscape Drawing Template and Sheet format.
- Propagate the settings to the Drawing sizes.
- Develop Linked Notes to SOLIDWORKS Properties and Custom Properties in the Sheet format.
- Insert a company logo and Third Angle icon with a relation in the Title block.
- Understand an Annotation Links Error.
- Create a Drawing Template without a Sheet format file and define its File Location in System Options.
- Create a sample model and drawing document with Custom Properties.
- Understand where the property information is being populated from: SOLIDWORKS Special Properties, Model Custom Properties, and Drawing Custom Properties.
- Save the Sheet format and define the File location in System Options.



During the initial SOLIDWORKS installation, you are requested to select either the ISO or ANSI drafting standard. ISO is typically a European drafting standard and uses First Angle Projection. The book is written using the ANSI (US) overall drafting standard and Third Angle Projection for drawings.



Download all needed model files from the SDC Publication website (<https://www.sdcpublications.com/Downloads/978-1-63057-485-7>).

## Engineering Drawing and Related Documentation Practices

Drawing Templates in this section are based on the American Society of Mechanical Engineers ASME Y14 2009 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 2009 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 Y14.1-2009	Decimal Inch Drawing Sheet Size and Format	ANSI Y14.1
ASME Y14.1M-2009	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-2009	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)

The book presents a portion of the ASME Y14 American National Standard for Engineering Drawing and Related Documentation Practices. Information presented in Chapters 5 - 9 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 2009.
- ISO or other International drawing standards.
- Older ANSI standards.
- Military standards.



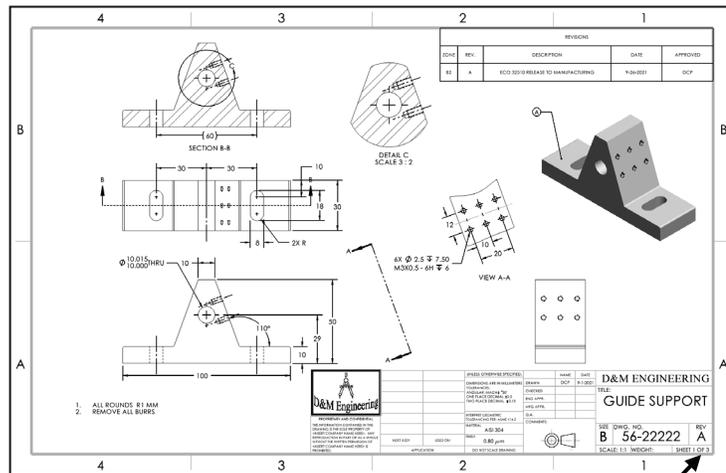
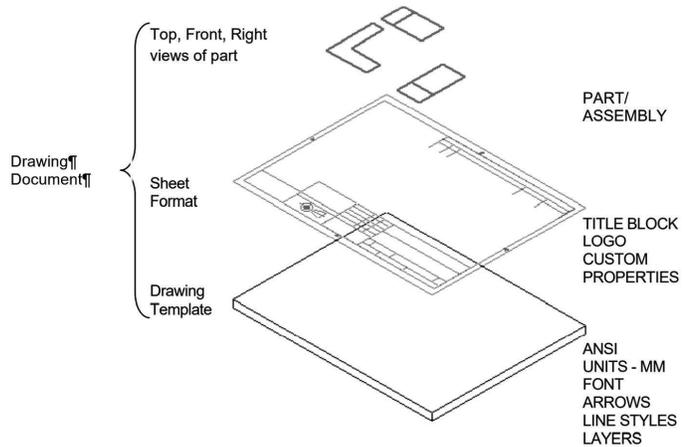
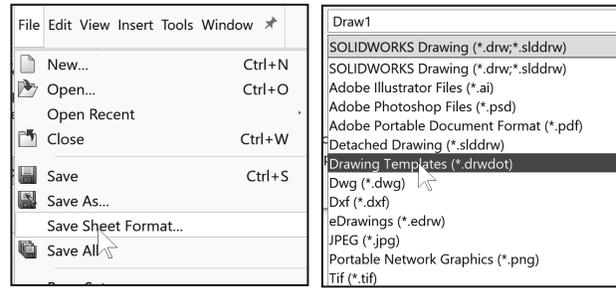
There is also the “We’ve always done it this way” drawing standard or “Go ask the Drafting supervisor” drawing standard.

### Structure of a Drawing Document

There are 2 primary pieces to a new Drawing document. These elements work together to produce the paper sheet, title block and border for drawings.

The Drawing document, Drawing Sheet and Sheet format each contain different properties and pieces of information.

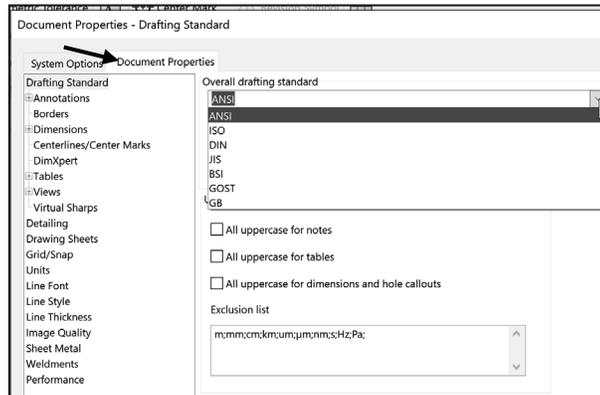
- Drawing Document:** Is the entire Drawing file. Drawing Templates are the files use to begin a new Drawing document. Drawing Templates can include both a Drawing sheet and a Sheet format file.
- Drawing Sheet:** Drawing views and Annotations, Sheet Format/Size, Sheet Scale, Type of Projection, Datum Label, etc.
- Sheet Format File:** The Sheet format file is incorporated into the Drawing Template. The Sheet format contains the border, Title block information, revision block information, company name, and or company logo information, Table Anchors, Custom Properties, and SOLIDWORKS Properties. Custom Properties and SOLIDWORKS Properties are shared values between documents.



### Drawing Document

The Drawing document contains Document Properties. These include important settings such as units, drafting standard, dimensions, annotations, line font, etc.

The default Documents Properties of a drawing are determined by the settings saved with the drawing template.

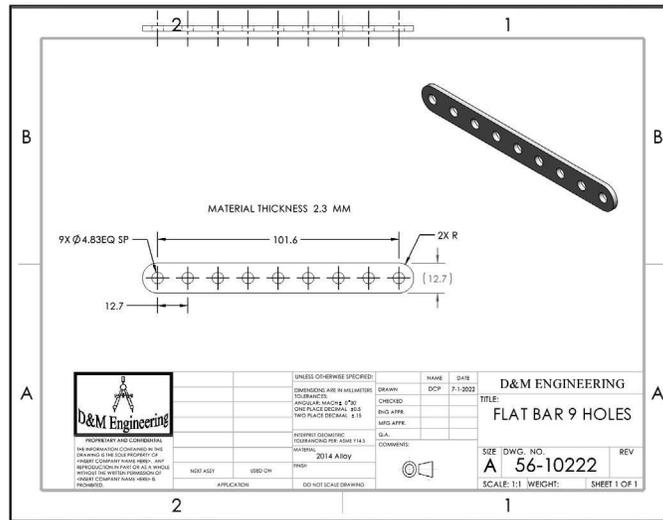


### Drawing Sheet and Format

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

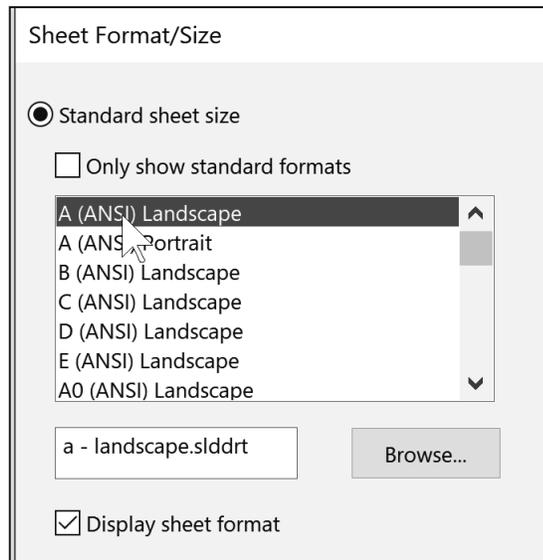
1. ASME Y14.1-2009 Decimal Inch Drawing Sheet Size and Format.
2. ASME Y14.1M-2009 Metric Drawing Sheet size.

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 Y14.1-2009 and ASME Y14.1M-2009 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 Y14.1-2009 Decimal Inch Drawing and ASME Y14.1-2009 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

### Drawing Sheet

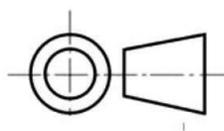
The drawing sheet represents the paper size and is the active area of the drawing when performing detailing tasks (views, dimensions, annotations, tables, display modes, centerlines, centermarks, etc.). Each drawing sheet includes properties that can be modified through the Sheet Properties dialog box.



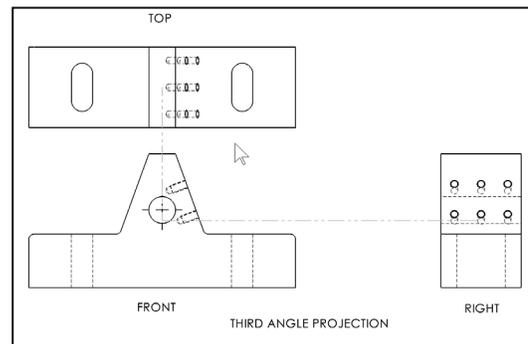
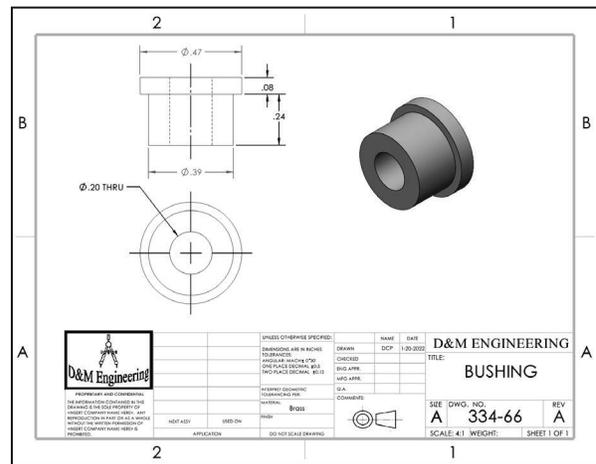
During the initial SOLIDWORKS installation, you are requested to select either the ISO or ANSI drafting standard. ISO is typically a European drafting standard and uses First Angle Projection.

The book is written using the ANSI (US) overall drafting standard and Third Angle Projection for all chapter drawings.

In the next section create a new drawing with the default SW Drawing Template. Utilize C size (ANSI) Landscape paper.



Third Angle Projection icon





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: Create a Drawing Document. Set a few Document Properties. Explore Sheet Properties.**

Create a new SOLIDWORKS Drawing. Set Overall drafting standard, Units, and Precision.

1. Click **New**  from the Menu bar toolbar.

2. Double-click **Drawing** from the Templates tab.

Create a C (ANSI) Landscape Sheet Format/Size drawing.

3. Select **C (ANSI) Landscape** from the Standard sheet size drop-down menu. View your options. A C (ANSI) Landscape drawing is either 22"x 17" or 279.40mm x 215.90mm depending on units. Use the default SW Sheet format (c - landscape.slddrt).

4. Click **OK** from the Sheet Format/Size dialog box.

Exit the Model View PropertyManager.

5. Click **Cancel**  from the Model View PropertyManager. Draw1 is the default drawing name. Sheet1 is the default first Sheet name. CommandManager, FeatureManager and Task Pane tabs will vary depending on system setup and Add-ins.

Set Document Properties for the Drawing Document. Set Overall drafting standard, Units, and Precision. Remember, Drawing Templates are the files used to begin a new Drawing Document. Drawing Templates can include both a drawing sheet and a sheet format file.

6. Click **Options**  from the Main menu.

7. Click the **Document Properties** tab.

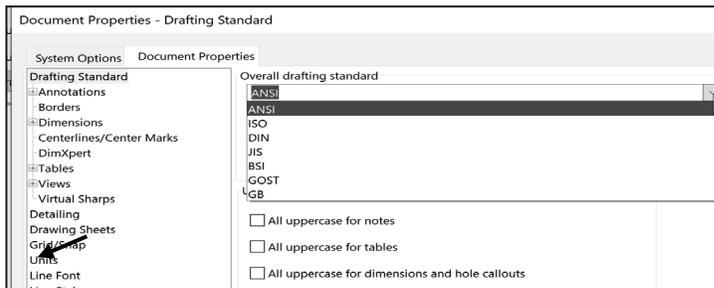
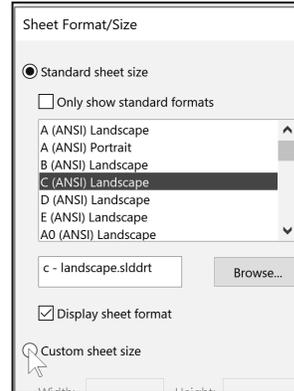
8. Select **ANSI** from the Overall drafting standard drop-down menu.

9. **View** your options.

10. **Explore** the various folders.



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**: Gosdarstuenny State Standard (Russian), **GB**: Guo Biao (Chinese).



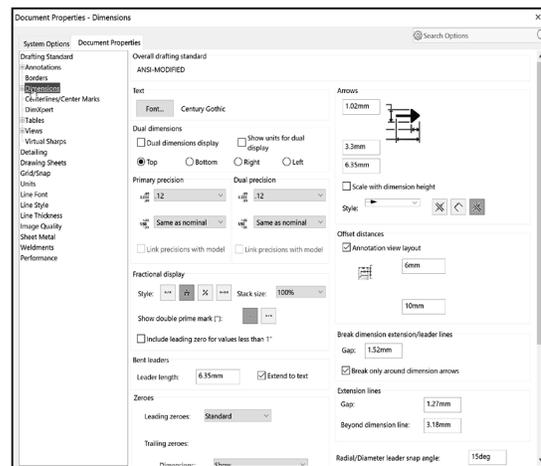
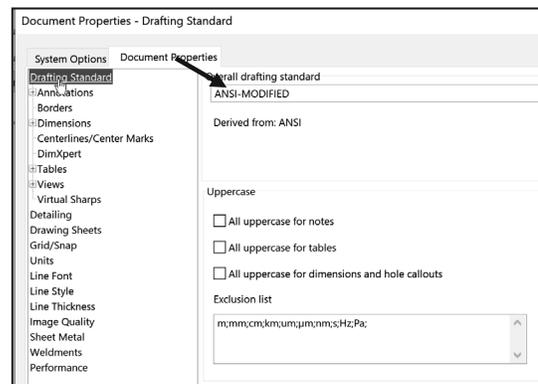
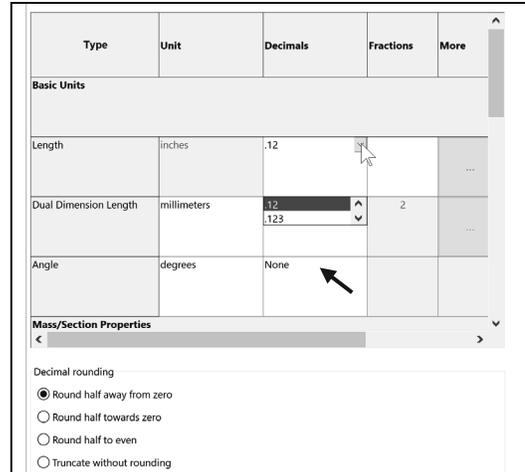
11. Click the **Units** folder.
12. **View** your options.
13. Click **MMGS** (millimeter, gram, second) Unit system.
14. Select **two places** for Length precision.
15. Select **None** for Angle precision.
16. Click the **Drafting Standard** folder.
17. **View** the options.

If you modify a document property from an Overall drafting standard, a modify message (ANSI-MODIFIED) is displayed as illustrated.

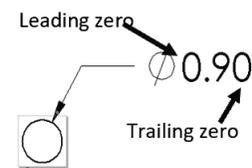
The Uppercase section controls the following options.

- **All uppercase for notes:** Set the default case as all uppercase for all new notes and balloons in a document.
- **All uppercase for tables:** Set the default as all uppercase for all new tables in a document.
- **All uppercase for dimensions and hole callouts:** Set the default as all uppercase for all new dimensions and hole callouts in a document.
- **Exclusion list:** List strings that you specify to exclude from automatic capitalization. To exclude strings such as units of measurement from capitalization, enter them in the box, separated by semicolons.

18. Click the **Dimension** folder.
19. **View** the options. Location to display Dual dimensions, set arrow head size, Leading zeroes, Trailing zeroes and more.



 Secondary units are displayed in parenthesis. Drawing dimensions are associated with the model. Use custom properties in the model and drawing to keep the association between documents.



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

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	.10mm	2.5mm
All other characters inside the Sheet boundary. Corresponds to the SW Dimension and Note font.	.12in.	3mm	.12in	3mm

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.

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

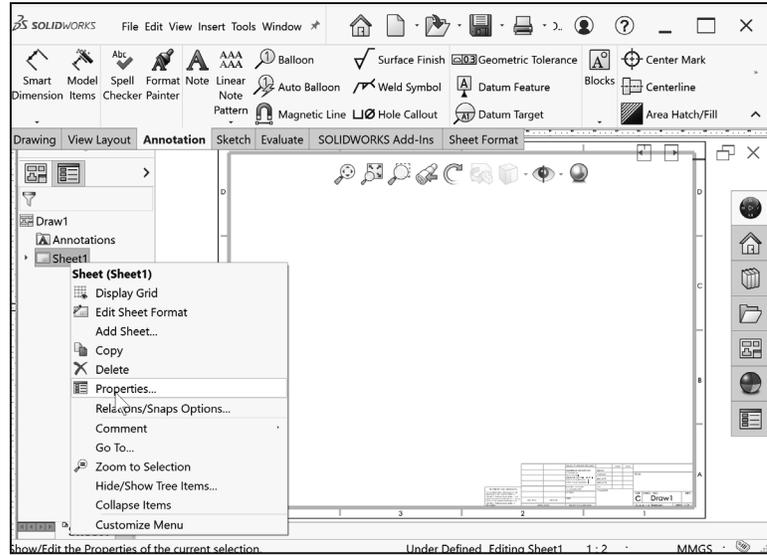
The ASME Y14.2M-1992(R1998) standard addresses the type and style of lines used in engineering drawings. Combine different Line Styles and use drawing layers to achieve the following types of ASME lines as illustrated.

Return to Sheet1.

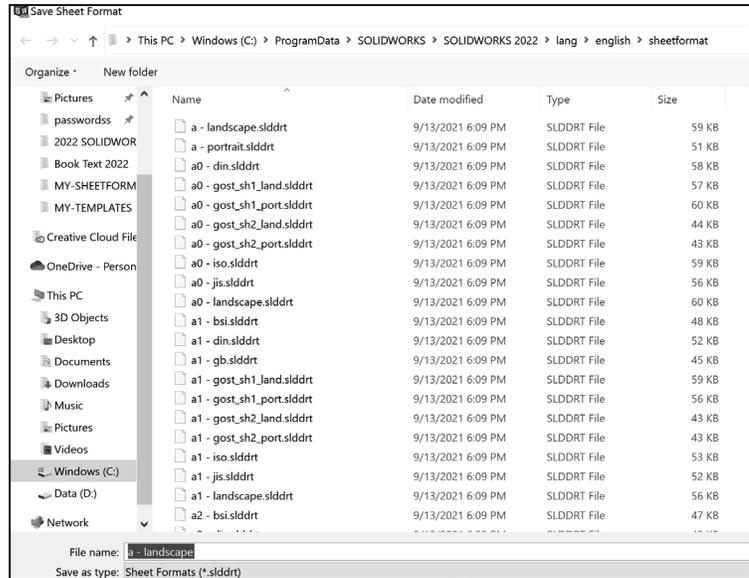
20. Click **OK**.

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.		Solid	Thick "Normal"
Hidden line displays the hidden edges or contours of a part.		Dashed	Thin
Section lining displays the cut surface of a part assembly in a section view.		Solid	Thin Different Hatch patterns relate to different materials
Center line displays the axes of center planes of symmetrical parts/features.		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.		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.		Phantom Solid	Thick Thick, "Normal"

Draw1 is the default drawing name. Sheet1 is the default first Sheet name. CommandManager, FeatureManager and Task Pane tabs will vary depending on system setup and Add-ins.



The Standard Sheet formats are located in the C:\ProgramData\SOLIDWORKS\SOLIDWORKS 2022\lang\english\sheetformat in a non-network system.



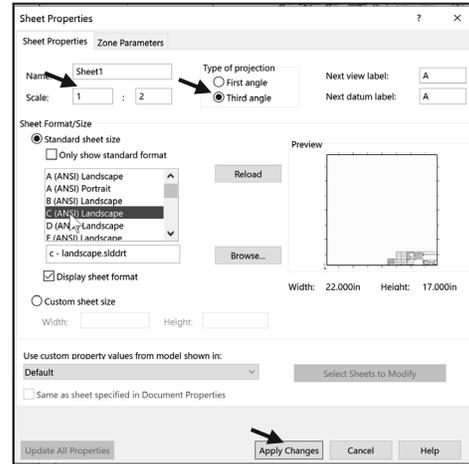


Utilize the Save As option to save a custom Drawing Template. Always select the Save as type option first, then select the Save in folder to avoid saving in default SOLIDWORKS installation directories.



Drawing Templates include at least one drawing sheet. Each drawing sheet has properties that are stored with the document template. These properties include a name for the sheet, default sheet scale, type of projection, and whether or not a sheet format is applied to the sheet.

Explore Sheet Properties. Set Third angle projection, C (ANSI) Landscape, and Sheet Scale.



21. Right click **Sheet1**.
22. Click **Properties**. The Sheet Properties dialog box is displayed. The default tab is Sheet Properties. The Zone Parameters tab provides the ability to set Zone Size and Margins.
23. Click **C (ANSI) Landscape** for Sheet Format/Size.
24. Click **Third angle** for Type of projection
25. Set Scale: **1:2**.
26. Click **Apply Changes** to close the dialog box. Apply Changes is active if you changed any of the default settings. Drawing1 is displayed.

## Sheet Format

Custom property data can be saved with a Sheet format to ensure the proper properties are created in the drawing document. Sheet formats exist as a separate files (\*.slddrt).

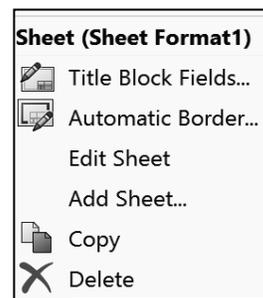
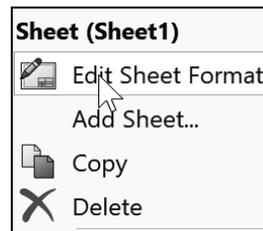
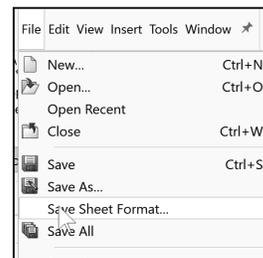
The Sheet format is NOT accessible when performing detailing tasks on the drawing sheet.

There are two major design modes used to develop a drawing:

- Edit Sheet Format and Edit Sheet.

**Edit Sheet Format** mode provides the ability to:

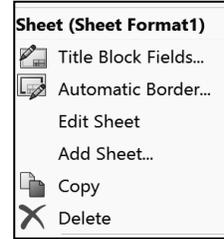
- Change the Title block size and text headings.
- Incorporate a Company logo.
- Define the Zone Tag Editor.
- Add Custom Properties, text and more.



Edit Sheet mode provides the ability to:

- Add or modify views.
- Add or modify dimensions.
- Add or modify notes and more.

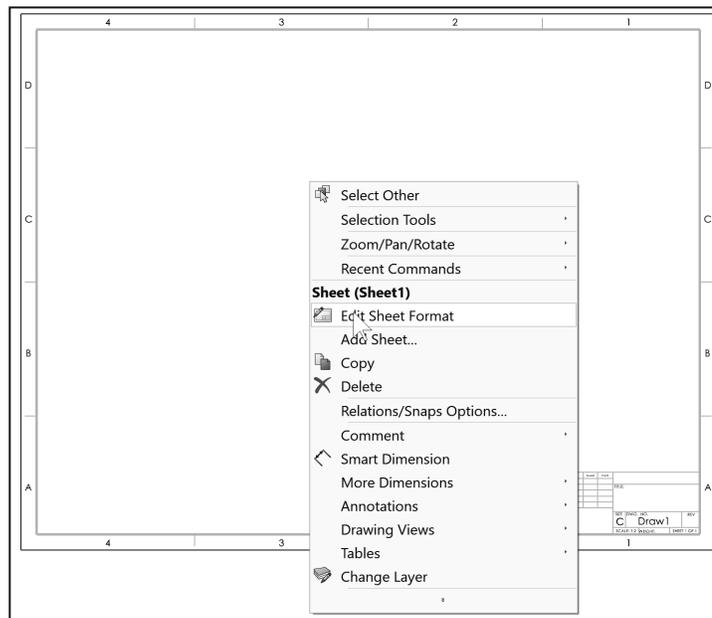
In the next section apply the Edit Sheet Format and Edit Sheet command.



**Activity: Apply the Edit Sheet Format and Edit Sheet Command**

Access Sheet Format1. Enter a Custom Property "\$PRP:"DrawnBy". Use the Edit Sheet Format and Edit Sheet command.

27. Right-click in the empty Sheet1 document.
28. Click Edit Sheet Format.
29. View Sheet Format1 and the links.
30. Zoom in on the Title block.
31. Hover over an empty block as illustrated. In this case the DRAWNBy box.



	NAME	DATE
DRAWN		
CHECKED		
ENG APPR.	\$PRP:"DrawnBy"	

This displays the Custom Notes Linked to Properties for Sheet Format1.



The format of notes linked to properties can be explored by hovering the cursor over the center of the title block. When editing a Sheet format, notes with links are displayed in blue. Notes that are static are displayed in black. You will address them later in the chapter.

PROPRIETARY AND CONFIDENTIAL THE INFORMATION CONTAINED IN THIS DRAWING IS THE SOLE PROPERTY OF <INSERT COMPANY NAME HERE>. ANY REPRODUCTION IN PART OR AS A WHOLE WITHOUT THE WRITTEN PERMISSION OF <INSERT COMPANY NAME HERE> IS PROHIBITED.			UNLESS OTHERWISE SPECIFIED:		NAME	DATE	TITLE: \$PRPSHEET:{Description}	
			DIMENSIONS ARE IN INCHES TOLERANCES: FRACTIONAL ± ANGULAR: MACH ± BEND ± TWO PLACE DECIMAL ± THREE PLACE DECIMAL ±	DRAWN				SIZE DWG. NO. REV C Draw1
			INTERPRET GEOMETRIC TOLERANCING PER:	CHECKED				
			MATERIAL \$PRPSHEET:{Material} FINISH \$PRPSHEET:{Finish}	ENG APPR.				
	NEXT ASSY	USED ON	APPLICATION	MFG APPR.				
			DO NOT SCALE DRAWING	Q.A.				
				COMMENTS:				

Enter a Custom Note Linked to Properties, "DrawnBy".

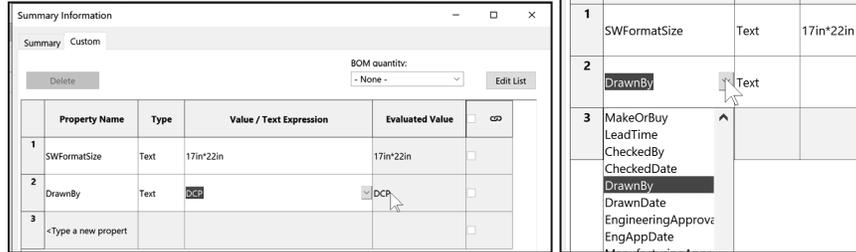
32. Click **File, Properties** from the Main menu. The Summary Information dialog box is displayed.

33. Click the **Custom** tab.

34. Click **DrawnBy** from the drop-down menu as illustrated.

35. **View** the options.

36. Click inside the **Value / Text Expression** box.



Enter your initials.

37. Example: **DCP**.

38. Click inside the **Evaluated Value** box.

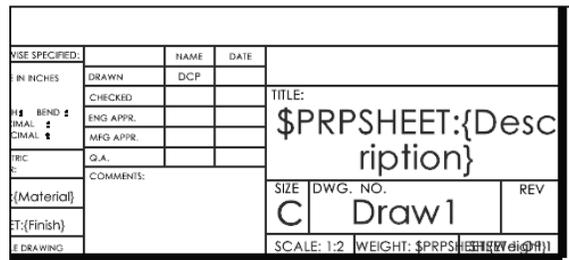
39. **View** the results.

Return to Sheet Format1.

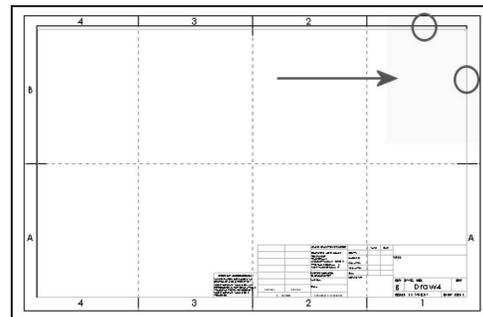
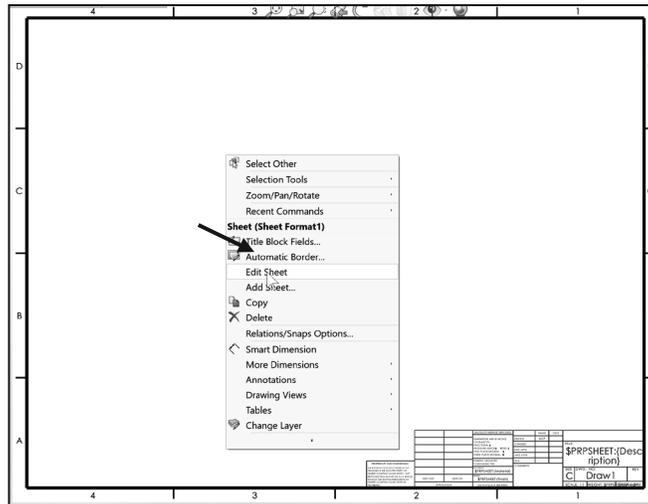
40. Click **OK** return to Sheet Format1.

Return to Sheet1.

41. Right-click **Edit Sheet**.



The Automatic Border tool lets you control every aspect of a sheet format's border, including zone layout and border size. Using the Automatic Border tool, borders and zones automatically update to match changes in the Zone Parameters tab of the Sheet Properties dialog box without having to manually edit the sheet format. You can also include Margin Mask areas where formatting elements such as labels and dividers are not shown. This is helpful when you want to mask an area on a sheet for notes.

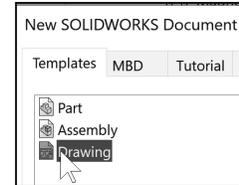
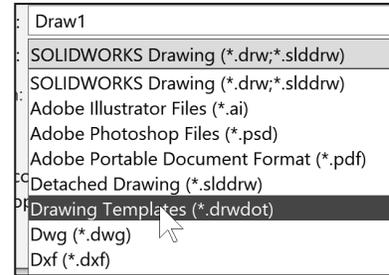


### Understand Drawing Templates

Drawing Templates are the files used to begin a new drawing document. They have a file extension of (\*.drwdot).

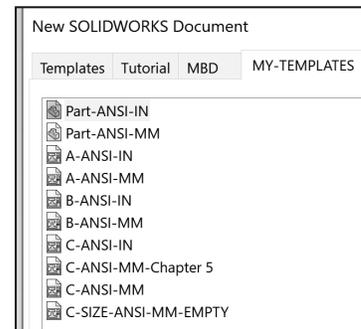
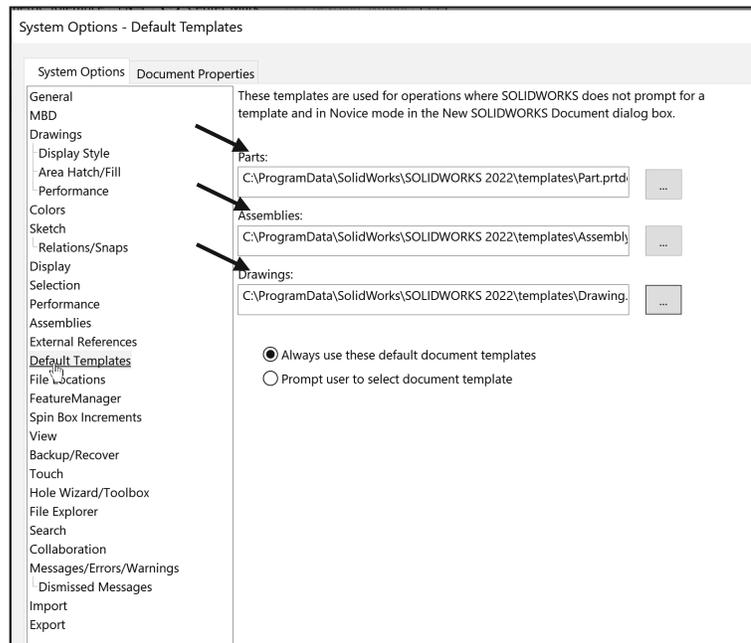
SOLIDWORKS starts with a default Drawing Template. The default Drawing Template is located in the C:\ProgramData\SOLIDWORKS\SOLIDWORKS 2022\templates folder on a non-network system. Note: the option “Prompt user to select document template”.

The Drawing Template can be displayed with or without the Sheet format. Combine the Sheet format with the Drawing Template to create a custom Drawing Template.

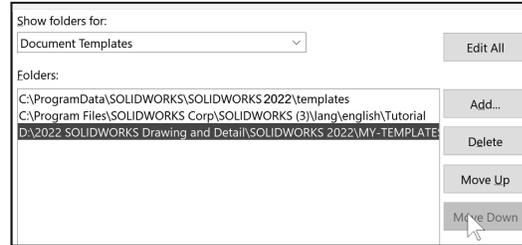


 Utilize the Save As option to save a custom Drawing Template. Always select the Save as type option first, then select the Save in folder to avoid saving in the default SOLIDWORKS installation directories.

Specify folders to search for different types of documents. Folders are searched in the order in which they are listed. The illustration displays a new MY-TEMPLATES tab which was created and is displayed in the Advanced New SOLIDWORKS Document dialog box.



The MY-TEMPLATES tab will not be displayed in the Advanced New SOLIDWORKS Document dialog box if the folder is empty. The MY-TEMPLATES tab will not be displayed if you do not address System Options - File Locations for the folder.



In the next section, save the Sheet format for Sheet1 to the SOLIDWORKS 2022/MY-SHEETFORMATS folder.

**Activity: Save Sheet Format. Create a Custom Sheet format.**

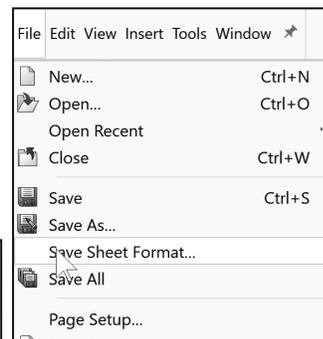
Save the Sheet Format. Use the SOLIDWORKS 2022/MY SHEETFORMATS folder.

42. Click **File, Save Sheet Format** from the Menu bar. The Save Sheet Format dialog box is displayed. The default SW location folder is displayed.

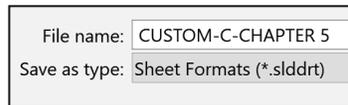
43. Select **SOLIDWORKS 2022/MY-SHEETFORMATS** for the Save in folder.

44. Enter **CUSTOM-C-CHAPTER 5** for File name.

45. Click **Save** from the Save Sheet Format dialog box.



In the next section, save the Drawing Template to the MY-TEMPLATES folder. Utilize the



Save As  option to save a

custom Drawing Template. Always select the Save As type option first, then select the Save in folder to avoid saving in the default SOLIDWORKS installation directories. The MY-TEMPLATES tab will not be displayed if you do not address System Options - File Locations for the folder. We will address this later in the book.

**Activity: Save a Custom Drawing Template with a Custom Sheet Format.**

Save the Custom Drawing Template with the Custom Sheet format.

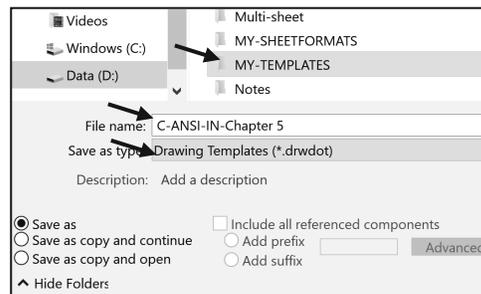
46. Click **Save As**  from the Main menu.

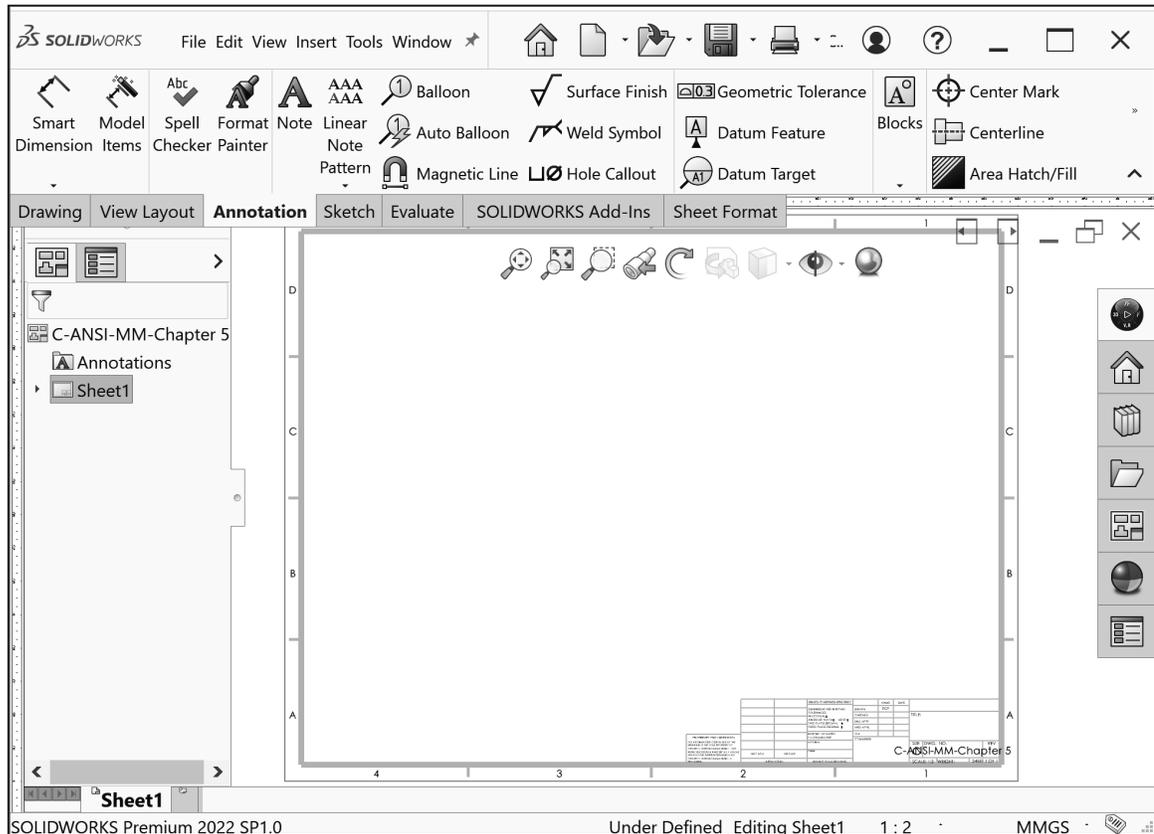
47. Click **Drawing Templates (\*.drwdot)** from the Save as type box. The default SW location templates folder is displayed.

48. Select **SOLIDWORKS 2022/MY-TEMPLATES** for the Save in folder. Note: These folders were downloaded in Chapter 4.

49. Enter **C-ANSI-MM-Chapter 5** for File name.

50. Click **Save**.





### Why are Drawings Structured this way?

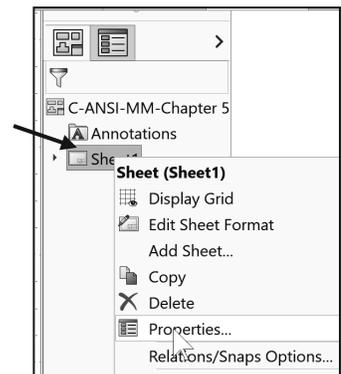
Sheet formats exist as separate, external files so that the Sheet size, title block and border information can easily be changed. For instance, if you begin a detailing project on an C-size sheet, but later decide A-size would be a better fit, you can simply modify the sheet properties to select a new format. If the Sheet format was a fixed entity in a Drawing Template you would have to copy your views to a new template to make this change.

In the next section, modify the Sheet format to A (ANSI) Portrait.

**Activity: Modify the Sheet Format size.**

Access Sheet Properties. Modify the Sheet format from C (ANSI) Landscape to A (ANSI) Portrait.

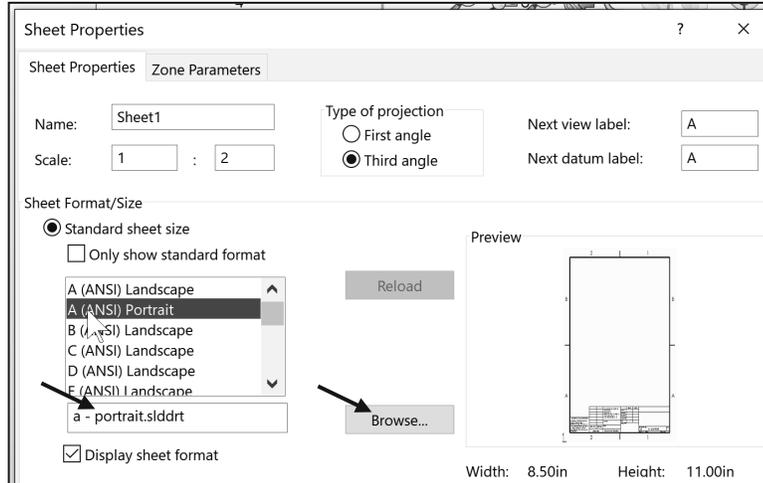
- 51. Right-Click **Sheet1** from the Drawing PropertyManager.
- 52. Click **Properties**. The Sheet Properties dialog box is displayed.



53. Click **A (ANSI) Portrait** as illustrated. Note the Sheet size: 8.50in x 11.00in.



Select a standard sheet size, (SOLIDWORKS installation directories) or click Browse and locate a Custom Sheet format file. Select Only show standard formats to display Sheet formats for the current drafting standard only. Otherwise, all formats from all standards are listed.



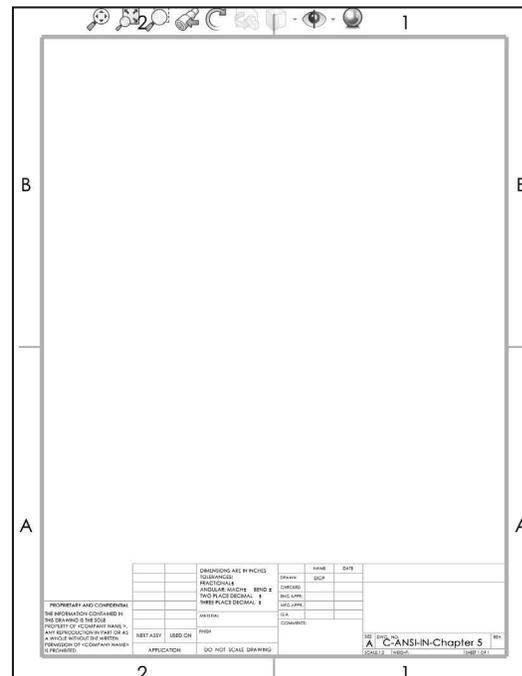
54. Click **Apply Changes**.

55. **View** the results. The Sheet size is updated and the new Sheet format is displayed.

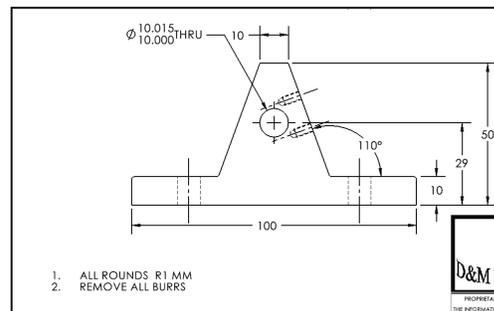
**Sheet Format Features.**

When editing or modifying the Sheet format, the following features of the Sheet format file are available:

**Title Block Sketch:** The Title block area is made up of sketch lines. They can be modified and constrained in the same way as sketch lines within a model. Standard formats contain Title blocks. The Title block in the A (ANSI) Portrait Sheet format contains the following lines and text as illustrated.

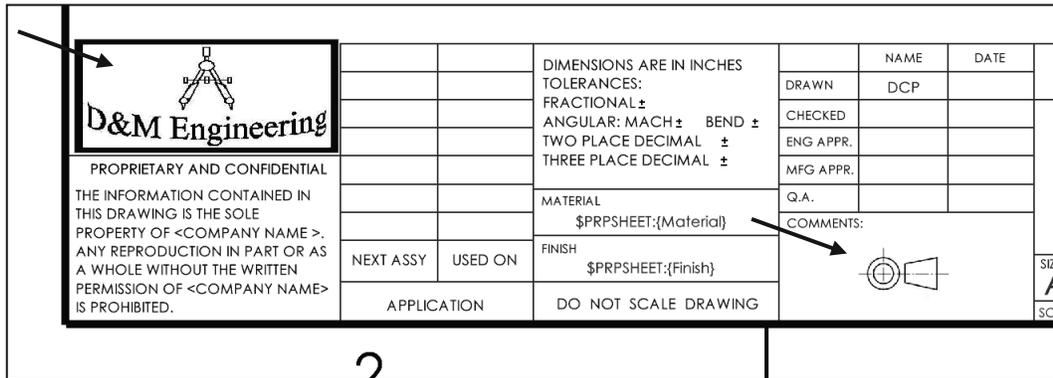
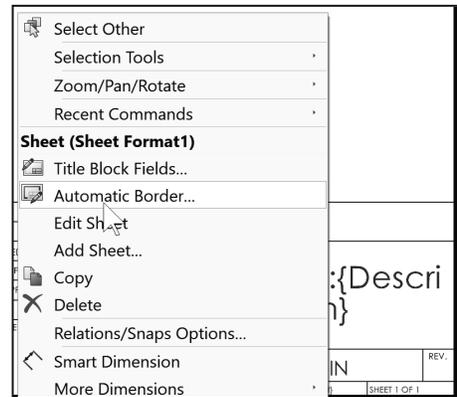


**Notes:** Sheet formats often contain static text notes as well as notes Linked to Properties. Linking notes to properties allows them to be automatically populated with information relevant to the current drawing. We will cover this in detail in the Linked to Properties section of the book.



**Border:** The Sheet format border includes the border surrounding the sheet as well as the zone lines and labels. We will use the zone lines and labels in various tables. The borders included in the SOLIDWORKS default Sheet formats are generated using the Automatic Border tool.

**Inserted Picture:** Although the SOLIDWORKS default Sheet format does not include pictures, it is common practice to add a Company logo or a Third Angle projection icon in the Comments box as illustrated.



Insert a company logo and the Third Angle Projection icon in the Title block.

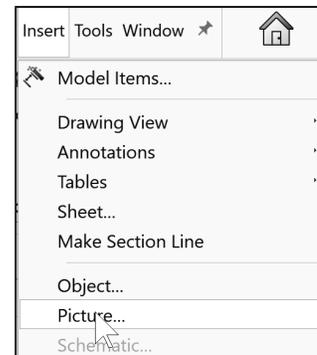
**Activity: Insert a company logo. Insert the Third Angle Projection icon.**

Edit Sheet Format1. Insert two pictures in the Title block. Insert them in the Edit Sheet Format mode.

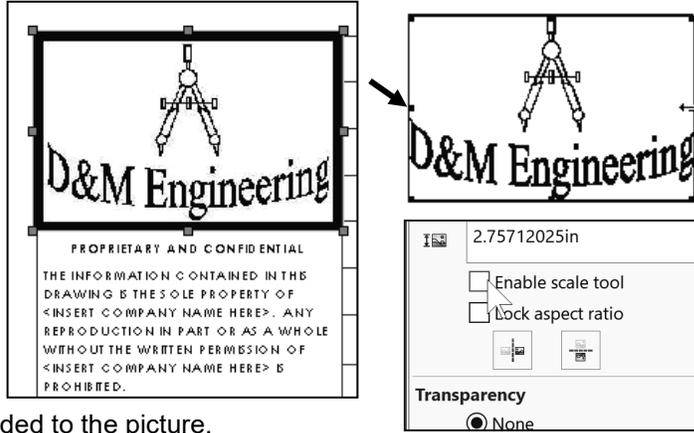
- 56. Right-click **Edit Sheet Format** in Sheet1.
- 57. Click the **Zoom to Area**  tool from the Heads-up View toolbar.
- 58. **Zoom in** on the Title block.
- 59. Click the **Zoom to Area**  tool to deactivate.

A Company logo is normally located in the Title block of the drawing. Create your own Company logo or copy and paste an existing picture. The Logo.jpeg file is provided in the SOLIDWORKS 2022/LOGOS folder.

- 60. Click **Insert, Picture** from the Main menu bar. The Open dialog box is displayed.

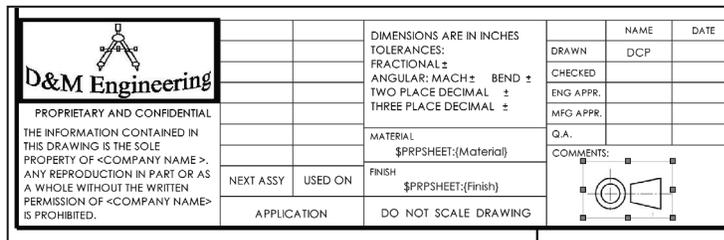


- 61. Select the **SOLIDWORKS 2022/LOGOS** folder.
- 62. Double-click the **Logo.jpg**. The Sketch Picture PropertyManager is displayed.
- 63. Uncheck the **Lock aspect ratio** box.
- 64. Uncheck the **Enable scale tool** box.
- 65. Drag the picture handles to size the **picture** to the left side of the Title block. Note: Text was added to the picture.



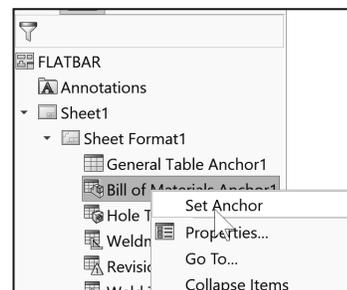
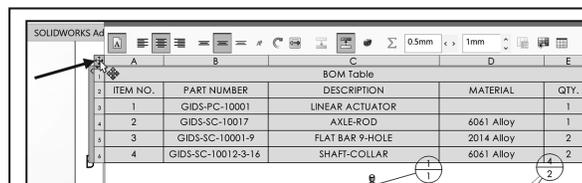
- 66. Click **OK** ✓ from the Sketch Picture PropertyManager.
- Insert the Third Angle Projection icon in the COMMENTS box.
- 67. **Zoom in** on the COMMENTS box in the Title block.
- 68. Click **Insert, Picture** from the Menu bar. The Open dialog box is displayed.

- 69. Select the **SOLIDWORKS 2022/LOGOS** folder.
- 70. Double-click the **THIRD ANGLE PROJECTION.jpg**. The Sketch Picture PropertyManager is displayed.



- 71. Uncheck the **Lock aspect ratio** box.
- 72. Uncheck the **Enable scale tool** box.
- 73. Drag the picture handles to size the **picture** to fit inside the COMMENTS box.
- 74. Click **OK** ✓ from the Sketch Picture PropertyManager.
- 75. **View** the results.

**Anchors:** Anchors position tables with respect to a drawing sheet format. Each type of annotation table has its own anchor point in a drawing sheet format.



**Note Linked to Properties.**

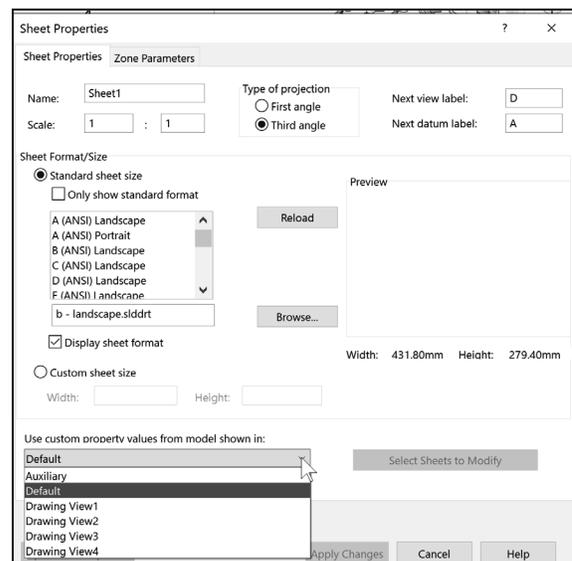
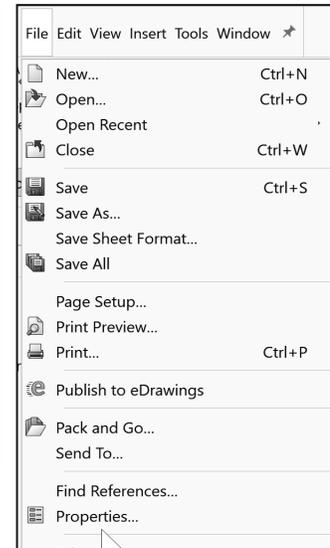
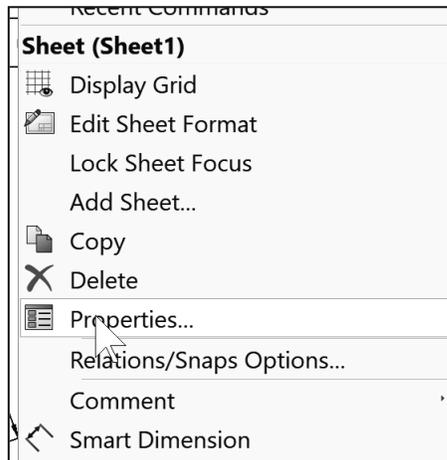
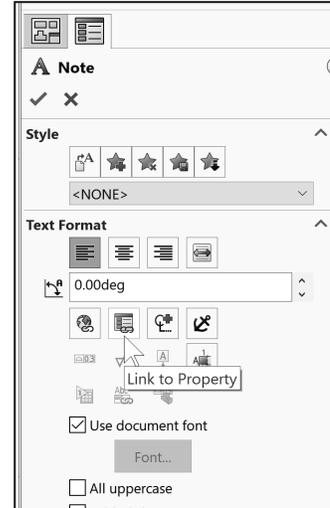
Links can be created to custom properties of the current document, Custom Properties from a referenced model, or to existing file information such as file or folder names. The different property links seen in a drawing title block are formatted as follows:

**\$PRP: “property name”.** Property links with this format are linked to Custom Properties of the current document - the drawing document. These are properties that are accessed from the File Properties dialog in the drawing.

**\$PRPSHEET: “property name”.** Property links with this format are linked to Custom Properties that are accessible

from the File Properties dialog in the part or assembly document used in the drawing views on the sheet.

For drawings that reference more than one model, Sheet Properties can be used to choose the proper model to populate these properties links.



**\$PRP: "SW - property name" or \$PRPSHEET: "SW-property name"**. Properties names that begin with SW are referred to as SOLIDWORKS Special Properties. SOLIDWORKS Special Properties are properties that exist in the SOLIDWORKS file by default. They do not require creation or input from the user; example: include properties such as the sheet scale, (SW-Sheet Scale) or the file Name (SW-File Name).

\$PRPSHEET:{Description}		
SIZE	DWG. NO.	REV.
A	C-ANSI-MM-Chapter 5	
SCALE:1:2	WEIGHT: \$PRPS{\$PRP:"SW-File Name"}	SHEET 1 OF 1

\$PRPSHEET:{Description}		
SIZE	DWG. NO.	REV.
A	C-ANSI-MM-Chapter 5	
SCALE:1:2	WEIGHT: \$PRPSHEET:{Weight}	SHEET 1 OF 1

\$PRP:"COMPANYNAME"		
\$PRPSHEET:{Description}		
SIZE	DWG. NO.	REV.
A	C-ANSI-MM-Chapter 5	
SCALE:1:2	WEIGHT: \$PRPSHEET:{Weight}	SHEET 1 OF 1

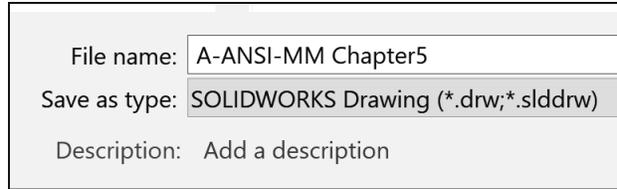
SIZE	DWG. NO.	REV.
A	C-ANSI-MM-Chapter 5	
SCALE:1:2	WEIGHT: \$PRPSHEET:{Weight}	SHEET 1 OF 1
SCALE:\$PRP:"SW-Sheet Scale"		

Return to Sheet1.

76. Right-click **Edit Sheet**.

Save the Drawing. Close all documents.

77. Click **File, Save As** from the Main menu.



78. Enter File name: **A-ANSI-MM Chapter5**.

79. Select the **SOLIDWORKS 2022/Chapter 5 Drawings** folder for Save As.

80. Click **Save**.

Close all open documents.

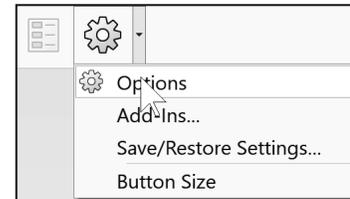
81. Click **File, Close** from the Main menu.

In the next section, display the MY-TEMPLATES tab in the New SOLIDWORKS Document dialog box. Set System Options - File Locations. Only perform this procedure if you are working on a non-network system.

**Activity: Add the MY-TEMPLATES tab to the New SOLIDWORKS dialog box.**

Display the MY-TEMPLATES tab in the New SOLIDWORKS Document dialog box.

82. Click **Options**  from the Main menu. The System Options General dialog box is displayed.

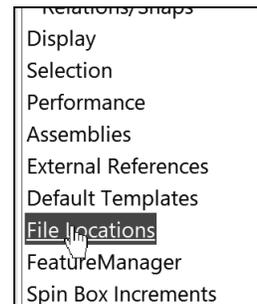


83. Click **File Locations** under the System Options tab.

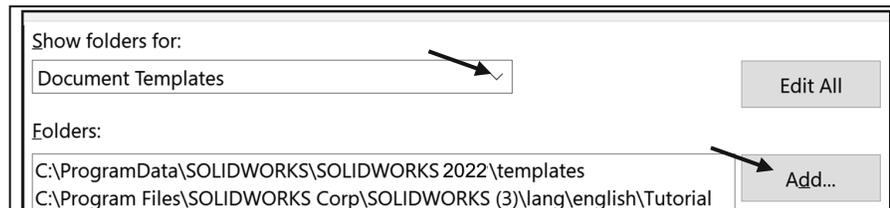
84. Select **Document Templates** from Show folders for.

85. Click the **Add** button.

86. Select the **SOLIDWORKS 2022/MY-TEMPLATES** folder.  
Note: Folders were downloaded in Chapter 4.



87. Click **Select Folder**.



88. If needed click **Move down** to have the MY-TEMPLATES folder as the last folder.

89. Click **OK** from the System Options dialog box.

90. Click **OK** from the SOLIDWORKS dialog box.

91. Click **Yes** to the SOLIDWORKS user account dialog box.

92. Click **Yes** again. An empty SOLIDWORKS window is displayed.

Display the MY-TEMPLATES folder in the Advance New SOLIDWORKS Document dialog box.

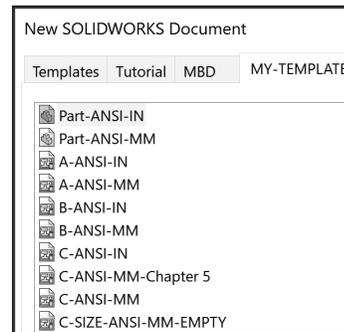
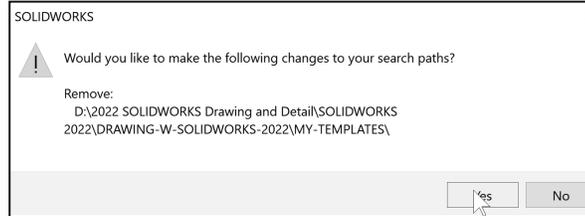
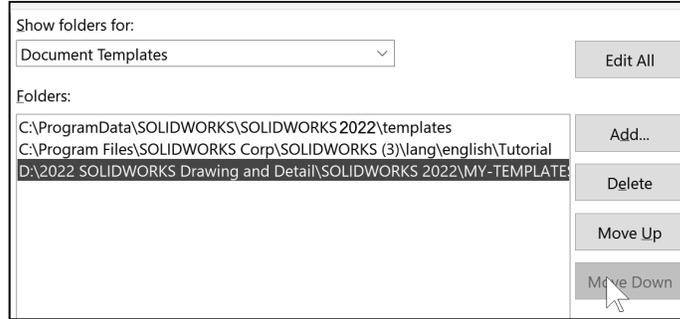
93. Click **File, New** from the Main menu. The New SOLIDWORKS Document dialog is displayed.

94. Double-click the **MY-TEMPLATES** folder tab.

95. **View** the existing templates. Save new Custom Drawing Templates in this location. Create new Drawing with Custom Templates using the MY-TEMPLATES tab.

96. Click **OK**. Note: Additional Templates are displayed.

97. **Close** all SOLIDWORKS documents.



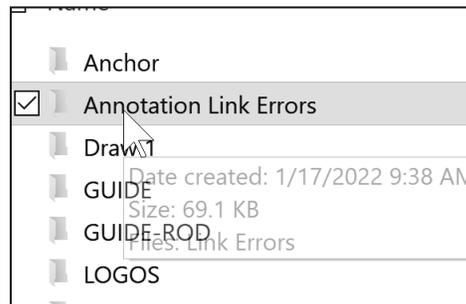
### Annotation Link Errors.

When a note is linked to property that does not exist, it is considered to be an error. By default, Annotation Links Errors are hidden from view. In the next section view annotation link errors.

**Activity: View Link Errors in a drawing.**

Open the Link Error drawing from the SOLIDWORKS 2022/Annotation Link Errors folder. View the Link Errors.

1. Select the **SOLIDWORKS 2022/Annotation Link Errors** folder.
2. Double-click the **Link Error** drawing from the Annotation Link Errors folder.
3. Click **View, Hide/Show, Annotation Link Errors** from the Main menu.
4. **View** the results.



	NAME	DATE	ERROR!:COMPANYNAME	
DRAWN	DCP	ERROR!:DrawnDate		
CHECKED	ERROR!:CheckedBy	ERROR!:CheckedDate		
ENG APPR.	ERROR!:CheckedBy	ERROR!:EngAppDate		
MFG APPR.	ERROR!:MfgApp	ERROR!:MfgAppDate		
Q.A.	ERROR!:QAApp	ERROR!:QAAppDate		
COMMENTS:				
SIZE		DWG. NO.	REV.	
A		Link Errors	ERROR!:Rev	
SCALE:1:2		WEIGHT: ERROR!:Weight	SHEET 1 OF 1	

Turn off the Annotation Link Errors.

5. Click **View, Hide/Show, Annotation Link Errors** from the Main menu. View the results.

Close the Link Error drawing.

	NAME	DATE		
DRAWN	DCP			
CHECKED				
ENG APPR.				
MFG APPR.				
Q.A.				
COMMENTS:				
SIZE		DWG. NO.	REV.	
A		Link Errors		
SCALE:1:2		WEIGHT:	SHEET 1 OF 1	

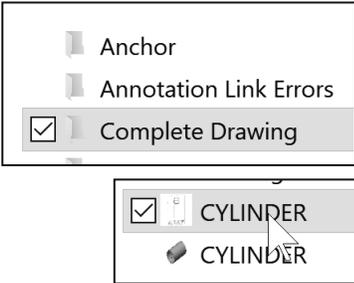
6. Click **File, Close** from the Main menu.

In the next section, explore a completed Title box.

**Activity: Explore a Complete Title Box - CYLINDER drawing.**

Open the CYLINDER drawing from the SOLIDWORKS 2022/Complete Drawing folder. View a complete Title box. The notes on this drawing are color coded to reflect where the property information is being populated from.

1. Double-click **CYLINDER drawing** from the SOLIDWORKS 2022/Complete Drawing folder. The CYLINDER drawing is displayed.
2. Click **View, Hide/Show**, check **Annotation Link Errors** from the Main menu.
3. Click inside the **Sheet1. View** the results. No errors are displayed.



DIMENSIONS ARE IN MILLIMETERS TOLERANCES: ANGULAR: MACH ± 0°30' ONE PLACE DECIMAL ±0.5 TWO PLACE DECIMAL ±0.15		NAME	DATE	D&M ENGINEERING  CYLINDER	
	DRAWN	DCP	03/14/22		
	CHECKED	DEF	03/17/22		
	ENG APPR.	FDC	03/18/22		
	MFG APPR.	BVC	03/22/22		
MATERIAL	1060 Alloy	Q.A.	LLP	04/01/22	
FINISH	0.80 μm	COMMENTS:		SIZE DWG. NO. 112.4 REV. A	
DO NOT SCALE DRAWING			SCALE:1:1 WEIGHT:0.25 \$PRP:"Number" SHEET 1 OF 1		

The notes in the drawing have been color coded to reflect where the property information is being populated from:

**GREEN:** SOLIDWORKS Special Properties.

**BLUE:** Model Custom Properties.

**RED:** Drawing Custom Properties.

In the next section, explore the custom properties of this drawing and model document.

**Activity: Explore Additional Custom Properties of the Drawing and Part.**

View the Custom Properties of this drawing. Deactivate the Annotation Link Errors option.

1. Click **View, Hide/Show**, uncheck **Annotation Link Errors** from the Main menu.
2. Click **File, Properties** from the Main menu. The Summary Information dialog box is displayed.

3. Click the **Custom** tab. These entries are displayed in red in the title box above..

Summary Information

Summary Custom

Delete BOM quantity: - None - Edit List

	Property Name	Type	Value / Text Expression	Evaluated Value	<input type="checkbox"/>	
1	SWFormatSize	Text	279.4mm*215.9mm	279.4mm*215.9mm	<input type="checkbox"/>	
2	CompanyName	Text	D&M ENGINEERING	D&M ENGINEERING	<input type="checkbox"/>	
3	Revision	Text	A	A	<input type="checkbox"/>	
4	DrawnBy	Text	DCP	DCP	<input type="checkbox"/>	
5	DrawnDate	Text	03/14/22	03/14/22	<input type="checkbox"/>	
6	CheckedBy	Text	DEF	DEF	<input type="checkbox"/>	
7	CheckedDate	Text	03/17/22	03/17/22	<input type="checkbox"/>	
8	EngineeringApprova	Text	FDC	FDC	<input type="checkbox"/>	
9	EngAppDate	Text	03/18/22	03/18/22	<input type="checkbox"/>	
10	ManufacturingAppr	Text	BVC	BVC	<input type="checkbox"/>	
11	MfgAppDate	Text	03/22/22	03/22/22	<input type="checkbox"/>	
12	QAApproval	Text	LLP	LLP	<input type="checkbox"/>	

OK Cancel Help

4. Click **Close**.

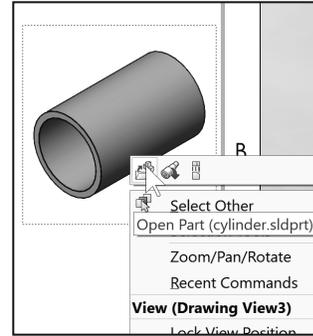
View the Custom Properties of the CYLINDER Part model.

5. Right-click in a **drawing view**.

6. Click **Open Part**. The CYLINDER part is displayed.

7. Click **File, Properties** from the Main menu. **View** the Properties of the CYLINDER part. These entries are displayed in blue in the Title box. Note the Finish and Material Properties Name.

8. Click **OK** from the Properties dialog box.



Properties

Summary Custom Configuration Properties Properties Summary

Delete BOM quantity: - None - Edit List

	Property Name	Type	Value / Text Expression	Evaluated Value	<input type="checkbox"/>	
1	Description	Text	CYLINDER	CYLINDER	<input type="checkbox"/>	
2	Material	Text	"SW-Material@CYLINDER.SLDPRT"	1060 Alloy	<input type="checkbox"/>	
3	Finish	Text	0.80	0.80	<input type="checkbox"/>	
4	Weight	Text	"SW-Mass@CYLINDER.SLDPRT"	254.469	<input type="checkbox"/>	
5	<Type a new propert				<input type="checkbox"/>	

Return to the CYLINDER drawing.

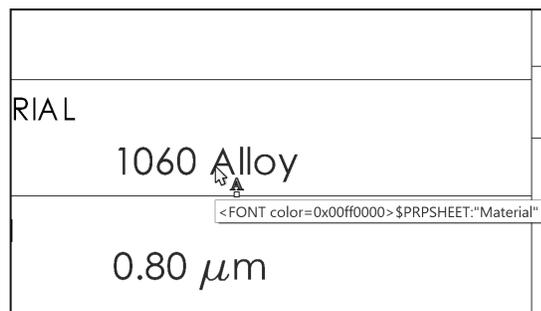
9. Click **Window, CYLINDER - Sheet1** from the Main menu. The CYLINDER drawing is displayed.

10. Right-click **Edit Sheet Format** in Sheet1. Do not click inside a drawing view.

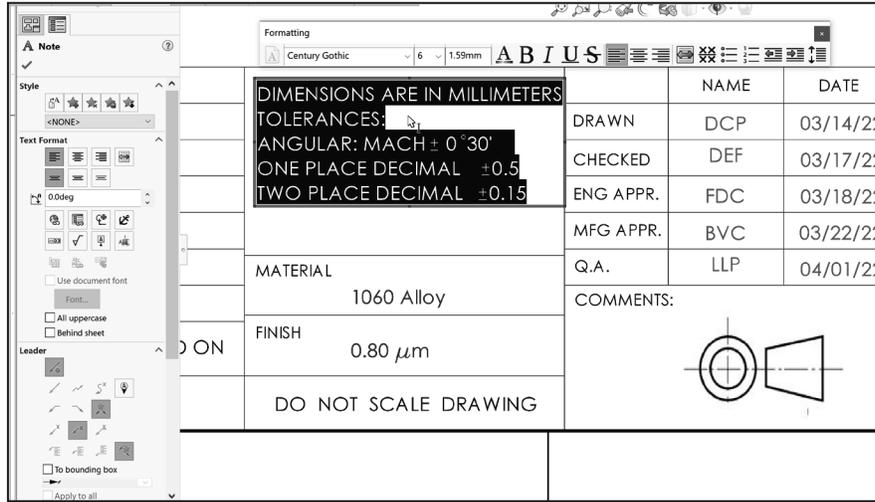
11. **Zoom in** on the Title box as illustrated.

12. Hover the **mouse cursor** over the MATERIAL box as illustrated.

13. **View** the Part link \$PRPSHEET. "Material".



14. **Double-click** inside the Tolerance block. The box turns black. The Note PropertyManager is displayed along with the Formatting toolbar. Use the Formatting toolbar to address Title box font text.



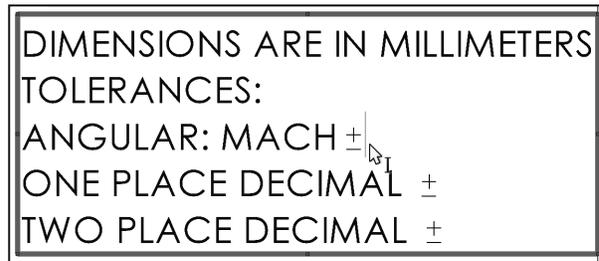
The Tolerance block provides information to the manufacturer on the minimum and maximum variation for each dimension on the drawing. If a specific tolerance or note is provided on the drawing, the specific tolerance or note will override the information in the Tolerance block.

General tolerance values are based on the design requirements and the manufacturing process.

 Create Sheet formats for different part types; examples: sheet metal parts, plastic parts and high precision machined parts. Create Sheet formats for each category of parts that are manufactured with unique sets of Title block notes.

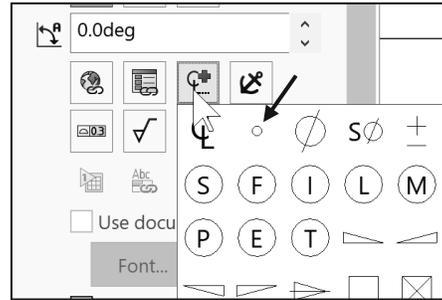
Modify the Tolerance block in the Sheet format for ASME Y14.5 machined millimeter parts. Delete unnecessary text. The FRACTIONAL text refers to inches. The BEND text refers to sheet metal parts. The Three Decimal Place text is not required for this millimeter part in the chapter.

- 15. Delete the **following items** as above in the Title box.
- 16. Click **inside the box** after ANGULAR: MACH ± as illustrated.



17. Enter 0.

18. Click the **Add Symbol**  button from the Text Format box. The Symbols dialog box is displayed.



 The **Link to Property**  icon provides access to drawing properties and component properties from any model in the drawing so you can add them to the text string.

 The **Lock/Unlock Notes**  icon provides the ability to fix the note in place. When you edit the note, you can adjust the bounding box, but you cannot move the note itself.

19. Select **Degree** ° from the Symbols dialog box.

20. Enter **30'** for minutes of a degree.

21. Enter **0.5** after ONE PLACE DECIMAL: ±.

22. Enter **0.15** after TWO PLACE DECIMAL±.

23. Click **OK** from the Note PropertyManager.

Note: Finish is 0.80 μm. To display a micrometer symbol in the CYLINDER drawing title box, Click **Edit Sheet format** in the CYLINDER drawing. Double-click inside the **FINISH** box. The Formatting toolbar is displayed. Click **SWGreck** from the Formatting toolbox Font. Press the **m** key to display the Greek letter μ. μ is the Greek letter for micro. Click **Century Gothic** for Font. Enter **m** for meter. The text reads μm for micrometer

DIMENSIONS ARE IN MILLIMETERS  
 TOLERANCES:  
 ANGULAR: MACH ± 0°30'  
 ONE PLACE DECIMAL ±0.5  
 TWO PLACE DECIMAL ±0.15

MATERIAL	1060 Alloy
FINISH	0.80 μm

0.80 μm

Various symbols are available through the Symbol dialog box. The ± symbol is located in the Modify Symbols list. The ± symbol is sometimes displayed as <MOD-PM>. The degree symbol ° is sometimes displayed as <MOD-DEG>.

Interpretation of tolerances is as follows:

- The angular dimension 110° is machined between 109.5° and 110.5°.
- The dimension 2.5 is machined between 2.0 and 3.0.
- The dimension 2.05 is machined between 1.90 and 2.20.

Return to Sheet1. Save the CYLINDER drawing.

24. Right-click **Edit Sheet**.

25. Click **Save** .

**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: Insert General Notes using a text file.**

Insert general notes from a text file. Copy and paste them in the drawing.

1. Double-click on the Notepad file, **SOLIDWORKS 2022/MY-SHEETFORMATS/GENERALNOTES.TXT**.

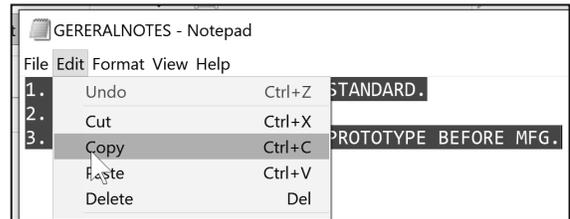
2. Click **Edit, Copy**.



Return to Sheet1 in the drawing.

3. Click **Note**  from the Annotation toolbar.

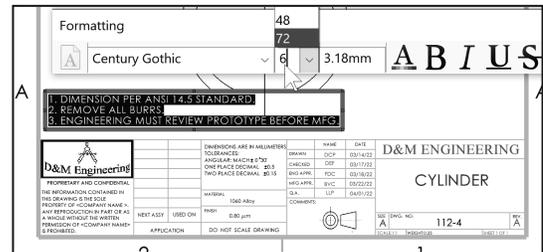
4. Click a **point** in the lower left-hand corner of Sheet1.



5. Click **inside** the Note text box.

6. Right click **Paste**. The three text lines are in the note box.

7. Click **6** for **Font size** from the Formatting dialog box.



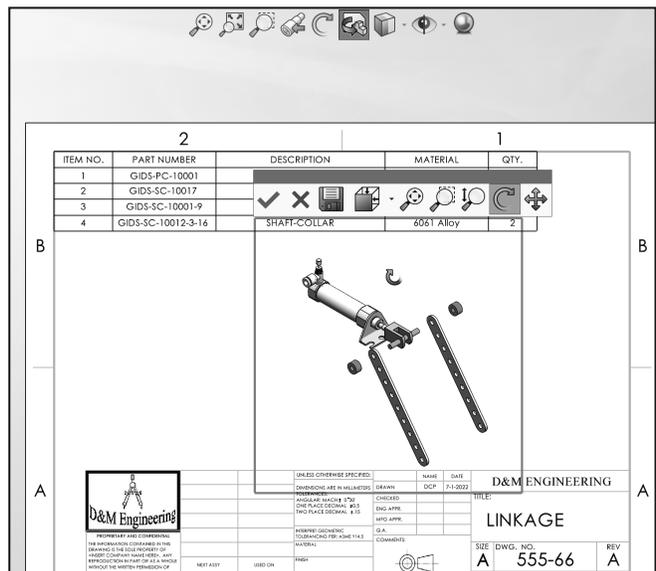
8. Click **OK**  from the Note PropertyManager.

9. Click **Save** .

**Display Styles / Modes**

Most 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.

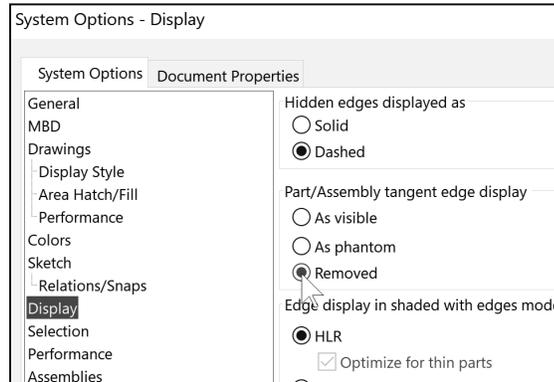
Wireframe and Shaded Display modes provide the best Graphic performance. Mechanical details require Hidden Lines Visible display and Hidden Lines Removed display. Select Shaded/Hidden Lines Removed to display Auxiliary Views to avoid confusion.



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 Tangent Edges for clarity.

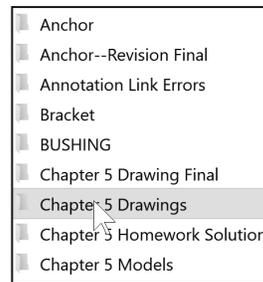


 Tangent Edges are displayed for educational purposes.

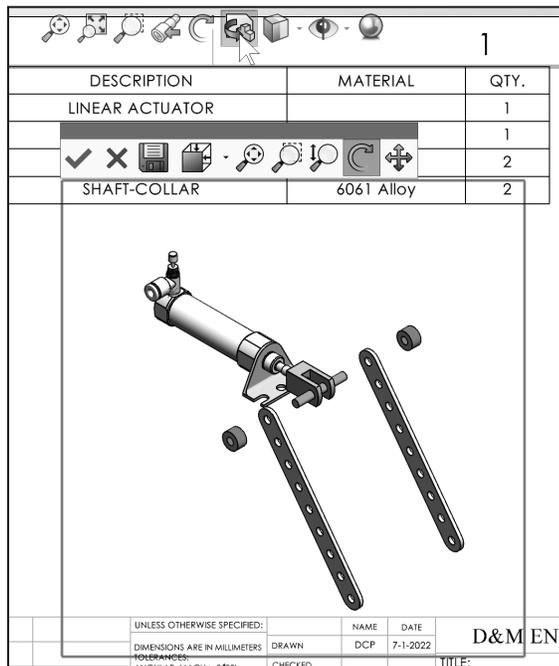
In the next section, explore the Custom properties of this drawing and model document.

**Activity: Explore the 3D Drawing tool and various Display modes.**

Open the LINKAGE drawing from the SOLIDWORKS 2022/Chapter 5 Drawing folder. Explore the 3D drawing tool.

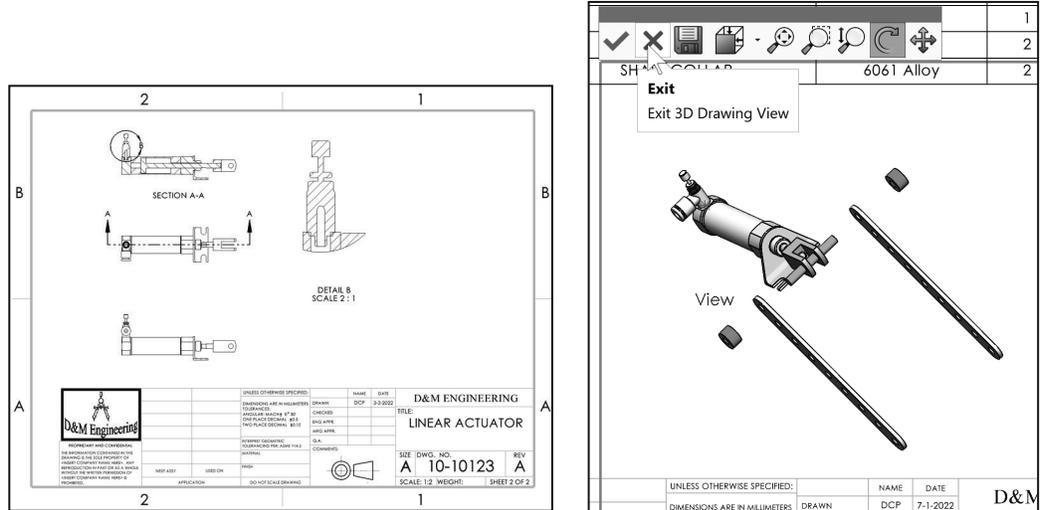


1. Double-click the **LINKAGE drawing** from the SOLIDWORKS 2022/Chapter 5 Drawing. The LINKAGE drawing is displayed. The LINKAGE drawing has two Sheets. Sheet1 is selected by default.
2. Click inside **Drawing view1**.
3. Click the **3D Drawing View** icon as illustrated in the Heads-up View toolbar.
4. **Rotate** the view as illustrated.
5. **View** the results. Return to the original Isometric view.
6. Click **Exit** .



Explore the LINKAGE Drawing.

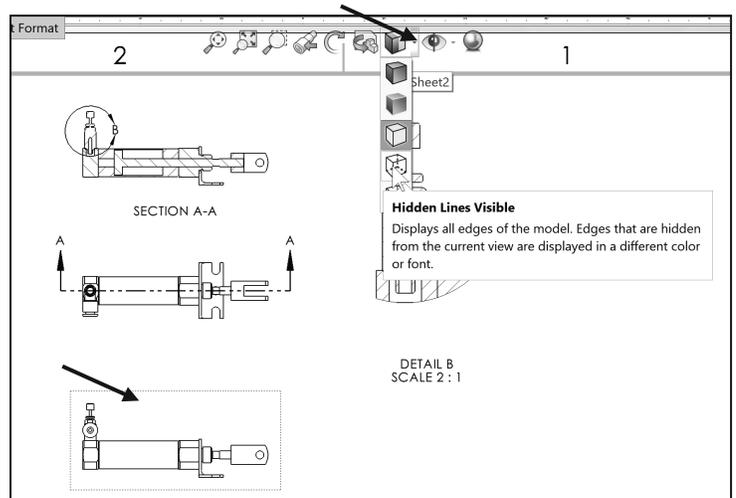
7. Click the **Sheet2** tab in the lower left bottom section of the drawing as illustrated. View Sheet2.



8. Click inside **Drawing View2**.
9. Click **Hidden Lines Visible** from the drop-down Heads-up View toolbar.
10. **View** the drawing view change. Return to Hidden Lines Removed.
11. Click **Hidden Lines Removed** from the drop-down Heads-up View toolbar.

Close all drawings and open documents. Do not save.

12. Click **File, Close** from the Main menu.
13. Click **Don't Save** from the SOLIDWORKS dialog box.



### Centerlines and Center Marks

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 past 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.



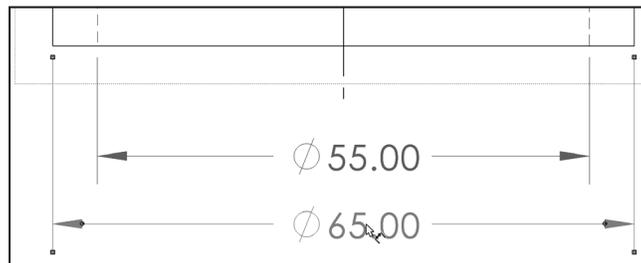
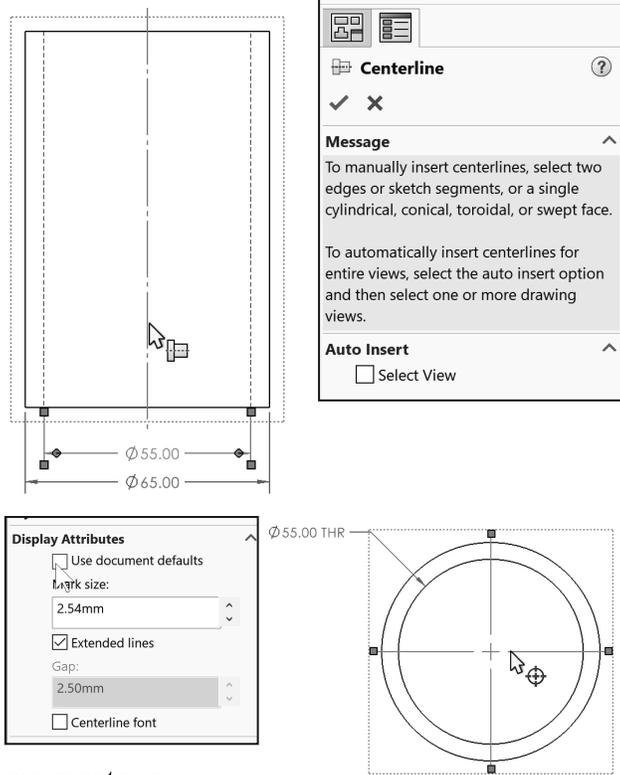
Center marks and Centerlines are annotations used to mark circle centers and describe the geometry size on the drawing.

### 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.

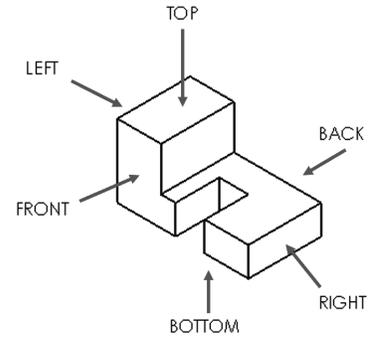


The values 1.5mm and 3mm are a guide. Base the gap and extension line on the drawing size and scale.



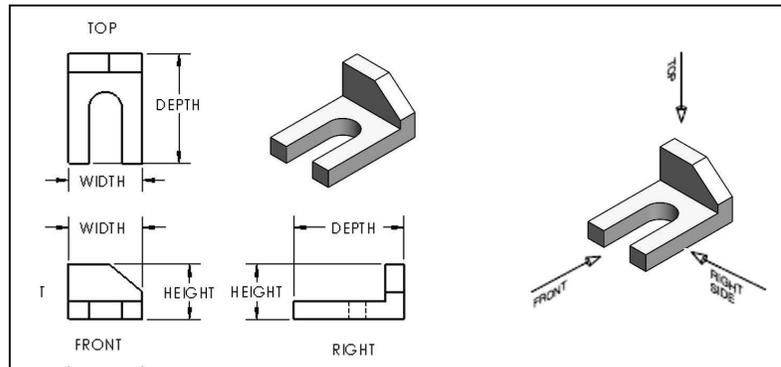
### Predefined and Projected Views

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 Drawing Template.



Insert the Top, Front, Right, and Isometric views into the Drawing. 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.

1. Create an **A-ANSI Landscape - MMGS, Third Angle** drawing.
2. Scale **1:2**. Precision **.12**.
3. Click **Insert, Drawing View, Predefined**  from the Main menu.
4. 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.

5. Click **Hidden Lines Removed** from the Display Style box.
6. Click **OK**  from the Drawing View1 PropertyManager.

Insert a Top Projected view.

7. Click the **View Layout** tab from the CommandManager.

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

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

10. Click a **position** directly above the Front view.

Insert the Right Projected view.

11. Click **Projected View**  from the View Layout toolbar.

12. Click inside the **Front** view.

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

Insert an Isometric Predefined view.

14. Click inside the **Front** view.

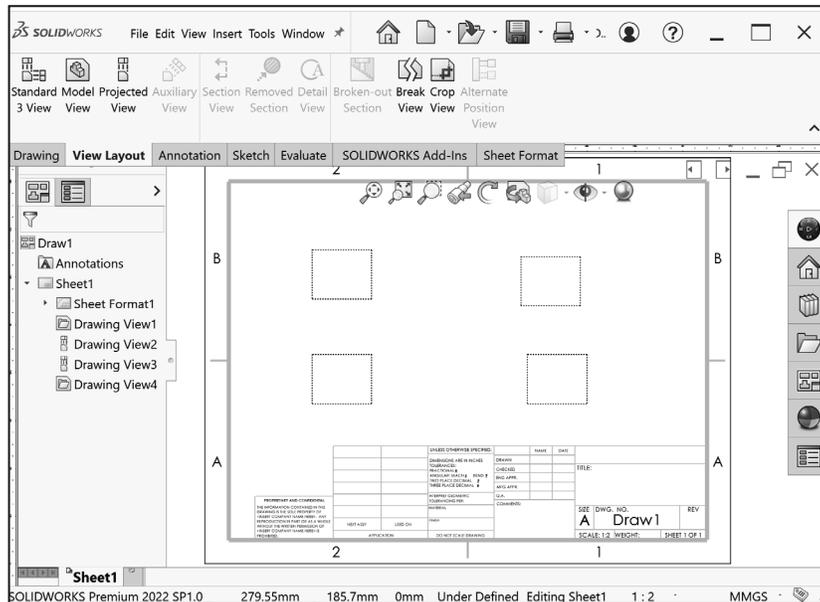
15. Click **Insert, Drawing View, Predefined**  from the Main menu. The Drawing View PropertyManager is displayed.

16. Click a **position** in the upper right corner of the Sheet as illustrated.

17. Click **\*Isometric** from the Orientation box.

18. Click **OK**  from the Drawing View4 PropertyManager.

19. Click **Save** . View the drawing FeatureManager. Note the view icons for the Predefined and Projected views.

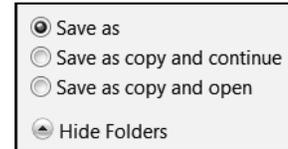


### Save As

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).

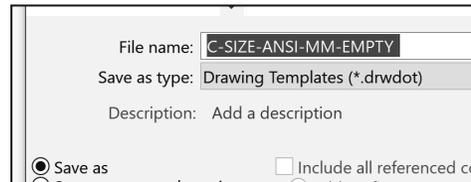
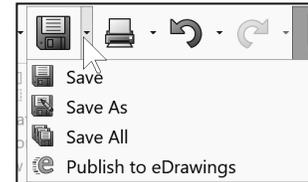


Test the Drawing Template located in the MY-TEMPLATES folder. Create a new Drawing document.

**Activity: Save As and Test New Drawing Template**

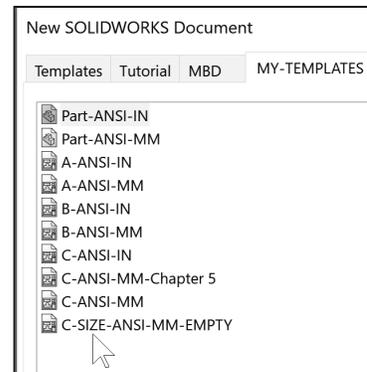
Save the empty Drawing Template.

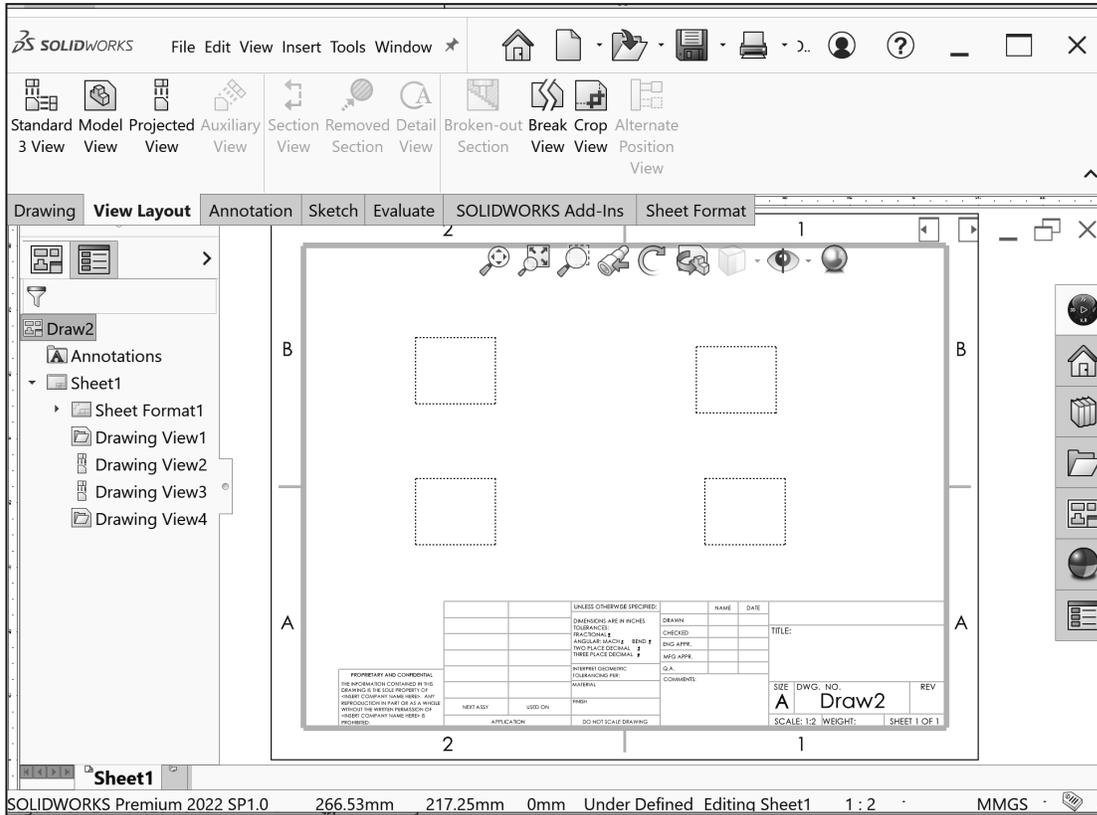
1. Click **Save As**  from the Main menu.
2. Select **Drawing Templates (\*.drwdot)** from the Save as type.
3. **Browse** and select the **SOLIDWORKS-2022\MY-TEMPLATES** for the Save in file folder.
4. Enter **C-SIZE-ANSI-MM-EMPTY** for the File name. The file extension for the Template is .drwdot.
5. Click **Save** from the Save As dialog box.
6. Click **Windows, Close All** from the Main menu.



Create a new drawing.

7. Click **New**  from the Menu bar toolbar.
8. Select **MY-TEMPLATES** tab from the New SOLIDWORKS Document dialog box.
9. Double-click **C-SIZE-ANSI-MM-EMPTY**.
10. Click **Cancel**  from the Model View PropertyManager. Draw2 is the current drawing document. Note the drawing view icons in the FeatureManager. Drag and drop the views if needed onto the sheet.





Save the drawing. Rename Draw2 to Link Note.

**11. Rename** Draw2 to Link Note.

**12. Save** the Link Note drawing to the SOLIDWORKS 2022/Chapter 5 Drawing folder. We will use this drawing in the next section.

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.

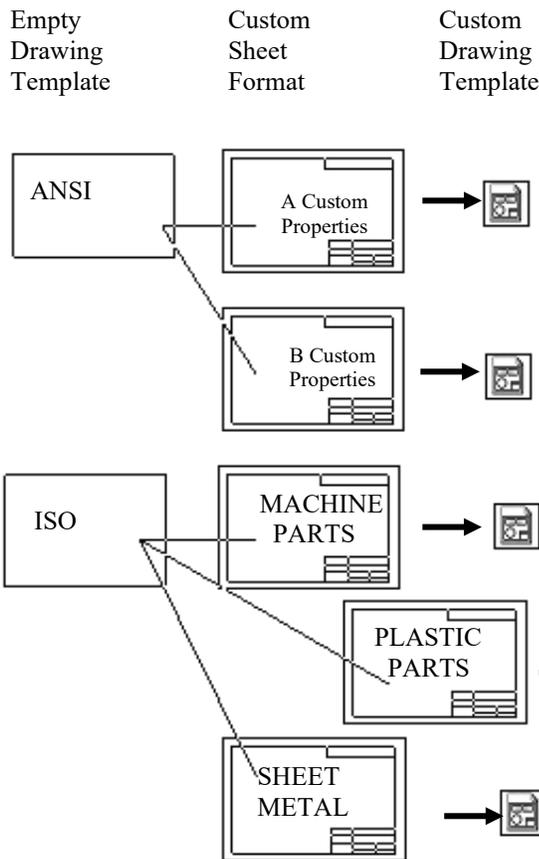
### Sheet Format

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 Y14.24 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.



### 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 C:\ProgramData\SOLIDWORKS\SOLIDWORKS 2022\lang\english, properties.txt.

Create your own User defined Property named variables. The properties.txt file is a hidden file. Insert the file path into your search bar to locate a hidden file.

```

properties - Notepad
File Edit Format View Help
Description
PartNo
Number
Revision
Material
Weight
Finish
StockSize
UnitOfMeasure
Cost - Total Cost
Cost - Material Cost
Cost - Manufacturing Cost
Cost - Material Name
Cost - Template Name
Cost - Stock Type
Cost - Stock Size
Cost - Cost Calculation Time
MakeOrBuy
LeadTime
CheckedBy
CheckedDate
DrawnBy
DrawnDate
EngIneerIngApproval
EngAppDate
ManufacturingApproval
MfgAppDate
QAApproval
QAAppDate
Vendor
VendorNo
Client
Project
Status
DateCompleted
CompanyName
Department
Division
Group
Author
    
```

### 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.

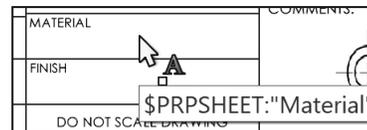
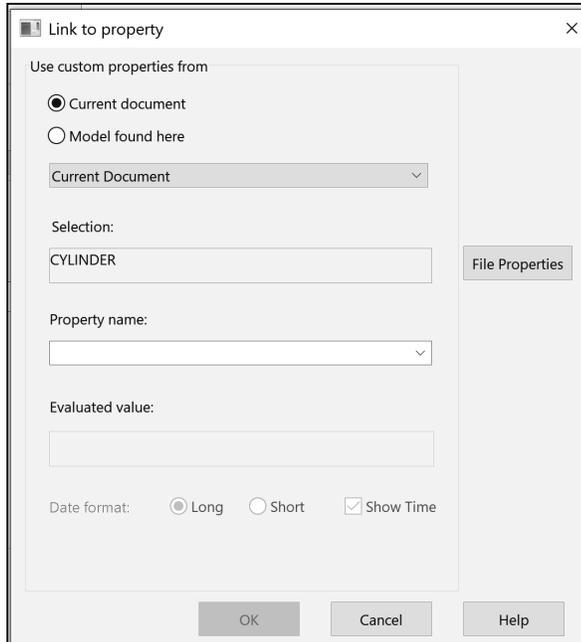
Linked Notes begin with the four different prefixes listed below:

Linked Notes that reference Custom Properties in the drawing utilize the prefix:

**\$PRP:** Enter double quotes to define the property name: Example:

-\$PRP:"Description"

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"



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.

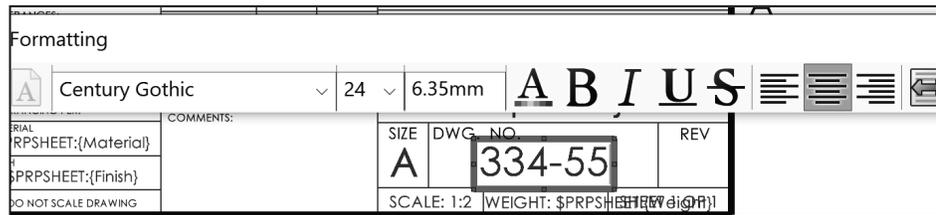
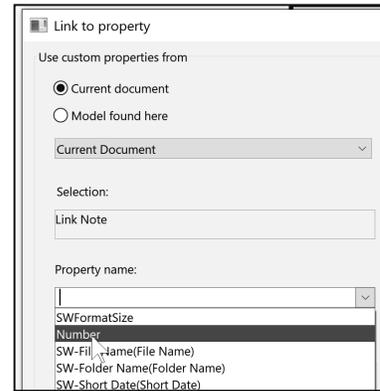
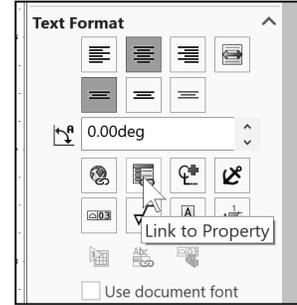
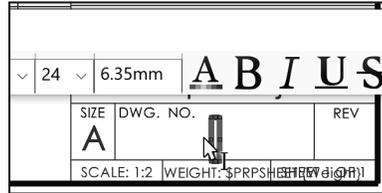
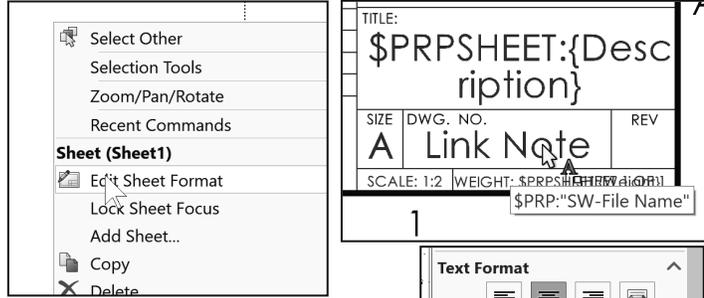
**Activity: Title Block and SW-File Name**

Create a Linked Note for the DWG NO System Property.

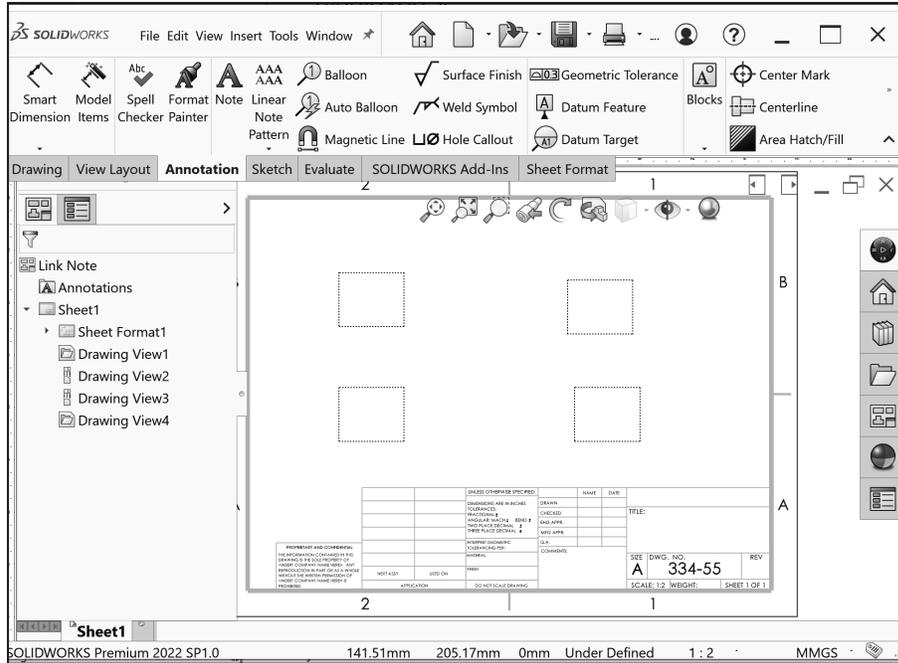
1. Click **File, Properties** from the Drawing Main menu. The Summary Information dialog box is displayed.
2. Select **Number** from the drop-down menu in the Property Name box as illustrated.
3. Click inside the **Value / Text Expression** box.
4. Enter **334-55**.
5. Click inside the **Evaluated Value** box.
6. Click **OK** from the Summary Information dialog box.

	Property Name	Type	Value / Text Expression	Evaluated Value
1	SWFormatSize	Text	215.9mm*279.4mm	215.9mm*279.4mm
2	Number	Text	334-55	334-55
3	Number Revision Material Weight Finish StockSize			

7. Right-click **Edit Sheet Format** in Sheet1.
8. **Zoom in** on the DWG. NO. box.
9. Hover over the box until you see **\$PRP:"SW-File Name"**.
10. Double-click **\$PRP:"SW-File Name"**.
11. **Delete** **\$PRP:"SW-File Name"**. The Note PropertyManager is displayed.
12. Click the **Link to Property** icon as illustrated. The Link to Property dialog box is displayed.
13. Select **Number** from the drop-down menu as illustrated.
14. Click **OK** from the Link to Property dialog box. Note the Formatting dialog box and the options. Use the same procedure above to address the other Link and Custom Properties in a drawing.



15. Click **OK** ✓ from the Note PropertyManager.
16. Right-click **Edit Sheet** in the graphic area.



Save the Drawing. View the results.

17. Click **Save**  .

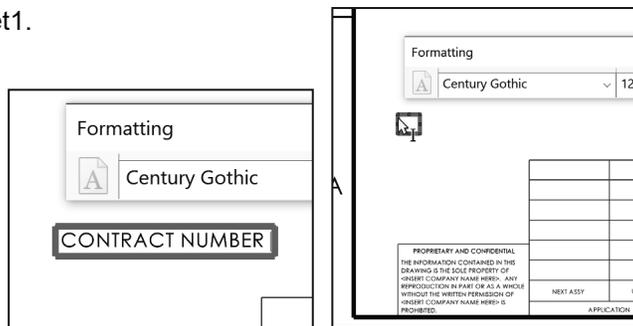
## User Defined Custom Property

Your company has a policy that a contract number must be contained in the Title block area. 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.

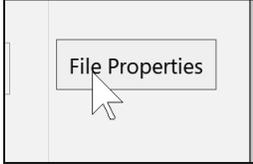
### Activity: User Defined Custom Property

Create a User defined Custom Property.

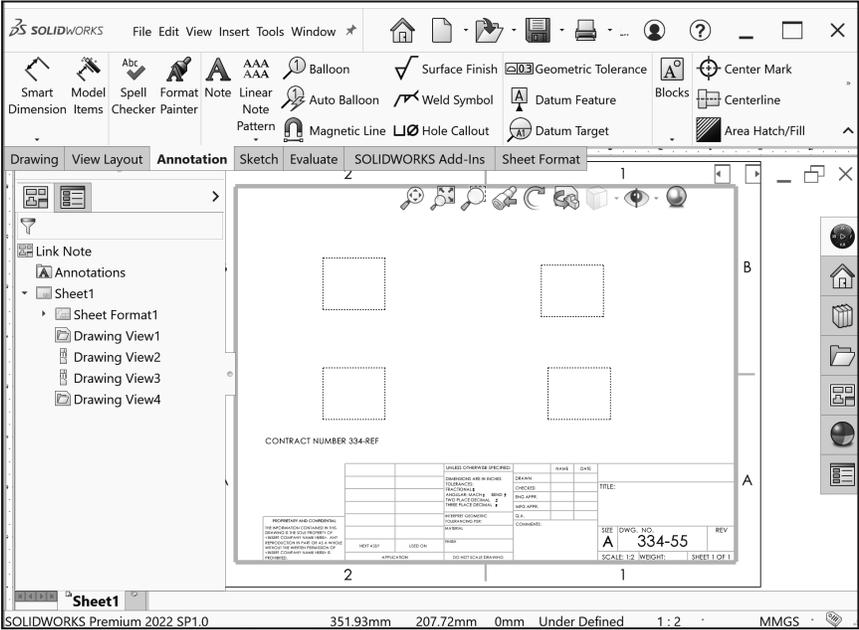
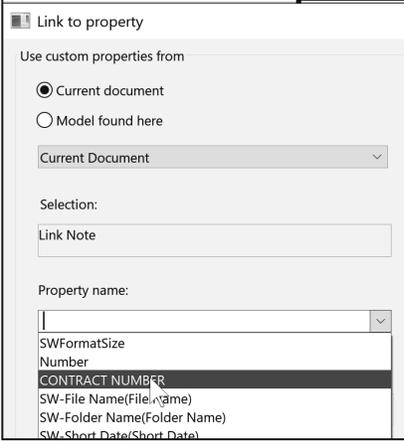
1. Right-click **Edit Sheet Format** in Sheet1.
2. Click **Note**  from the Annotation toolbar.
3. Click a **point** in the left-hand corner as illustrated.
4. Enter **CONTRACT NUMBER**.
5. Press the **space bar**.



6. Click the **Link to Property**  icon from the Text Format box.
  7. Click the **File Properties** button.
  8. Click the **Custom** tab.
  9. Click inside the **Property Name** box.
  10. Enter **CONTRACT NUMBER** for Name.
  11. Click inside the **Value / Text Expression** box.
  12. Enter **334-REF** for Value.
  13. Click inside the **Evaluated Value** box.
  14. Click **OK** from the Summary Information box.
  15. Select **CONTRACT NUMBER** in the Property Name text box.
  16. Click **OK** from the Link to Property box. **View** the results.
  17. Click **OK**  from the Note PropertyManager.
  18. Right-click **Edit Sheet**.
- Save the drawing. Close all documents.
19. Click **Save** .
  20. **Close** all open documents.



	Property Name	Type	Value / Text Expression	Evaluated Value
1	SWFormatSize	Text	215.9mm*279.4mm	215.9mm*279.4mm
2	Number	Text	334-55	334-55
3	CONTRACT NUMBER	Text	334-REF	334-REF
4	<Type a new property>			



## Save Sheet Format and Drawing Template

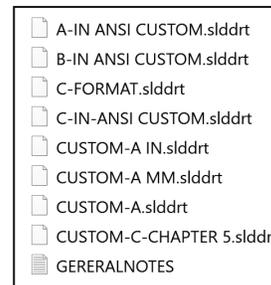
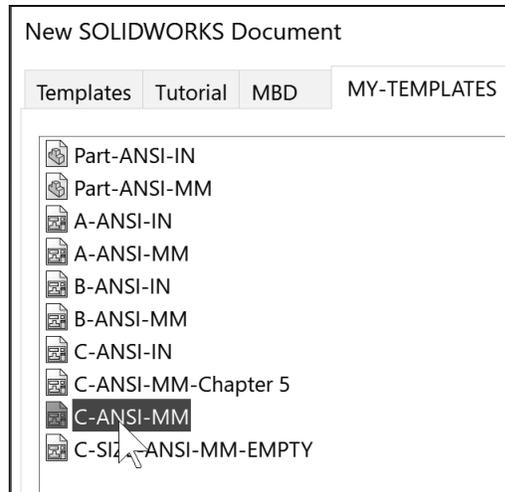
A Custom Sheet format (\*.slddrt) and Custom Drawing Template (\*.drwdot) utilize two different commands to save the current drawing document (.drw). Utilize the File, Save Sheet Format option to create a Custom Sheet format. The Custom Sheet formats are stored in the MY-SHEETFORMATS folder in this book.

Utilize the Save As command and select the Drawing Template option to create the Custom Drawing Template. Combine the Custom Sheet format with the Custom Drawing Template.

You created the Custom C-ANSI-MM-Chapter 5 and C-SIZE-ANSI-MM-EMPTY template with the standard SOLIDWORKS Sheet.

Note: There are additional Custom Sheets and Formats shown in the illustration.

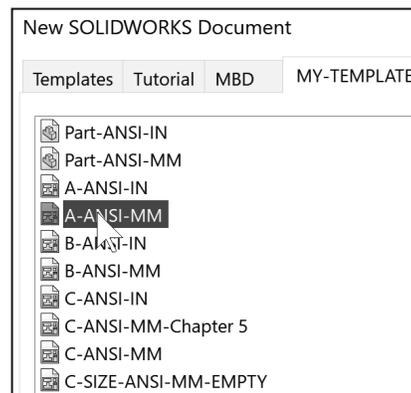
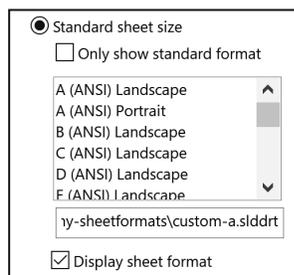
Save the Sheet format and Drawing Templates in the Edit Sheet mode. Insert Drawing views into the drawing in Edit Sheet mode. Views can't be displayed in the Edit Sheet Format mode.

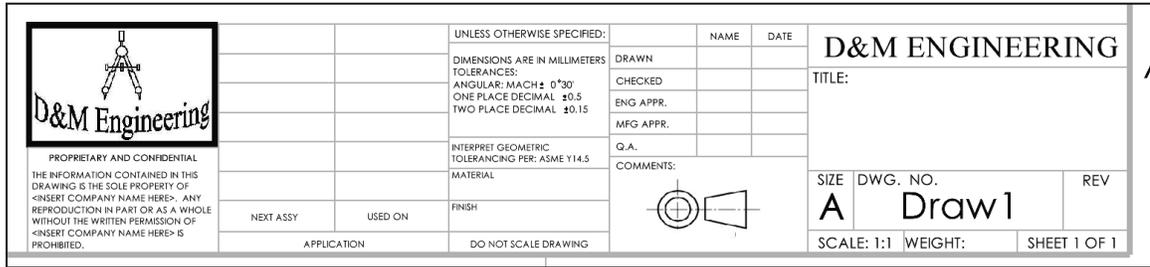


### Activity: Open a Custom Template with a Custom Sheet Format

Create a new drawing using the Custom A-ANSI-MM Template.

1. Click **File, New** from the Main menu.
2. Click the **MY-TEMPLATES** tab.
3. Double-click **A-ANSI-MM**.
4. **View** the results. The Custom A-ANSI-MM Template use the CUSTOM-A Sheet format.
5. Click **Cancel**  from the Model View PropertyManager.
6. **View** the results.



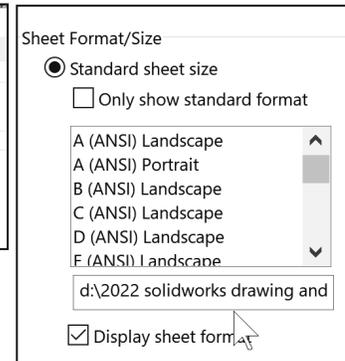
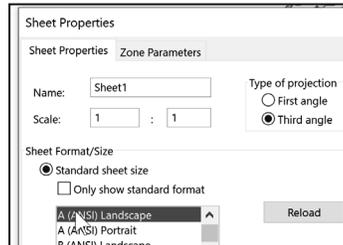
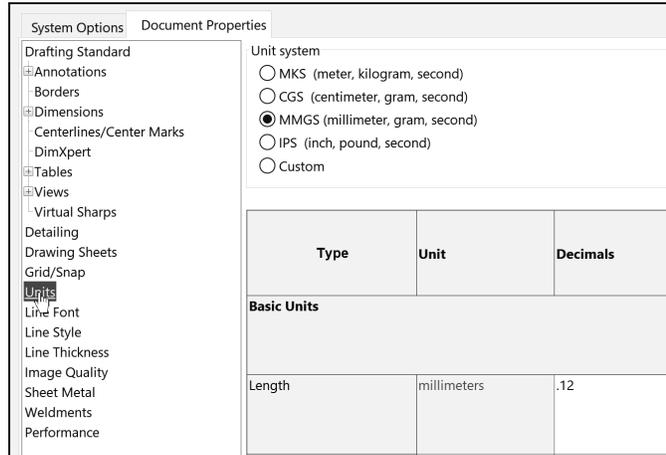
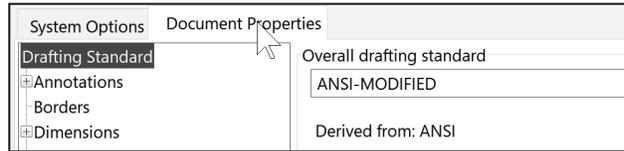


The Custom Title box was incorporated in the Custom Sheet (CUSTOM-A). The logos, Dimensions are in MILLIMETERS and the TOLERANCES are set per ASME Y14.5 along with the Company Name (D&M ENGINEERING).

The Custom Template incorporated the Sheet Scale, Units, Precision, Drafting Standard and Angle of Projection for the drawing views.

View the Custom Drawing Properties.

7. Click **Options** , **Document Properties** from the Main menu.
8. **View** the Overall drafting standard.
9. Click the **Unit** folder.
10. **View** units (MMGS) and precision (.12).
11. Click **OK** from the Document Properties dialog box.
12. Right-click **Sheet1** in the Drawing PropertyManager.
13. Click **Properties**. View Scale, Type of Projection and Sheet location.
14. Click **OK** from the Sheet Properties dialog box.
15. **Close** the drawing.



## Chapter Summary

The purpose of this chapter was to provide a deeper understanding of how SOLIDWORKS drawing documents and templates are created and used. Create an awareness on the structure of a Drawing Document. Provide a general knowledge of the ASME Y14 2009 Engineering Drawing and Related Documentation Practices.

You learned about the elements which construct a drawing document and distinguished between System Options and Document Properties as they relate to drawings, drawing templates and sheets.

You created a new SOLIDWORKS File Location for a drawing template and developed Linked Notes to SOLIDWORKS Properties and Custom Properties in the Sheet format.

You also worked with Annotation Links Error.

**Questions**

1. Explain the Structure of a Drawing Document.
2. Name four items that are contained in the Sheet format file.
3. Identify the paper dimensions (inches) required for an A (ANSI) Landscape size Sheet.
4. Identify the paper dimensions (inches) required for an A4 (ANSI) Landscape Sheet.
5. Name the three Drafting Standards.
6. Identify the primary type of projection utilized in a drawing in the United States.
7. Identify the primary type of projection utilized in a drawing in Europe.
8. Identify the location of the SW default Templates.
9. Name four Display Modes for a drawing view.
10. Identify two Unit Systems supported by SOLIDWORKS.
11. Identify 4 Custom Properties which are contained in a Title block.
12. The Drawing template ends with the SOLIDWORKS file extension \_\_\_\_\_.
13. A Sheet format ends with the SOLIDWORKS file extension \_\_\_\_\_.
14. Describe the procedure to insert a picture into the Sheet format.
15. True or False. You need to create a Custom Drawing Template before you create a Custom Sheet format.

**Exercises**

**Exercise 5.1:**

Create an A-ANSI-IN Landscape drawing template. Third Angle, Scale 1:2. Set Precision as illustrated.

Name Custom Drawing Template: A-ANSI-IN.

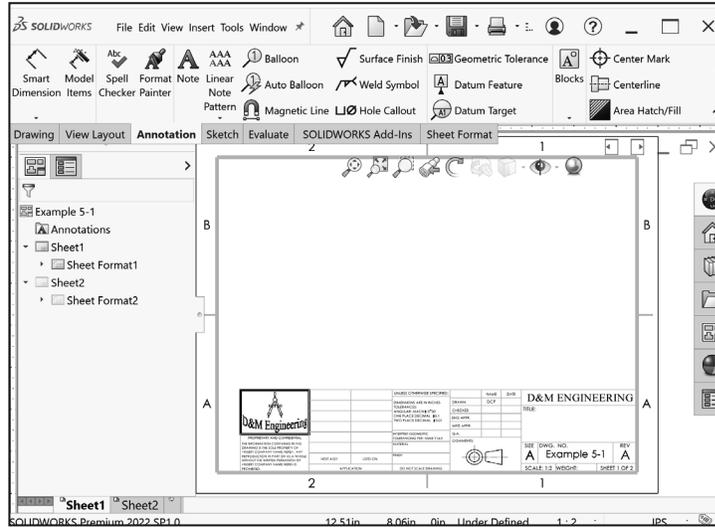
Create a Custom Sheet format. Name the Custom Sheet format: A-IN ANSI CUSTOM.

Incorporate all of the illustrated Title block information and logos. Note: Create and insert your school or company name and logo.

Use the Custom Template and Custom Sheet format to create a new drawing.

Name the new drawing Exercise 5.1.

Add a second sheet using the Custom Sheet format: A-IN-CUSTOM FORMAT.



UNLESS OTHERWISE SPECIFIED:		NAME	DATE	<b>D&amp;M ENGINEERING</b>	
DIMENSIONS ARE IN INCHES TOLERANCES: ANGULAR: MACH ± 0°30' ONE PLACE DECIMAL ±0.1 TWO PLACE DECIMAL ±0.01		DRAWN	DCP		
INTERPRET GEOMETRIC TOLERANCING PER: ASME Y14.5		CHECKED		SIZE	DWG. NO.
MATERIAL		ENG APPR.		A	Example 5-1
FINISH		MFG APPR.		SCALE: 1:2	WEIGHT:
DO NOT SCALE DRAWING		Q.A.		REV	A
		COMMENTS:		SHEET 1 OF 2	

Basic Units		
Length	inches	.12
Dual Dimension Length	millimeters	.12 .123
Angle	degrees	None

Mass/Section Properties

**Exercise 5.2:**

Create a C-ANSI-IN Landscape drawing template. Third Angle, Scale 1:2.

Set Precision as illustrated.

Name Custom Drawing Template: C-ANSI-IN.

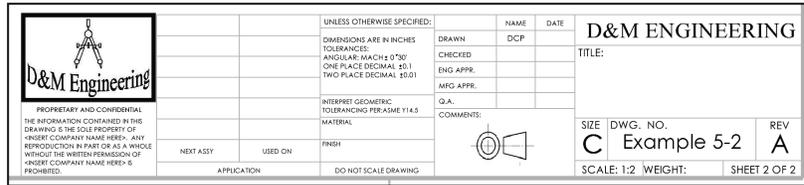
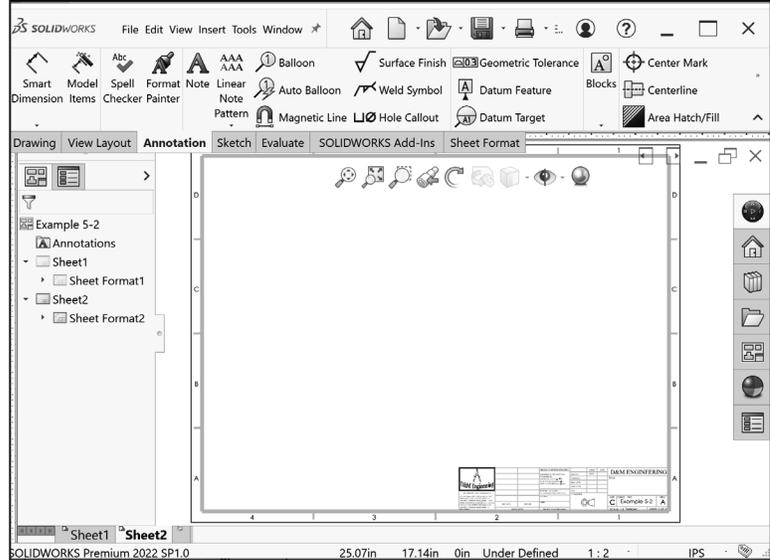
Create a Custom Sheet format. Name the Custom Sheet format: C-IN-ANSI CUSTOM.

Incorporate all of the illustrated Title block information and logos. Note: Create and insert your school or company name and logo.

Use the Custom Template and Custom Sheet format to create a new drawing.

Name the new drawing Exercise 5.2.

Add a second sheet using the Custom Sheet format: C-IN-CUSTOM FORMAT.



Basic Units			
Length	inches	.12	
Dual Dimension Length	millimeters	.12 .123	
Angle	degrees	None	
Mass/Section Properties			

**Exercise 5.3:**

Create a B-ANSI-IN Landscape drawing template.

Third Angle, Scale 1:2.

Set Precision as illustrated.

Name Custom Drawing Template: B-ANSI-IN.

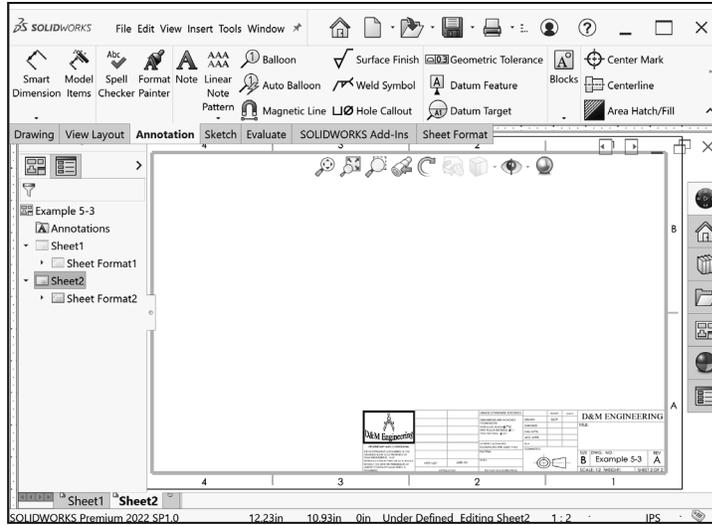
Create a Custom Sheet format. Name the Custom Sheet format: B-IN ANSI CUSTOM.

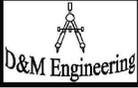
Incorporate all of the illustrated Title Block information and logos. Note: Create and insert your school or company name and logo.

Use the Custom Template and Custom Sheet format to create a new drawing.

Name the new drawing Exercise 5.3.

Add a second sheet using the Custom Sheet format: B-IN-CUSTOM FORMAT.



 <p>PROPRIETARY AND CONFIDENTIAL THE INFORMATION CONTAINED IN THIS DRAWING IS THE SOLE PROPERTY OF D&amp;M ENGINEERING. ANY REPRODUCTION IN PART OR AS A WHOLE WITHOUT THE WRITTEN PERMISSION OF D&amp;M ENGINEERING IS PROHIBITED.</p>	UNLESS OTHERWISE SPECIFIED: DIMENSIONS ARE IN INCHES TOLERANCES: ANGULAR MATCH ±0°30' ONE PLACE DECIMAL ±0.1 TWO DECIMAL ±0.01	DRAWN: DCP CHECKED: ENG APPR. MFG APPR. Q.A. COMMENTS:	NAME DATE	<p><b>D&amp;M ENGINEERING</b></p> <p>TITLE:</p> <p>SIZE DWG. NO. REV <b>B Example 5-3 A</b></p> <p>SCALE: 1:2 WEIGHT: SHEET 2 OF 2</p>
	INTERPRET GEOMETRIC TOLERANCING PER: ASME Y14.5			
	MATERIAL:			
	FINISH:			
NEXT ASSY	USED ON			
APPLICATION	DO NOT SCALE DRAWING			

Basic Units		
Length	inches	.12
Dual Dimension Length	millimeters	.12 .123
Angle	degrees	None
Mass/Section Properties		