A Hands-On Introduction to SOLIDWORKS® 2024

Text and Video Instruction

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CHAPTER 2

BASIC DRAWINGS IN SOLIDWORKS®

CHAPTER OUTLINE

CHAPTER SUMMARY

In this chapter you will learn how to create an orthographic projection in SOLIDWORKS®. You will also explore new feature-based commands such as Revolve and the Hole Wizard. By the end of this chapter, you will be able to create an orthographic projection of your part, show a pictorial on the print and edit the contents of the title block.
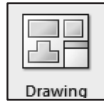
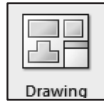## 2.1) CREATING DRAWINGS IN SOLIDWORKS®

It is very simple to create an orthographic projection of a part in SOLIDWORKS®. Once a solid model is created, it just takes a few clicks to create an orthographic projection and pictorial of the model. One thing to keep in mind while using SOLIDWORKS® to create the part print is that the program does not always follow the ASME standard. So, it is your job to inspect the part print and make any adjustments to the drawing that is required for it to conform to the ASME standard.

2.2) NEW DRAWING, USER INTERFACE, AND SETUP

2.2.1) New drawing

To create a part print or orthographic projection of a part, first a model the object must be created. Then, to create a drawing of the part select the new document icon and then drawing. SOLIDWORKS® gives many options to choose from when creating a drawing. For example, sheet size, the number and the type of views can all be chosen. Also, the user can sketch on the drawing if content needs to be added.

2.2.2) Selecting sheet size

When creating a drawing, SOLIDWORKS® will first ask for a sheet size to be chosen. (This is the physical size of the paper where the drawing will be placed.) Each sheet size (e.g. A, B, C, D) has a different physical size. These sizes are controlled by the ASME Y14.110 standard for inch sizes and ASME Y14.110M for millimeter sizes. Figure 2.2-1 shows the *Sheet Format/Size* window and the different options available.
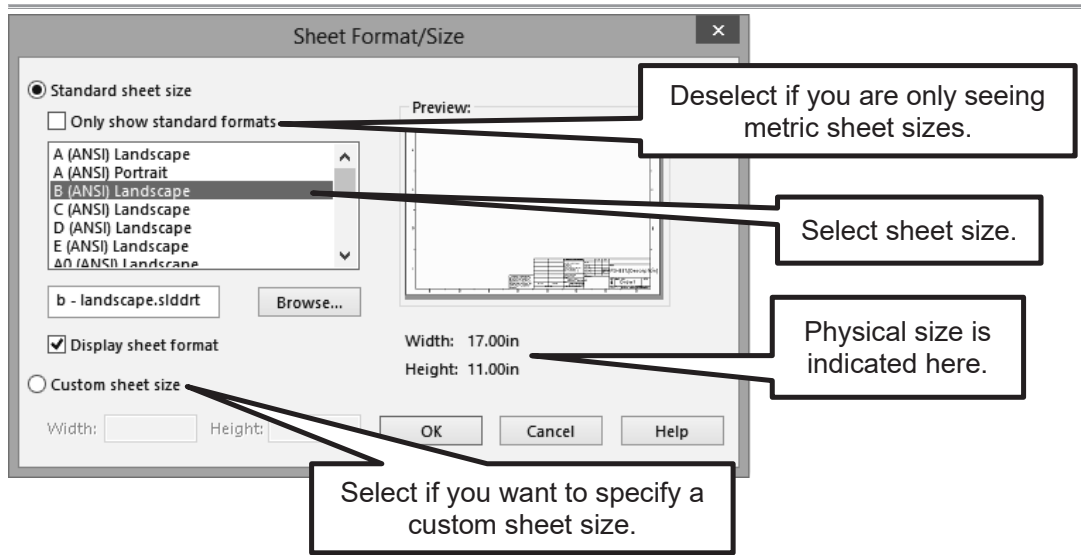
Figure 2.2-1: Selecting sheet size

2.2.3) Model view

A *Model View* window will appear which will allow the user to browse for a part or select an open part to create the drawing (see Figure 2.2-2). After the part is selected, SOLIDWORKS® will allow the selection of the number and type of views for display. The display style and scale of each view can also be chosen (see Figure 2.2-3). SOLIDWORKS® also gives the user the ability to create views after the drawing space has been entered by using the commands located in the **View Layout** tab.
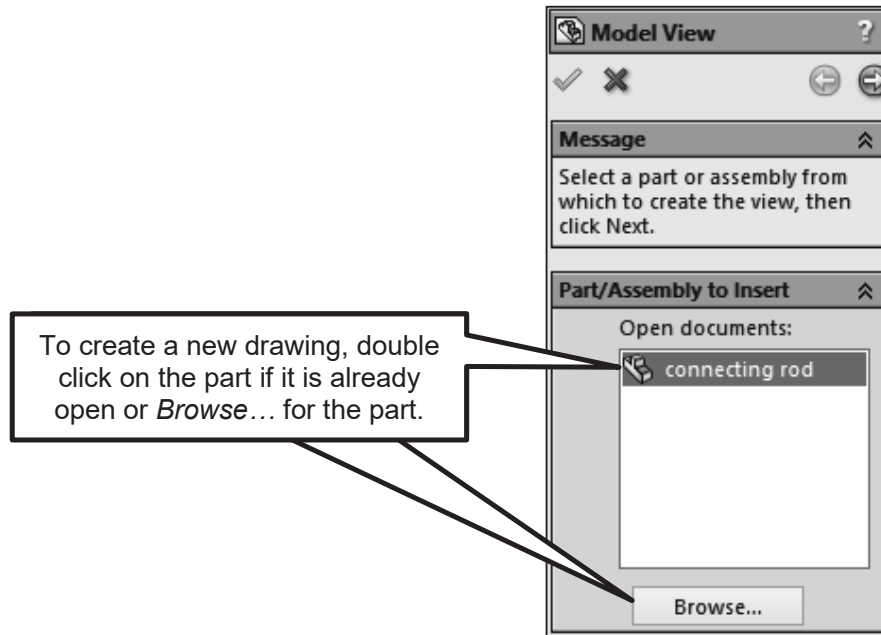
To create a new drawing, double click on the part if it is already open or *Browse…* for the part.

Figure 2.2-2: *Model View* Window

Allow multiple views

Choose views

Choose display style

Select pictorial type (Isometric is used if none are checked)

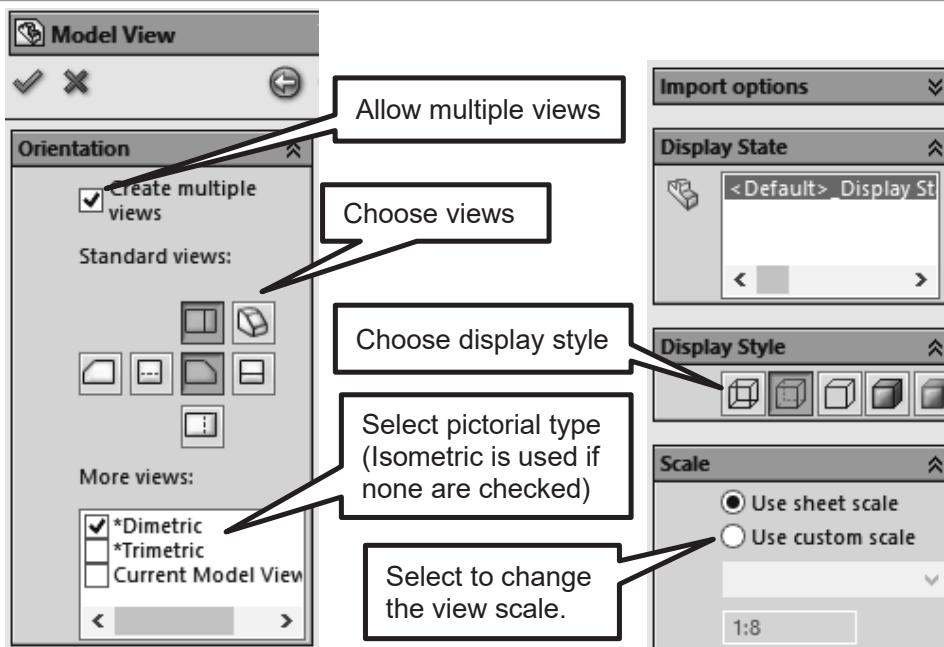Select to change the view scale.

Figure 2.2-3: Model View Options

2.2.4) Drawing user interface

The drawing user interface has commands specific to creating a part print. The commands available allows the user to create views, sketch on them, and add dimensions and annotations. Figure 2.2-4 shows the drawing user interface. The different tabs available in the Command Manager are:

- **Drawing:** The *Drawing* tab contains frequently used commands.
- **View Layout:** The *View Layout* tab contains commands that gives the ability to add views to the drawing such as section views, detailed views and auxiliary views.
- **Annotation:** The *Annotation* tab contains commands that allow the views to be annotated, such as adding dimensions, notes, and GDT symbols.
- **Sketch:** The *Sketch* tab allows the user to sketch on and modify the drawing.
- **Evaluate:** The *Evaluate* tab contains commands that allow the user to check spelling, measure distances and mark up the drawing.
- **Sheet Format:** The *Sheet Format* tab contains commands that allow for editing of the sheet format, title block and border.
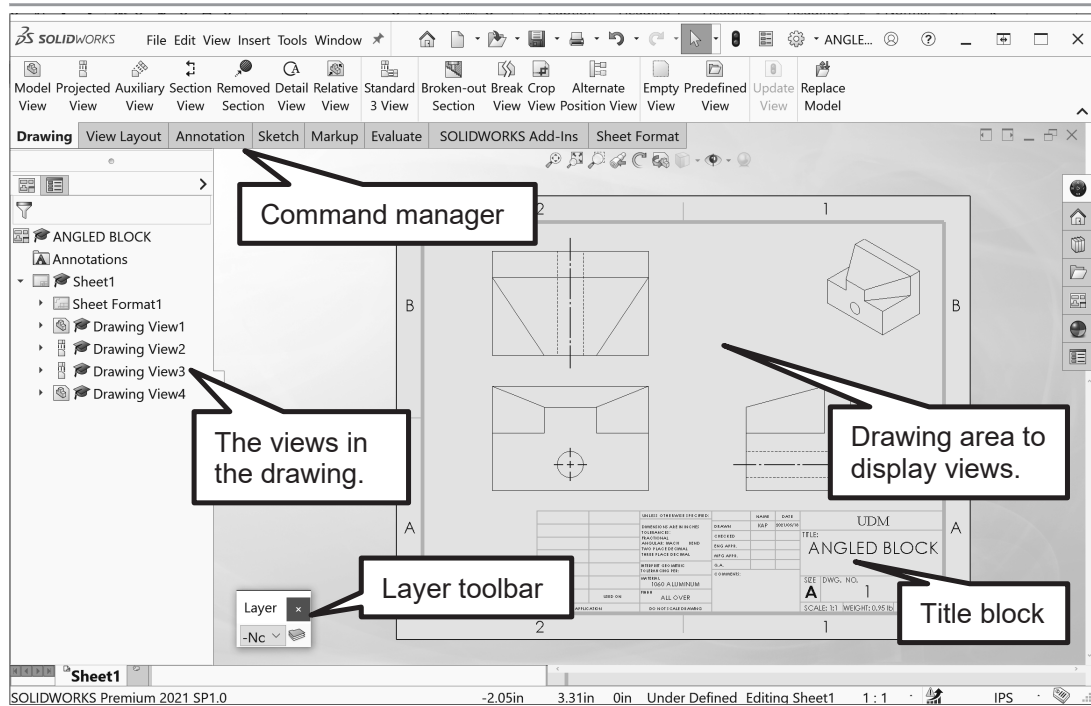

Figure 2.2-4: Drawing user interface

2.3) DRAWING VIEWS

When a new drawing is started, you select the model file that you wish to create the orthographic projection from and then you select the views you want to show. After you have entered the drawing or paper space, you can add additional views using the commands located in the **View Layout** *tab* (see Figure 2.5-1). Projected views, auxiliary views, section views, and detail views are some of the options available here.
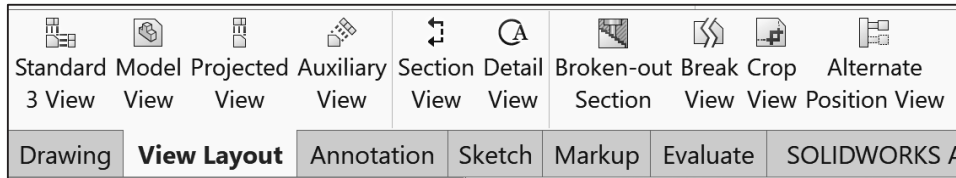

Figure 2.5-1: View Layout tab

2.3.1) Standard and projected views

If it is found that the user needs to start over with the views selected or just need to add an additional projected view, the **Standard 3 View** and **Projected View** commands can be used.

- **Standard 3 View** **:** This command creates the three standard orthographic views (i.e. front, top, right side). These views can be created using either third-angle or first-angle projection.

- **Projected View** **:** A projected view is an orthographic view created off of an existing view. The projected view can be created using either third-angle or first-angle projection.

2.4) ANNOTATIONS

Drawings may be annotated by using the commands located in the **Annotation** tab. Annotation commands include adding dimensions, notes, surface finish symbols, GDT notes and symbols, centerlines, and center marks. The *Annotation tab* is shown in Figure 2.4-1.
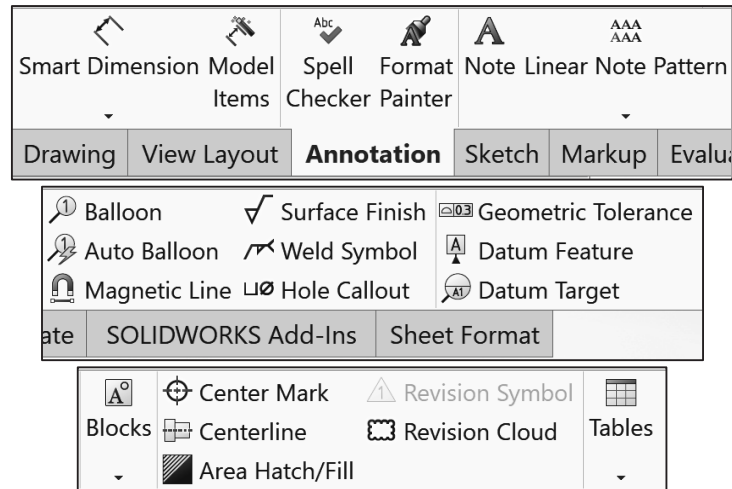

Figure 2.4-1: Annotation tab

2.4.1) Centerlines and center marks

When creating an orthographic projection, SOLIDWORKS® generally adds the center marks automatically, but does not add the centerlines. The user can add centerlines using the *Centerline* command. In many cases, even though SOLIDWORKS® has added center marks, these will need to be modified.

- **Centerlines** [Centerline] may be added automatically by selecting the **Select View** toggle and then selecting the view which will add the center lines to the entire view. They may also be added individually by selecting the two lines that define the centerline.
- **Center Marks** [Center Mark] may be added by selecting the circular feature where the center mark is wanted to be added. The *Center Mark* properties may be adjusted by changing the parameters in the *Center Mark* window (See Figure 2.3-1). You need to **deselect** the **Use document defaults** to do this. The length of the center mark may be adjusted by clicking on the center mark and dragging the grip boxes.
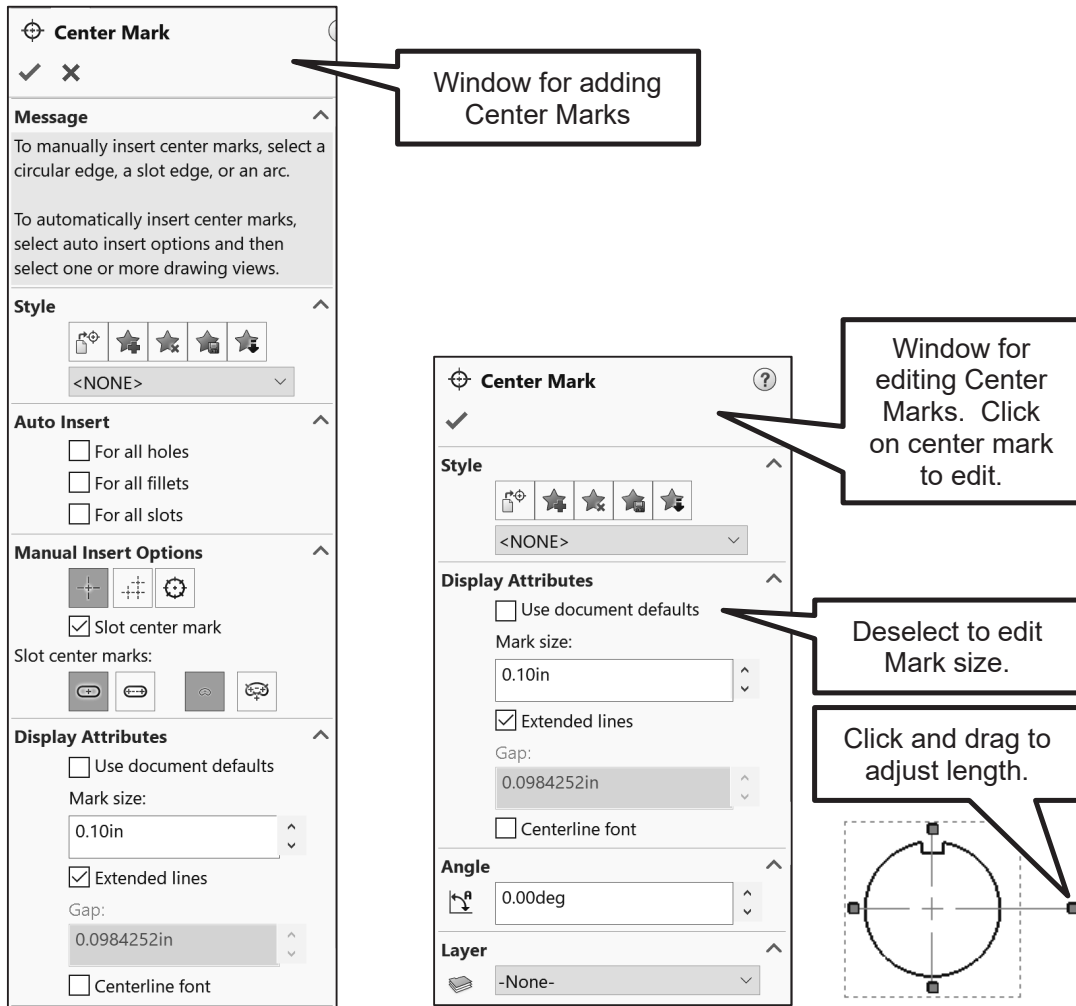
Figure 2.3-2: Center Mark window

2.4.2) Model Items

The *Model items* command allows the user to insert dimensions, annotations, and reference geometry from the model into the drawing. See Figure 2.3-3 to see all of the options available. After inserting model items, they can be deleted (**DELETE** key), dragged to another view (**SHIFT + drag**) or copied to another view (**CTRL + drag**). They also can be repositioned. For a smooth motion try using the **ALT** key while dragging. Each dimension also has grip boxes that will allow the ends of each extension line to be adjusted.
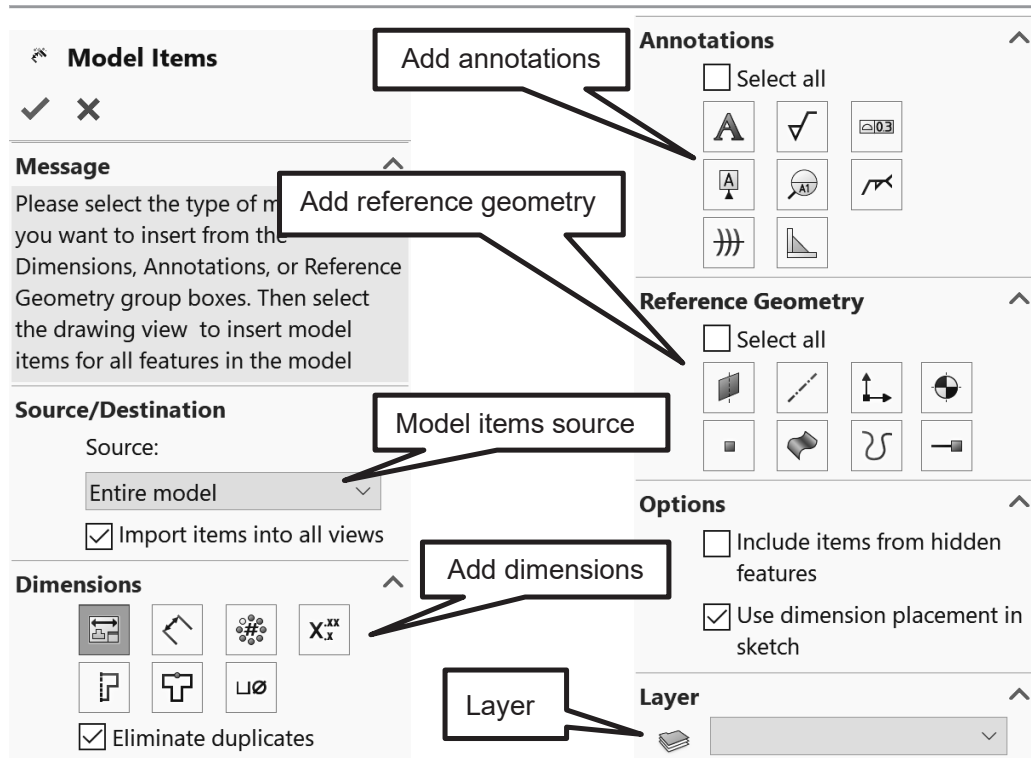

Figure 2.3-3: Model Items options

2.5) LAYERS

Layers are like transparencies, one placed over the top of another. Each transparency/layer may contain a different line type or a different part of the drawing. One layer may be used to create visible lines, while another layer may be used to create hidden lines. Assigning a different line type to each layer helps control and organize the drawing. If the Layer toolbar is not active, go to **View-Toolbars-Layer** to activate it. To create a new layer or edit a layer's properties, click on **Layer Properties** in the *Layer* toolbar. A *Layers* window will appear which will allow for the creation and editing of layers (see Figure 2.5-3).
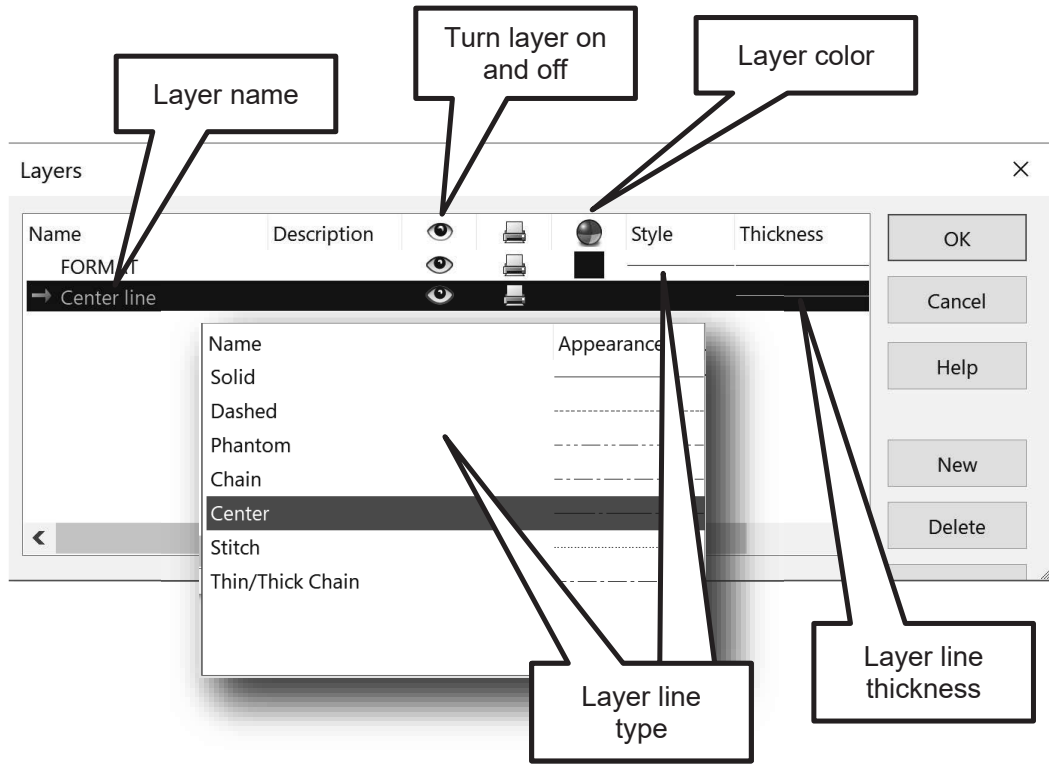
Figure 2.5-3: Layer Window

2.6) ANGLED BLOCK PRINT TUTORIAL

2.6.1) Prerequisites

To complete this tutorial, the user should have completed the listed tutorial and be familiar with the listed topics.

- Chapter 1 – Angled block tutorial
- Passing familiarity with orthographic projection.
- Ability to read dimensions.

2.6.2) What you will learn?

The objective of this tutorial is to introduce you to the SOLIDWORKS' drawing capabilities. In this tutorial, you will be creating a part print of the angled block that you modeled in Chapter 1. The part print is shown in Figure 2.6-1. Specifically, you will be learning the following commands and concepts.

Drawing

- New drawing
- Sheet size
- Standard 3 view
- Display style
- Center line
- Center mark
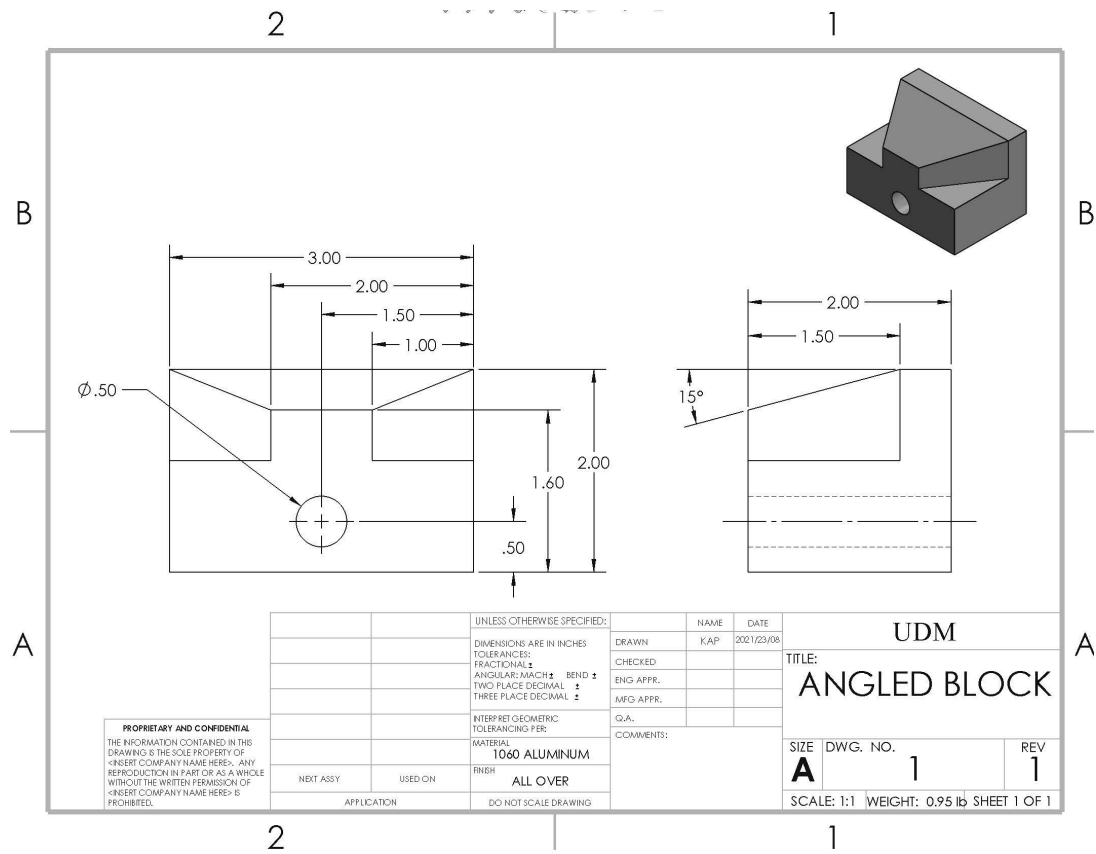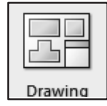- Edit sheet format
- Edit sheet

Figure 2.6-1: Angled block detail drawing

2.6.3) Orthographic Projection

It is often the case that CAD programs do not follow the ASME drawing standards when creating a part print. Therefore, it is important that the user reviews the print and decides what is correct and changes what needs to be adjusted.

1) **Open ANGLED BLOCK.SLDPRT**.

2) Start a **New Drawing** .

3) Select a **Sheet Size** of **A (ANSI) Landscape**. If only the metric sheet sizes can be seen, deselect the *Only show standard formats* check box.

> ➢ See section 2.2.2 to learn about **Selecting Sheet Sizes**.

4) Set the drafting standard to **ANSI** and set all the text to **upper case** for notes, tables, and dimensions. (*Options* – *Document Properties – Drafting Standard*)

5) Set the units to **IPS** (i.e. inch, pound, second) and set the **Decimals = .12**. Also, select the rounding option, **Round half to even**. (*Options* – *Document Properties – Units*)

6) **Save** the drawing as **ANGLED BLOCK.SLDDRW**. Remember to **Save** often throughout this tutorial.
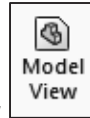
Model View

7) In the **View Layout** tab, select **Model View** . In the *Model View* window, double click on the **ANGLE BLOCK** part and then choose the following.
   a. Orientation: Select **Create multiple views** and choose to create a **Front**, **Top, Right side** and **Isometric** pictorial views.
   b. Display style = **Hidden Lines Visible**.
   c. Scale = **Use sheet scale**.
   d. ✓

> ➢ See section 2.2.3 to learn about the ***Model view*** options.

8) Click and drag the views to the approximate locations shown.
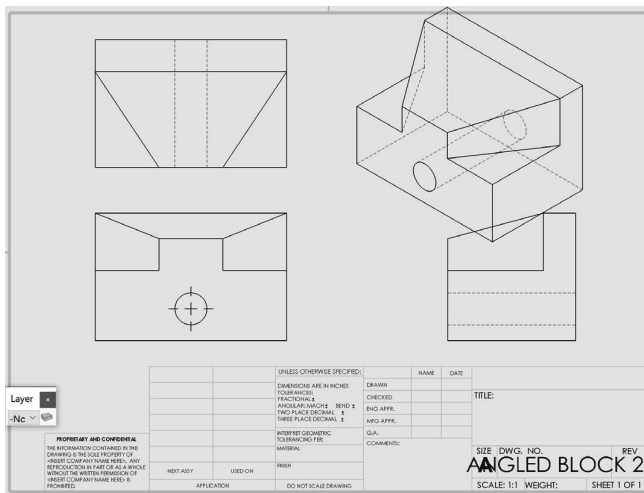
Model View

✓ ✗ ⬅

Reference Configuration ∧

🔣 Default ⌄

Orientation ∧
   ☑ Create multiple views
   Standard views:

A	◈
□	—
A	A
□	

   More views:

☐ *Trimetric
☐ *Dimetric
☐ Current Model View

Mirror ⌄

Import options ⌄

Display State ∧
   <Default>_Display State 1

Display Style ∧
⬡ ⬡ ◇ ◼ ◼

Scale ∧
   ◉ Use sheet scale
   ○ Use custom scale
   1:1 ⌄

9) Notice that the isometric pictorial is very large. So, the size needs to be reduced. Select the pictorial view in the drawing area and set the display style as **Shaded with Edges** and use a **Use custom scale** and a **1:2** scale.
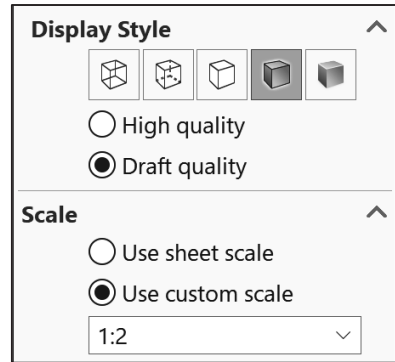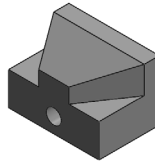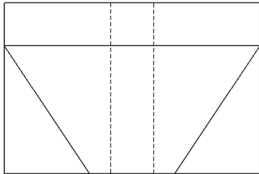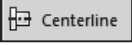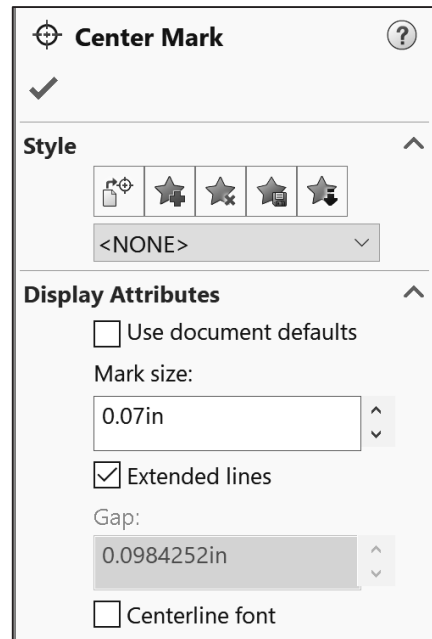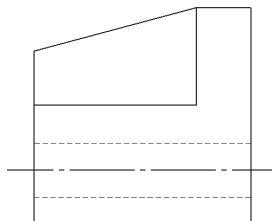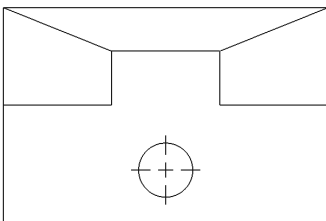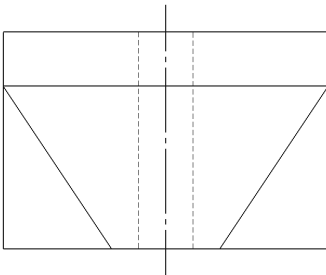
Display Style

○ High quality
● Draft quality

Scale

○ Use sheet scale
● Use custom scale

1:2
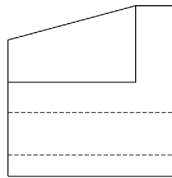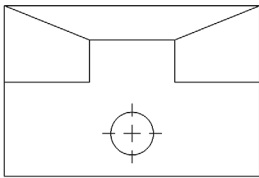
10) Add missing centerlines. Click on **Centerlines** ⊞ Centerline in the *Annotation* tab. Select the **Select View** check box, and then select the **top view** and **right-side view**. Click **OK** in the *Centerline* window.

11) Adjust the center mark in the front view by clicking on it. Deselect the **Use document defaults** check box and adjust the *Mark size* to **0.07in**.

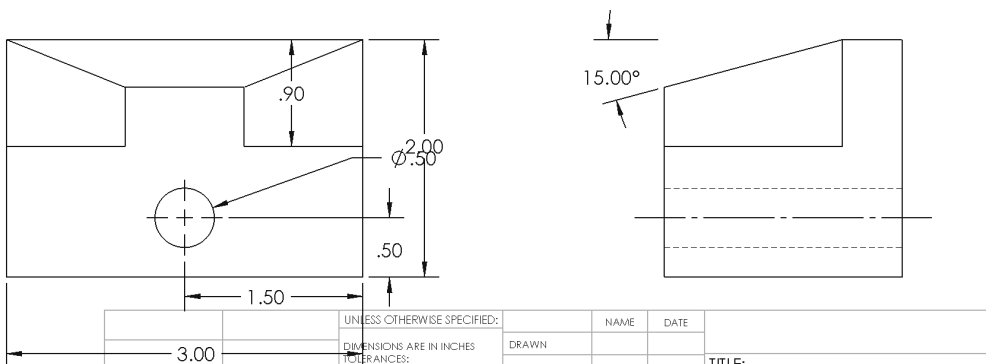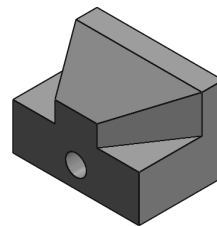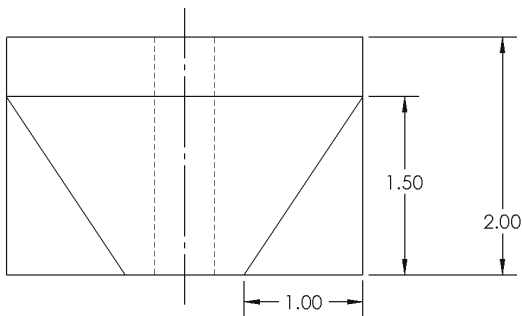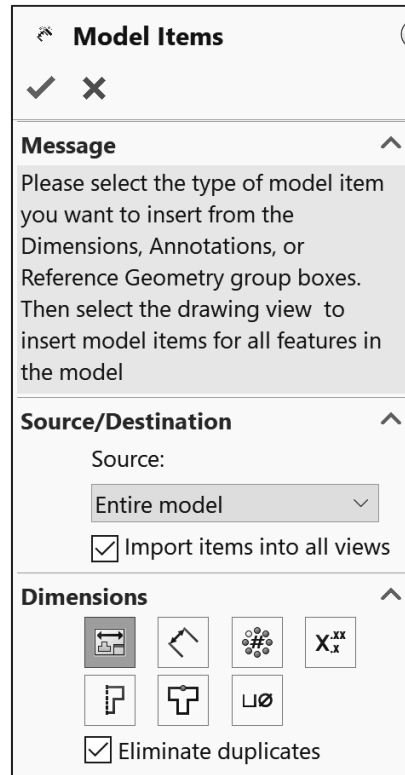> ➢ See section 2.4.1 to learn about *Centerlines & Center Marks*.

⊕ **Center Mark** ?

✓

Style

<NONE>

Display Attributes

☐ Use document defaults

Mark size:

0.07in

☑ Extended lines

Gap:

0.0984252in

☐ Centerline font
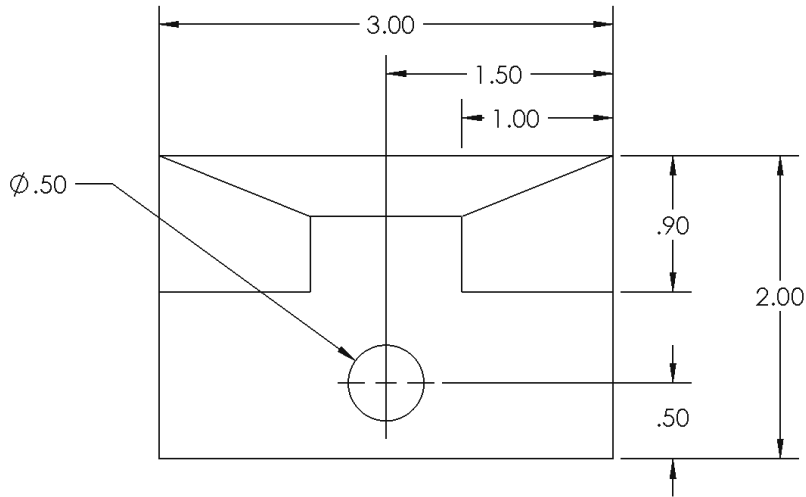
2.6.4) Dimensions

1) Add **Model Items** (*Annotation* tab) [Model Items] to the drawing using the following settings. Ignore the fact that the dimensions are jumbled and overlapping. They will be moved to more appropriate locations.
   a) Source = **Entire model**
   b) Activate the **Import items into all views** check box.
   c) Dimensions = **Marked for drawing dimensions**
   d) Activate the **Eliminate duplicates** check box.

> ➢ See section 2.4.2 to learn about the **Model items** options.
> ➢ Note: Don't worry, the dimensions may look messy. They may look similar to what is shown, or they may look entirely different. It is okay. This is where a good knowledge of Engineering Graphics comes in handy. Later, the print will be changed so that it is easily readable and presentable.

Model Items

Message

Please select the type of model item you want to insert from the Dimensions, Annotations, or Reference Geometry group boxes. Then select the drawing view to insert model items for all features in the model

Source/Destination

Source:

| Entire model ⌄ |

☑ Import items into all views

Dimensions

☑ Eliminate duplicates

2) The *Front* view will be focused on first. Click and drag the dimensions to the approximate locations shown in the figure. If a dimension is not currently in the front view, click on the dimension, hold the **SHIFT** key down and drag it to the front view.

3) **Delete** the **.90** dimension and add the **1.10** and **2.00** dimensions using **Smart dimension** .
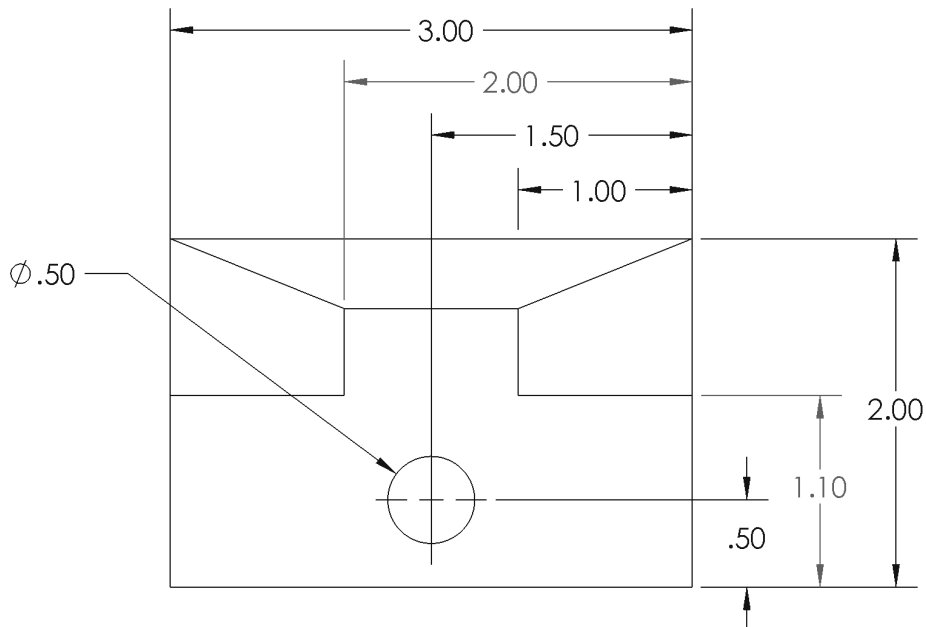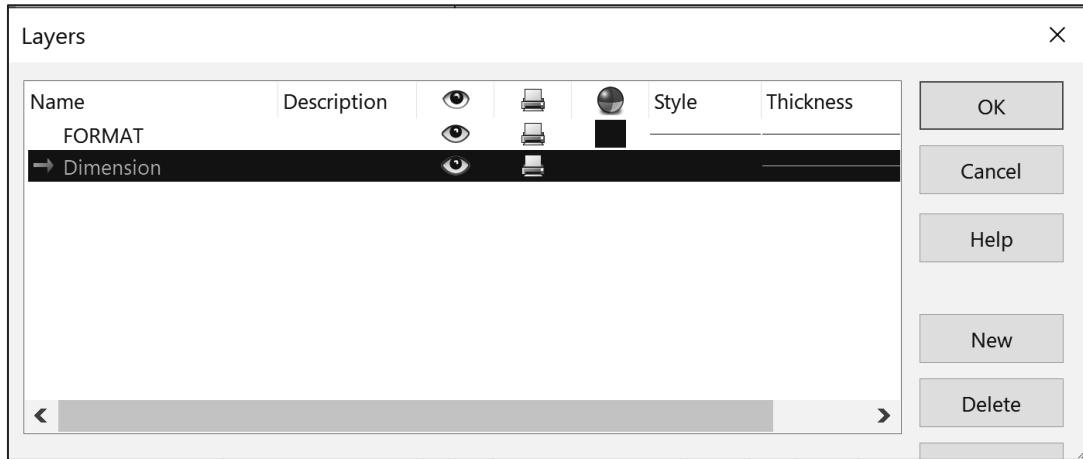
4) Notice that the added dimensions are grey. To make them black a layer must be created and the dimensions must be placed on it. If the *Layers* toolbar is not showing (it should be in the bottom left corner), activate it (**View – Toolbars – Layer**).

5) Create a **New** ⬙ layer called **Dimension**.

| Layers								✕
Name	Description	👁	🖶	⬤	Style	Thickness		**OK**
FORMAT		👁	🖶	⬛				**Cancel**
→ Dimension		👁	🖶					**Help**
								New
‹						›		**Delete**

➢   See section 2.5 to learn about *Layers*.

6) Select the 2.00 and 1.10 dimension and then select the dimension layer in the Layer toolbar. This will place those two dimensions on that layer. The dimension should now be black.

Layer	✕
Dim ⌄	⬙

7) Notice that the gaps between some of the extension lines and the object are not ideal. To change this, click on the dimension. Grip boxes will appear. Drag the grip boxes to an appropriate location. For examples, see the figure.
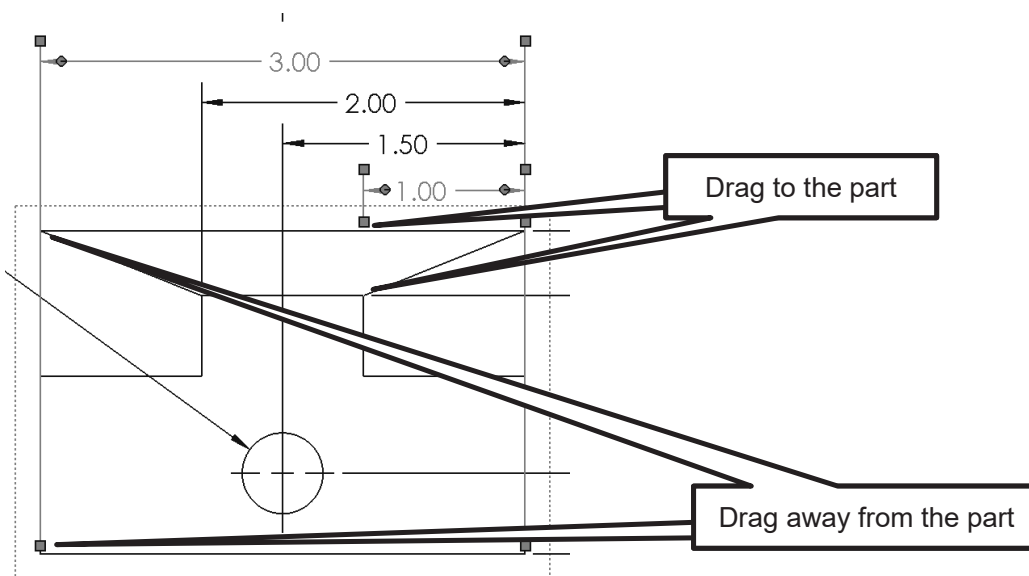
8) Drag the dimensions and the dimension text into the correct locations. Holding the **ALT** key will produce a smooth drag. The final *Front* view should look like the figure.
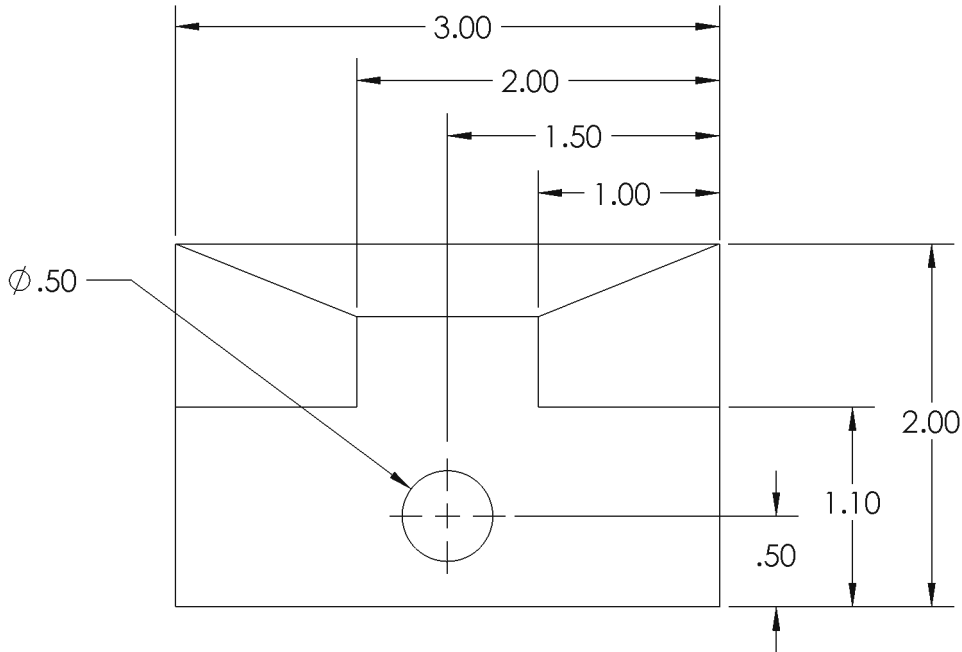

9) Using the **SHIFT** key, drag the dimensions in the *Top* view into the *Right-Side* view. Then, make any adjustments to make the Right-Side view look like the figure.
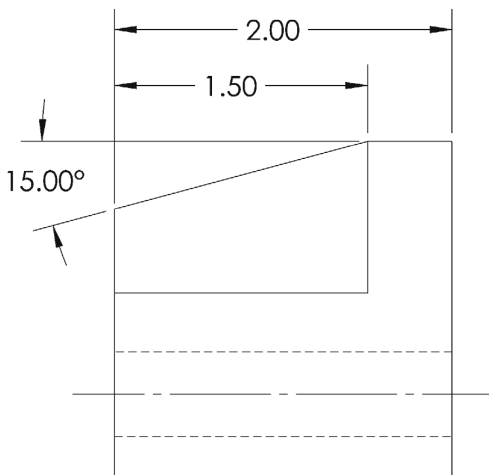
10) If the angle dimension has decimals, click on it and fix it in the *Dimension* options window.
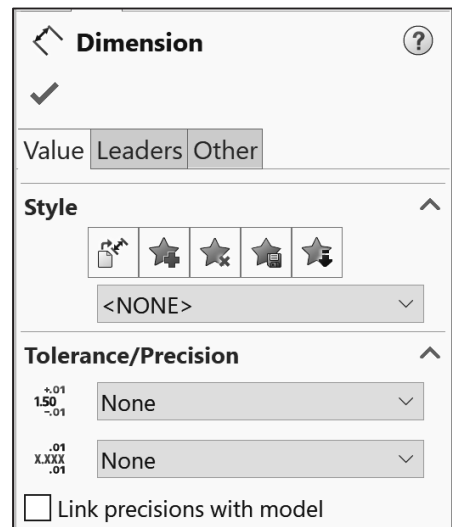
11) Notice that there are no dimensions on the *Top* view. We can keep it, but we would have to change the scale of the views. In this case, let's just delete the *Top* view. The pictorial will fill in any information lost by deleting the *Top* view.
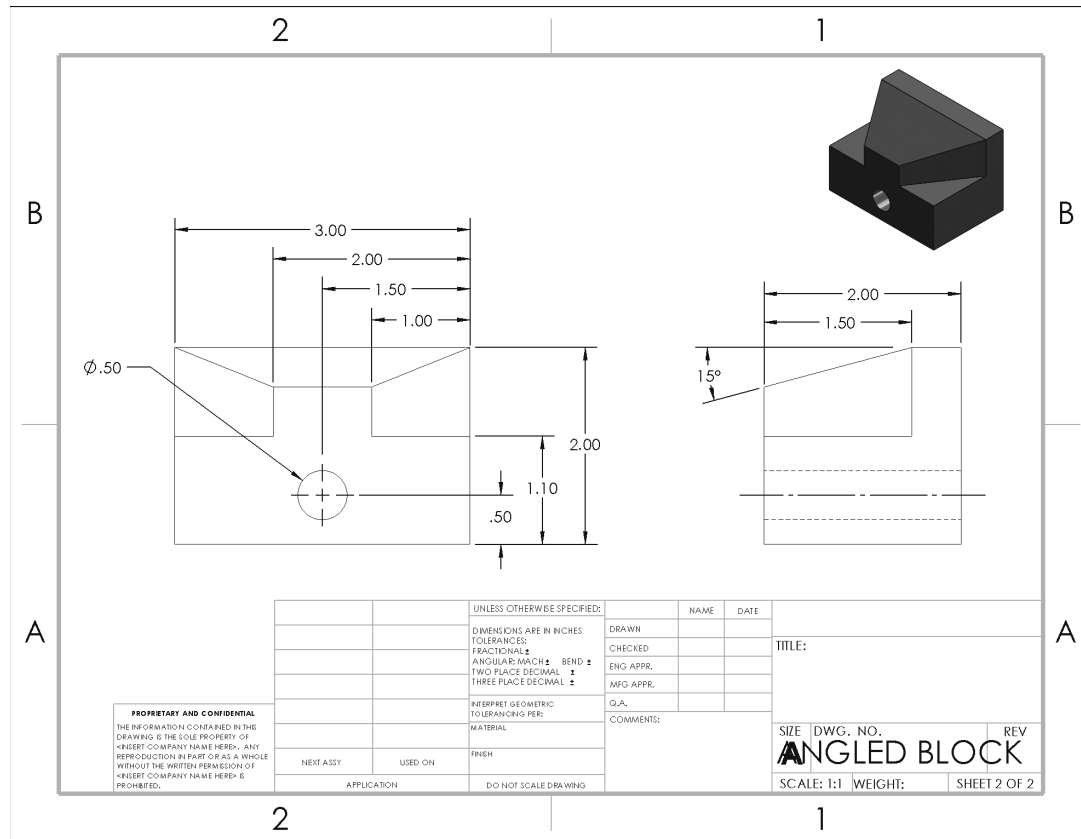
2.6.5) Filling in the title block

1) **Right-click** on the drawing somewhere outside the views but inside the drawing area and select **Edit Sheet Format**. Notice that the drawing will disappear.

2) Move the mouse around the title block and notice that every once in a while a text symbol will appear. This indicates a text field. To enter text, just double click on the field.
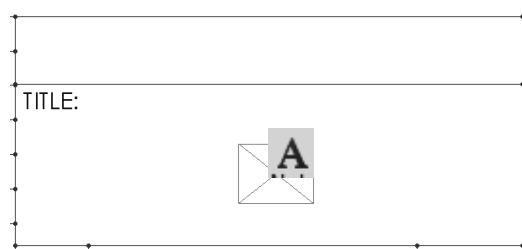
3) Enter the following text. Note that all text should be capitalized. You can use the formatting window to increase or decrease the text size as needed.
   a) TITLE = **ANGLE BLOCK**
   b) Above the title place your school's name.
   c) DWG. NO. = **1**
   d) REV = **1**
   e) WEIGHT = Enter the value listed in the *Mass Properties*
   f) DRAWN NAME = your initials
   g) DRAWN DATE = enter the date in this format (YYYY/MM/DD)
   h) MATERIAL = **1060 ALUMINUM**
   i) FINISH = **ALL OVER**

4) Get out of the title block by **right-clicking** on the drawing and selecting **Edit Sheet**.

		UNLESS OTHERWISE SPECIFIED:		NAME	DATE		UDM	
		DIMENSIONS ARE IN INCHES	DRAWN	KAP	2021/05/18			
		TOLERANCES: FRACTIONAL	CHECKED			TITLE:		
		ANGULAR: MACH BEND TWO PLACE DECIMAL	ENG APPR.			ANGLED BLOCK		
		THREE PLACE DECIMAL	MFG APPR.					
		INTERPRET GEOMETRIC TOLERANCING PER:	Q.A.					
			COMMENTS:					
		MATERIAL 1060 ALUMINUM				SIZE	DWG. NO.	REV
NEXT ASSY	USED ON	FINISH ALL OVER				A	1	1
	APPLICATION	DO NOT SCALE DRAWING				SCALE: 1:1	WEIGHT: 0.95 lb	SHEET 1 OF 1

5) **Save** your drawing.

2.7) DEFINING A FRONT VIEW TUTORIAL

2.7.1) Prerequisites

To complete this tutorial, the user should have completed the listed tutorial and be familiar with the listed topics.

- Chapter 2 – Angled block part print tutorial
- Passing familiarity with orthographic projection.

2.7.2) What you will learn?

The objective of this tutorial is to learn how to change the defined front view. The orthographic projection for a simple U-block is shown in Figure 2.7-1. The front view is not ideal. Specifically, the user will be learning how to change the view by learning the following commands and concepts.
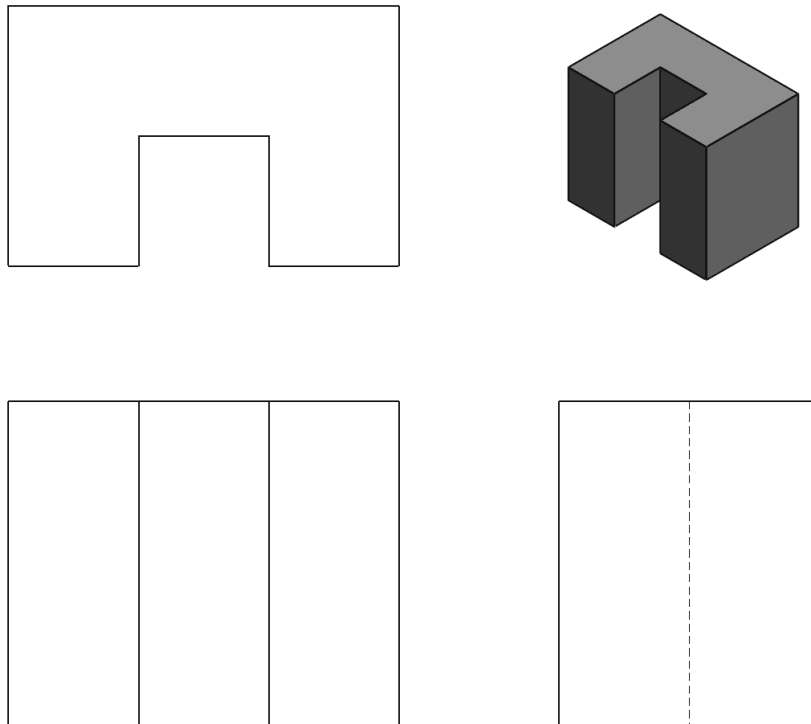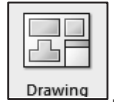
Drawing

- Update Standard View

Figure 2.7-1: U-Block initial orthographic projection

2.7.3) The initial Orthographic Projection

1) **Open** the part file **U-BLOCK.SLDPRT** that came with this book.

2) Start a **New Drawing** [Drawing].

3) Select a **Sheet Size** of **A4 (ANSI) Landscape.**

➢   See section 2.2.2 to learn about *Selecting Sheet Size*.

4) Double-click on **U-BLOCK** in the *Model View* window.

5) Create a **Front, Top,** and **Right-side view** along with an **Isometric pictorial** with a *Display style* of **Hidden Lines Visible**. Use the sheet scale.

6) Change the *Scale* of the pictorial to **1:2** and the *Display style* to **Shaded with Edges**.

7) Notice that the view that should be the Front view is currently the Top view. We need to change this.
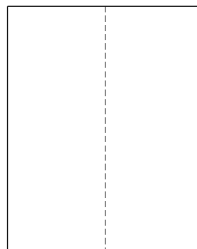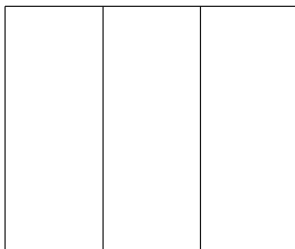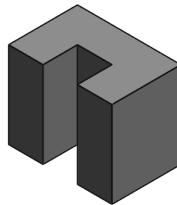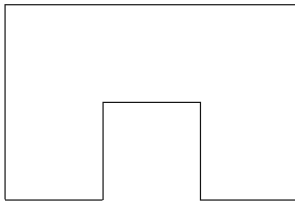
2.7.4) Changing the Front View

1) Enter the **U-BLOCK.SLDPRT** model file (**Window – U-BLOCK**).

2) Hit the **SPACE** bar to activate the *View Cube* and view the block from the view shown in the figure.

Click
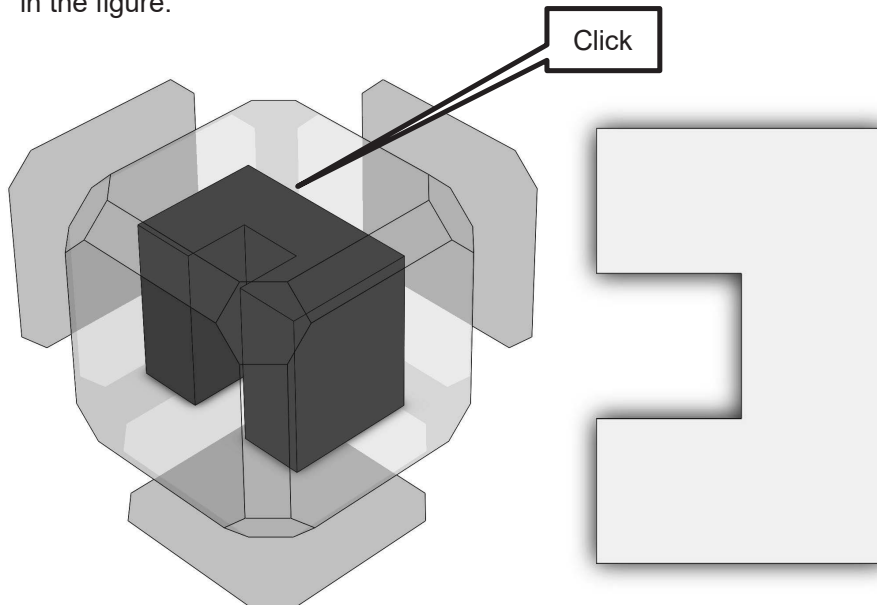
3) Notice that the view is not in the correct orientation for the *Front* view. Hold down the **SHIFT** key and then hit the **Left arrow – Up arrow – Right arrow** in sequence. This should place the view in the proper orientation. If it does not, continue figuring out the proper arrow sequence until the *Front view* looks like the figure.
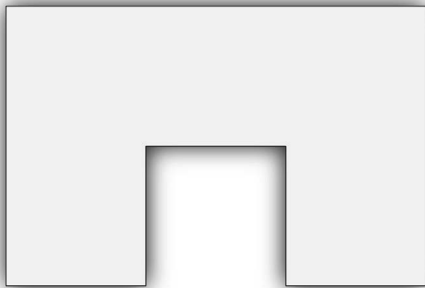
4) Hit the **SPACE BAR**. In the *Orientation* window, click on **Update Standard Views** and then click on **Front** view . Click **Yes** if a warning window appears.
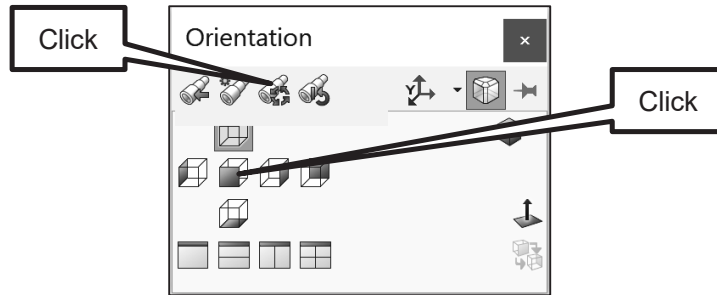
5) Enter the **U-BLOCK.SLDDRW** drawing file. Notice that the views have updated.
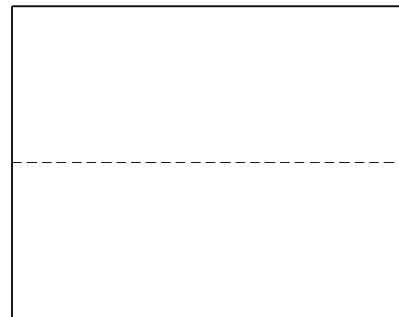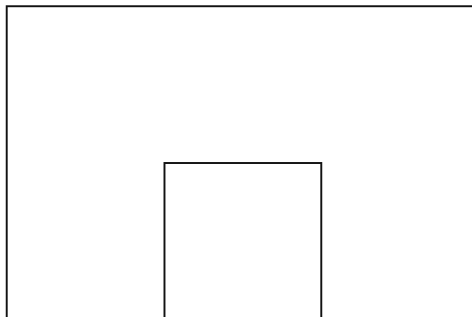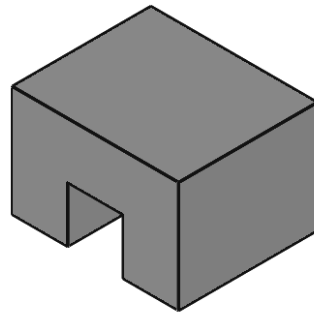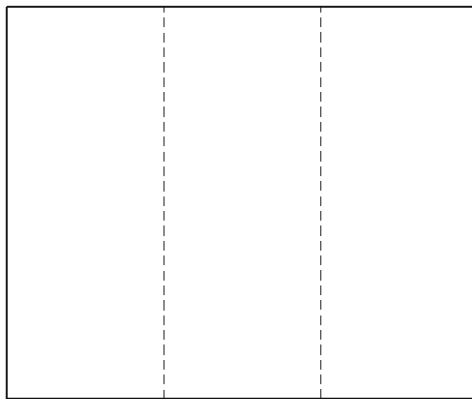
NOTES:

CREATING BASIC DRAWINGS IN SOLIDWORKS® PROBLEMS

P2-1)   Use SOLIDWORKS® to create the part print that looks exactly like the detailed drawing of the model shown in P1-1. Fill in the appropriate information into your title block. Enter the weight of the part into the title block of the drawing.

P2-2)   Use SOLIDWORKS® to create the part print that looks exactly like the detailed drawing of the model shown in P1-2. Fill in the appropriate information into your title block. Enter the weight of the part into the title block of the drawing.

P2-3)   Use SOLIDWORKS® to create the part print that looks exactly like the detailed drawing of the model shown in P1-3. Fill in the appropriate information into your title block. Enter the weight of the part into the title block of the drawing.

P2-4)   Use SOLIDWORKS® to create the part print that looks exactly like the detailed drawing of the model shown in P1-4. Fill in the appropriate information into your title block. Enter the weight of the part into the title block of the drawing.

P2-5)   Use SOLIDWORKS® to create the part print that looks exactly like the detailed drawing of the model shown in P1-5. Fill in the appropriate information into your title block. Enter the weight of the part into the title block of the drawing.

P2-6)   Use SOLIDWORKS® to create the part print that looks exactly like the detailed drawing of the model shown in P1-6. Fill in the appropriate information into your title block. Enter the weight of the part into the title block of the drawing.

P2-7)   Use SOLIDWORKS® to create the part print that looks exactly like the detailed drawing of the model shown in P1-7. Fill in the appropriate information into your title block. Enter the weight of the part into the title block of the drawing.

P2-8)   Use SOLIDWORKS® to create the part print that looks exactly like the detailed drawing of the model shown in P1-8. Fill in the appropriate information into your title block. Enter the weight of the part into the title block of the drawing.

P2-9)   Use SOLIDWORKS® to create the part print that looks exactly like the detailed drawing of the model shown in P1-9. Fill in the appropriate information into your title block. Enter the weight of the part into the title block of the drawing.

P2-10)   Use SOLIDWORKS® to create the part print that looks exactly like the detailed drawing of the model shown in P1-10. Fill in the appropriate information into your title block.   Enter the weight of the part into the title block of the drawing.

P2-11)   Use SOLIDWORKS® to create the part print that looks exactly like the detailed drawing of the model shown in P1-11. Fill in the appropriate information into your title block.   Enter the weight of the part into the title block of the drawing.

P2-12)   Use SOLIDWORKS® to create the part print that looks exactly like the detailed drawing of the model shown in P1-12. Fill in the appropriate information into your title block.   Enter the weight of the part into the title block of the drawing.

P2-13) Use SOLIDWORKS® to create the part print that looks exactly like the detailed drawing of the model shown in P1-13. Fill in the appropriate information into your title block.   Enter the weight of the part into the title block of the drawing.

P2-14) Use SOLIDWORKS® to create the part print that looks exactly like the detailed drawing of the model shown in P1-14. Fill in the appropriate information into your title block.   Enter the weight of the part into the title block of the drawing.