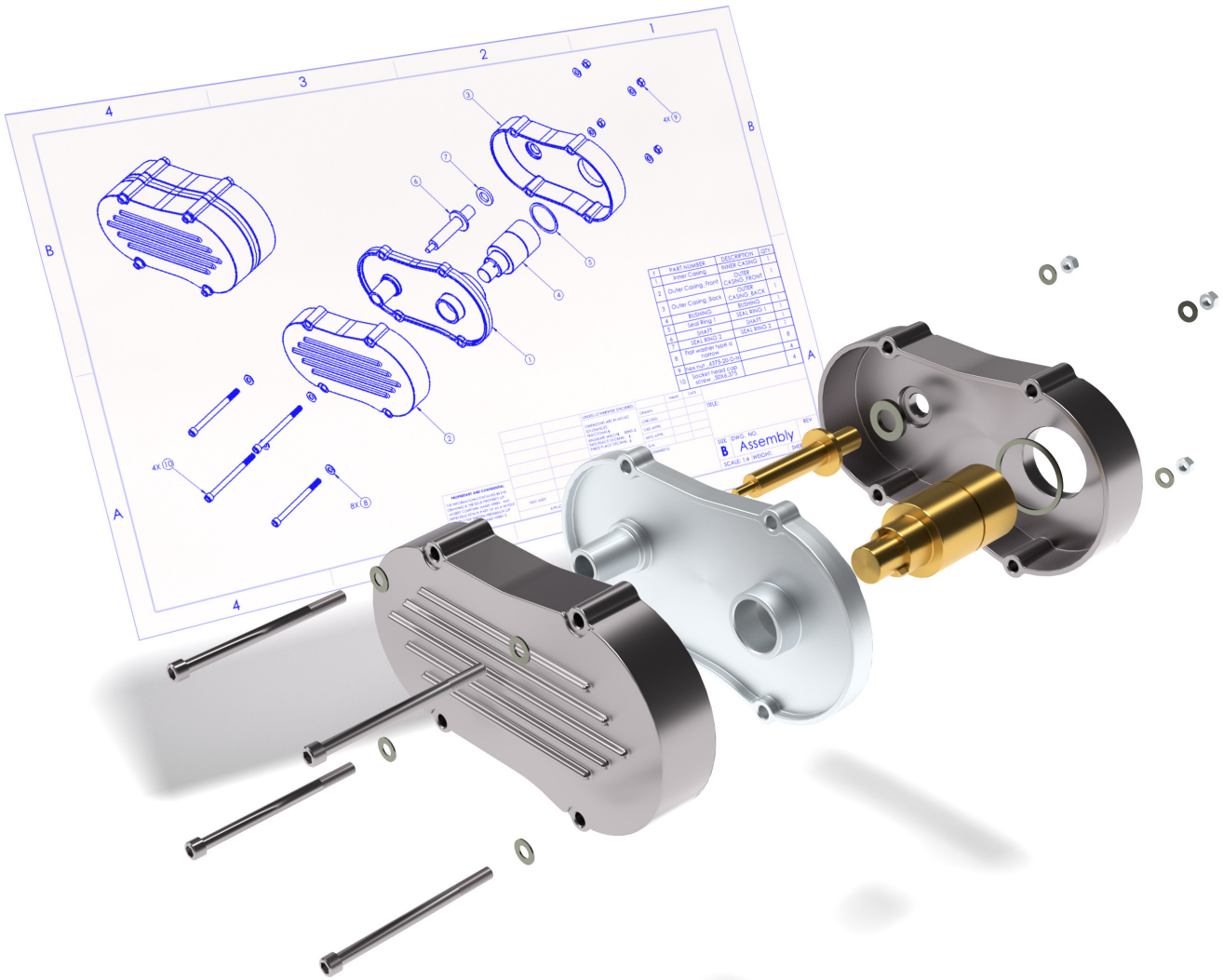
New Chapter Covering the CSWPA-DT Exam

Mastering Modern CAD Drawings with SOLIDWORKS® 2025

Applying ASME Standards to Engineering Drawings



Lani Tran, CSWE

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Chapter 2: Detailing a Machined Part

Stainless Steel Flange

Machined parts are parts manufactured using material removal such as a lathe, a mill, or a router for machining operation.

Machining parts is an excellent alternative to other manufacturing operations such as molding, casting, extrusion, etc.

Machined parts do have some advantages compared to 3D printed, injection molded, or casted parts such as:

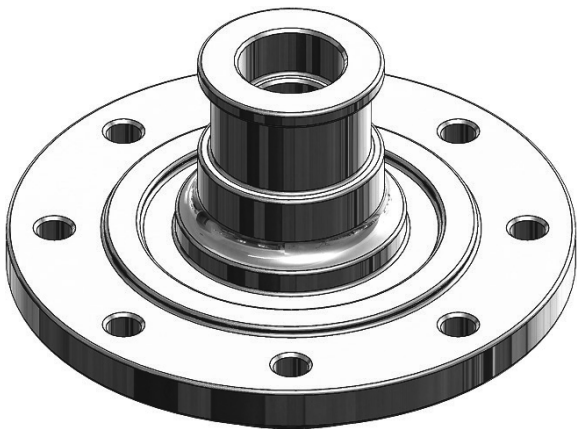
* They do not require any special tooling; you just need to clamp down the work piece and start machining.

* Unlike injection molded or sheet metal parts, machined parts can have different wall thickness and can be flipped and cut from different sides as needed.

* Machined parts can hold tighter tolerance and can produce higher accuracy and precision.

A wider variety of material options are also available for machined parts.

This chapter discusses the process of creating an engineering drawing for the machined flange.



1. Opening a part document:

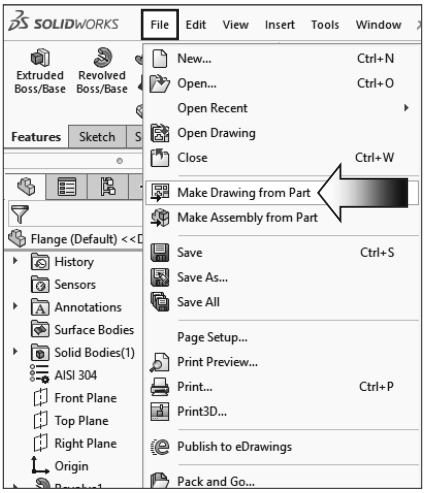
Select: **File, Open**.

Browse to the Training Folder and open the part document named: **Flange.sldprt**.

The material **AISI 304** has already been assigned to the part. AISI stands for American Iron and Steel Institute. This material is the most common stainless steel used in the industry.

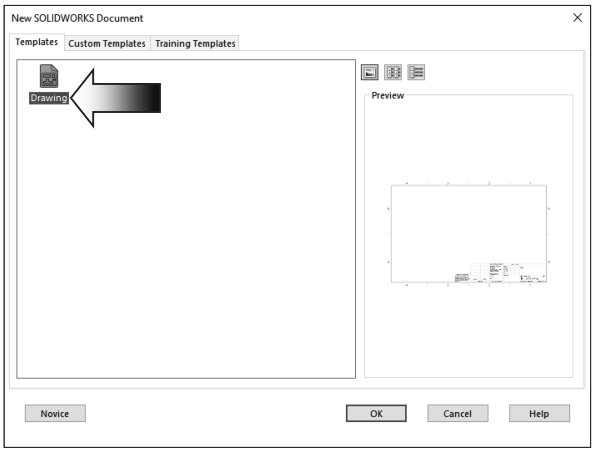
2. Setting the drawing properties:

A part, or a 3D model, is designed and created in a 3D environment and a detailed drawing is created on a sheet of paper, in a 2D environment.



To generate a drawing from a part, first we have to transfer the part to a drawing sheet.

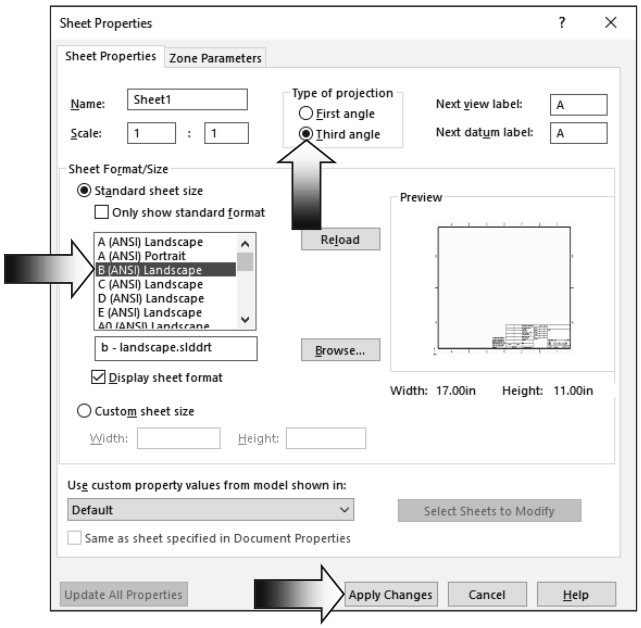
Select: **File, Make Drawing From Part** (arrow).



Select **Drawing** (or Draw) template from the New SOLIDWORKS Document dialog box.

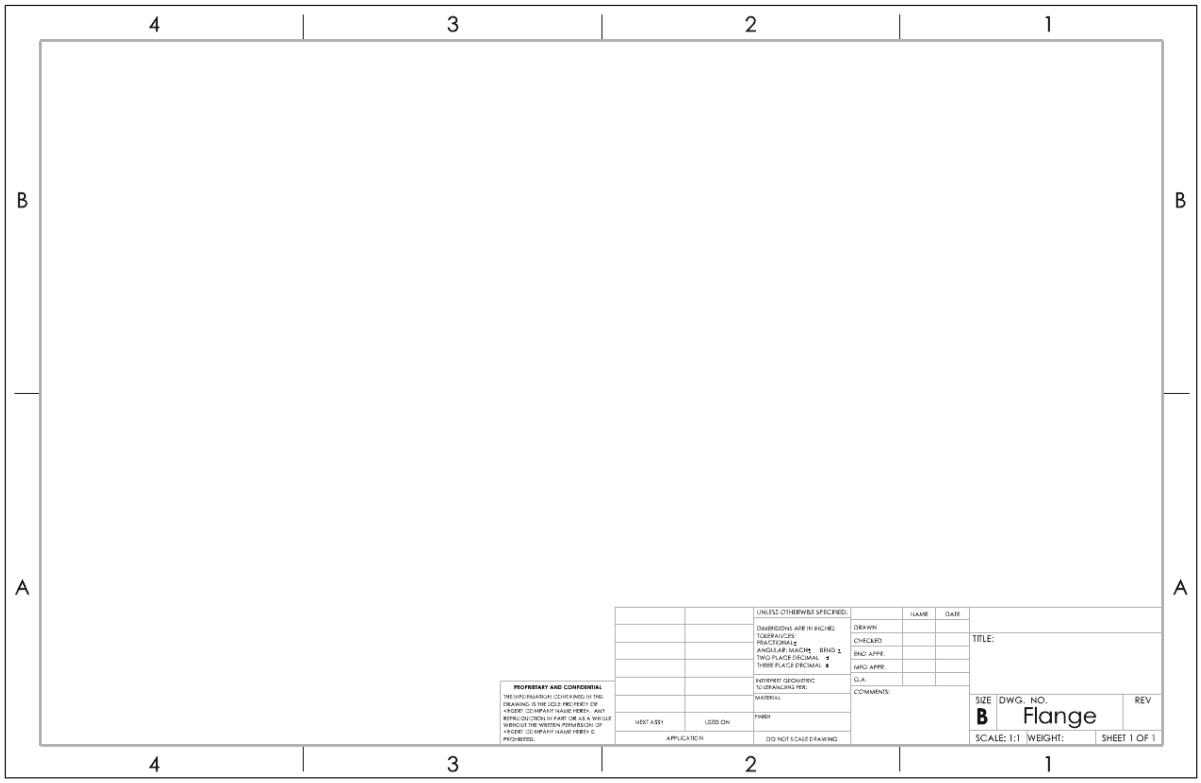
In the Sheet Properties dialog box, select the following:

* Scale: **1:1**
* Type Of Projection: **Third Angle**
* Sheet Format/Size: **B (ANSI) Landscape**
* Next View Label: **A**
* Next Datum Label: **A**
* Display Sheet Format: **Enabled**



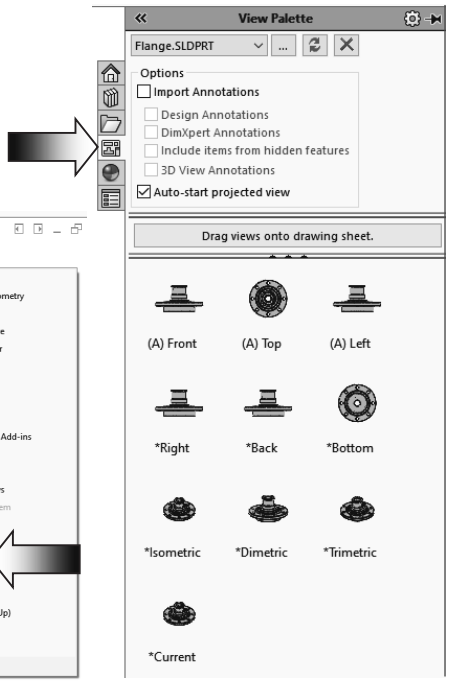
Click **Apply Changes**.

The **B (ANSI) Landscape** drawing template is opened (the paper size is 11” x 17”).

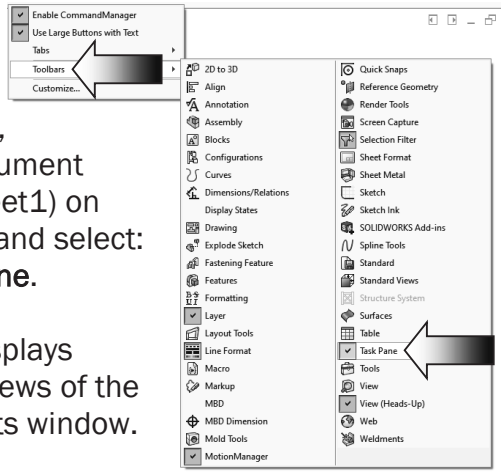


3. Creating an engineering drawing:

SOLIDWORKS offers several options to create the drawing views. We will take a look at the View Palette option first.



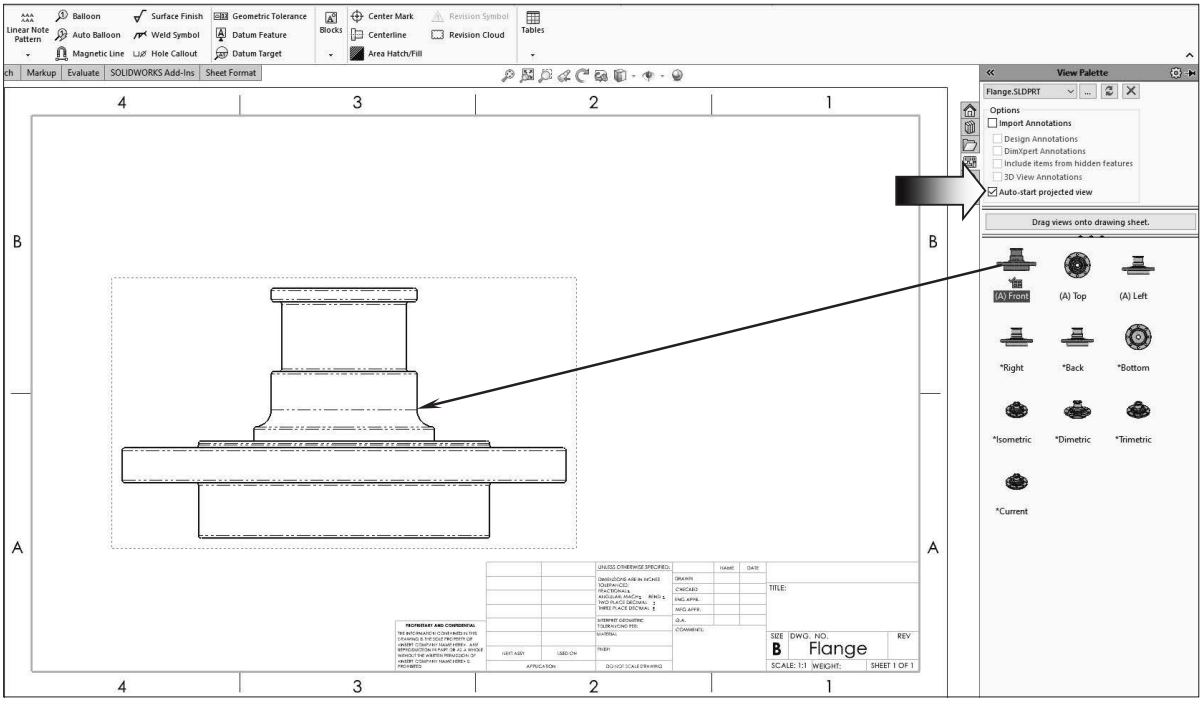
If the Task Pane is not visible on the upper right side of the screen, right-click the document name (Flange-Sheet1) on top of the screen and select: **Toolbars, Task Pane**.



The Task Pane displays several drawing views of the current model in its window.

In the Task Pane, under Options, enable only one checkbox: **Auto-Start Projected View** and clear all other checkboxes.

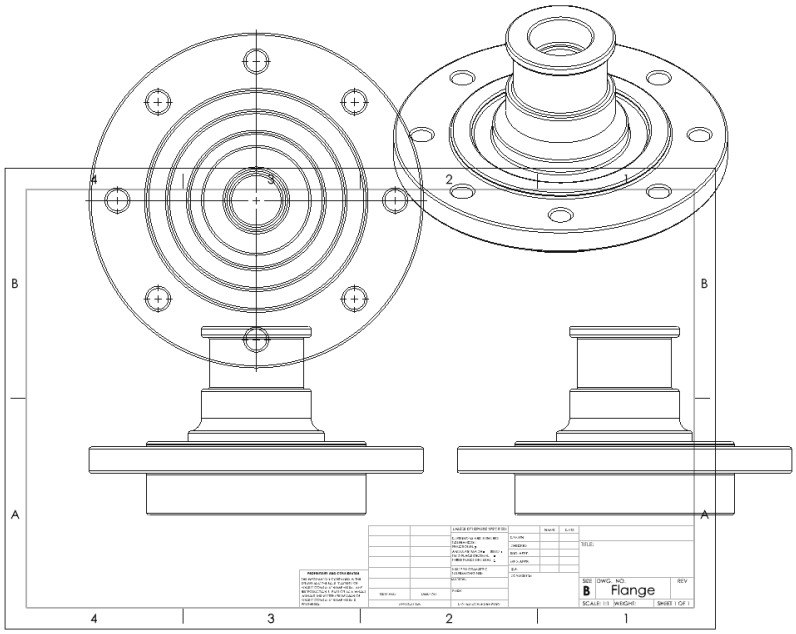
Locate the **Front** drawing view and drag/drop it onto the drawing as shown below.



The **Projected View** command is activated automatically to allow for more views to be created.

Move the mouse pointer upwards; when the preview of the **Top** drawing view appears, click the mouse to make the Top drawing view.

Also add the **Right** and the **Isometric** views.

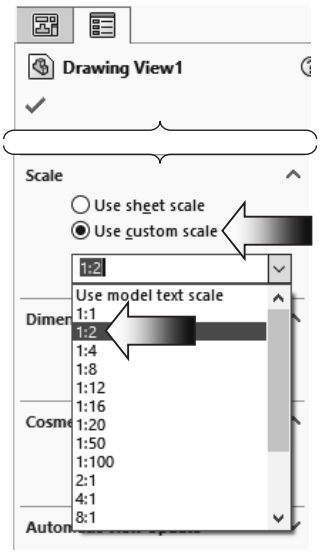


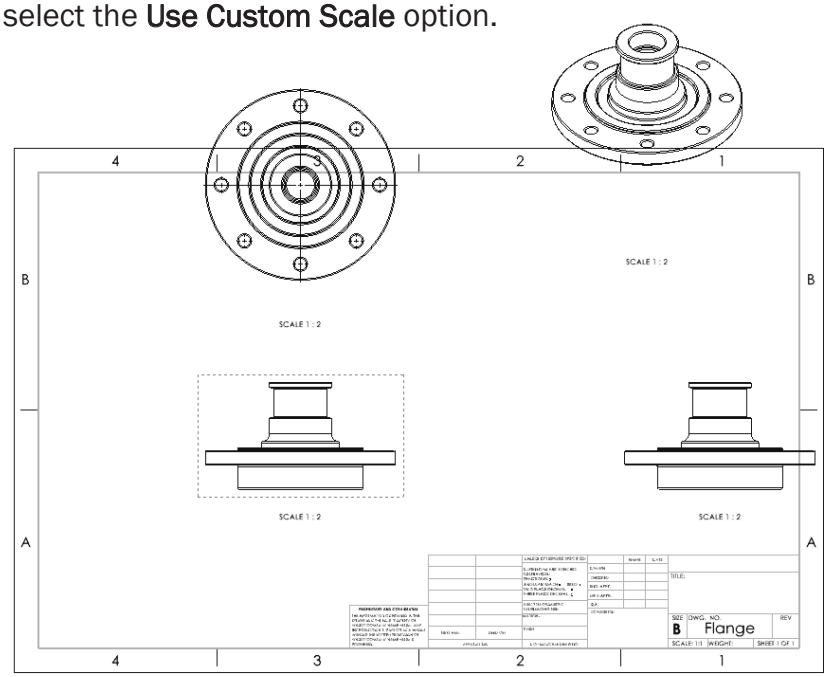
4. Changing the scale of the drawing Views:

The Front drawing view was added to the drawing first; therefore, it is considered the parent view. Changing the scale of the parent view will also update the other drawing views as well.

Click the dotted border of the **Front** drawing view (or click any where inside the Front drawing view) to access the scale options.

In the **Scale** section, select the **Use Custom Scale** option.





Expand the **Scale** drop-down arrow and select: **1:2** (one-half scale).

Click **OK**.

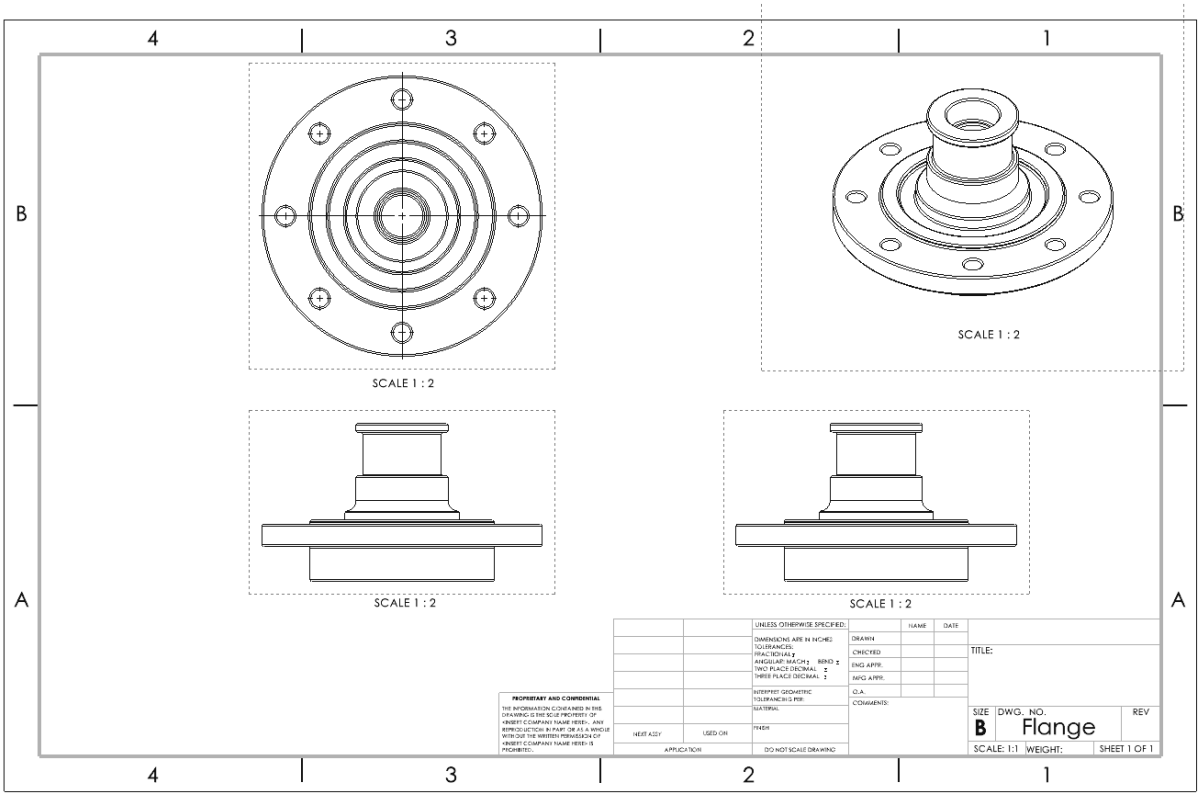
The scale of the Front drawing view and the others are changed to one-half the actual size.

The scale option only resizes the drawing views so that they will fit in the current B-size drawing, the dimensions remain at 1 to 1, or full size.

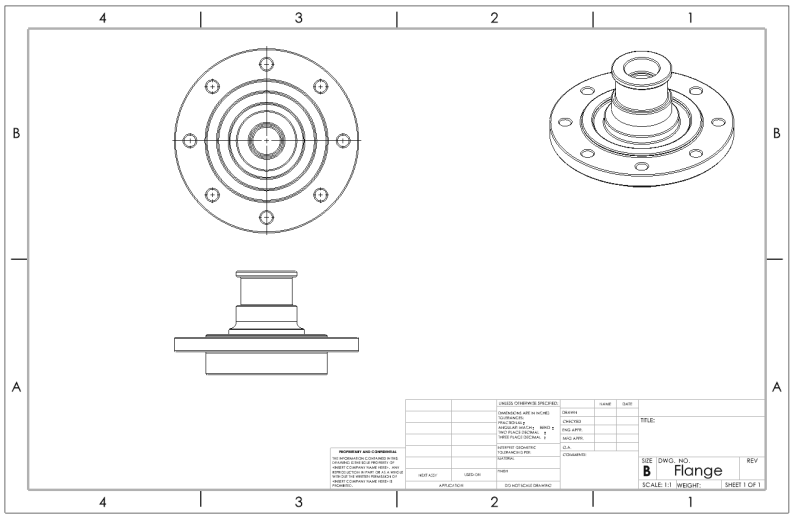
5. Rearranging the drawing Views:

Drag the dotted border of each drawing view to rearrange them.

Move the drawing views to the approximate locations as shown below.



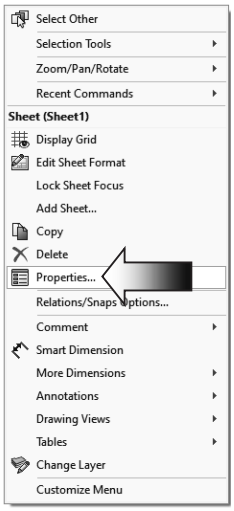
Delete the **Right** drawing view; we will replace it with a section view in the next couple steps.



Also, delete the scale callout "**Scale 1:2**" below each drawing view. We will change the main Sheet Scale in the next step.

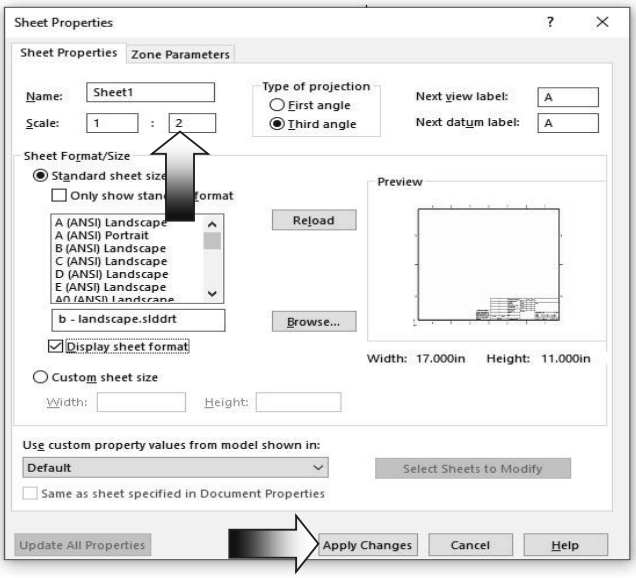
6. Modifying the sheet scale:

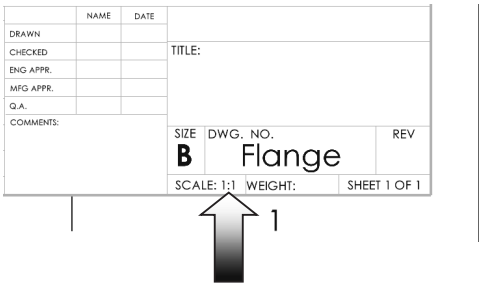
The Sheet Scale that was entered in step 2 is the default scale for all new drawing views. The user can change the scale of any drawing views by accessing the Scale Properties on the FeatureManager tree and entering the custom scales.



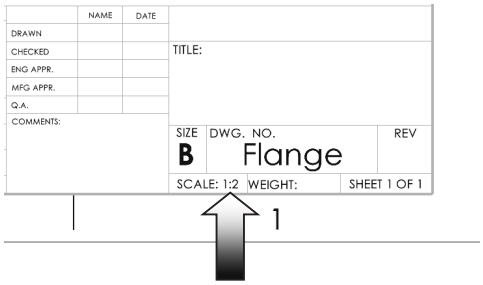
Right-click anywhere inside the drawing and select **Properties**.

Change the Sheet Scale to **1:2** and click **Apply Changes**.

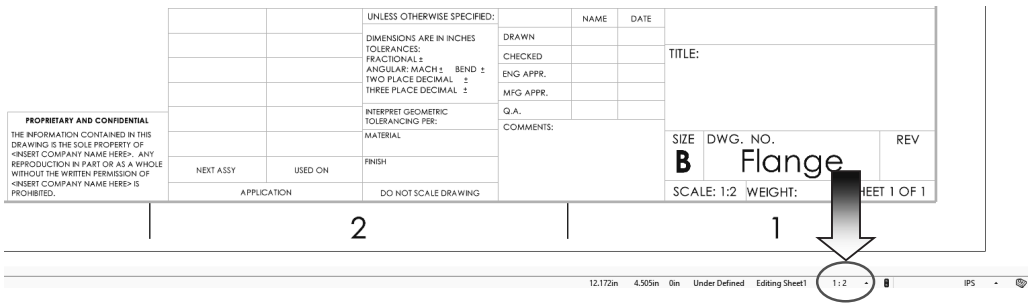




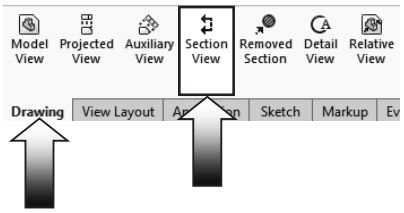
The Sheet Scale is updated in the Title Block.



The status bar at the bottom right displays a pop-up to select a list of commonly used scales. If you needed a custom scale, it requires going into the Sheet Properties (above), where the scale values can be input.



7. Creating a section view:

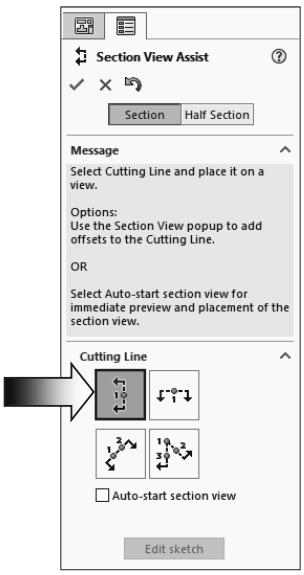


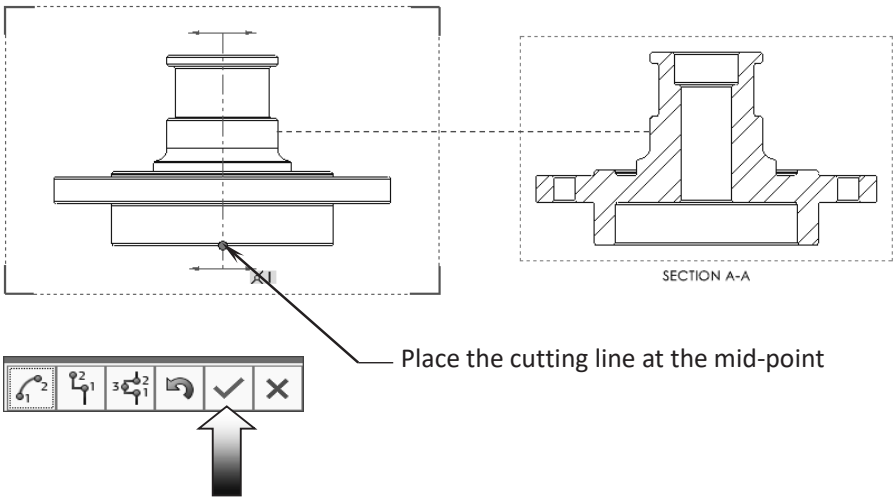
Section Views are used to clarify the interior construction of a part that cannot be clearly described by hidden lines in exterior views. By taking an imaginary cut through the object and removing a portion, the inside features may be seen more clearly.

Select the dotted border of the **Front** drawing view.

Switch to the **Drawing** tab and click **Section View**.

For Cutting Line, select the **Vertical** option.





Hover the mouse pointer over the bottom **horizontal edge** and click the mouse when the **mid-point** pops up.

Click the **green check** to accept the placement of the cutting line.

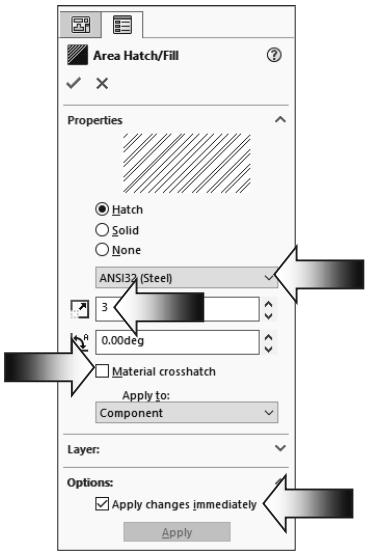
Place the section view on the right side of front drawing view.

Click **OK**. (The material crosshatch will be discussed in the next step.)

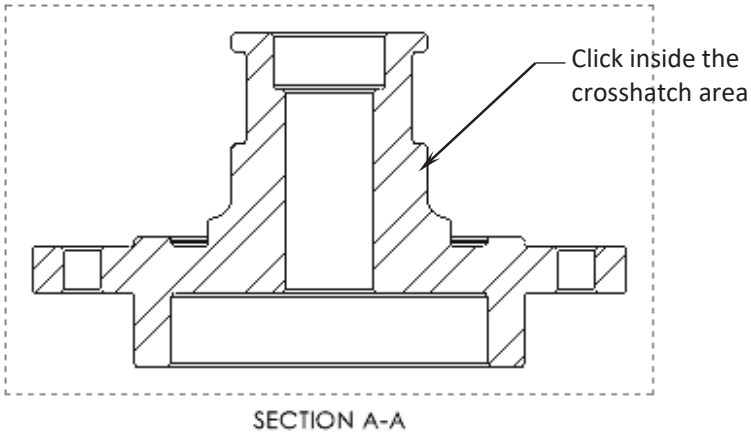
8. Modifying the crosshatch properties:

In a section view, the crosshatch lines represent the cutting surface(s) and the material of the part. They should be clear, easy to see and not to be mistaken with other object lines. One way to achieve that is to increase the density of the crosshatch.

Click inside the crosshatch area to access the **Area Hatch/Fill** options.



Clear the **Material Crosshatch** checkbox.



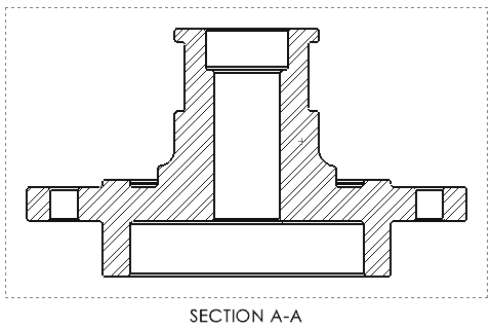
For Hatch Pattern, select **ANSI 32 (Steel)**.

For Hatch Pattern Scale, change it to **3**.

In the Options section, enable the checkbox: **Apply Changes Immediately**.

Click **OK**.

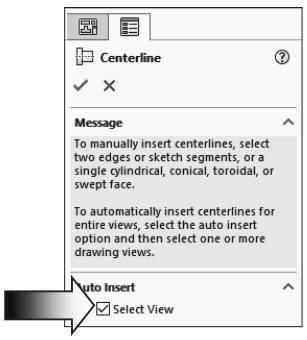
The crosshatch lines are more dense.
It is now much easier to see the sectioned areas than before.

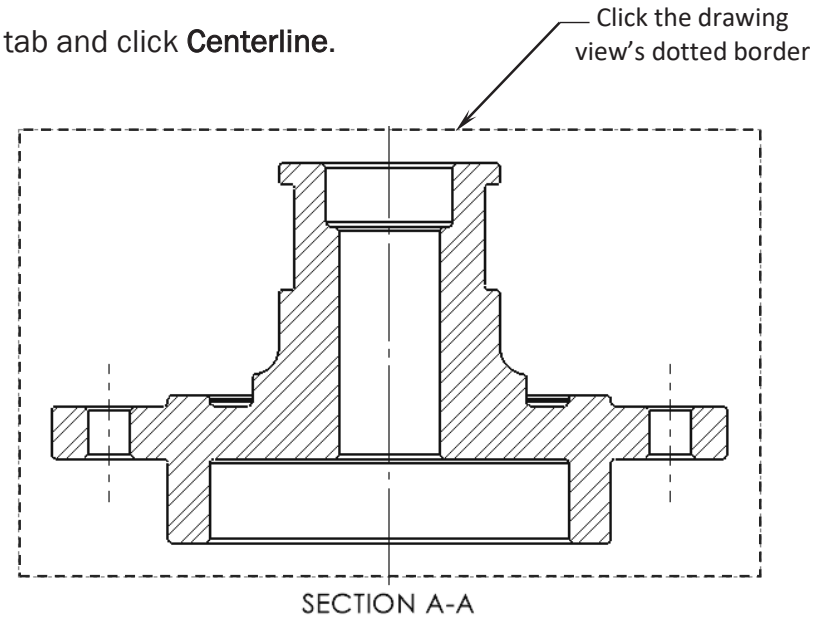


9. Adding the centerlines:

Depending on the Document Property settings, both Centerlines and Center Marks can be added automatically to the drawing views. In this case, only the center marks were added to the Top drawing view but the centerlines were not. We will add them manually to the section view.

Switch to the **Annotation** tab and click **Centerline**.





Enable the **Select View** checkbox.

Click the **dotted border** of the section view. Centerlines are automatically added to all holes.

Click **OK**.

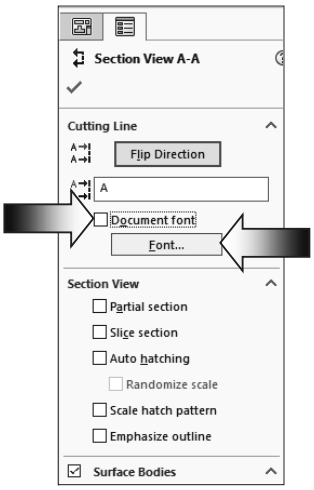
10. Changing the Font Size:

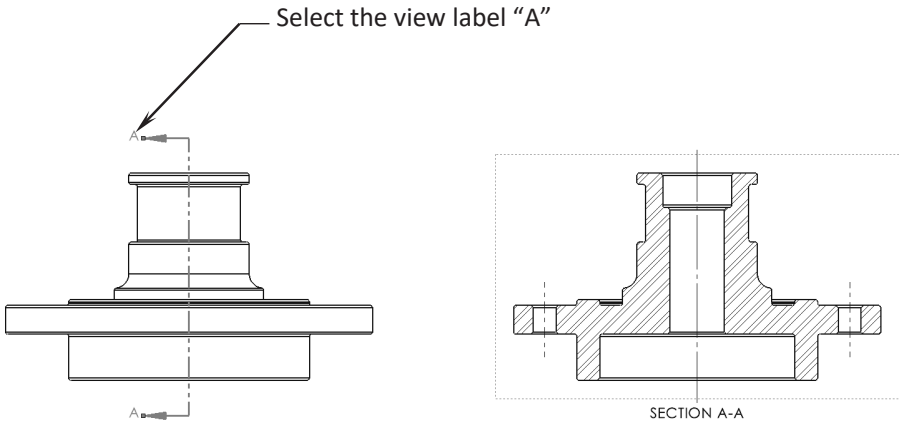
For clarity, the view label such as Section A-A, View B, etc., should be changed to a larger font size than the other annotations, so that they can be seen more easily.

The current font size is 12 points; we will change it to 14 points, and Bold.

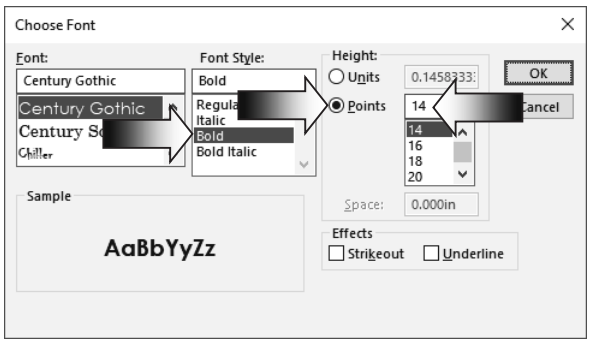
Select the view label “**A**” as noted below.

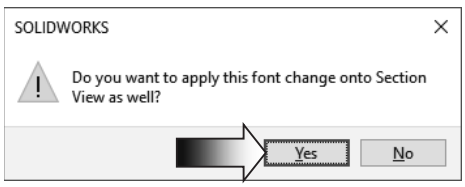
From the FeatureManager tree, under the Cutting Line section, clear the checkbox for **Document Font**, and click the **Font** button (arrow).



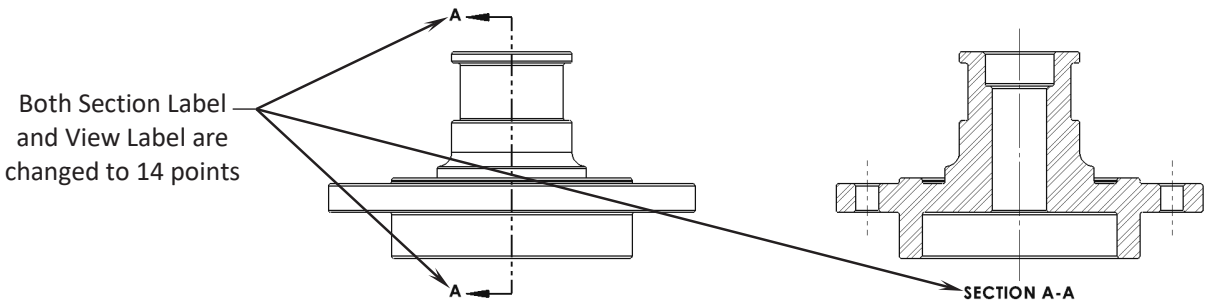


Select the following: **Bold, Points, 14 points** and click **OK**.





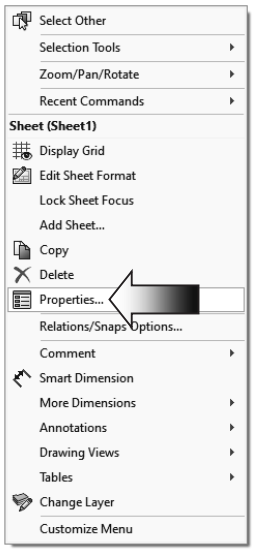
Click **YES** to also change the font of the section view label.

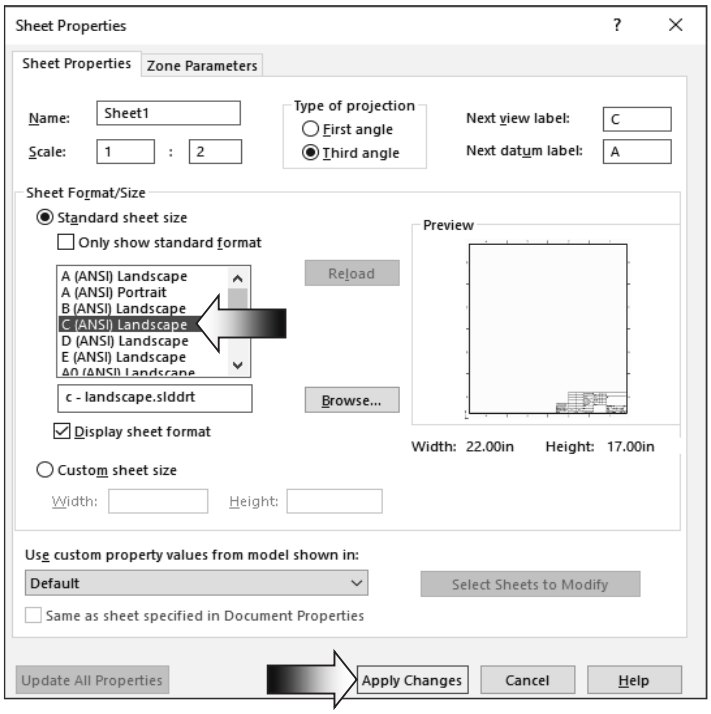


11. Changing the drawing paper size:

It seems we may not have enough room to add dimensions and annotations. We do not want to make the drawing views scale any smaller as it will make small features in the part more difficult to see. Instead, we will change the size of the drawing paper to the next size up, the C-Size 22” X 17”.

Right-click anywhere inside the drawing and select **Properties**.



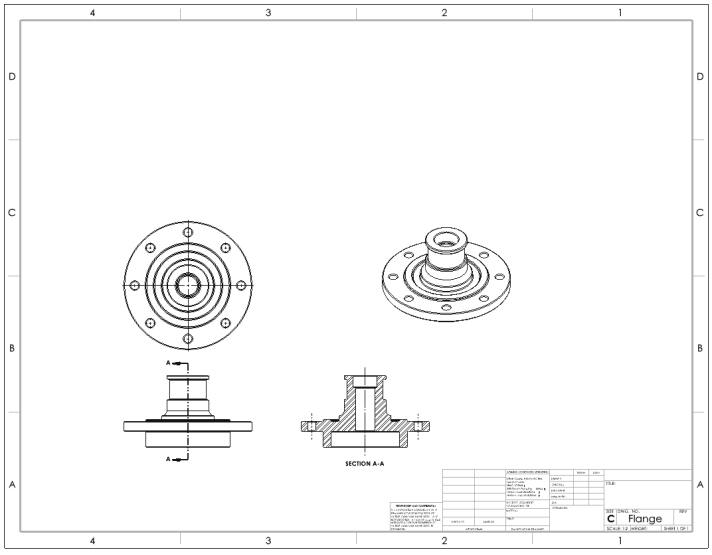


Under the Sheet Format/ Size, select:
C (ANSI) Landscape
(the paper size is 22.0” x 17.0”).

Click **Apply Changes.**

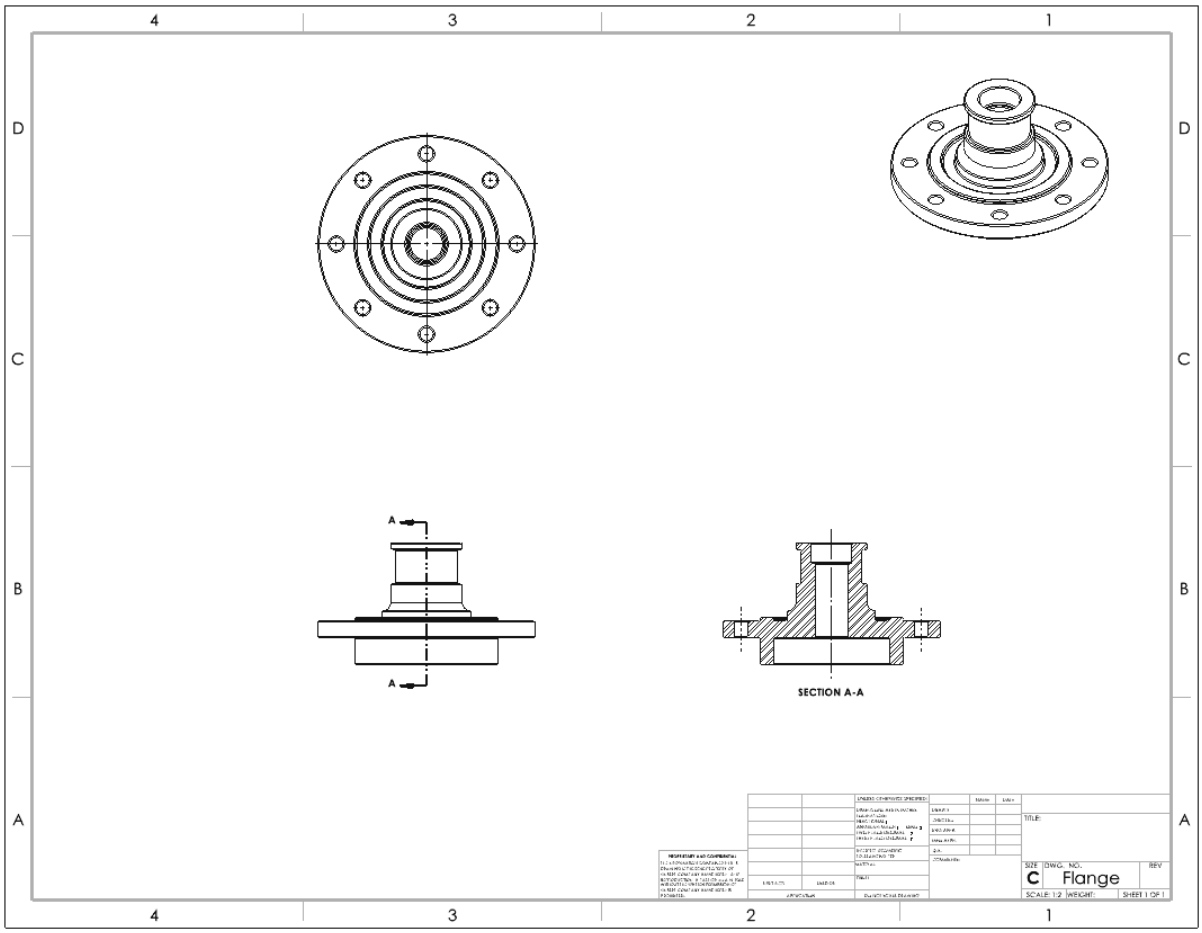
The C-Size drawing format is loaded. We now have a lot of extra room to add dimensions and annotations.

But first, let us rearrange the drawing views.



Move the drawing views by dragging their dotted borders.

Space them out similar to the image shown below. There should be plenty of room to allow for dimensions and annotations to be added.

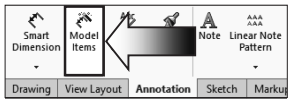


For training and learning purposes, it would be easier to focus on one topic at a time such as creating the drawing views, inserting the model dimensions, and adding the datums and tolerances.

We will add the General Notes, the Revision Block, and fill out the information in the Title Block towards the completion of this drawing.

Remember to save your work every once in a while.

12. Inserting the model dimensions:

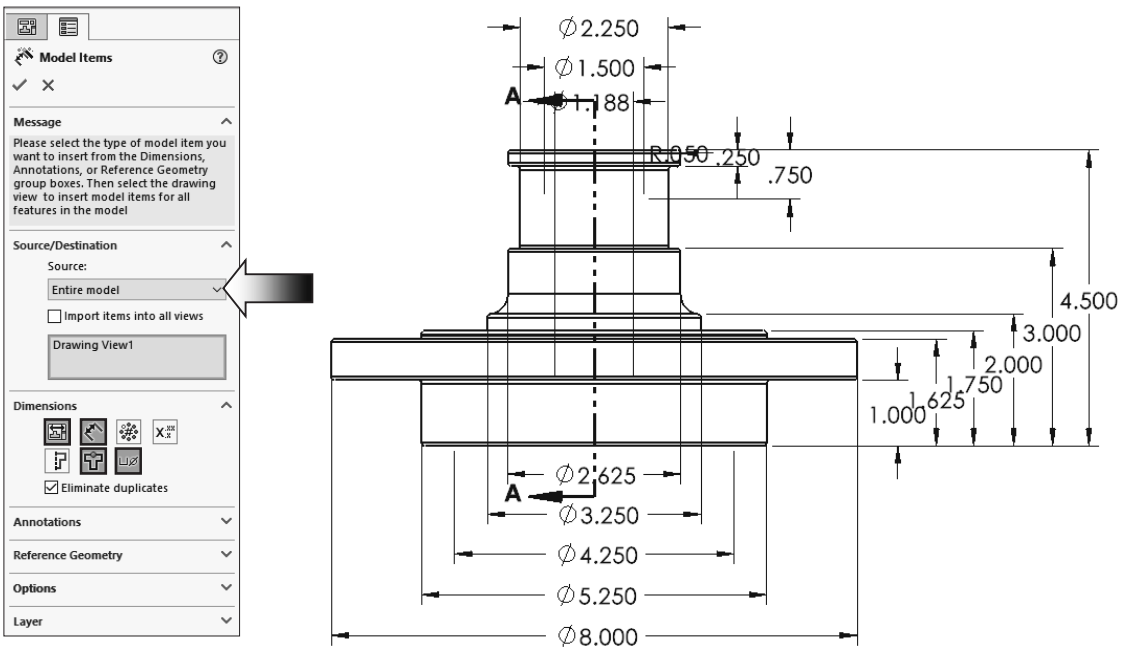


To maintain the parametric relationship between the 2D drawing and the 3D model, the driving dimensions in the model should be inserted into the drawing views. This way, any changes done to the model will be populated to the drawing automatically.

Select the dotted border of the **Front** drawing view.

Switch to the **Annotation** tab and click **Model Items**.

For Source, select **Entire Model**.



For Dimensions, select:

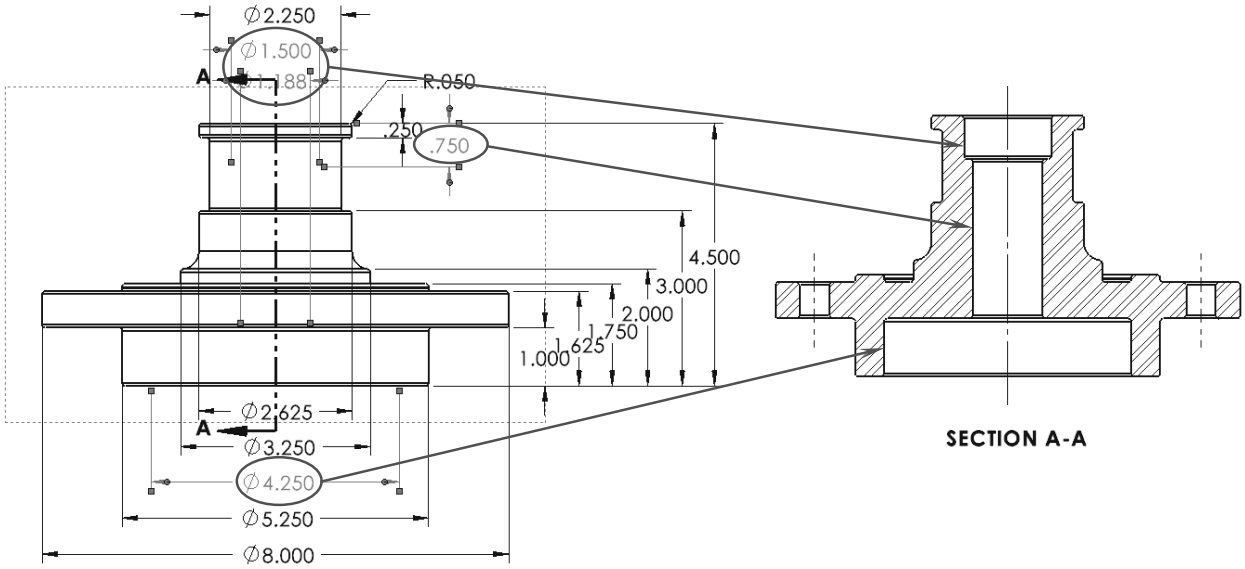
* Marked For Drawing
* Hole Wizard Locations
* Not Marked For Drawing
* Hole Callout

Click **OK** to export the model dimensions into the selected drawing view.

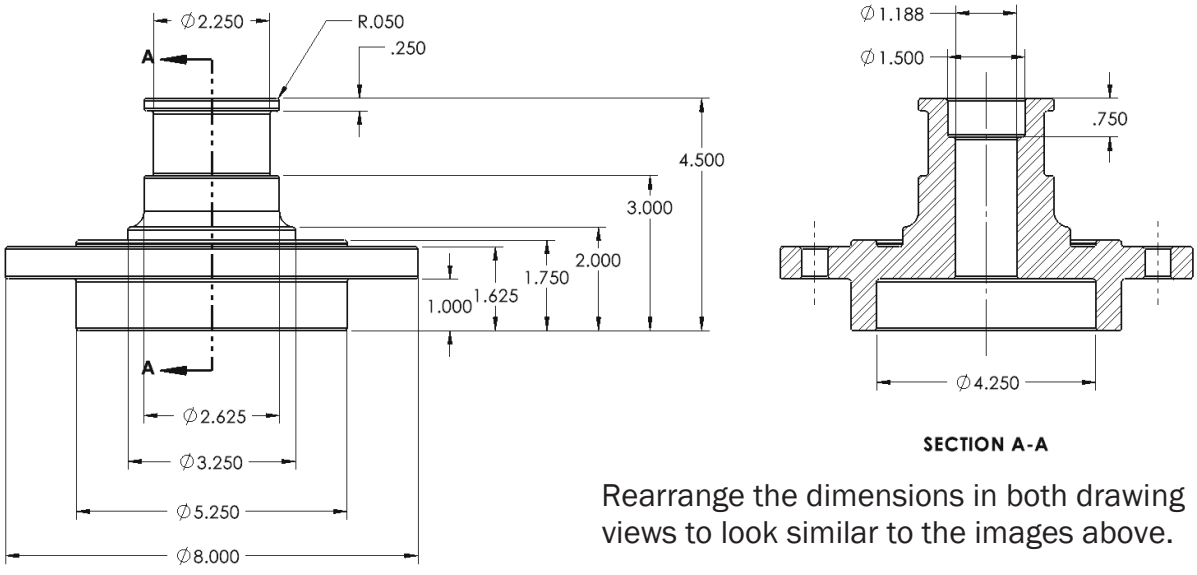
13. Moving dimensions:

Dimensions can be moved from one drawing view to another and still maintain their associations to the model.

Hold the **Control** key and select 3 dimensions: **Ø1.500**, **Ø 1.188**, and **Ø 4.250**.

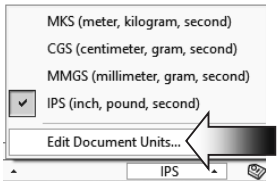


Now hold the **Shift** key and drag/drop the 3 selected dimensions to anywhere inside the **section view**.



Rearrange the dimensions in both drawing views to look similar to the images above.

14. Removing dimension overlaps with Break-Lines:



To further enhance the clarity of a drawing, the overlaps between the dimension leader lines should be replaced with small gaps. This can be done quite easily with a couple simple steps.

Click the small arrow at the bottom right corner of the screen, next to the IPS units, and select: **Edit Document Units**.

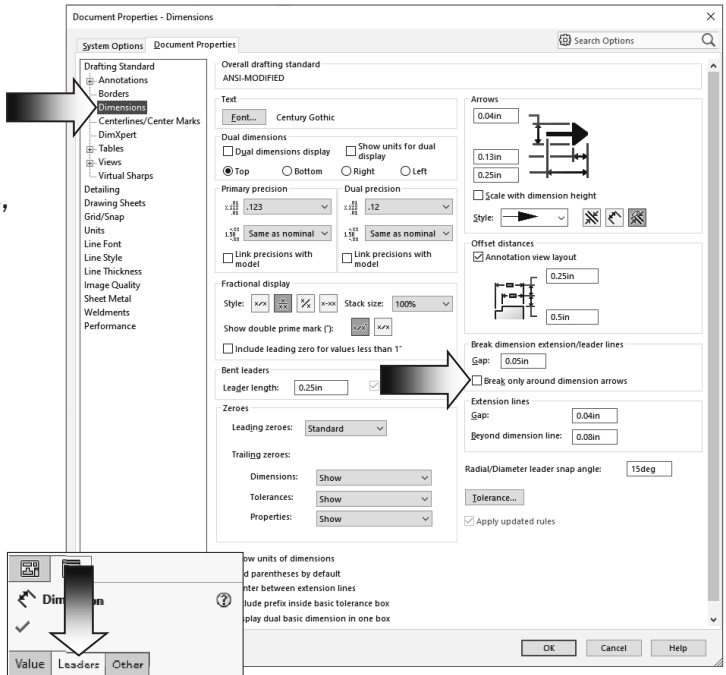
On the left pane, select the **Dimensions** option.

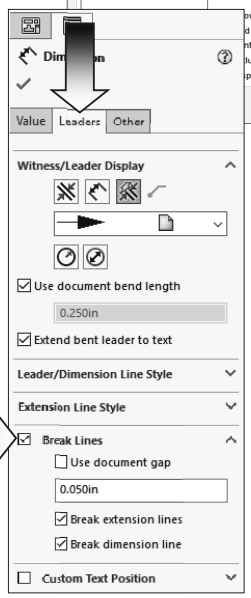
On the right pane, clear the checkbox for **Break Only Around Dimension Arrow**.

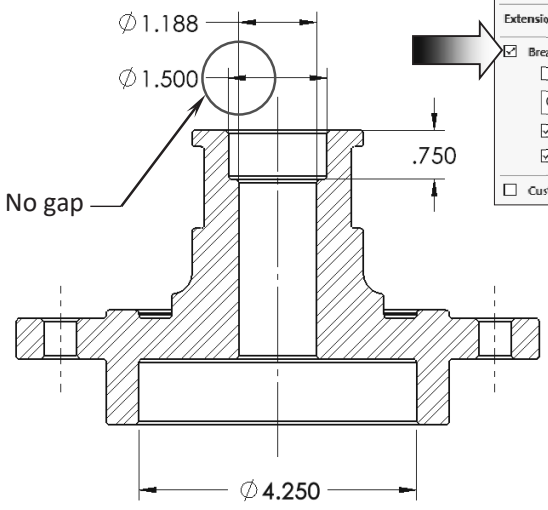
Click **OK**.

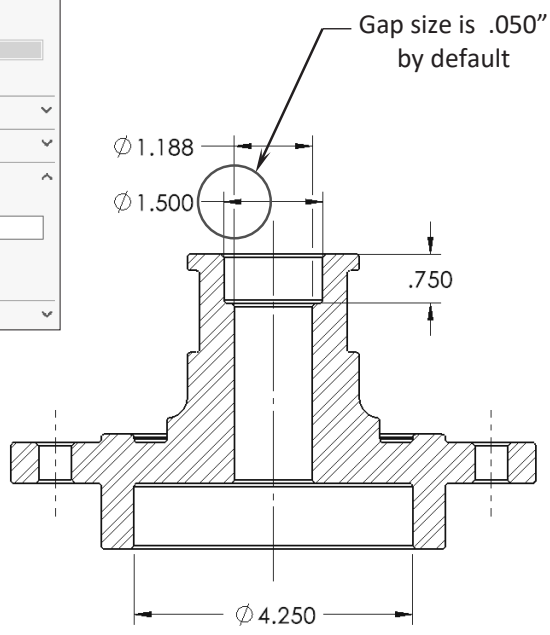
Select the dimension **Ø1.188** and click the **Leaders** tab.

Enable the **Break Lines** checkbox and click **OK**.



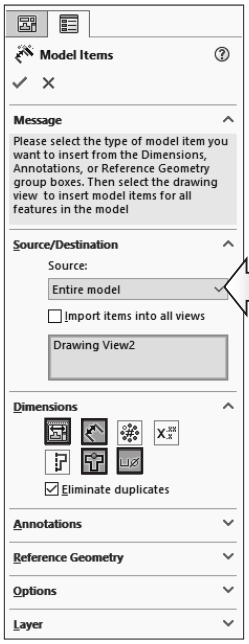






15. Inserting other dimensions:

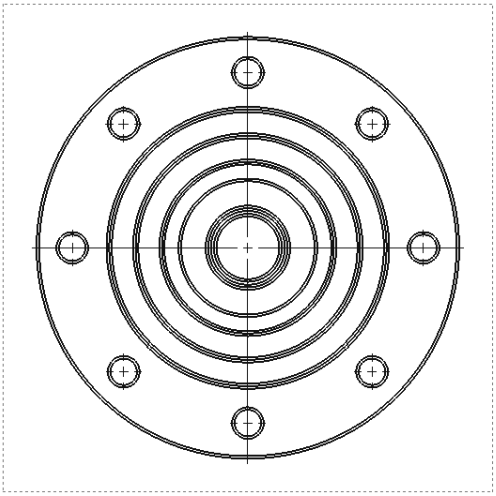
Zoom in on the **Top** drawing view and select its dotted border.



Click **Model Items** on the **Annotation** tab.

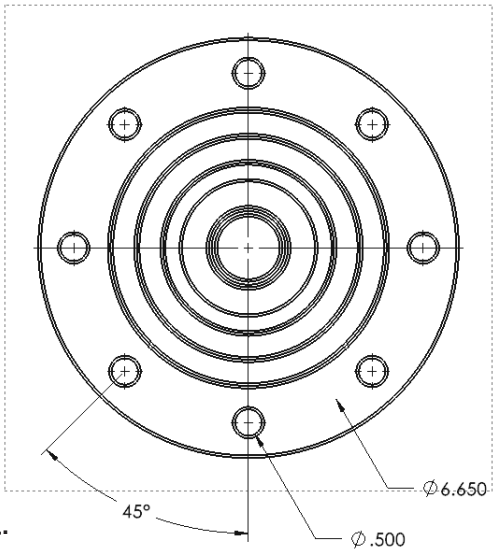
The options that were selected from the previous step should still be active.

Click **OK**.

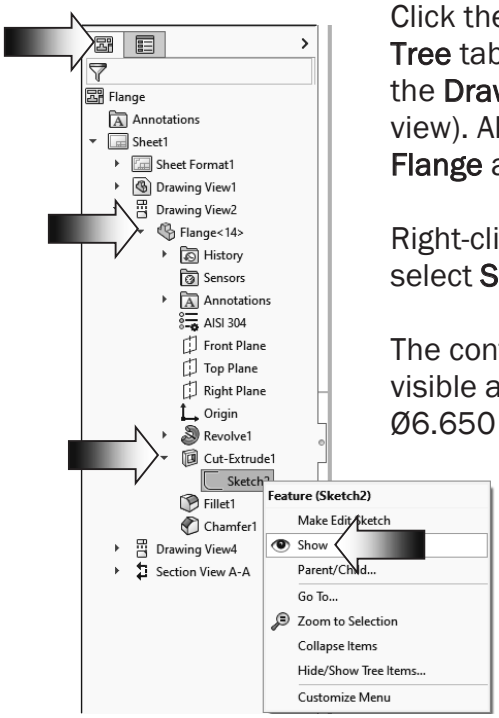


Three dimensions were added, but the Ø6.650 must be attached to the Bolt-Circle of the hole Pattern.

Make sure the dotted border of the **Top** drawing view is still selected.

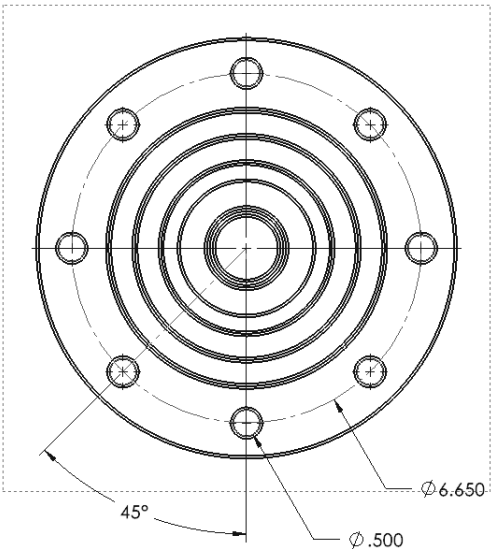


Click the **DrawingManager Tree** tab (arrow); expand the **Drawing View2** (Top view). Also expand the part **Flange** and the **Cut-Extrude1**.



Right-click **Sketch-2** and select **Show**.

The contruction circle is now visible and the dimension Ø6.650 is attaching to it.

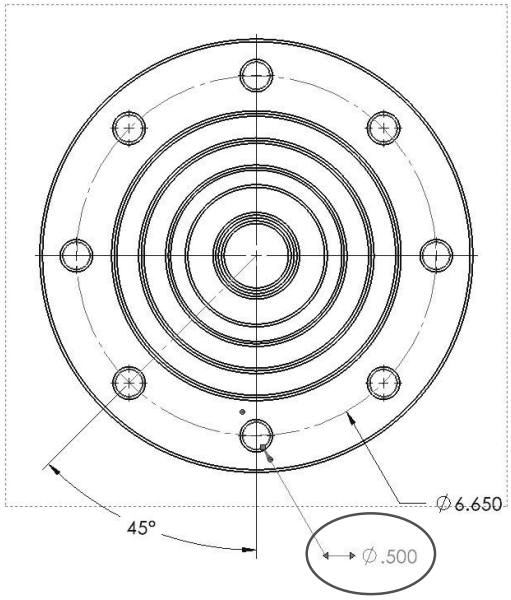


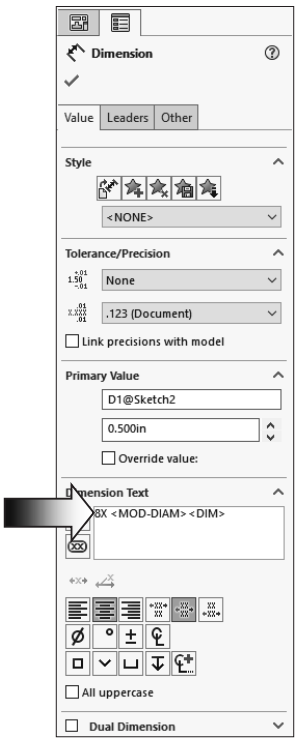
Click on the name **Drawing View 2** and press: **Shift+C** to collapse the tree.

16. Adding callouts:

Callouts such as Depth and Number of Instances will be added at this time.

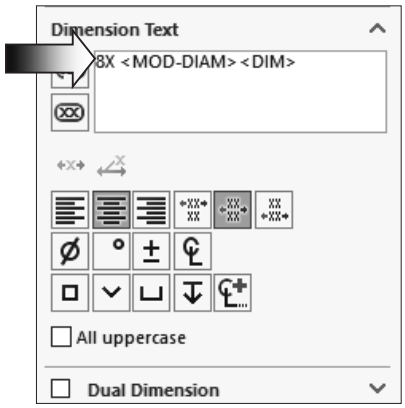
Select the dimension **Ø.500** in the Top drawing view.

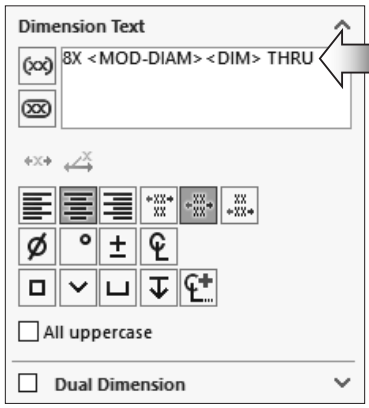




Locate the **Dimension Text** section and place the mouse cursor <u>before</u> the <MOD-DIAM><DIM> and Enter: **8X**.

<u>Do not</u> click OK just yet.





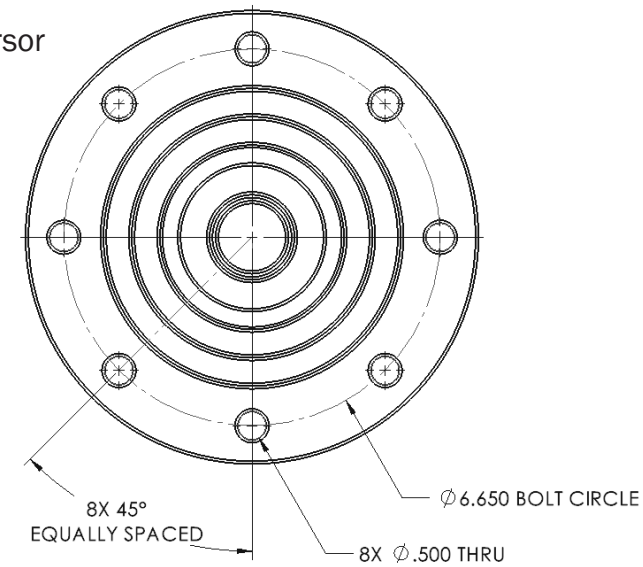
Next, click to place the mouse cursor <u>after</u> the <DIM> and enter: (space) **THRU**.

Repeat the last step and update the following callouts:

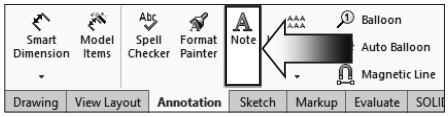
8X 45°
EQUALLY SPACED

Ø6.650 BOLT CIRCLE

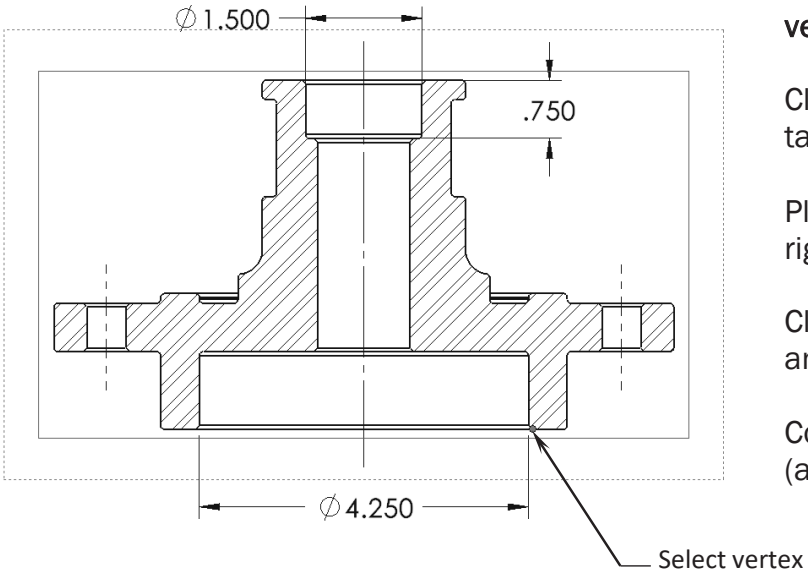
Click **OK**.



17. Adding additional callouts:



Sometimes the chamfer dimensions do not get exported properly, and even if they did, they may appear at some odd locations. To overcome this issue, we will use a note to callout the chamfer dimensions instead.



SECTION A-A

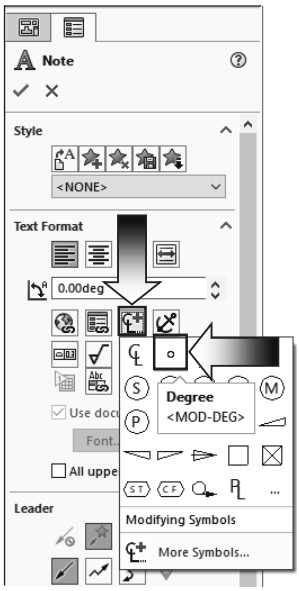
In the section view, select the **vertex** as indicated below.

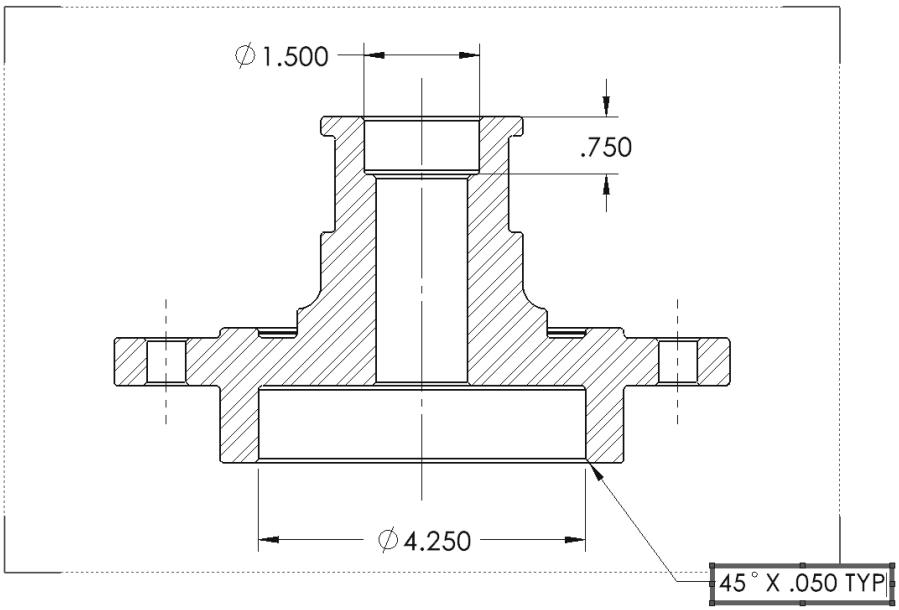
Click **Note** on the **Annotation** tab.

Place the note on the lower right side and enter: **45**

Click the **Add Symbol** button and select the **Degree** symbol

Continue to type: **X 050 TYP** (after the degree symbol).





SECTION A-A

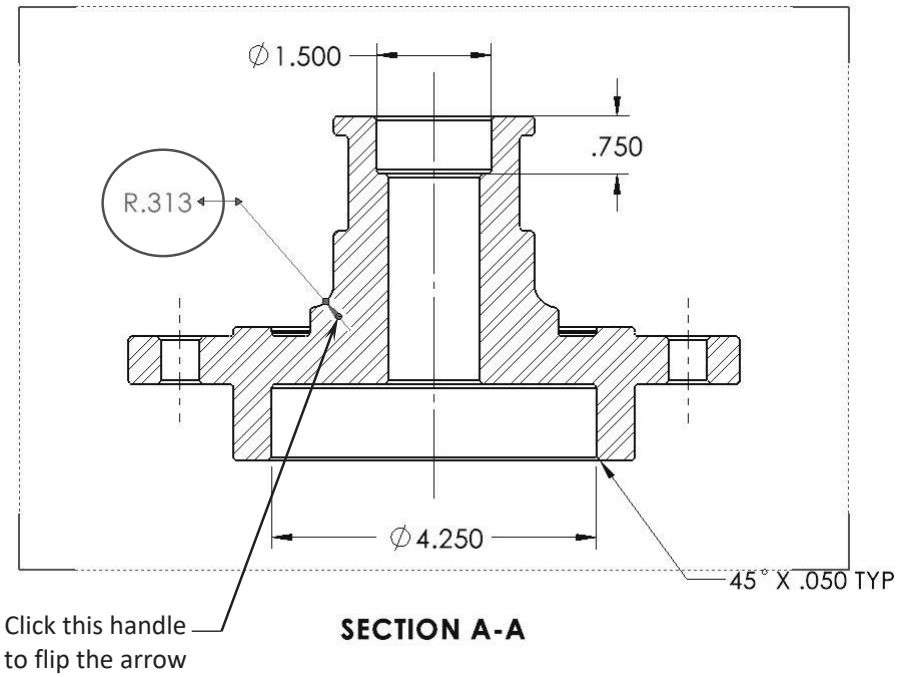
Click **OK**.

Check your note against the one shown here.

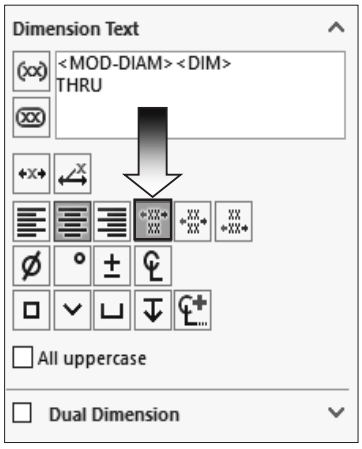
Select the **Smart Dimension** tool on the **Annotation** tab.

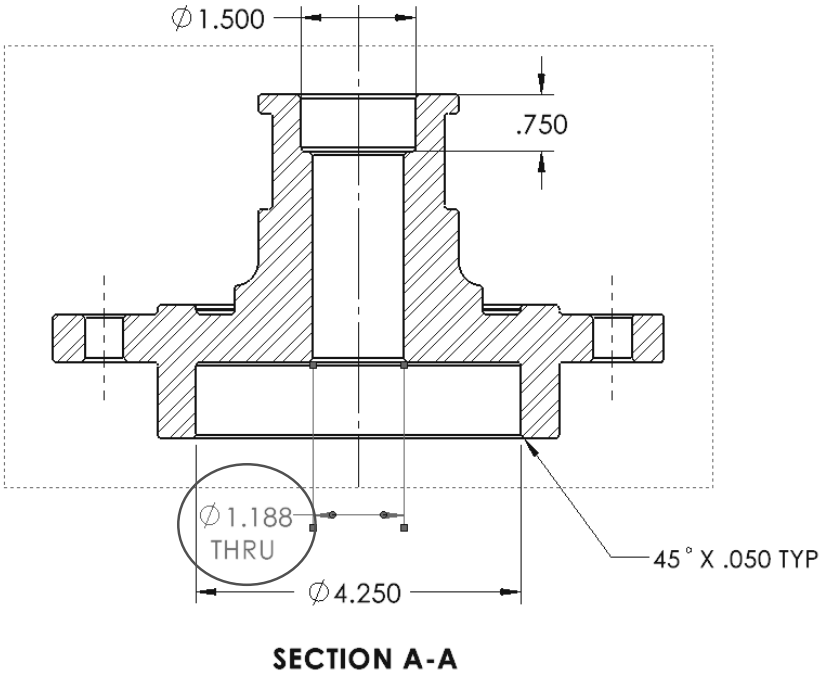
Add the **radius** dimension as shown in the image on the right.

Click the **round handle** to flip the dimension arrow outward.



Add the text **"THRU"** under the dimensions **Ø1.188** (circled) and select the **Top Justify** button (arrow) to align the callout to the top.





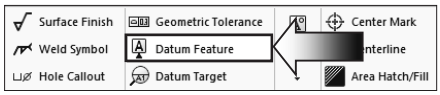
Click **OK**.

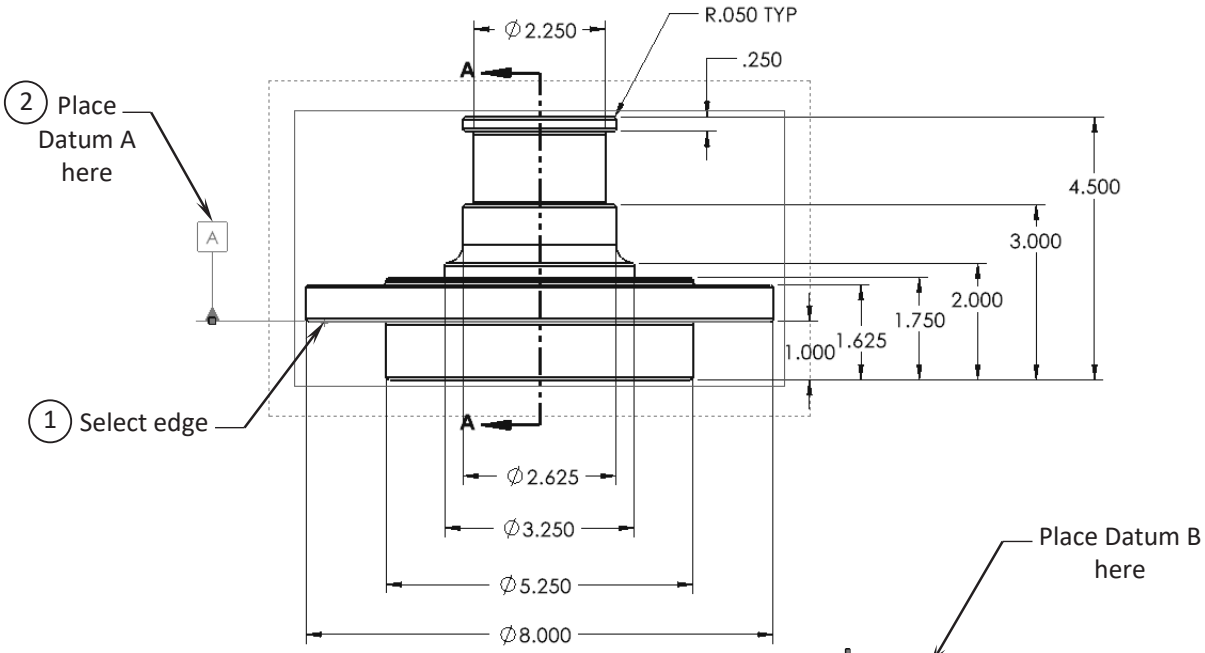
18. Adding datums:

In an engineering drawing, datums are used with geometric dimensioning and tolerancing on an object used to create a reference system for measurement.

In short, a datum is used as a reference point, surface, or axis on an object against which measurements are made.

Switch to the **Annotation** tab and click: **Datum Feature**.



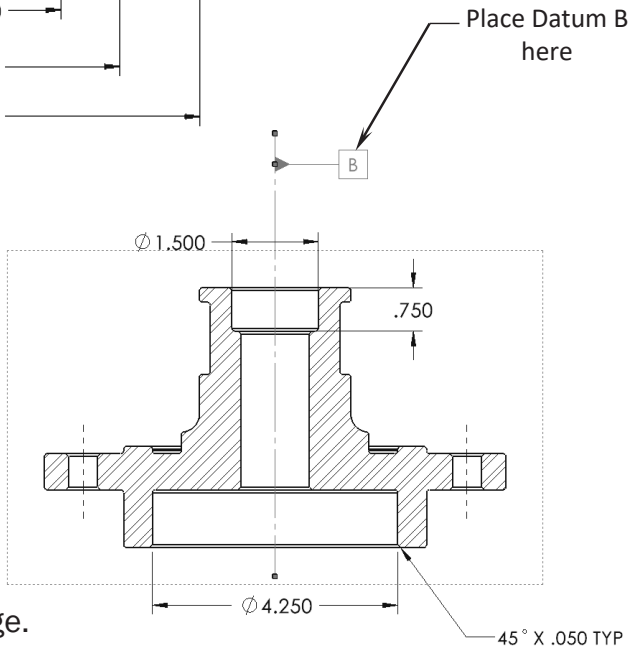


For **Datum A**, (click 1) select the bottom edge of the flange in the **Front** drawing view.

Move the cursor outwards and click (click 2) to place Datum A as shown in the image above.

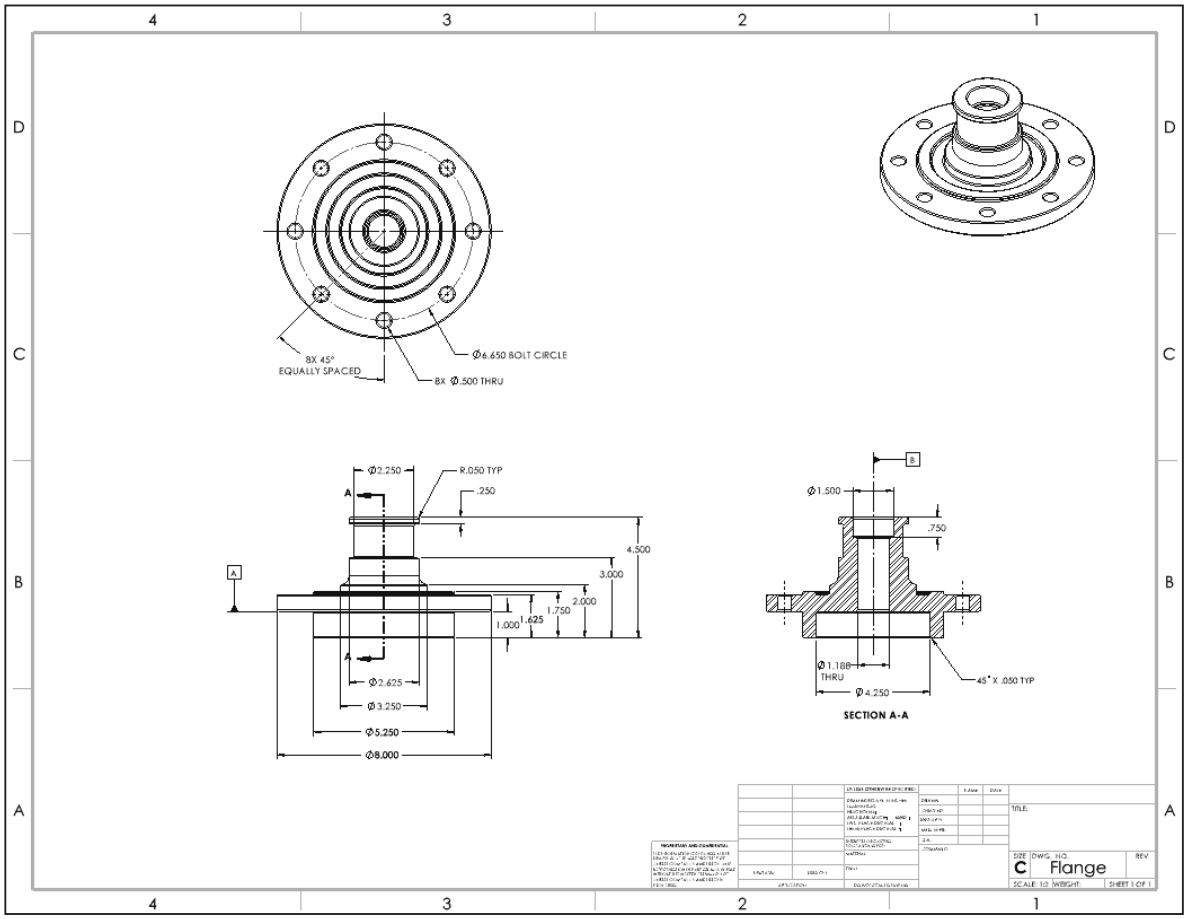
For **Datum B**, select the centerline in the middle of the section view.

Place Datum B as noted in the image.



SECTION A-A

Your drawing should look similar to the one shown below. Make any corrections as needed.



Remember to save your work every once in a while.

We will create the General Notes, the Revision Block, and fill out the Title Block information at this time.

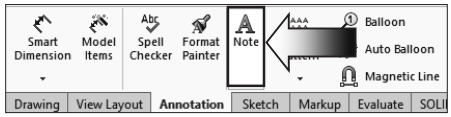
The Tolerance and precision topics will be discussed and added to the drawing in the next chapter.

19. Adding the General Notes:

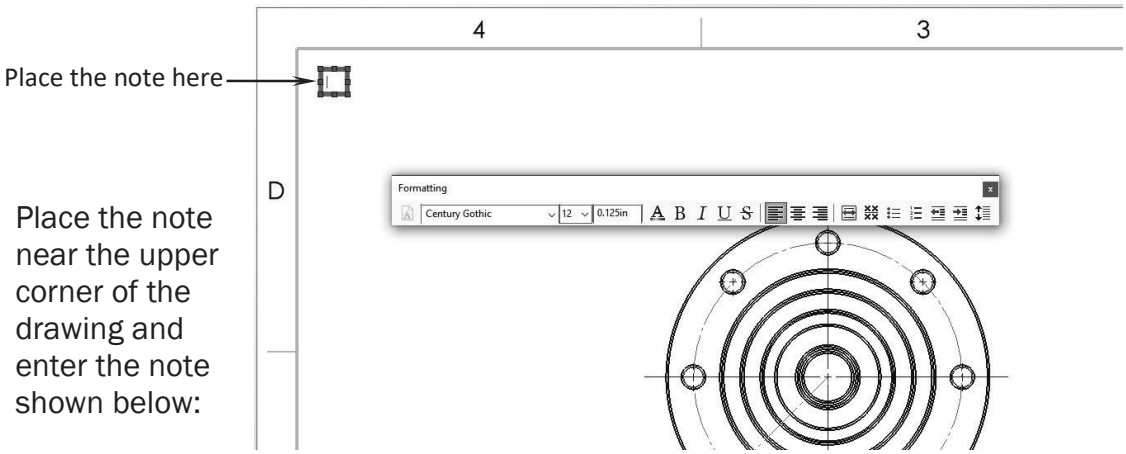
General notes provide information and direction to the manufacturers by clarifying design details or construction practices.

The general and specific notes should not be confused with the information found in the bill of materials, title block, revision chart, or the drawing specifications. The general notes are usually located in the upper left corner of the drawing.

Switch to the **Annotation** tab and click **Note**.



Zoom in closer to the upper left corner of the drawing.



Place the note near the upper corner of the drawing and enter the note shown below:

NOTES: UNLESS OTHERWISE SPECIFIED
1. INTERPRET DRAWINGS AND DIMENSIONS PER ASME ANSI-Y14.5
2. DIMENSIONS ARE IN INCHES, 3 DECIMAL PLACES
3. MATERIAL: STAINLESS STEEL 304 OR EQUIVALENT
4. BREAK ALL EDGES, TUMBLE, AND DEBURR PARTS
5. SHIPPING PACKAGE TO BE LABELED WITH PART NUMBER, MANUFACTURER'S NAME, LOT NUMBER, AND QUANTITY
6. PARTS TO BE CLEAN AND FREE OF LUBRICANT OR DEBRIS

Click **OK**.

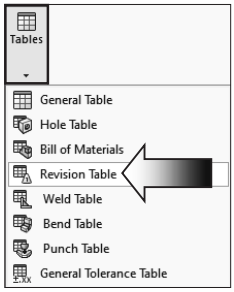
20. Adding the Revision Table:

The Revision Table or revision block, located in the upper right corner of the drawing, shows details about the changes that were made to roll the revision.

The Revision Table includes the revision, the description of what changes were made, the date of the revision, and approval of the revision.

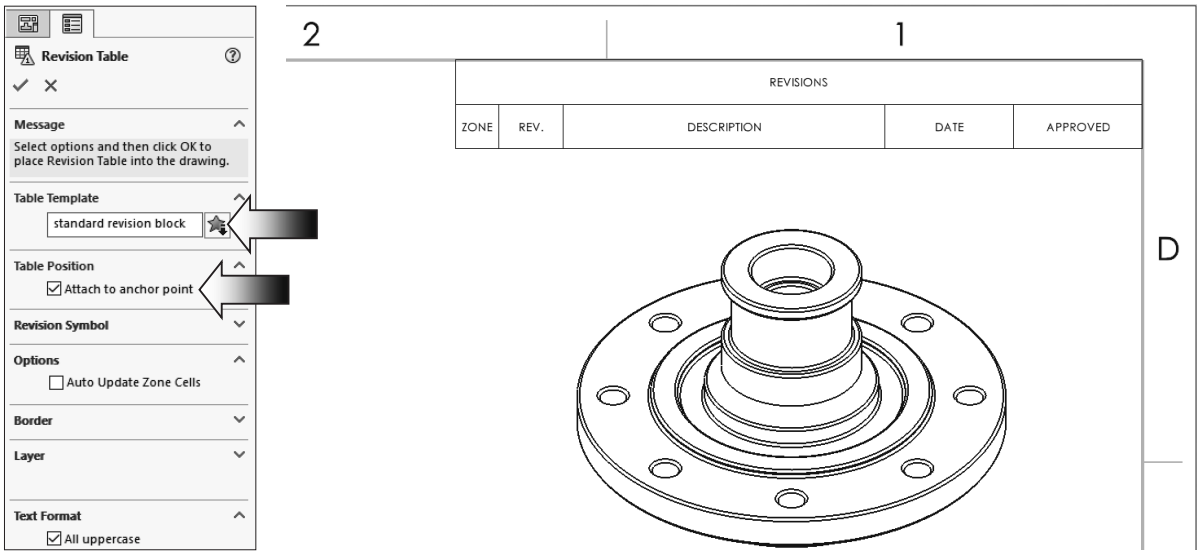
Zoom in closer to the upper right corner of the drawing.

Switch to the **Annotation** tab, expand the **Tables** drop-down list and select: **Revision Table** (arrow).



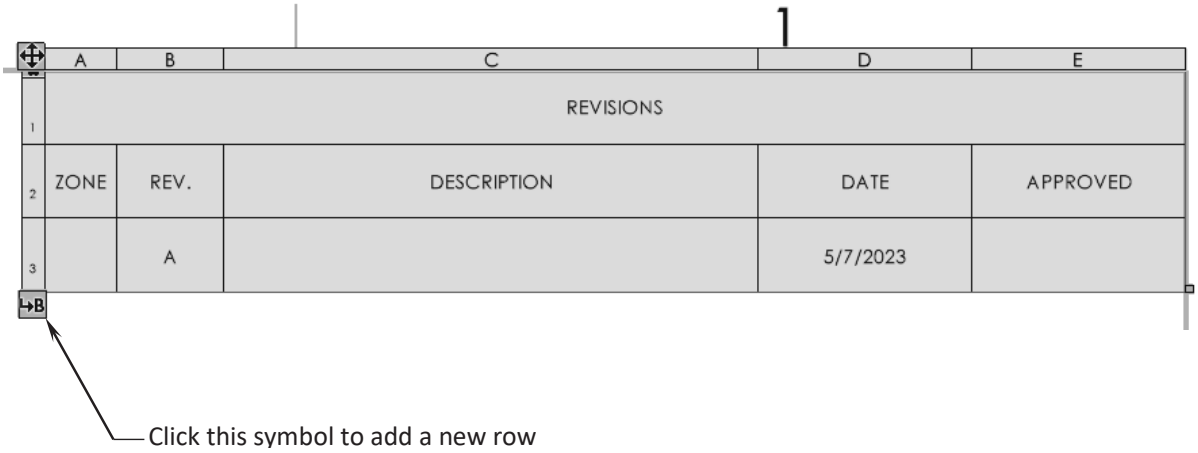
For Table Template, use the default **Standard Revision Block**, which can be found in the C:\Program Files\SolidWorks Corp\SolidWorks 20XX\Lang\English.

Enable the checkbox for **Attached to Anchor Point**. This option will snap/lock the Revision Table to the upper right corner of the drawing automatically.



Click **OK**.

Hover the mouse pointer over the Revision Table and click square ↳B at the lower left corner, to add a new row.



	A	B	C	D	E
1	REVISIONS				
2	ZONE	REV.	DESCRIPTION	DATE	APPROVED
3		A		5/7/2023	

↳B

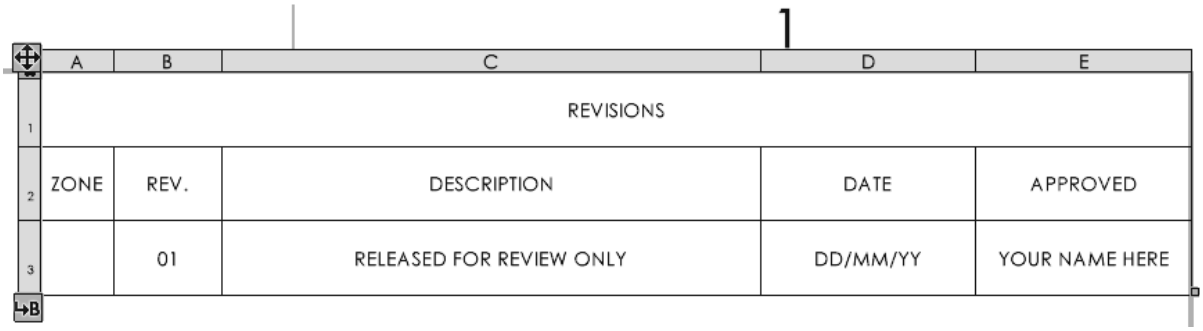
Click this symbol to add a new row

Double-click the letter **A** and change it to **01**.

Double-click in the **Description** field and enter: **RELEASED FOR REVIEW ONLY**.

Double-click in the **Date** field and enter today's date.

Double-click in the **Approved** field and enter your name there.



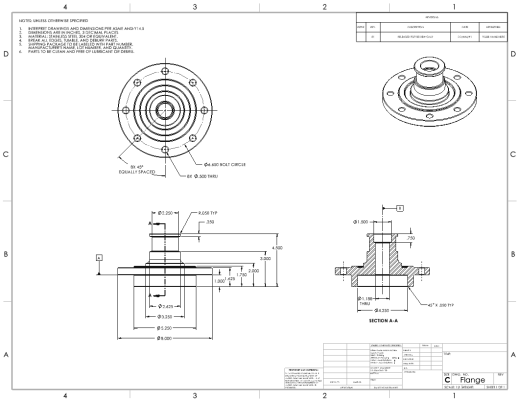
	A	B	C	D	E
1	REVISIONS				
2	ZONE	REV.	DESCRIPTION	DATE	APPROVED
3		01	RELEASED FOR REVIEW ONLY	DD/MM/YY	YOUR NAME HERE

↳B

Zoom-To-Sheet (or press F) and rearrange the drawing views so that they look more presentable and easy to read.

The Isometric view should be kept at the upper right corner of the drawing to comply with ANSI standards.

We will now take a look at how to fill out the information in the Title Block.



21. Filling out the Title Block:

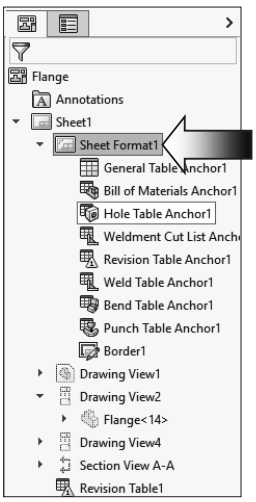
By default, every SOLIDWORKS drawing will have 2 layers to start with; more layers can be added at any time during the creation of the drawing.

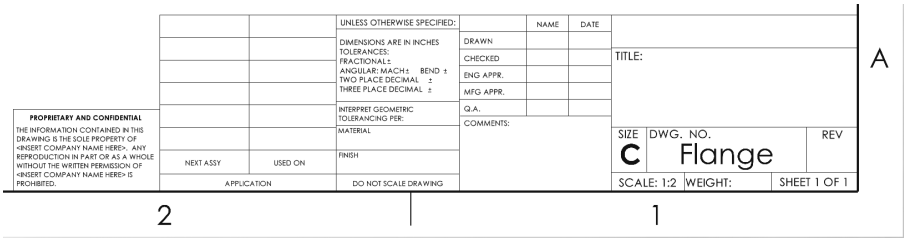
The “Front Layer” is called the **Sheet**, and the “Back layer” is called the **Sheet Format**.

The Sheet is where drawing views, dimensions and annotations are created, and the Sheet Format is where the information that belongs to the Title Block is kept.

We will need to edit the Sheet Format in order to fill out the information in the Title Block.

From the FeatureManager tree, expand **Sheet1** and double-click the **Sheet-Format1** to activate the back layer.

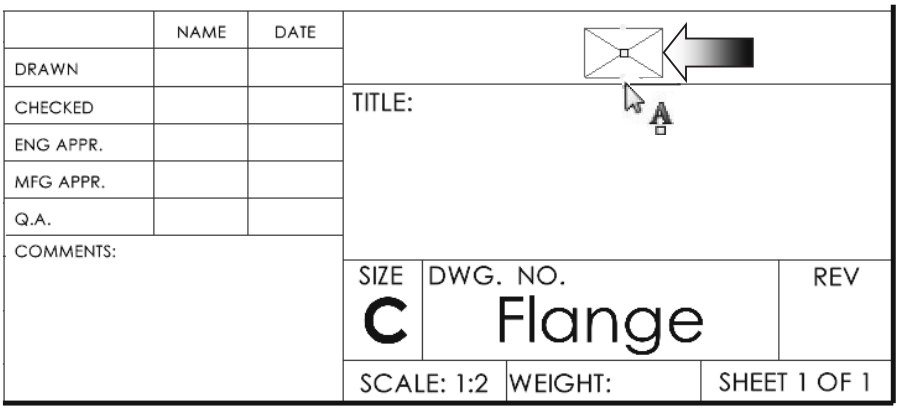




The Sheet Format is activated. The lines and text in the Title Block can be modified at this time.

Hover the mouse cursor over the middle of the Company field until the symbol appears. This is the symbol of a blank note; there is a blank note in every field by default.

Double-click this blank note and enter the name of your company (or school) here.



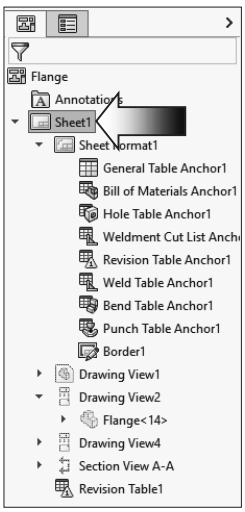
Continue adding or modifying the information in the Title Block as shown below.

UNLESS OTHERWISE SPECIFIED:		NAME	DATE	SOLIDWORKS CORP.
DIMENSIONS ARE IN INCHES TOLERANCES: FRACTIONAL ± ANGULAR: MACH ±1° BEND ± TWO PLACE DECIMAL ±.01 THREE PLACE DECIMAL ±.005	DRAWN	YOUR NAME	MM-DD-YY	TITLE: STAINLESS STEEL FLANGE
	CHECKED			
	ENG APPR.			
	MFG APPR.			
INTERPRET GEOMETRIC TOLERANCING PER:	Q.A.			SIZE C | DWG. NO. 012-34-567 | REV 01
MATERIAL S.S. 304	COMMENTS:			
FINISH				
DO NOT SCALE DRAWING				SCALE: 1:2 | WEIGHT: | SHEET 1 OF 1

<u>Note:</u> The information provided in this lesson is intended for training and learning purposes only.

22. Switching back to the Sheet:

After the Title Block is filled out, we should go back to the Sheet (the Front Layer) prior to saving the drawing.



From the FeatureManager tree, double-click **Sheet1** to activate it.

When the Sheet layer is active, the information in the Title Block is no longer editable.

If you need to edit the Sheet Format again, simply double-click Sheet Format1 on the FeatureManager tree.

UNLESS OTHERWISE SPECIFIED:		NAME	DATE	SOLIDWORKS CORP.
DIMENSIONS ARE IN INCHES TOLERANCES: FRACTIONAL ± ANGULAR: MACH ±1° BEND ± TWO PLACE DECIMAL ±.01 THREE PLACE DECIMAL ±.005	DRAWN	YOUR NAME	MM-DD-YY	TITLE: STAINLESS STEEL FLANGE
	CHECKED			
	ENG APPR.			
	MFG APPR.			
INTERPRET GEOMETRIC TOLERANCING PER:	Q.A.			SIZE C | DWG. NO. 012-34-567 | REV 01
MATERIAL S.S. 304	COMMENTS:			
FINISH				
DO NOT SCALE DRAWING				SCALE: 1:2 | WEIGHT: | SHEET 1 OF 1

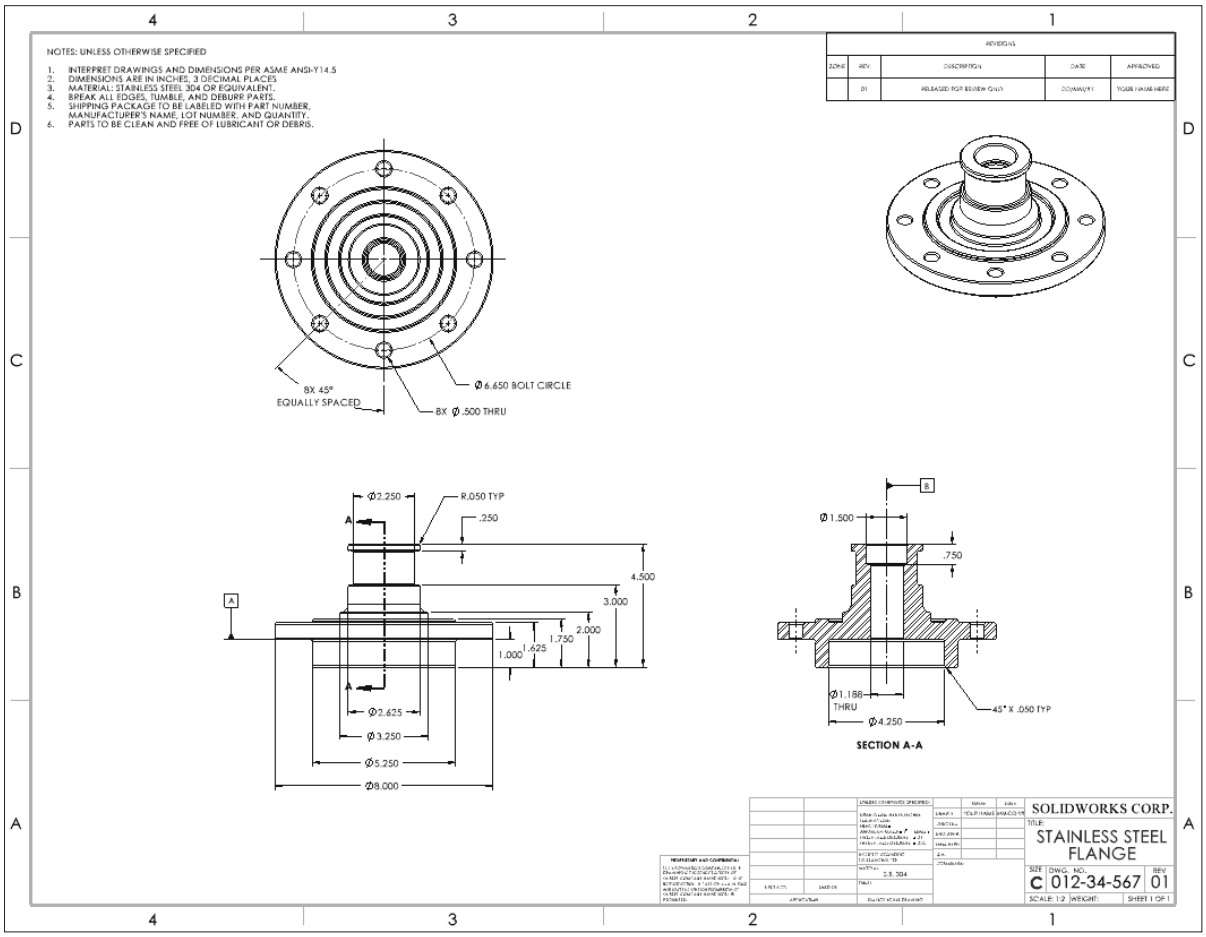
The 1st half of the lesson is completed. Let us save our work at this time.

23. Saving your work:

Select: **File, Save As.**

Enter: **Detailing a Machined Part.slddrw** for the file name.

Click **Save.**



Close all documents.

The Tolerance and precision topics (GD&T) will be discussed and added to the same drawing, in the next chapter.

Exercise: Detailing a Machined Part

Base Block Drawing

The exercises are intended to help you apply what you have learned from the previous lessons. They come with some instructions, but they are not as detailed as the lessons. The purpose is to give you an opportunity to explore and create drawings on your own, at your own pace.

1. Opening a part document:

Select **File, Open**.

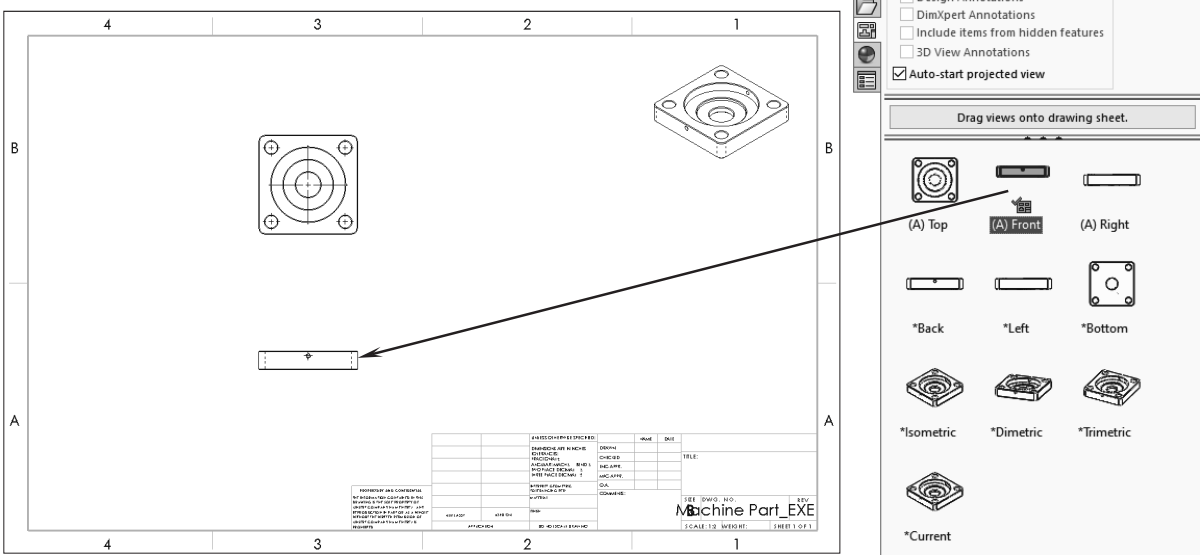
Browse to the Training Folder and open the part document named: **Machined Part_EXE.sldprt**.

The material **5052-H32** has already been assigned to the part.

2. Transferring to a drawing:

Create a drawing from the model using the following:

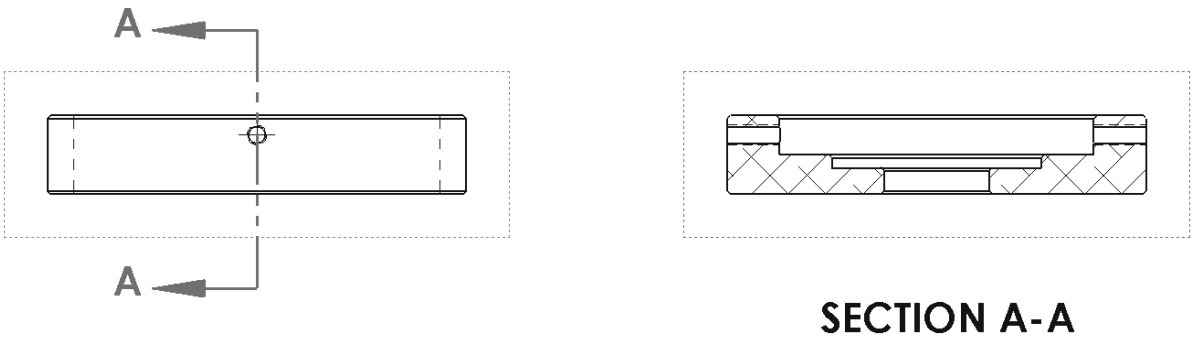
* **B (ANSI) Landscape**
* **3rd Angle Projection**
* **Scale: 1:2**
* Add the **Front**, **Top**, and **Isometric** views from the **View Palette**.



3. Adding a Section View:

Switch to the **Drawing** tab and click **Section View**.

Select the **Vertical Cutting Line** and place the line on the center of the hole.



Place the Section View on the right side of the Front view. Verify that the section arrows are pointing to the left as shown in the image above.

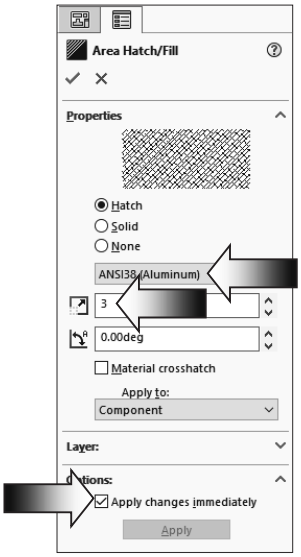
4. Modifying the Hatch Pattern:

Click inside the hatch area to access the **Hatch Properties**.

For Hatch Pattern, select **ANSI38 (Aluminum)**.

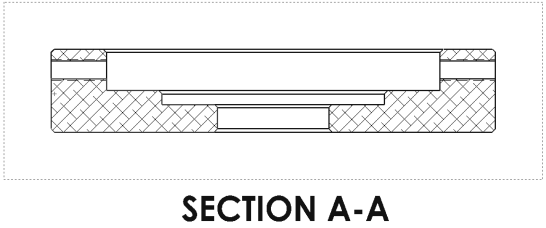
For Scale, enter **3**.

Enable the checkbox **Apply Changes Immediately**.



Click **OK**.

Compare your hatch pattern against the image shown on the right.

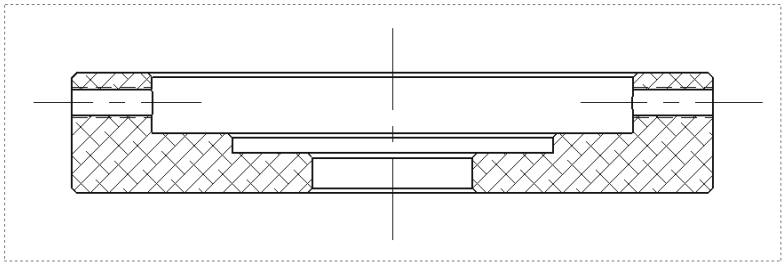


5. Adding Centerlines:

Switch to the **Annotation** tab and click **Centerline**.

Add the **3 centerlines** shown in the image below to the section view.

Adjust the length of the centerlines by dragging the handles at the ends, if needed.

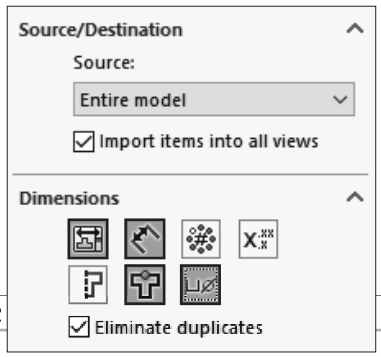


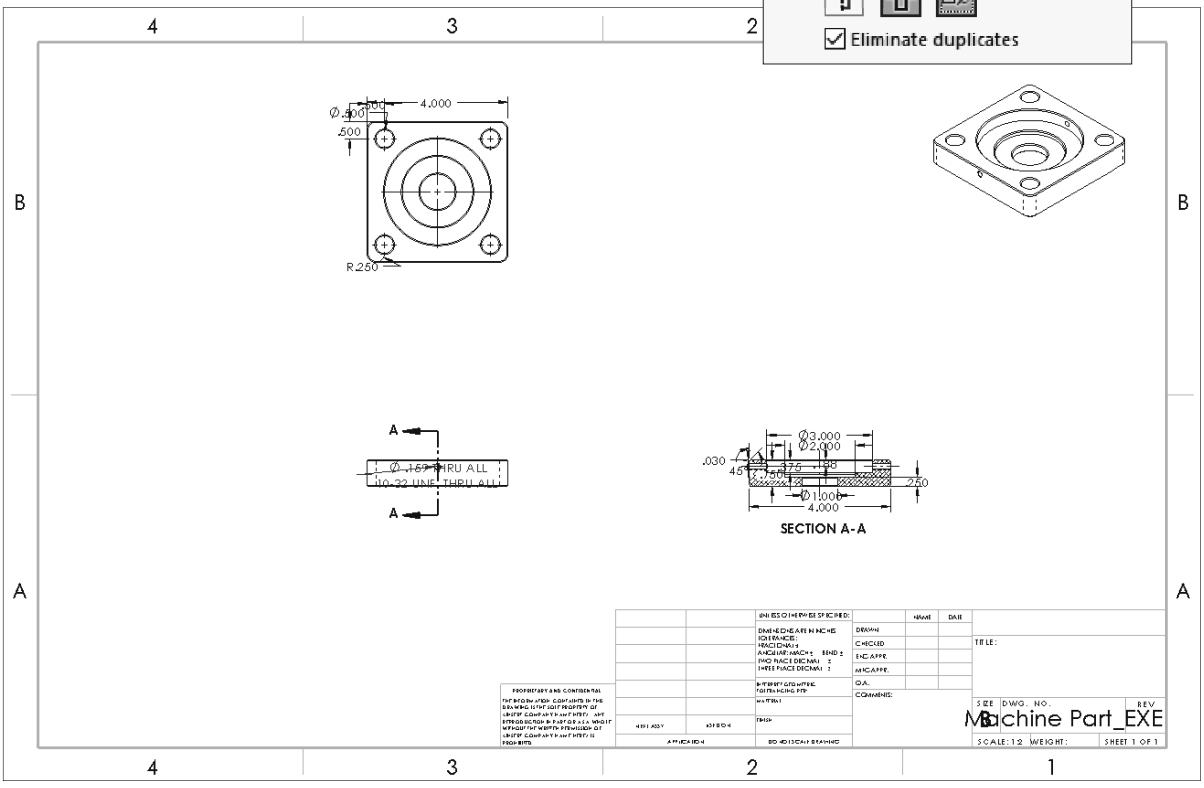
SECTION A-A

6. Adding the Model Dimensions:

Click **Model Items** on the **Annotation** tab.

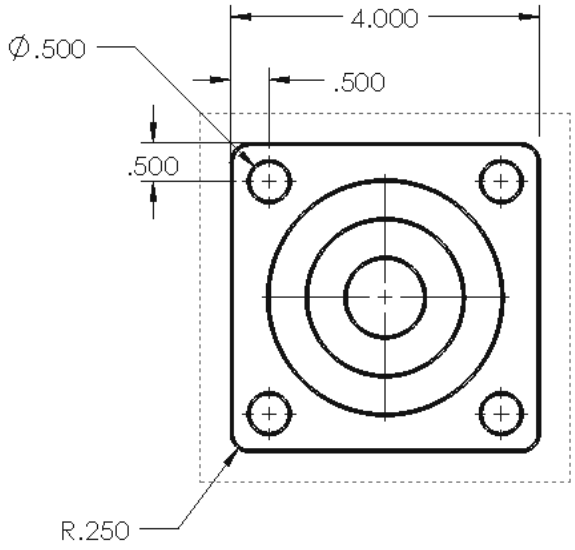
Select the options shown in the dialog box on the right and click **OK**.





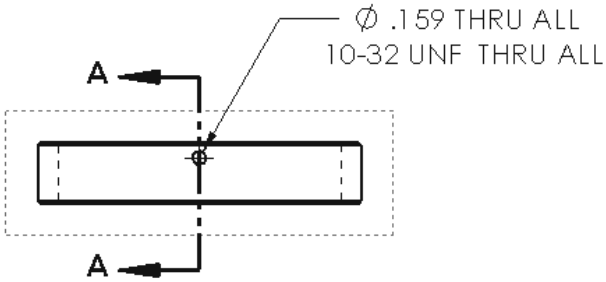
7. Rearranging dimensions:

Zoom in on each drawing view and rearrange the dimensions similar to the images shown below.



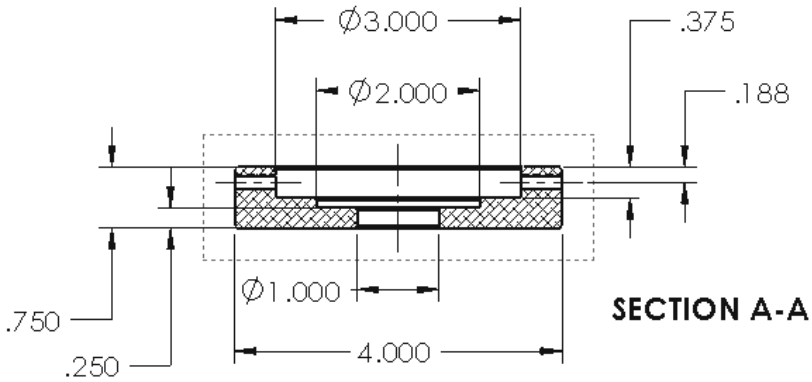
Start with the **Top** drawing view, move the dimensions so that they are easier to read.

Next, zoom closer to the **Front** drawing view.



The hole callout will be modified in the next step.

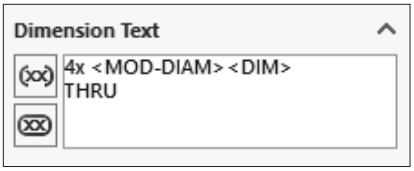
Zoom in on the **Section View**.

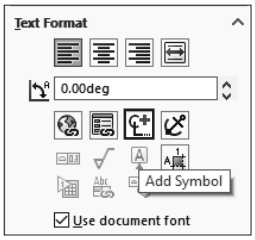


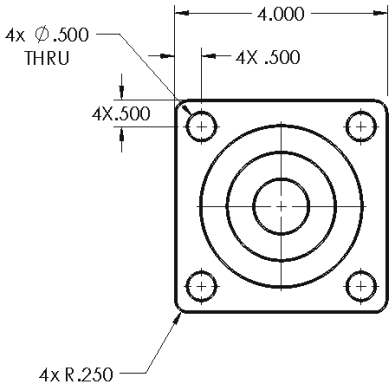
Rearrange the dimensions similar to the image shown on the right.

8. Modifying the dimensions:

Add the number of instances to the dimensions using the **Dimension Text** box as shown here.

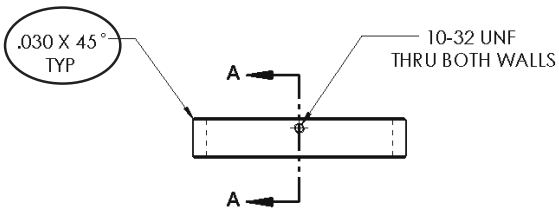


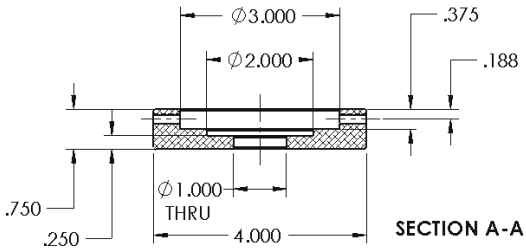




Add a note for the chamfer dimension (circled).

Click the Add Symbol button and select the degree symbol from the list.





9. Adding the General Notes:

Click **Note** from the **Annotation** tab, and enter the notes below:

NOTES: UNLESS OTHERWISE SPECIFIED
1. INTERPRET DRAWINGS AND DIMENSIONS PER ASME ANSI-Y14.5
2. DIMENSIONS ARE IN INCHES, 3 DECIMAL PLACES
3. MATERIAL: ALUMINUM 5052-H32
4. BREAK ALL EDGES, TUMBLE, AND DEBURR PARTS
5. SHIPPING PACKAGE TO BE LABELED WITH PART NUMBER, MANUFACTURER'S NAME, LOT NUMBER, AND QUANTITY
6. PARTS TO BE CLEAN AND FREE OF LUBRICANT OR DEBRIS

Place the General Notes on the lower left corner of the drawing.

10. Filling out the Title Block:

From the FeatureManager tree, expand **Sheet1** and double-click the **Sheet-Format1** to activate the back layer.

Enter the information shown below, in each appropriate section.

UNLESS OTHERWISE SPECIFIED:		NAME	DATE	SOLIDWORKS CORP.
DIMENSIONS ARE IN INCHES TOLERANCES: FRACTIONAL ± ANGULAR: MACH ± BEND ± TWO PLACE DECIMAL ± THREE PLACE DECIMAL ±	DRAWN	YOUR NAME	MM-DD-YY	
	CHECKED			TITLE: MACHINED PART EXE
	ENG APPR.			
	MFG APPR.			
INTERPRET GEOMETRIC TOLERANCING PER:	Q.A.			
MATERIAL 5052-H32	COMMENTS:			SIZE B / DWG. NO. 012-34-568 / REV 01
FINISH				
DO NOTSCALE DRAWING				SCALE: 1:2 / WEIGHT: / SHEET 1 OF 1

Double-click **Sheet1** on the FeatureManager tree to switch back to the Sheet.

11. Saving your work:

Make any corrections as needed prior to saving the drawing document.

Select **File, Save As**.

Enter **Machined Part_EXE.slddrw** for the file name.

Click **Save**.

Close the part and the drawing documents.