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## **Introduction**

### **CATIA Version 6 Drafting**

Upon completion of this course, the student should have a full understanding of the following topics:

- Creating drawings
- Creating views
- Modifying views
- Creating detail, broken, auxiliary, and section cut views
- Dimensioning and marking up view geometry
- Creating text
- Using symbols, details, and patterns in a drawing
- Creating and modifying basic geometry
- Using folding lines and multi-view projections to create view geometry
- Creating and using backgrounds

## Drafting

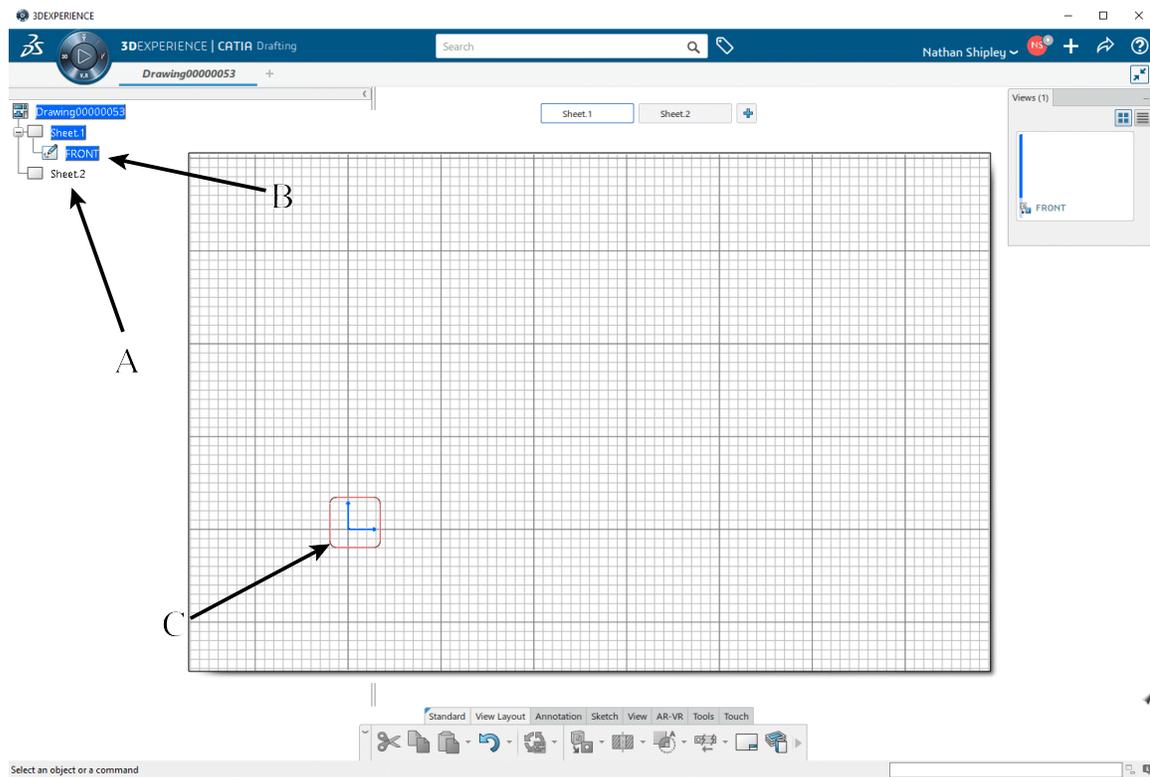
Currently, it is necessary for designers to create paper drawings and layouts of their parts and assemblies. This course will cover the steps necessary to create multiple view drawings and detail sheets of parts and assemblies. This will include all annotations and dimensions that are necessary to finish a completed drawing.

The Drafting workbench includes a variety of icons that allow for the creation of these drawings. It is normally considered that the workflow should be from 3D to 2D. There are some tools for generating 2D geometry independent of a 3D model, but it is not a common occurrence.

You should already have a good understanding of Part Design upon starting this course. This will complete a basic cycle of events from designing your own parts to creating a multi-view drawing. The course will demonstrate creating a multi-sheet drawing of an assembly, as well as the detail drawings of the individual parts. Unfortunately, the industry still requires and depends heavily upon paper drawings. There has been much discussion on “paper-less” environments where the end users would just pull up the design on the computer and analyze the areas of interest. In the future, that may be the case. But until then, drafting is an extremely important aspect of your V6 knowledge.

At any point, it will be necessary to create drawings for technical publications and handbooks. This class will explore the options available to export pictures from CATIA to wordprocessors that then could be used to create technical documents, or even web documents.

## Drawing Screen



This is how the drawing screen looks with two sheets and one empty view. Normally, when starting a new drawing, you will only have one sheet and no views.

- A Each sheet created in a drawing is represented in the specification tree. It is possible to create many different sizes of sheets.
- B Each view created in a drawing is represented in the specification tree under the sheet that the view belongs to.
- C Each view created has a view axis that represents the orientation of the current view. It is possible to hide or show the view axis.

## Section Views and Section Cuts

Section views and section cuts show the profile of a part at a specific location. The areas that contact the cutting lines are filled with a pattern. If the part has a material applied to it, the pattern will be based upon that material. The difference between a section view and a section cut is that a section view will show the geometry that lies beyond the cut line, whereas a section cut will only show what exists at the cut line. The cut line can be a single line, or it can be several lines with joggles to show a cut through the same part at different locations. Also available are aligned section views and aligned section cuts. The aligned styles allow the joggles to be at angles other than 90 degrees. They also display how the part appears at every line of every joggle.

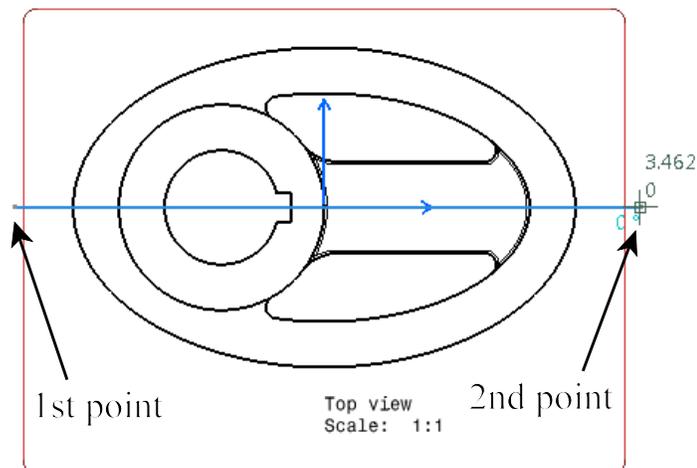
**Open the DRFT - Section Views and Cuts drawing.** You should notice that the front view is active since the outline is red. You should also notice by the symbols in the tree that the views are not synchronized with the 3D part.

**Press the third mouse button on a view frame and choose the *Load 3D Data* option, then update the drawing.** If you do not synchronize the views with the part, the view creation icons will not be selectable.

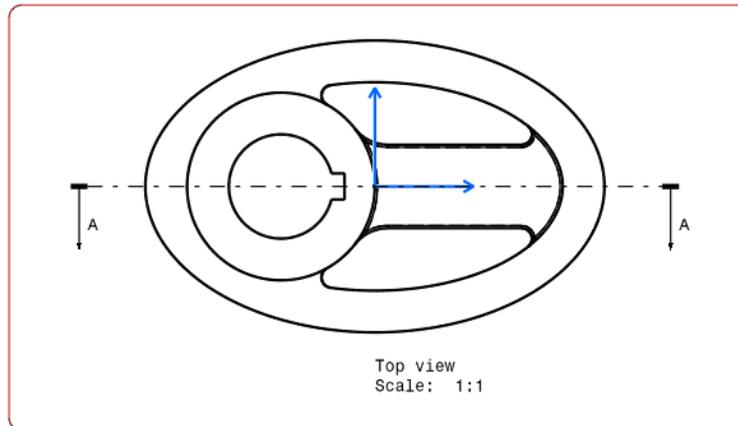
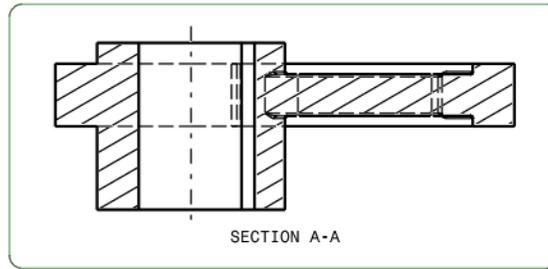
**Double select on the top view.** The frame of the top view turns red to indicate that it is active.

**Select the Offset Section View icon.**  You will create a single line through the top view.

**Click to define the starting point of the section cut line as shown below, then double-click to define the end point.** A single click will continue the cut line, creating a joggle at the selected locations. Ensure you double-click where you want the cut line to end. If at any time you make a mistake, select the Undo icon or press Ctrl + Z as many times as necessary to take you back to a previous selection. 



**Position the section view above the top view as shown below.** Section arrows appeared on your cut line. The frame color for section views is green.



*Note: You have the option of selecting a circular edge to define the cut line. If you select on a circle, the cut line will automatically pass through that circle's center point.*

**Delete the section view by selecting its frame and pressing the Delete key.** You will create the section view using existing geometry to help locate and orientate the cut line.

**Turn on the Create Detected Constraints icon from the Sketch section.**  It is located in the sub-toolbar of the Snap to Point icon. This will create any detected constraints.

**Select the Offset Section View icon.**  You will locate the cut line using a circle on the drawing.

When one of the sectioning icons is selected, the *Tools Palette* toolbar appears. Its options assist with orientating the cut line.

**Tools Palette**

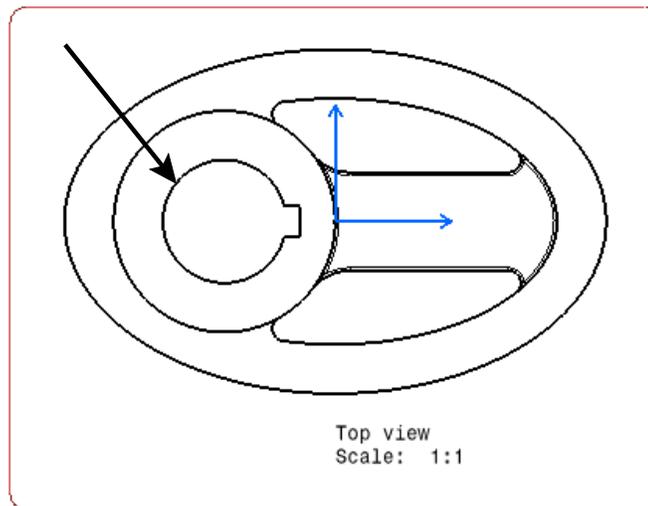


 Parallel

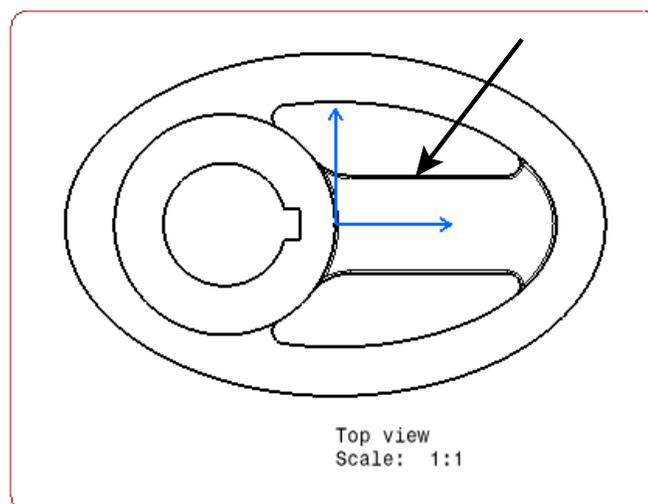
Orientates the cut line parallel to the selected geometry

	Perpendicular	Orientates the cut line perpendicular to the selected geometry
	Angle	Orientates the cut line at a specific angle to the selected geometry
	Offset	Offsets the cut line a specific distance from the selected geometry

**Select the inner circle of the cutout as shown below.** The cut line snaps to the center of the circle, but it can still be rotated to any angle.



**Ensure the Parallel icon is selected in the *Tools Palette* toolbar and select the top, horizontal line shown below.**  The cut line is forced to be parallel to the selected line.

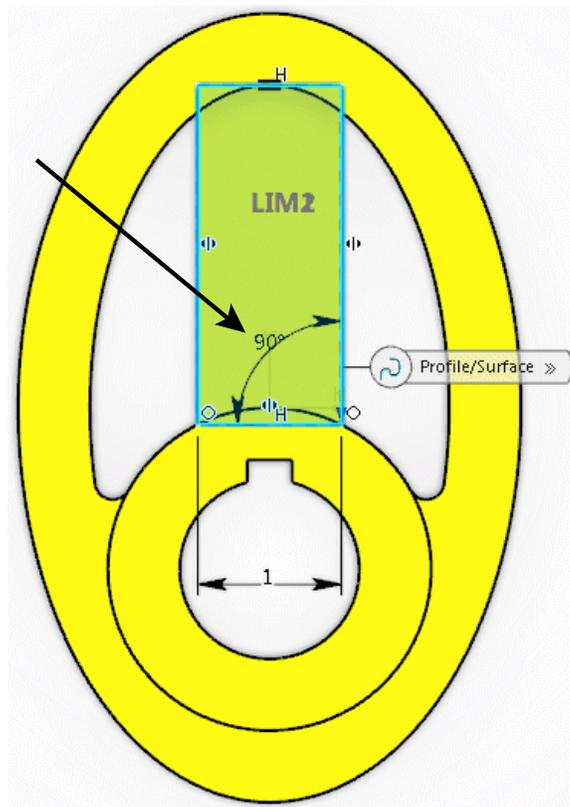


**Double-select to define the end of the line around the same location as before.** Now you have to position the view.

**Drag the section view above the top view as you did earlier.** By having the Create Detected Constraints icon turned on when defining the cut line parallel to the geometry, it will remain associated. You will change the angle of the pad to see how it affects the cut line.

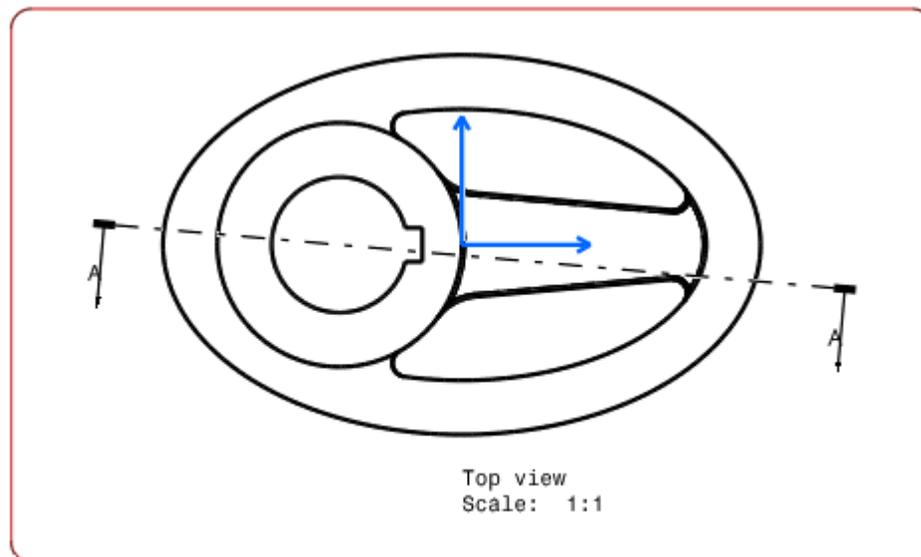
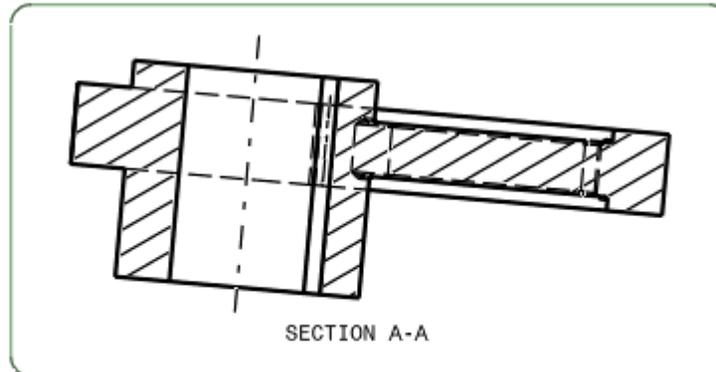
**Press the third mouse button on any view frame and choose the *Open 3D Data* option.** The part that the views are linked to opens.

**Double-select *Pad.3* in the part window and change the 90 degree constraint to 85.** You can double-click on the constraint to change it without having to enter the sketch. The pad is now tapered.



Select **OK**.

Go back to the drawing and select the **Update Current Sheet** icon.  The cut line changed, and so did the resulting view.

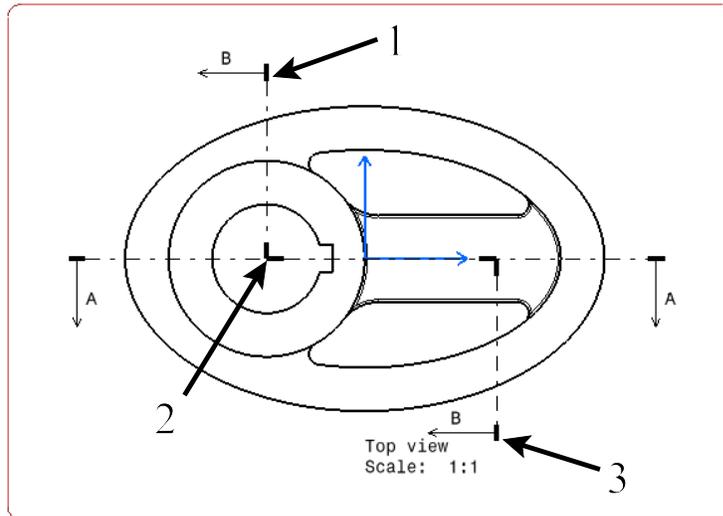


Change the pad angle back to 90 degrees in the part, then return to the drawing and update it. The section view is at its original orientation again.

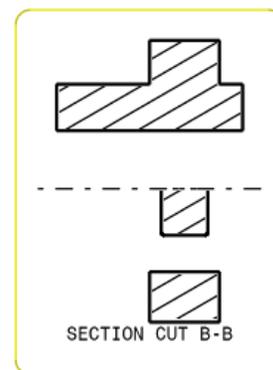
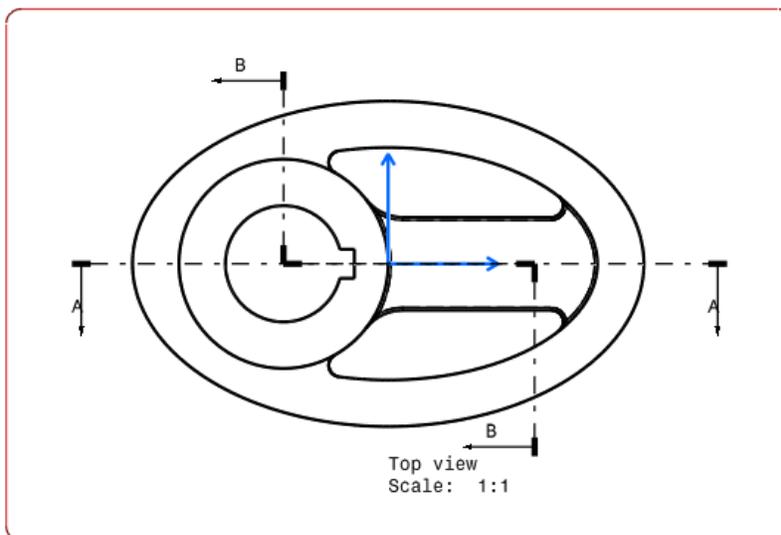
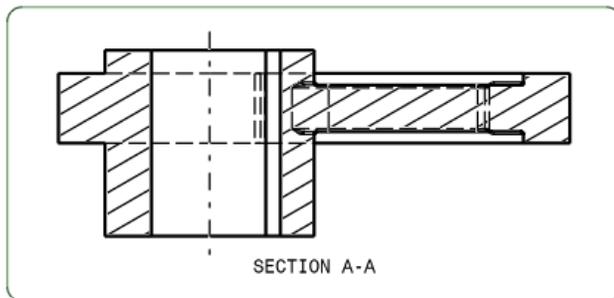
Turn off the **Create Detected Constraints** icon.  You will now create a section cut.

Select the **Offset Section Cut** icon.  It is located within the sub-toolbar of the Offset Section View icon. This will allow you to define cut lines as well, but the view will only contain the foreground of the part up to the cut line.

**Define your cut line similar to the one shown below.** The numbers show the order of selection. Make sure you double select your last location.



**Position your view to the right of the top view as shown below.** The frame color for section cuts is yellow.

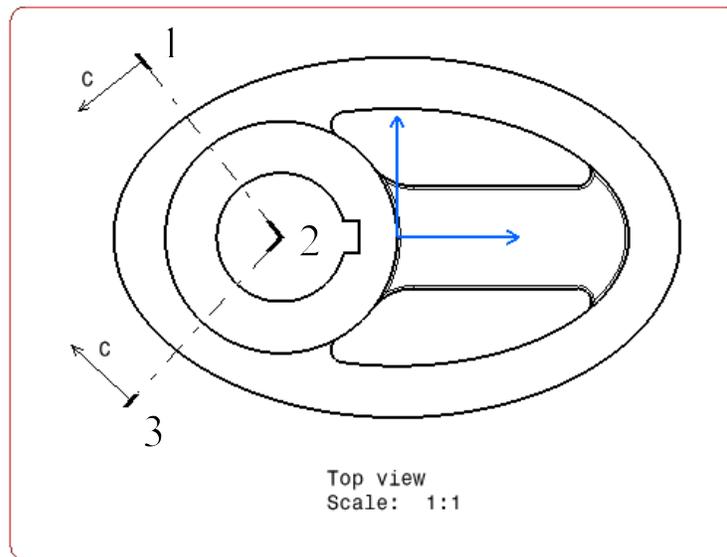


*Note: Comparing the section cut to the section view, you will see that the section cut does not include anything past the cut line, whereas the section view does.*

You are going to create an aligned section view now.

**Select the Aligned Section View icon.**  It is located within the sub-toolbar of the Offset Section View icon. With this tool, you will create cut lines that are not at 90 degree angles.

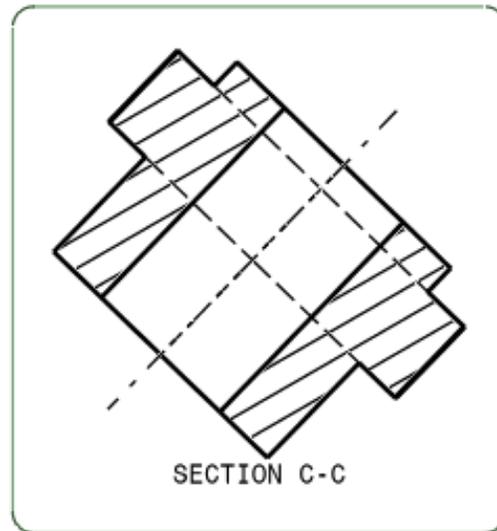
**Define your cut line as shown below.** The numbers show the order of selection. Make sure you double select your last location.



*Note: The other sectioning lines have been hidden for clarity in the above image.*

**Position the new view up and to the right of the front view.**

Change the *Properties* of the aligned section so that *Hidden Lines* are not shown. It should appear similar to the diagram below.



Creating an aligned section cut is done in the same manner. The difference is that, like an offset section cut, it only shows the foreground of the part up to the cut line. As seen above, an aligned section view shows both the foreground and the background. Feel free to experiment with the Aligned Section Cut icon on your own. 

So far, you have been defining your cut line in the drawing. However, you also have the option of defining an element in the 3D part as the cut line. This is shown next.

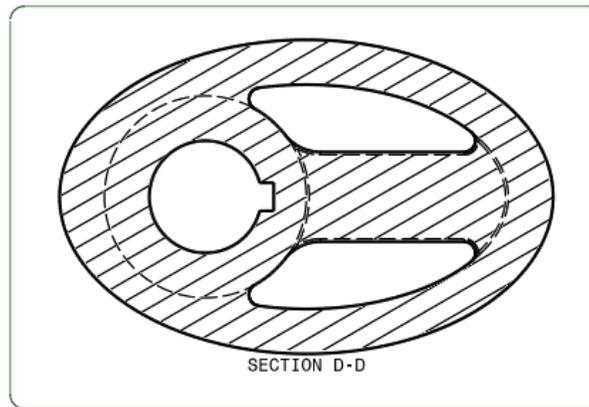
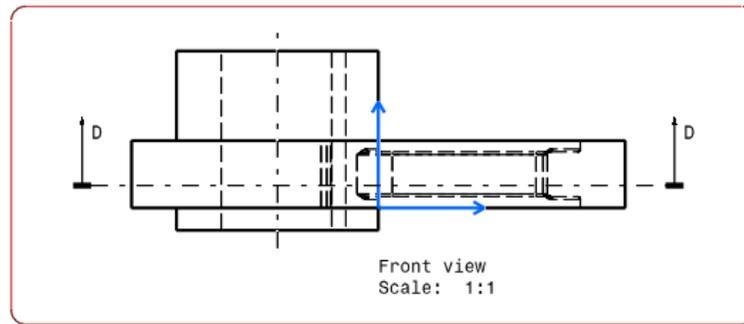
**Create a plane in the part that is offset upward from the xy plane 0.25 inches.** This will be the element to define the cut line.

**Switch to the drawing and update it.**

**Activate the front view, then select the Offset Section View icon.**  You will need to update the views since you made a change to the part.

**Switch back to the part and select the new plane.** The cut line automatically appears in the front view. It is not associated or linked to the plane, and can be modified in the drafting workbench.

**Position the section view under the front view as shown below.**



You can also create a sketch in your part that can be used to define a cut line. When you use a sketch, the cut line is associated, or linked, to the sketch. If you want to modify the cut line, you must open the part and modify the sketch, then update your views. If you erase the sketch that was used to define the cut line, the cut line will no longer be linked, and will be converted to a normal cut line that can be modified within the drafting workbench.

There are also options for **Curve Section View** and **Curve Section Cut** which are meant to utilize a 3D curve in the geometric part to define the section. These will not be covered.

**Save the drawing and close all documents.**

## Dimensioning

This section will discuss the various methods of creating and generating dimensions. You will also explore how to modify dimensions.

### Creating Dimensions

Dimensioning can be a little overwhelming due to the amount of options, but once you get a good understanding of those options, dimensions are fairly easy to create. This exercise will focus on creating dimensions manually.

**Open the DRFT - Basic Dimensioning drawing.** You will experiment with the different dimensioning tools.

### Length/Distance

Select the **Dimensions** icon from the *Annotation* section.  The *Tools Palette* toolbar appears. The options in the toolbar are also available by pressing the third mouse button while creating a dimension.



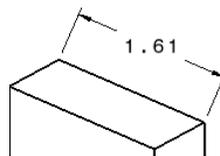
Projected Dimension

The orientation of the dimension is based upon the location of the mouse cursor



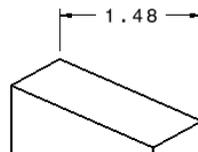
Force Dimension on Element

The orientation of the dimension is forced parallel to the selected element



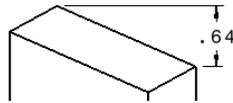
Force Horizontal Dimension...

The orientation of the dimension is forced horizontal in the view



**Force Vertical Dimension...**

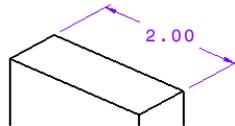
The orientation of the dimension is forced vertical in the view

**Force Dimension Along a...**

The orientation is forced either parallel, perpendicular, or at a specific angle to a selected element; additional icons are available in the *Tools Palette* toolbar with this option

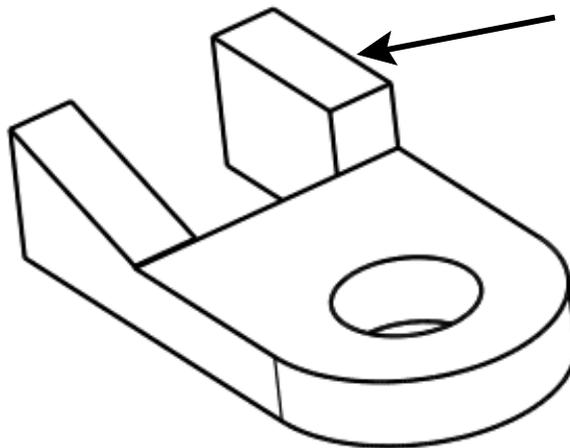
**True Length Dimension**

The dimension shown is the actual dimension of the element in the part; the purple color signifies a true length

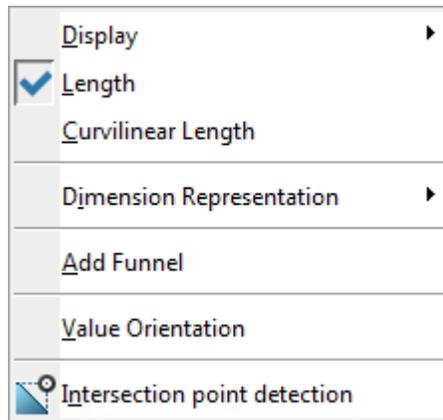
**Intersection Point Detection**

Creates dimensions by using intersection points; the icon must be pre-selected in order for the dimension to use intersection points

In the isometric view, select the line shown below. A length dimension appears. Before positioning it, you will look at some of the options that can be adjusted during its creation.



**Press the third mouse button.** A contextual menu appears with the following options.

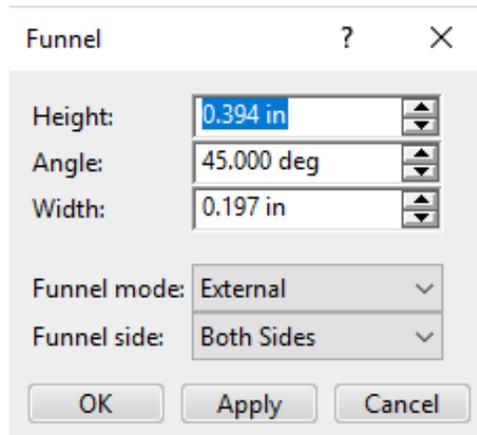


*Dimension Representation*

These options appear as icons in the *Tools Palette* toolbar when you select certain dimension icons

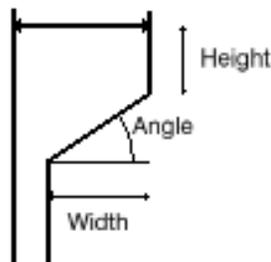
*Add Funnel*

Adds a funnel on the extension lines in order to account for too much or too little space on either side of a dimension



*Height, Angle, Width*

Refers to the height, width, and angle of the funnel

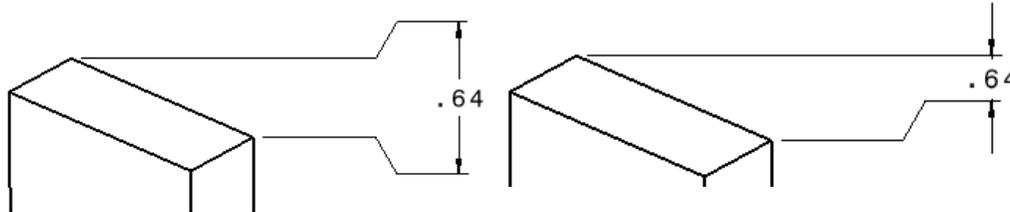


*Funnel mode*

Determines whether the funnel goes out (*External*) or in (*Internal*)

*Funnel side*

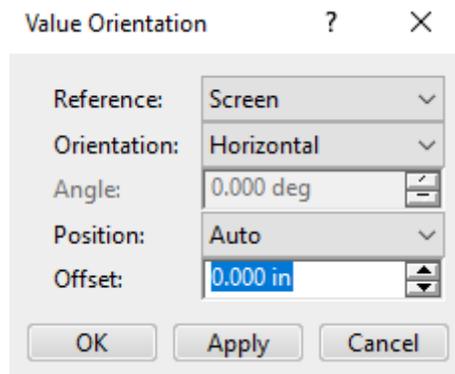
Displays the funnel in one of three ways: 1) *Both Sides*, 2) *Left or Bottom*, 3) *Right or Top*



*External, Both Sides*

*Internal, Left or Bottom*

*Value Orientation*



*Reference*

The orientation of the value can reference the *Screen*, the *View*, or *Dimension Line*

*Orientation*

The dimension can be orientated either *Horizontal*, *Vertical*, or *Fixed Angle* when referencing the *Screen* or *View*. It can be orientated either *Parallel*, *Perpendicular*, or *Fixed Angle* when referencing the *Dimension Line*

*Angle*

This option is only available when defining a *Fixed Angle*

*Position*

Positions the dimension value either *Inside* the extremities or *Outside* the extremities. It defaults to *Auto*, which will switch the dimension value based on the room available between the extremities.

*Offset*

The value will be positioned a specified distance away from the dimension line

Select **Dimension Representation**, then **True Length Dimension**. This will create a dimension representing the true length of the element.

Press the third mouse button and choose **Add Funnel**. You are going to create a funnel for this dimension. The *Funnel* window appears.

Enter the following parameters: **Height = 0.5**, **Angle = 30**, **Width = 0.25**, **Funnel mode = External**, and **Funnel side = Both Sides**. This defines the funnel.

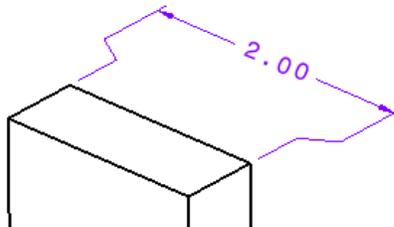
Select **OK**. The funnel is generated.

Press the third mouse button again. The **Add Funnel** option is no longer available. Instead, there is an **Edit Funnel** option and a **Remove Funnel** option.

Select **Value Orientation**. The *Value Orientation* window appears.

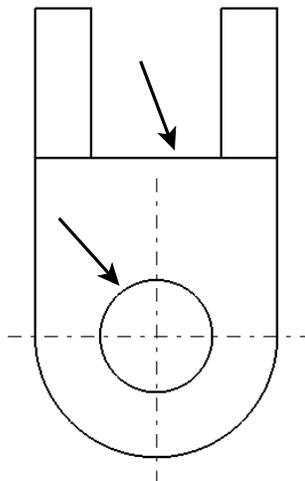
Enter the following parameters: **Reference = Dimension Line** and **Orientation = Parallel**, then select **OK**. The orientation of your dimension value changes.

Position the dimension up and to the right of the line and select at that location. The dimension is created, and should appear similar to the one shown below. The dimension will be purple if you have the Elements' Analysis icon selected, since it is a true dimension. This icon is located in the *View* section.

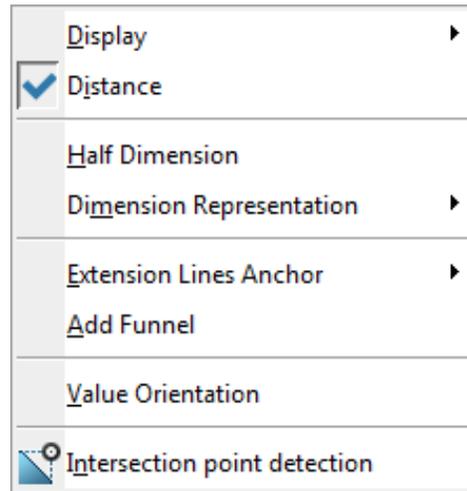


Select the **Dimensions** icon again.  This is the same icon you just used.

Select the circle and the line indicated below. A distance dimension appears.

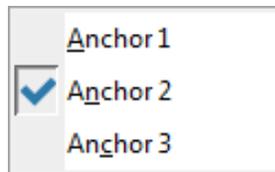


**Press the third mouse button.** Two new options appear. For now, only one will be discussed.



*Extension Lines Anchor*

Determines where the extension lines will be based from



*Anchor 1*

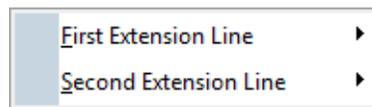
A location along the circle

*Anchor 2*

This is always the center of the circle

*Anchor 3*

A location along the circle



*First Extension Line*

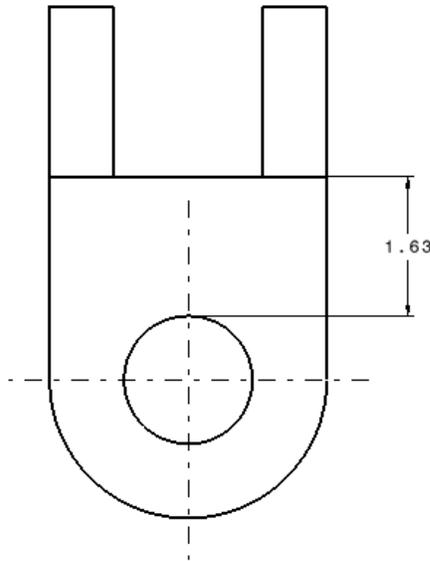
Appears when two circles are selected for the dimension; this determines the location of the first extension line

*Second Extension Line*

Appears when two circles are selected for the dimension; this determines the location of the second extension line

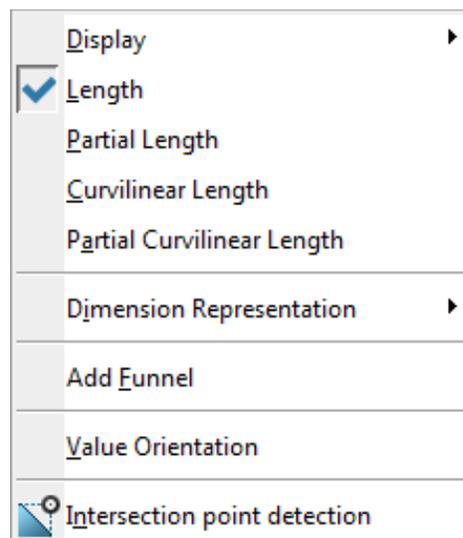
Select **Extension Lines Anchor**, then select **Anchor 3**. A location on the circle which is closest to the line is chosen.

Click to locate the dimension on the sheet as shown below.



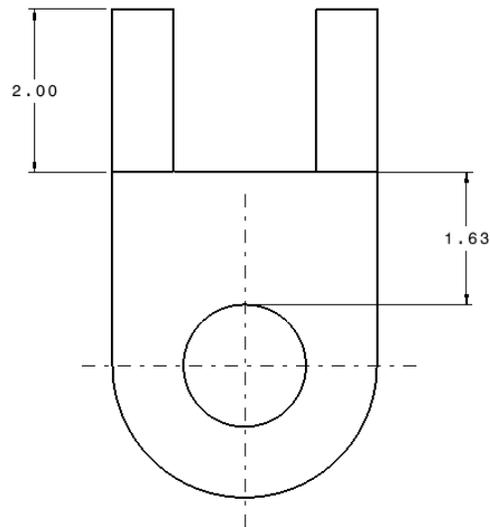
Next, you will create a simple length constraint.

Select the **Dimensions** icon, then select the **upper-left vertical line in the top view** and press the **third mouse button**.  A contextual menu appears as shown here.



There are three new options: *Partial Length*, *Curvilinear Length*, and *Partial Curvilinear Length*. The *Partial Length* option enables you to create a dimension from locations on a linear element other than the endpoints. The other two options will be discussed in a separate exercise.

Click on the sheet to locate the dimension as shown here.



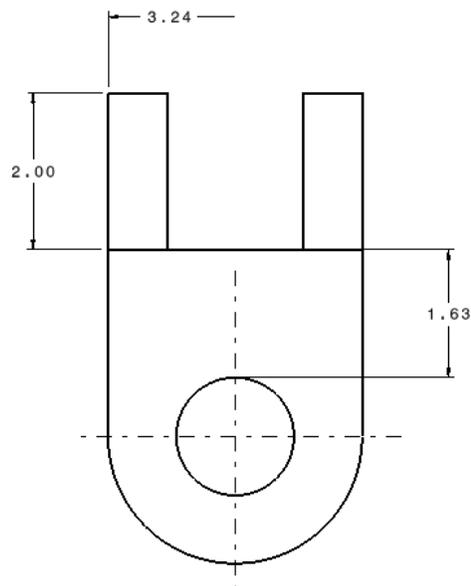
Select the **Dimensions** icon again.



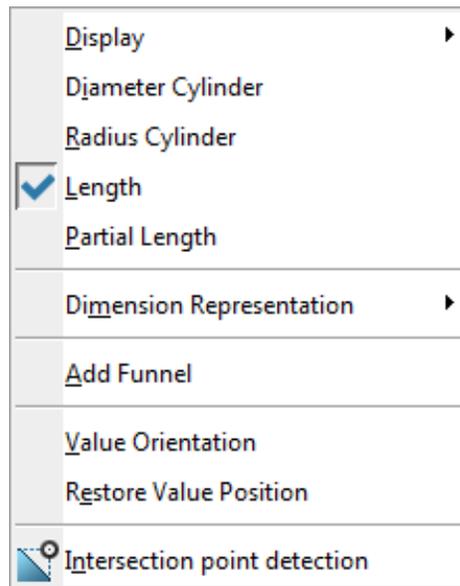
Select the **vertical center line** and the **leftmost vertical line in the top view**. You are going to create a half dimension which will automatically double the value and remove the extension at the center line.

Press the **third mouse button** and select *Half dimension*. This is a toggle option, meaning that all dimensions created after this will also be half dimensions until you turn it off.

Specify a location for the dimension. The value doubled and there is only one extension line visible.



Select the **Dimensions** icon and click on left-most vertical line in the right view, then press the third mouse button.  The following contextual menu appears.



A couple of new options appear: *Diameter Cylinder* and *Radius Cylinder*. These options are available because CATIA knows the line you selected belongs to a cylindrical face.

**Press Esc to exit the dimension creation mode and delete all of the dimensions.** You can do this by clicking on them and pressing the Delete key.

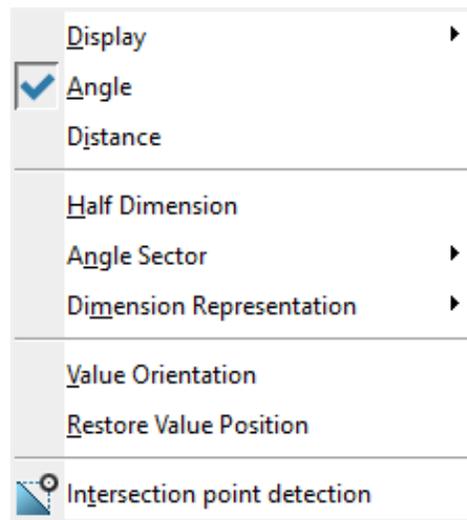
## Angle

Select the **Dimensions** icon, then select on the angled line and the top horizontal line in the right view.  Unless you have already turned *Half dimension* off, you need to do so now.

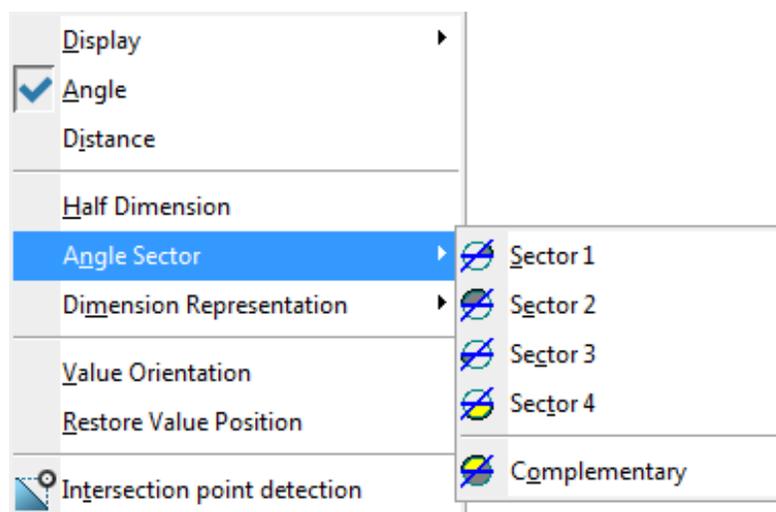
Press the third mouse button and select *Half dimension*. This will toggle the *Half dimension* option off.

Press the third mouse button again and select *Angle*.

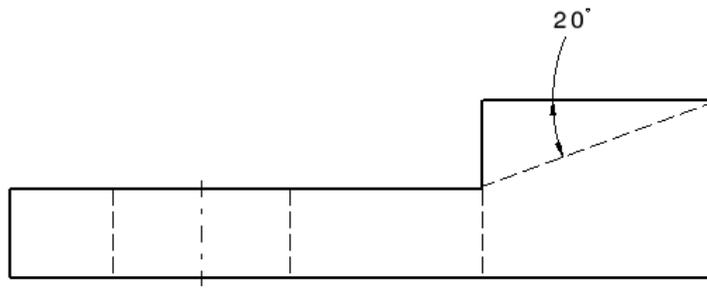
Press the third mouse button again. Most of these options have already been covered.



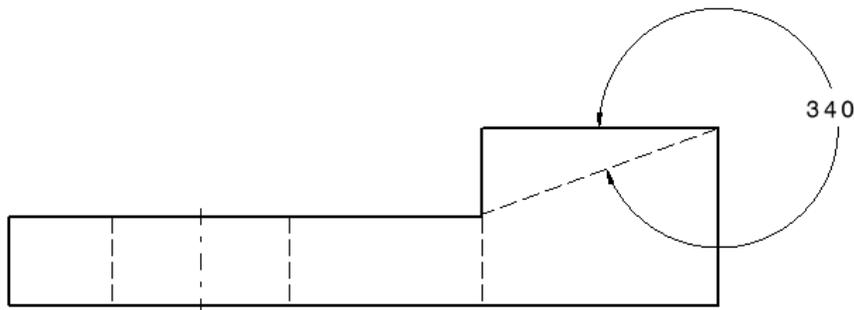
The *Angle Sector* sub-menu allows you to specify which sector you would like to create the dimension for.



Specify a sector and a location to obtain the same dimension shown below.



The *Complementary* option within the *Angle Sector* sub-menu will give you the opposite angle of the initial value. This is shown below.

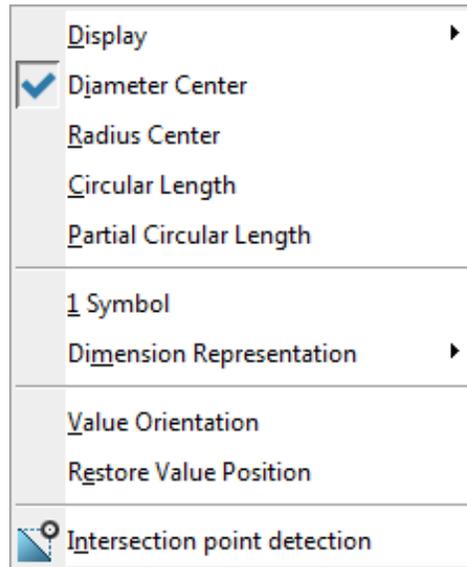


*Note: Although CATIA allows you to do so, you should not typically dimension to hidden lines. Refer to Appendix B in this manual for general, drafting standards.*

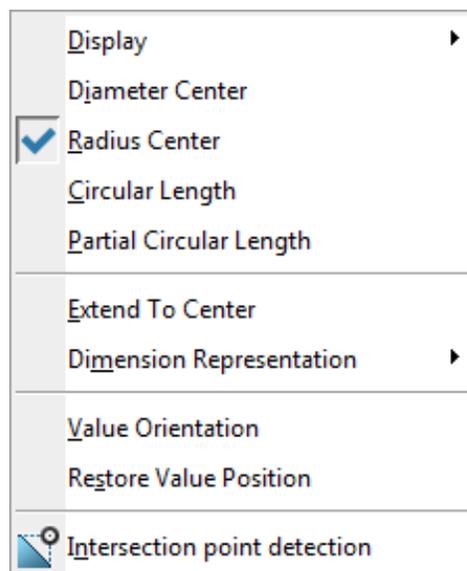
## Radius

Select the **Dimensions** icon.  You will create a radius dimension, which is usually used for partial circles.

Select the large arc in the top view, then press the third mouse button. If the dimension is set to *Diameter Center*, the following menu appears.



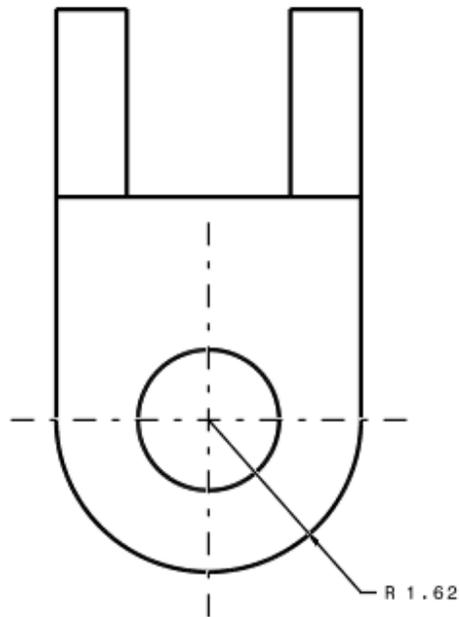
If the dimension is set to *Radius Center*, the following menu appears.



Select the **Radius Center** option if it is not already chosen, then select **Extend to Center**. The extension line extends to the center point of the arc.

*Note: When using the Diameter Center option, Extend to Center will automatically be activated if you press the third mouse button within the circle. Pressing the third mouse button outside of the circle turns it off. You must also have the 1 Symbol option selected in order to see the Extend to Center option.*

Specify the location of the dimension as shown below.

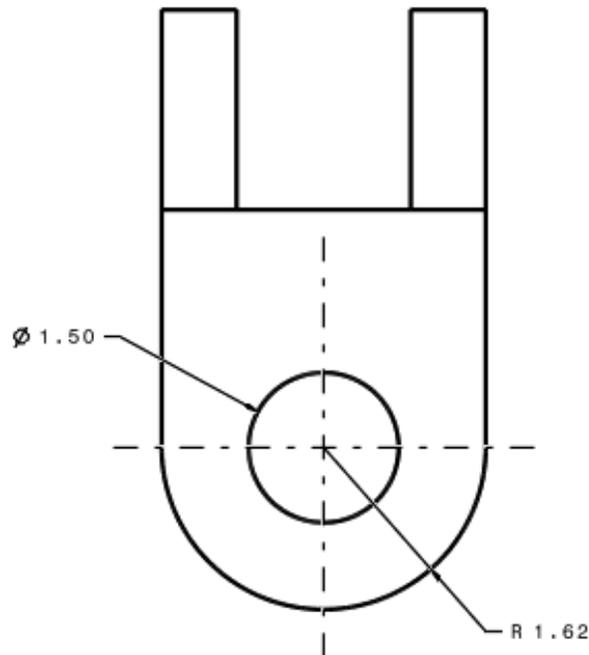


## Diameter

Select the **Dimensions icon**.  You will create a diameter dimension, which is usually used for complete circles.

Select the **full circle in the top view** and press the **third mouse button**, then select the **Diameter Center** option if not already selected, then the **1 Symbol** option. This allows your dimension to have only one arrow instead of two.

Specify the location of the dimension as shown below.



Save the drawing and close all documents.

## Annotations

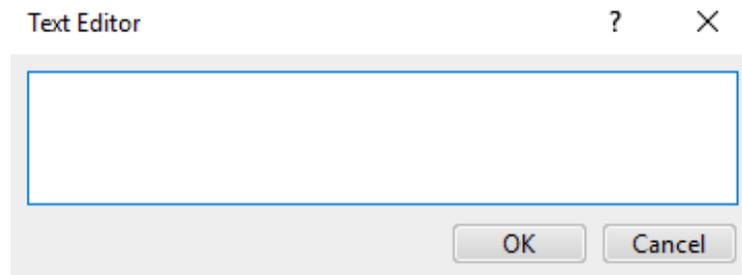
Annotations and symbols are normally required to finish a drawing. Annotations are used to convey notes and information to the downstream user, while symbols are used to represent items, such as part numbers and welds. This section will cover the various tools available to create both annotations and symbols.

### Creating Text

**Open the DRFT - Annotations drawing and part.** This will allow you to work with items in both windows. The drawing contains four views.

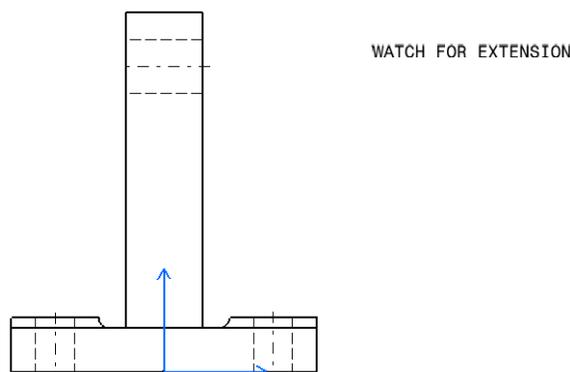
**Activate the front view.**

**Select the **Text** icon, then select a location in the front view to the right of the extension.** **Abc** This defines the anchor point for the text. The *Text Editor* window appears.



**Enter WATCH FOR EXTENSION and select *OK*.** The text appears. You can move the text to any location by clicking and dragging it with the first mouse button.

**Move the text to the approximate location shown below.**



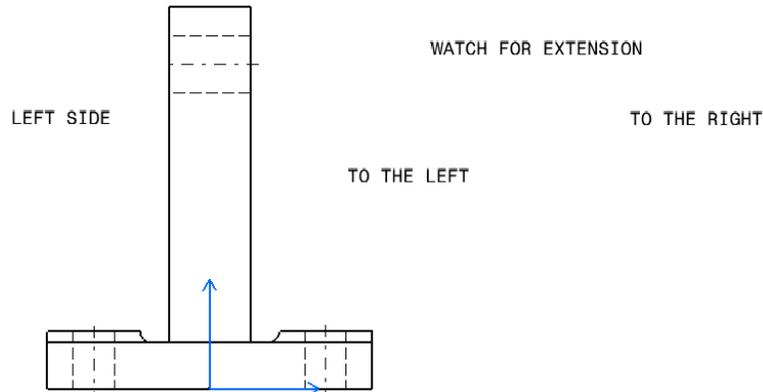
**Create text that says TO THE RIGHT underneath and to the right of the last one.**

**Create text that says TO THE LEFT underneath and to the left of the first piece of text.** Now you will create text that is associated to geometry.

Select the **Text** icon, then select the left, vertical line of the extension.  The *Text Editor* window appears.

Key in **LEFT SIDE** and select **OK**. The text appears.

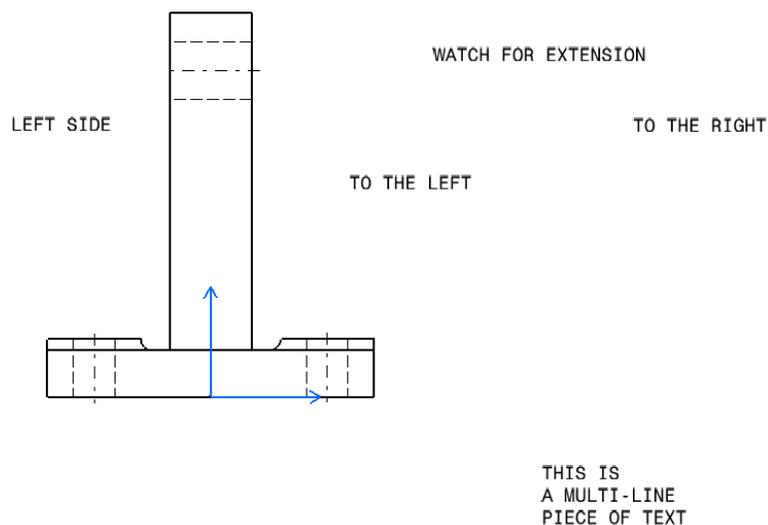
Move the text to the left of the extension as shown below.



All of the text has the same graphical properties. You can set the properties of the text before you create it by using some of the icons in the top toolbar area. You will investigate these options when you modify text.

Select the **Text** icon, then position the text below and to the right of the geometry in the view. 

Key in **THIS IS** and press **Shift-Enter**, then key in **A MULTI-LINE** and press **Shift-Enter**, then key in **PIECE OF TEXT** and select **OK**. This should create a string of text that spans three lines as shown below. You have to press the Shift key along with Enter in order to add an additional line of text.



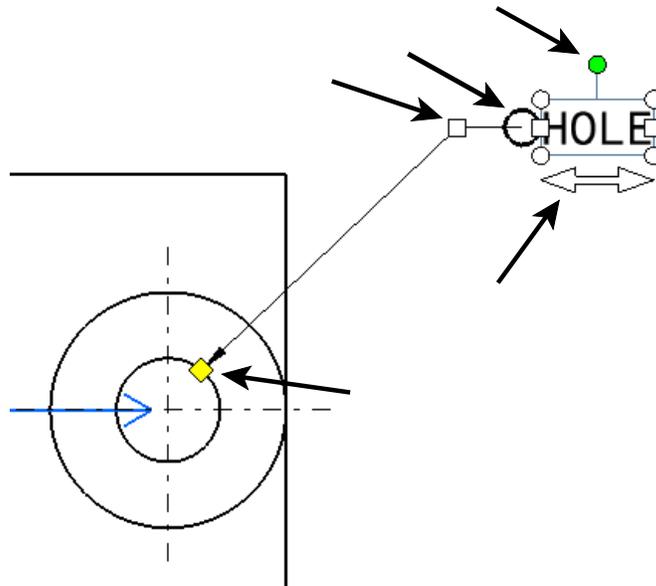
**Activate the top view.** You are now going to create text with a leader.

**Select the Text with Leader icon.**  It is located within the sub-toolbar of the Text icon. You will select an element to position the anchor point of the arrow head.

**Select the small circle on the right.** You should see a dashed frame appear. You can move it in order to position the anchor point of the text.

**Position the dashed frame up and to the right, then click on the sheet to define the anchor point for the text.** The *Text Editor* window appears.

**Enter HOLE and select *OK*.** The text with leader is created, and some positioning manipulators appear.



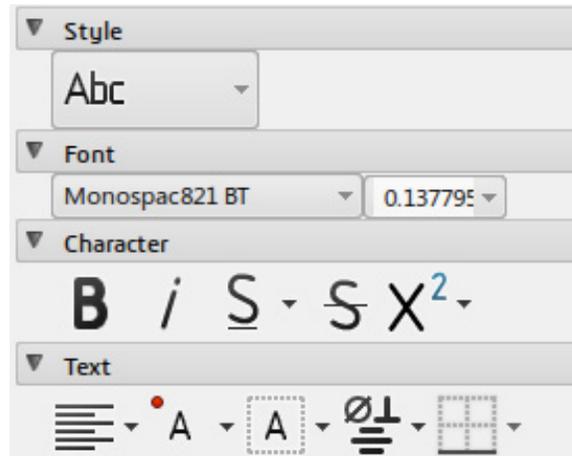
You have just learned how to create text. You will now explore the modification options.

## Modifying Text

There are many options to modify text, a lot of these options you can perform before you create the text. Most of the common options are located in the *Object Properties* menu.

**Right select in the display and select *Display, Object Properties*.** The *Object Properties* menu will appear.

### Object Properties for Text



These options allow you to change the font type and the font size, and whether or not the text is bold, italic, underlined, overlined, or scored. You can also make the text a subscript or a superscript. Other icons include text justification, anchor point, frame, and symbols. You will try out each option.

### Font properties

**Activate the front view and select *THIS IS A MULTI-LINE PIECE OF TEXT*.** The text related icons become available.

*Note: If the box is showing less than three decimal places, press the third mouse button while on the box and select Set precision. A Set precision window will appear. Change the Precision to 3 and select OK.*

**Change the font type to *Times New Roman*.** The text changes font.

**Change the font size to 0.276.** This option might contain more decimal places; select the option that is closest. The text becomes larger.

**Select the Bold icon.** **B** The text should appear bold. This is a toggle. Therefore, you can select it again to turn the bold off.

**Select the Bold icon again.** **B** The bold is removed.

**Select the Italic icon.** *i* The text appears in italics.

Select the **Italic** icon again.  The italics is removed.

Select the **Underline** icon.  All three lines of text appear underlined.

Select the **Underline** icon again.  The underline is removed.

Select the **Overline** icon.  It is located within the sub-toolbar of the Underline icon. All three lines of text appear with an overline.

Select the **Overline** icon again.  The overline is removed.

Select the **Strikethrough** icon.  All three lines of text are struck through.

Select the **Strikethrough** icon again.  The strike through is removed.

Select the **Superscript** icon.  The text appears as a superscript.

Select the **Superscript** icon again.  The superscript is removed.

Select the **Subscript** icon.  It is located within the sub-toolbar of the Superscript icon. The text appears subscripted.

Select the **Subscript** icon again.  The subscript is removed.

You can apply more than one property to text at any given time. It is possible to have a piece of text that is bold, italic, underlined, appears as a superscript, and has a strikethrough, all at the same time. The only options that cannot be applied simultaneously are the underline/overline and the superscript/subscript.

## Justification

You can justify text to the left, the center, or the right. The justification will only be evident when there are multiple lines in the same piece of text.

**Select the Center Justification icon.**  It is located within the sub-toolbar of the Left Justification icon. The text is left-justified by default.

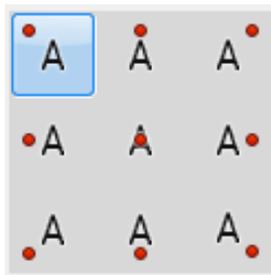
**Select the Right Justification icon.**  It is located within the sub-toolbar of the Left Justification icon. The text now aligns to the right.

**Select the Left Justification icon.**  The text aligns to the left.

## Anchor point

You can position text with respect to the anchor point. The text will move around the anchor point based upon which icon is selected.

**Select the down-arrow beside the Anchor Point icon.**  The anchor point options appear. Currently, the text is positioned so that the anchor point is in the upper-left corner. You will change the anchor point a couple of times.



**Select the center Anchor Point option.**  The text moved up and to the left in order for the anchor point to be in the center.

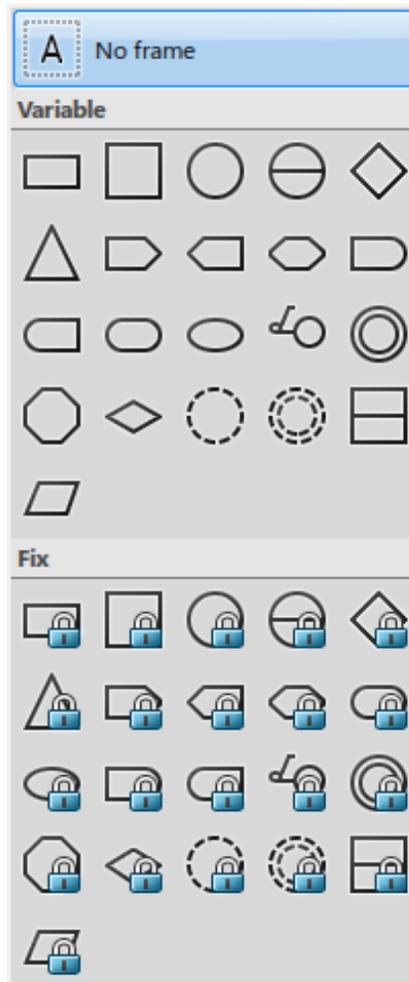
**Select the bottom-right Anchor Point option.**  The text moved up and to the left again in order for the anchor point to be in the lower right corner. Continue to experiment with the different anchor point locations.

**Select the top-left Anchor Point option.**  Your text returns to its original location.

## Frame

You have the option of adding a variety of frames to your text. You will explore a few of them.

Select the down-arrow beside the **Frame** icon.  The frame options appear.



The frames with the yellow locks are fixed-size frames defined in the standards file. These become useful when your company has a set size for specific frames and you must fit your text within the frame by modifying the text size. The other frames are variable-sized frames that will adjust to fit the text.

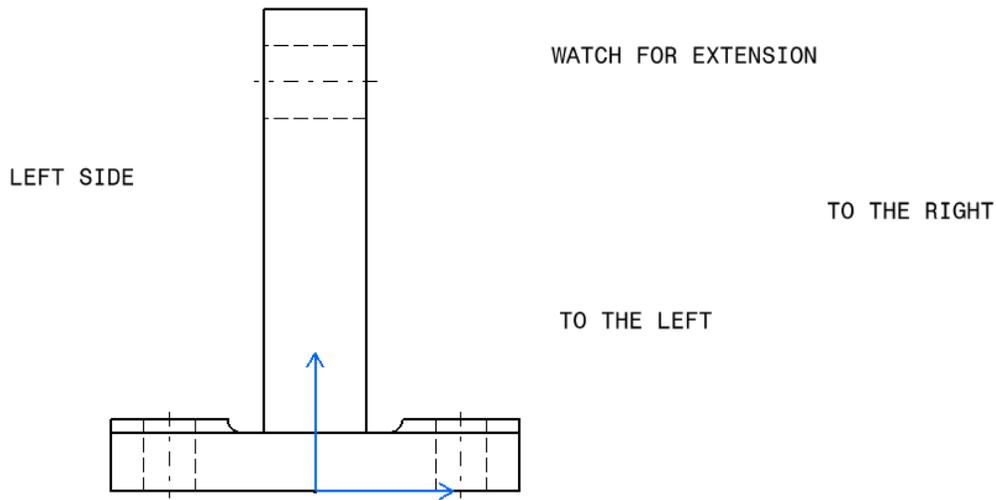
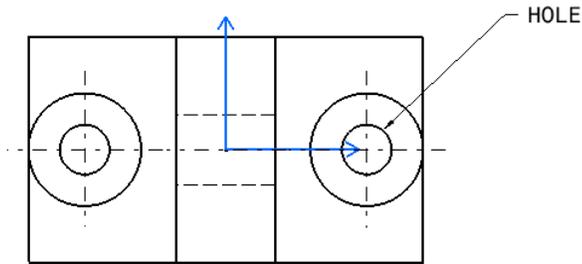
Select the **rectangular Frame** option.  A rectangular frame appears around the text.

Select the **circular Frame** option.  A circular frame appears around the text.

Select the **left flag Frame** option.  Feel free to experiment with the other frame options, but set the frame to this option before continuing.



Select on an empty area of the sheet to release the text. Your views should look similar to the diagrams shown below.



THIS IS  
A MULTI-LINE  
PIECE OF TEXT

Close the *Object Properties* menu. Keep in mind this can be turned on and off at your convenience.