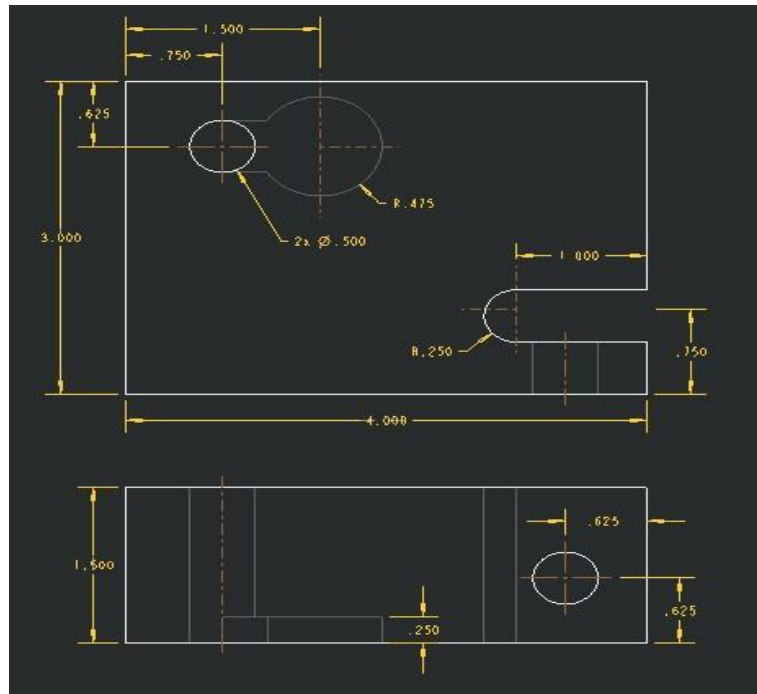


# Creo Tutorial: Offset Section

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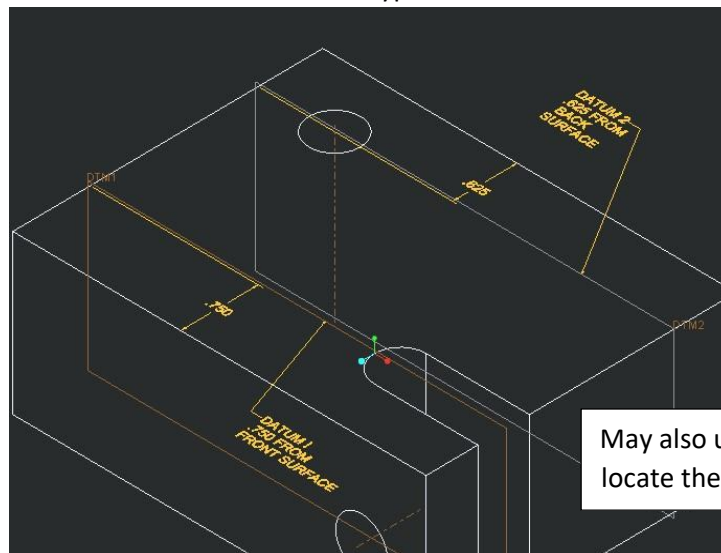
1. Create the following Part



2. Creating Cutting Plane Line

There are multiple ways to create an offset section. The method below has the user create datums to draw the cutting plane line on. This is helpful so when drawing the cutting plane line the designer does not need to worry about the location of the cutting plane line and will just need to reference the created datums to draw the line onto.

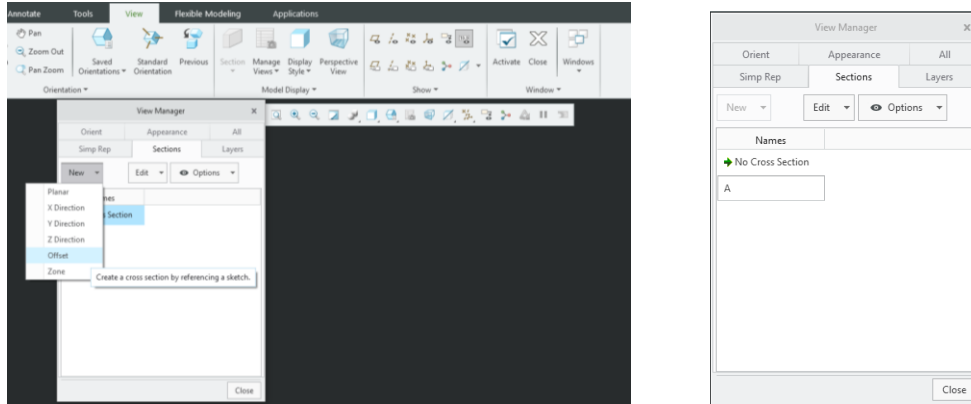
- a. Model Tab > Plane Icon > Select Surface to Offset from > Type in Offset > Select Ok



May also use Centerlines to locate the cutting plane line to

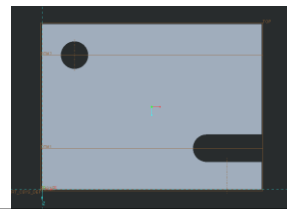
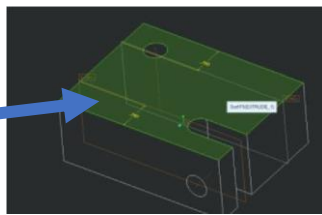
3. **Drawing the Cutting Plane Line:** Creo allows the designer to draw the cutting plane line (edge of the imaginary knife) within a 2D Sketch.

- a. View Tab > Manage Views > Sections > New > Offset > Name the Cut A (NOTE: 1<sup>st</sup> Cut on any part should be Labeled A; unless otherwise specified by the company or client)



- a. Select the surface to draw the cutting plane line on > The view plane will rotate perpendicular to the viewer and place the view into a sketch

Select Top Surface to Draw Cutting Plane Line On

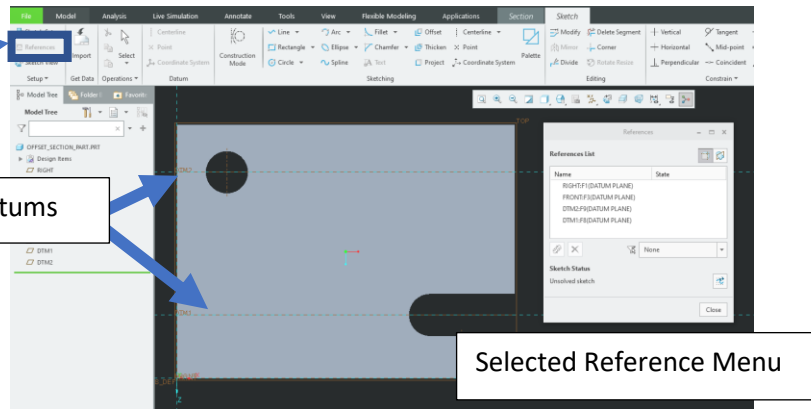


View Rotates into a Sketch

- b. Select Reference Tool > Select the Datums that were created for the cutting plane

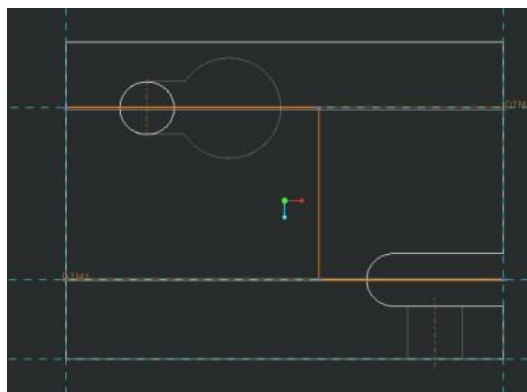
1. Select Reference Tool

2. Select Created Datums



- c. Draw the Cutting Plane using the line tool.

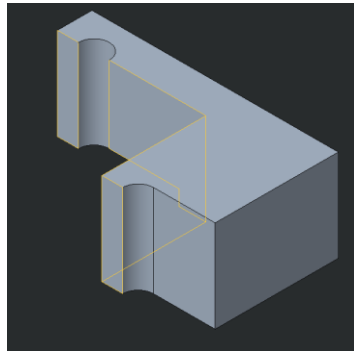
- i. Changing View to Hidden Lines makes easier to see all features
- ii. NOTE: Only draw lines that represent the Cutting Plane; This means an open profile will be created.
- iii. NOTE: Cutting Plane must start/end on the edge of the part or extend beyond. DO NOT start/stop in the middle of the part.
- iv. NOTE: All Cutting Plane Lines should be Vertical or Horizontal



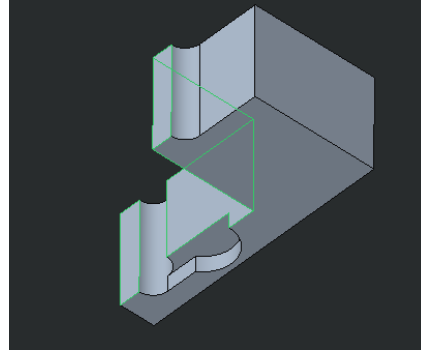
- d. Green Check when Complete Sketch
- e. Check the view that it is cutting correctly > Green Check to Accept Section View

For Example

- i. When cutting through a hole the cutting plane cuts through the whole hole showing the complete diameter
- ii. That features are cut through properly



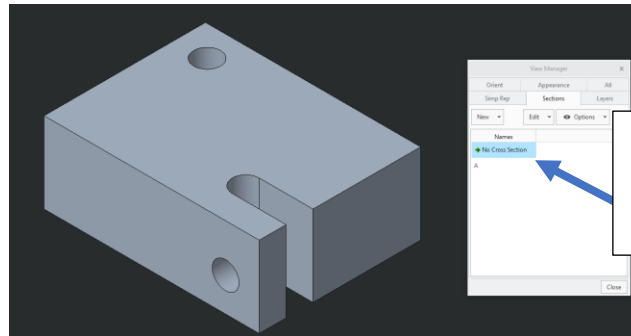
Isometric View



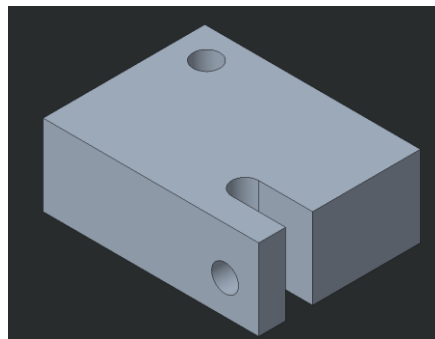
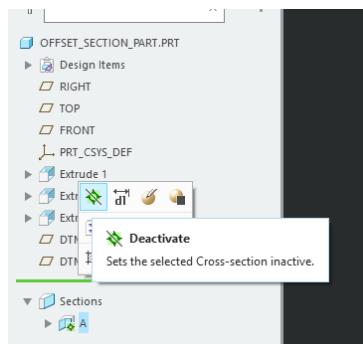
Bottom Isometric View

- f. Deactivating the Section View

Option 1: In the Manage Views Menu > Section Tab > Double Click on No Cross Section



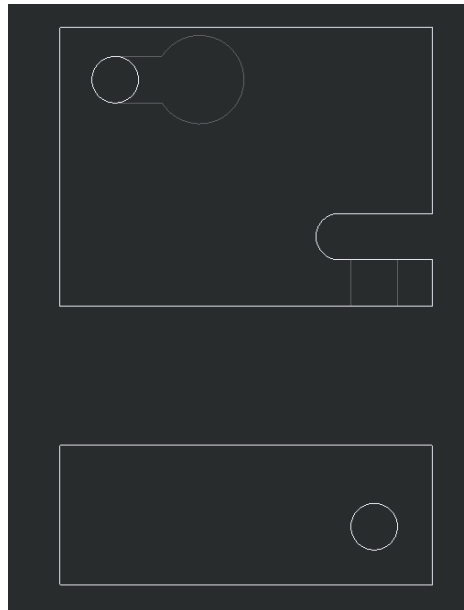
Option 2: From the Model Tree: Left Click On Section A > Select the Green Diamond to Activate or Deactivate the Section View



## Layout

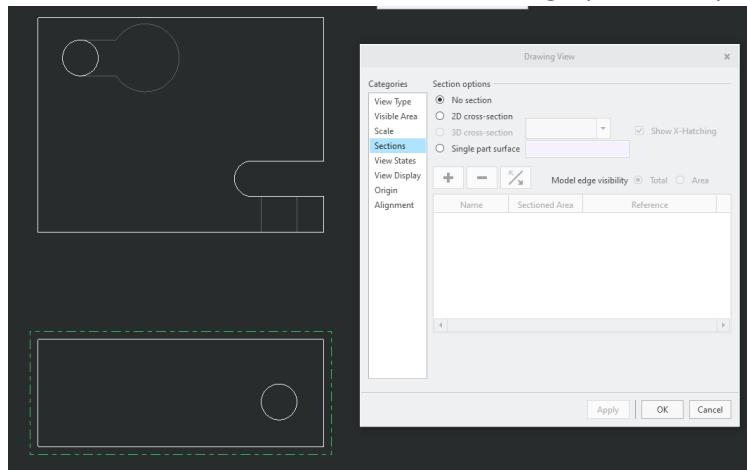
In this tutorial the section view showing interior details will be the Front View. Note on this part the front hole is cut away. Designers need to make a decision on whether not to create new view or draw in using sketching tools the feature in order to locate and size the feature. This tutorial will show how to draw/sketch in the hole that is cut away.

1. Place Top > Set as Hidden Lines > Tangent Edges > None
2. Project Front View (Section View) from Top View > Set as NO Hiddent >Tangent Edges = None

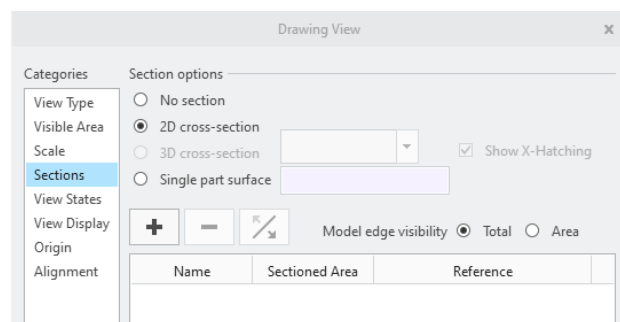


## 3. Set Section View

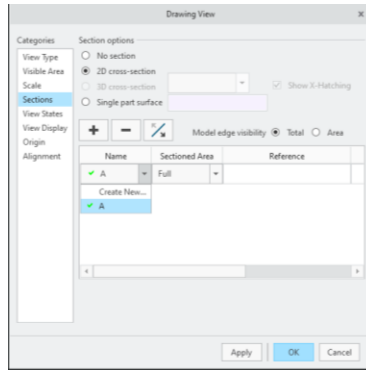
- a. Go into the Properties of the Front View > Select Section Category from the pop-up menu



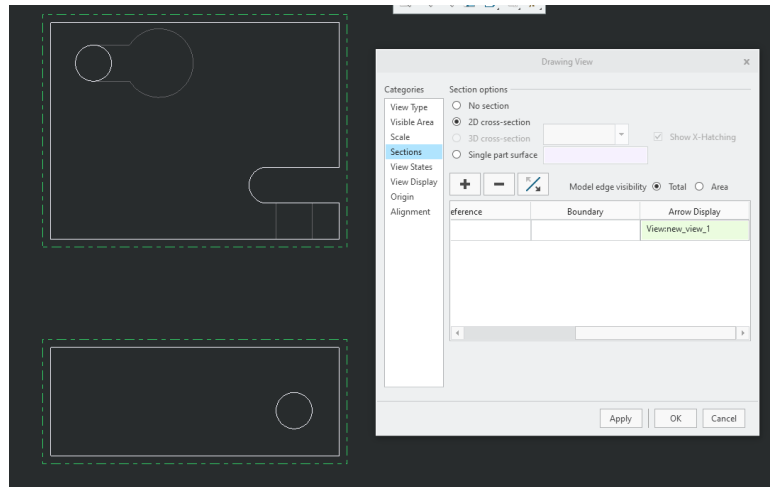
- b. Select 2D Cross-Section



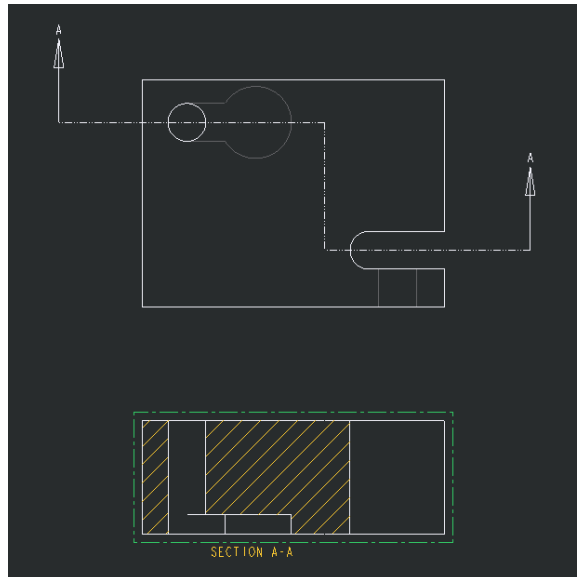
- c. Select + Sign > Select Section A with the Green Check next to the name.  
NOTE: Green Check represents the section view can be seen in the view; Red X would represent the view cannot be seen in the view



- d. Placing Cutting Plane Line: Using the slider, drag to the right to Arrow Display > Select the Empty Below Arrow Display > Select the Top View



- e. Select Apply > Select OK



f. Modify the Dimensioning Settings

i. Select File > Prepare > Drawing Properties > Select Detail Options: Change

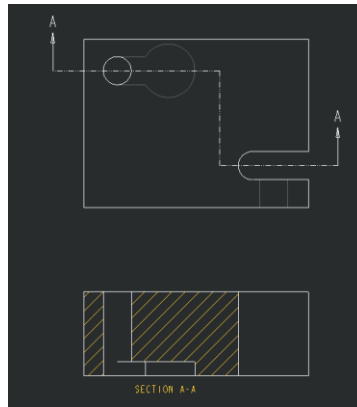
1. Change the Following

a. crossec\_arrow\_length = .09

b. cross\_arrow\_width = .03

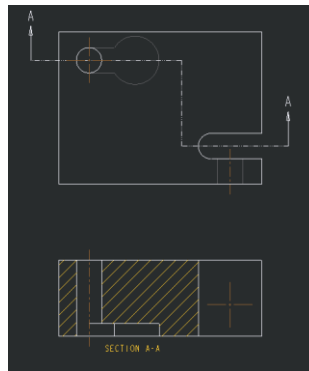
2. Apply Settings > Exit Menu

ii. Change Text Size of Section Markers (A's) to .1 Tall



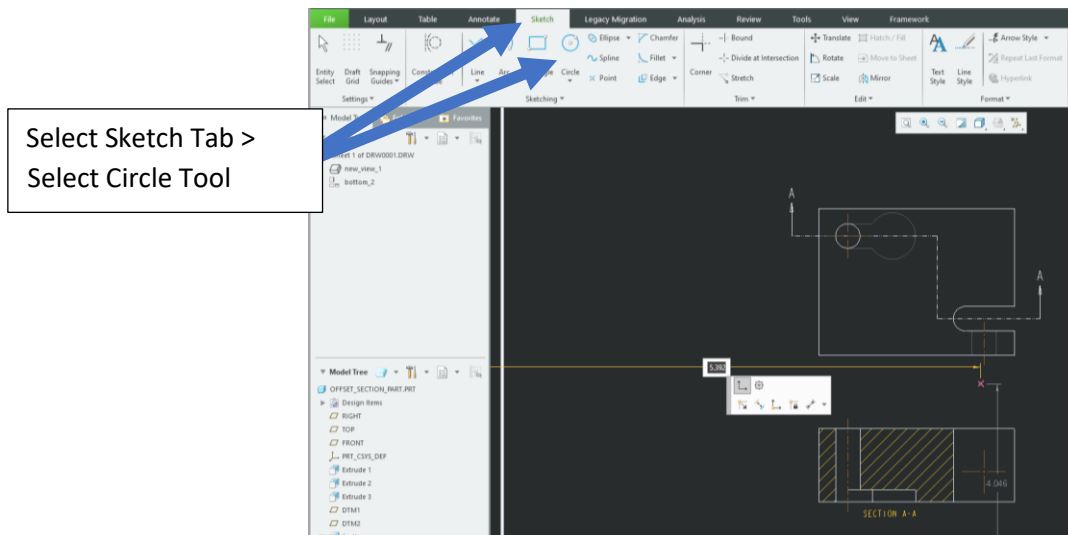
g. Place Centerlines

NOTE: Centerlines will appear for features that cannot be seen in the view. For this tutorial the Centerline for the hole that is cut away will show in the front view > Keep this centerline



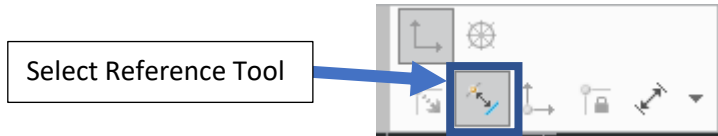
4. Adding Missing Feature: Process of adding/sketching missing features can be challenging. With the use of reference drawing in missing features can become less challenging

a. Select Sketch Tab > Select Circle Tool

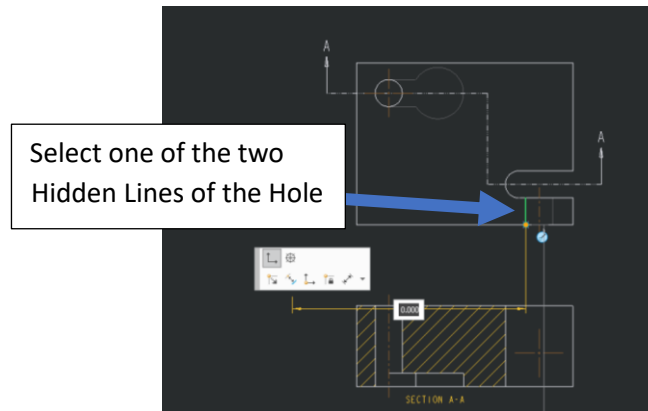


b. Selecting References

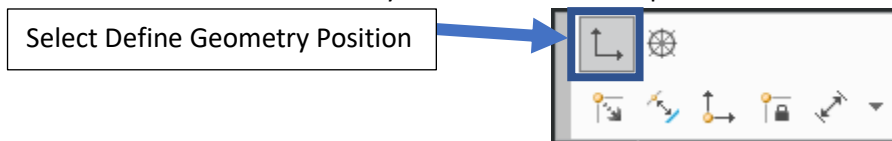
i. Select Change Reference Dimension for Feature from the Pop-Up Selection Menu



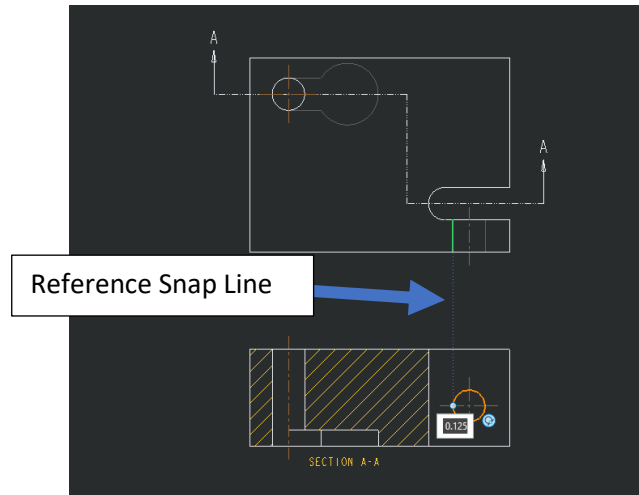
ii. Select one of the two hidden lines of the hole from the Top View: This will allow the designer to snap to the radius of the hole



iii. Select Define Geometry Position from the Pop-Menu



iv. Place Cursor on the Centerline for the circle > Left Click > Drag Mouse horizontally in the direction of the select hidden line from the Top View; a reference snap should appear > Left Click to Place Circle



**NOTE:** Creo allows the designer to type in a dimensional value for the feature. This works well when the views scale is Full. At any other scale the designer would need to calculate the scale change on the dimension to get the correct size of the feature. Utilizing reference and snap lines removes the need to calculate a features size based on the views scale

v. Relate the Circle/Hole to the Front View, so the feature will take the scale of the view and move with the view

