Overview

This drawings functionality allows you to create 2D drawings from your Fusion 360 designs and supports core drawing tools, which give the ability to generate PDF and DWG documentation of your Fusion 360 model. When you create a drawing, it is created as a derived document of a Fusion 360 model, and it shows up in the Data Panel as a unique derived item in the active project.

Launch Video

Learning Objectives

- About Fusion 360 Drawings
- Create a Drawing of a model
- Create Views
- Create & Edit Annotations
- Drawing Settings and Preferences
- Output the Drawing

Introduction to Drawing Views

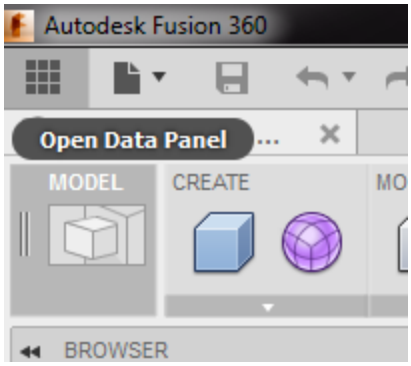
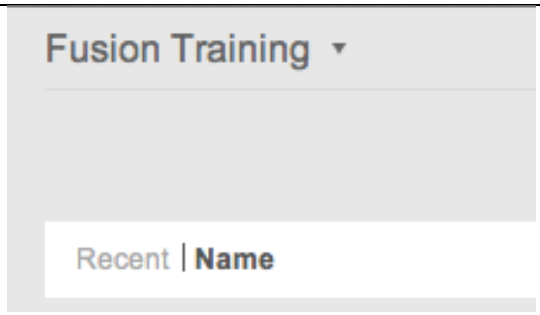
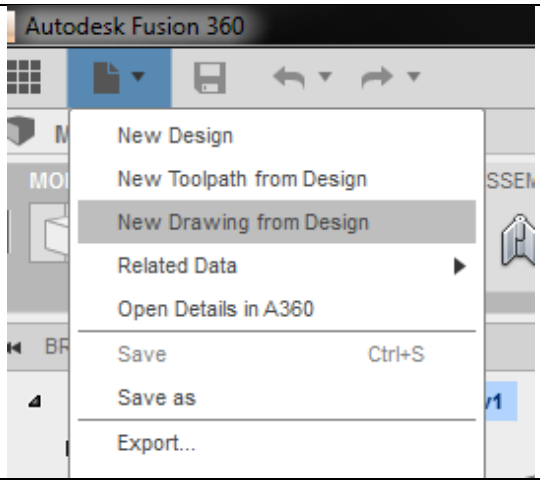
A drawing view is an object that contains a 2D projection of a 3D model.

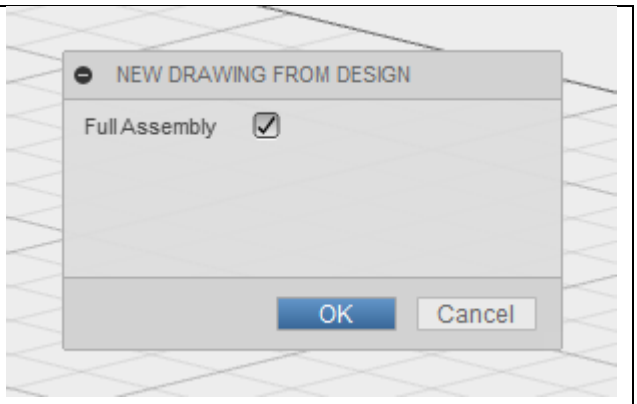
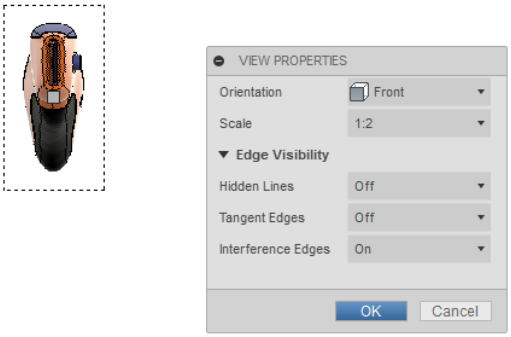
When you create a drawing from the Fusion 360 modeling environment, the system automatically launches a new tab of the Drawing workspace and generates a 2D projection of the components you select. The drawing view generated is referred to as a **base view**. Once you place the base view in the drawing, you can generate orthogonal and isometric projected views from it.

Projected views inherit the properties of the base view by default. When you change the properties of the base view, the projected view properties also change. However if you override a property of a projected view, that property stops following the changes you make to the base view.

Note: When creating a drawing, the system picks up settings such as the projection angle, annotation format, and the drawing border and title block from Preferences.

Create a Drawing: In this section you will open the design file for the Utility Knife and learn how to create a new drawing of the assembly.

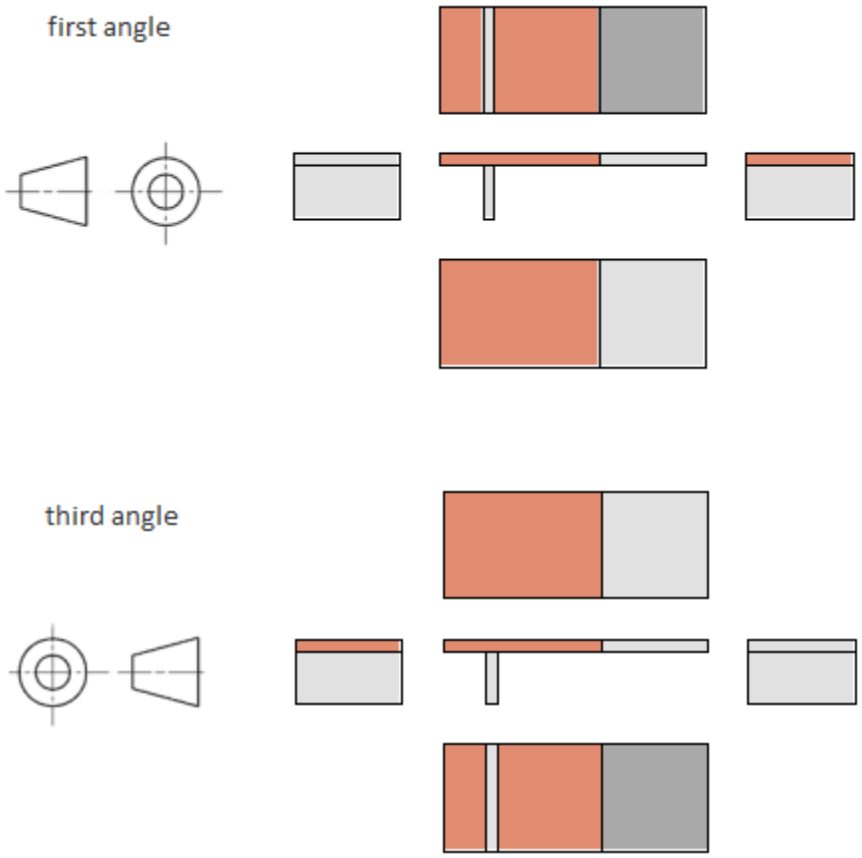
Step 1 – Open the Data Panel *In this module we will be using the* **08_Drawings Utility Knife** *file to complete the exercise. If you haven't set up a new project and uploaded the necessary designs, please follow the steps in the Introduction module.* 1. Click on the icon in the upper left to open the Data Panel. 2. The Data Panel will slide open	
Step 2 – Open the design 1. At the top left of the Data Panel, select the project where you uploaded **08_Drawings Utility Knife**. 2. Double-click on the design called **08_Drawings Utility Knife** to open the design in Fusion 360. 3. When the design has opened in your modeling window, click on the '**x**' icon to close the Data Panel.	
Step 3 – Initiate a new Drawing 1. Click on the File dropdown menu from the top menu bar. 2. Select **New Drawing from Design** from the file menu dropdown.	

Step 4 – Choose Assembly 1. Select **Full Assembly** from the dialog and click **OK** to initiate the drawing. 2. Notice that a new tab is automatically generated in Fusion 360 of the Drawing workspace. Note: If you un-check "Full Assembly" from the New Drawing dialog, you have additional controls to pick any set of components to create a drawing from if the full assembly is not what you are looking to document.	
Step 4 – Commit a Base View 1. Move your cursor around the screen and see the Base View preview is attached to the cursor. 2. Click on the top left quadrant of the sheet to place the view 3. Click **OK** to commit the view. Note: Notice that after the view is committed, the shaded preview matures into a 2D line drawing of the view.	

About Projected Views & View Options

Projected views maintain a parent-child relationship with the base view it was generated from. They inherit their properties from the parent base view. If necessary, you can override them after you create the projected view.

The projection angle defines the method employed to generate projected views.



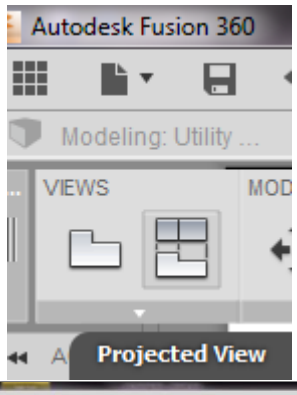
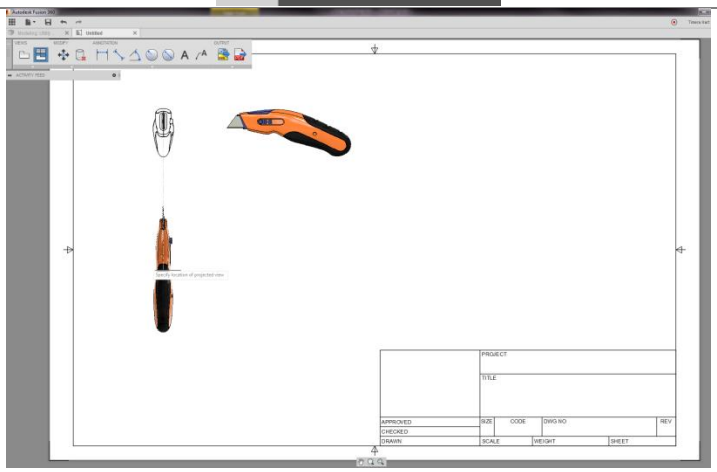
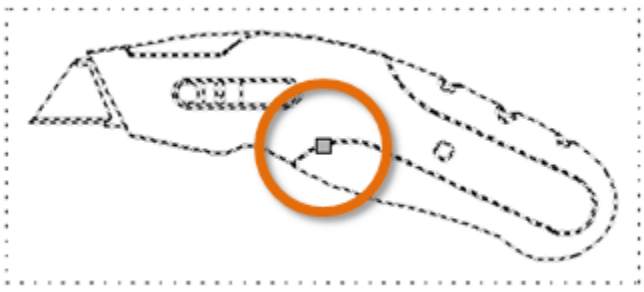
First Angle Projection

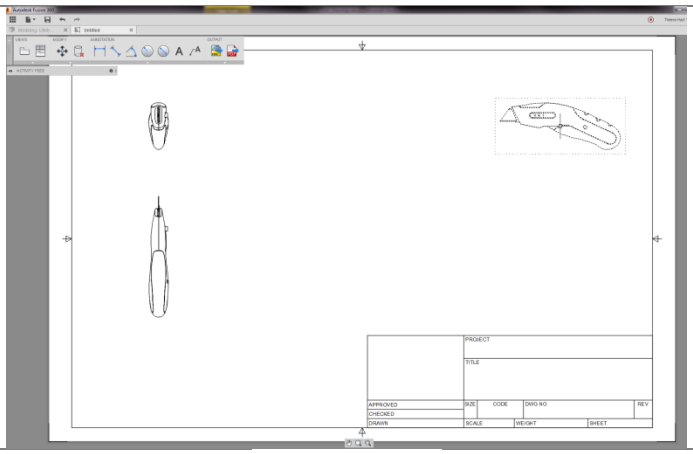
When you use first angle projection, projected views placed to the right of a base view depict the appearance when viewing it from the left. Projected views placed below the base view depict the appearance from above. The ISO drafting standard specifies that drawings use first angle projection.

Third Angle Projection

When you use third angle projection, projected views placed to the right of a base view depict the appearance when viewing it from the right. Projected views placed below the base view depict the appearance from below. The ANSI drafting standard specifies that drawings use third angle projection.

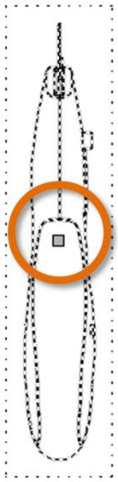
Finish the layout: Now that you have created a Base View of the model assembly, you will learn to create projected views and edit their properties to create a complete drawing layout.

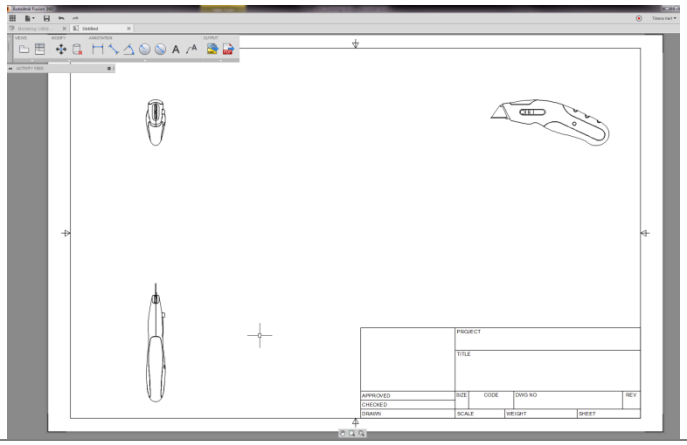
Step 1 – Initiate the **Projected View** command 1. Click **Views > Projected View**. 2. Select the existing Base View as the parent view that the projected views will be created from and associated to.	
Step 2 – Place the views 1. Drag the cursor to the right of the base view, and notice that the projected view is previewed based on this alignment. 2. Select to the right of the existing base view to place a projected view. 3. Select to the bottom of the existing base view to place a second projected view. 4. Press **Enter** to finish the task Note: Projected views inherit all its properties from the parent. When the properties of the parent view change, the corresponding properties on the projected view also change.	
Step 3 – Move the right view 1. Click anywhere inside the selection boundary of the right projected view to activate it. 2. Click the **center grip** to drag the view. 3. Move the view to a new location at the far right of the layout.	



Step 4 – Move the bottom view

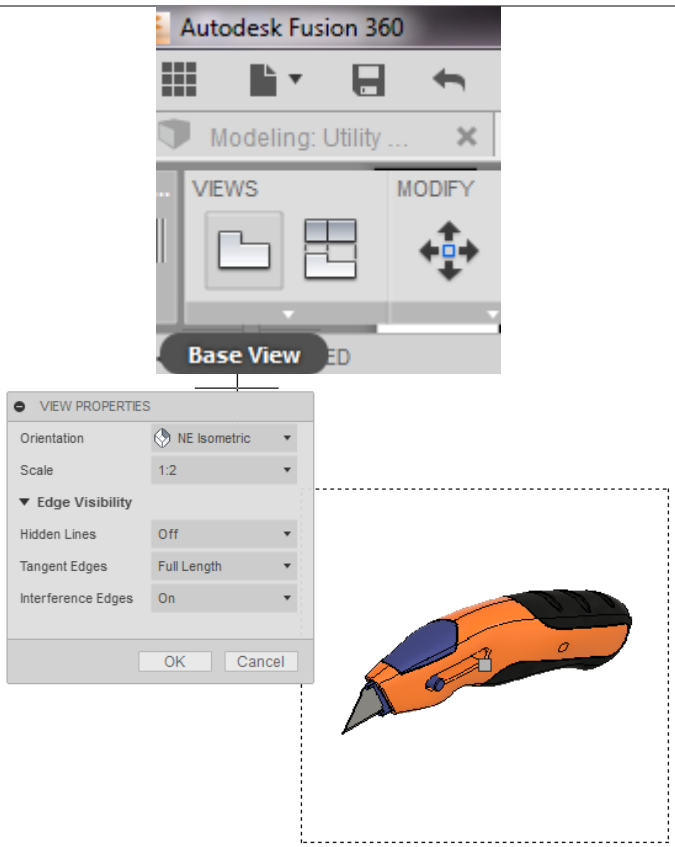
1. Click anywhere inside the selection boundary of the bottom projected view to activate it.
2. Click the **center grip** to drag the view.
3. Move the view to a new location at the left bottom of the layout.





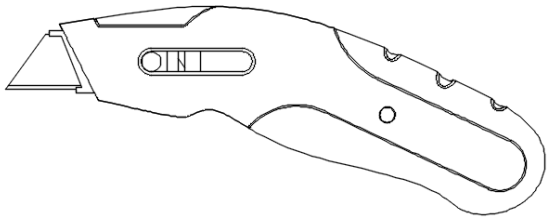
Step 5 – Create an isometric base view

1. Click **Views > Base View**.
2. Click to place the view in the center of the sheet layout.
3. Set the Orientation to **NE Isometric.**
4. Click **OK** to commit the view

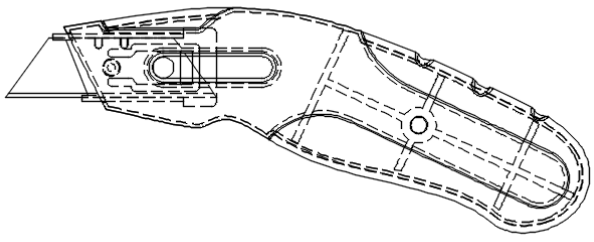


View Properties

Hidden Lines – Select On or Off from the drop-down list to display hidden lines within the selected base view. The Hidden-line representation suppresses or exposes lines, edges and other objects that are located behind other three-dimensional objects. This view property can be particularly helpful when trying to visually communicate the inner workings or dimensions of a complex assembly or part.

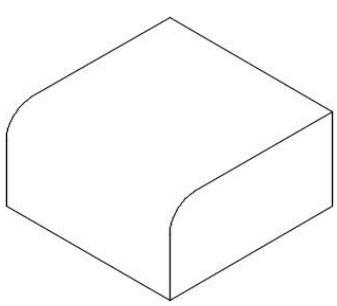


Hidden Lines OFF

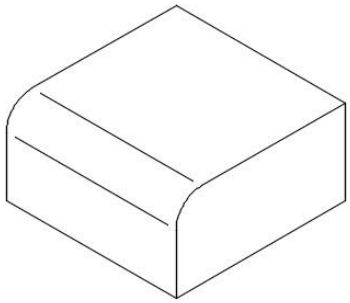


Hidden Lines ON

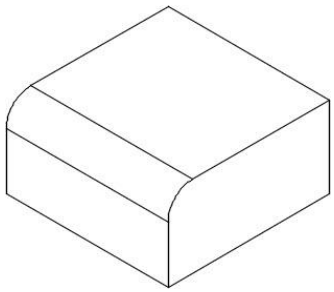
Tangent Edges – Select Full length, Shortened or Off from the drop-down list to display Tangent edges within the selected base view. Tangent edges mark the transition between a flat surface and a rounded edge, most commonly seen as filleted edges. Tangent edges can be set to Full Length, Shortened, or Off.



Tangent Edges Off

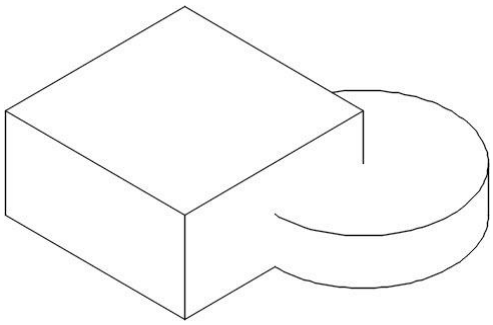


Tangent Edges Shortened

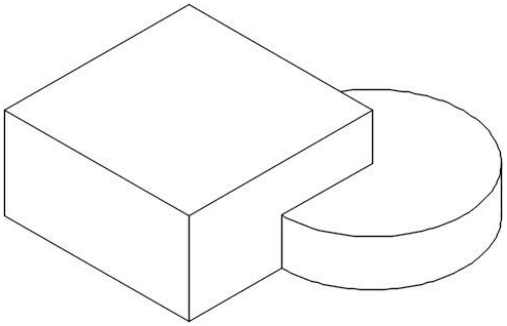


Tangent Edges Full Length

Interference Edges – Select On or Off from the drop-down list to display of Interference edges within the selected base view. An interference edge occurs when two faces of two components intersect. When Interference Edges are turned on, an edge is displayed that shows where the two components meet. When selected, associated drawing views are to display both hidden and visible edges that were previously excluded due to an interference condition (press, or interference fit conditions, threaded fasteners in tapped holes where the hole feature is modeled with the minor diameter).



Interference Edges Off



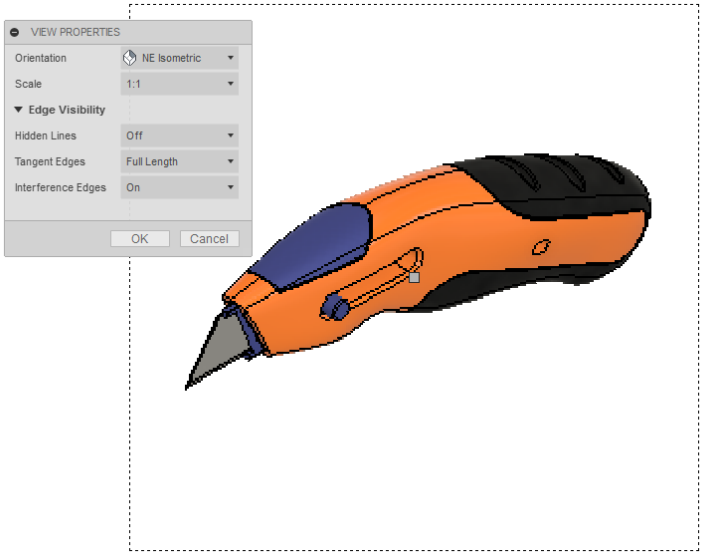
Interference Edges ON

Edit the layout views: Now that you have created a base view and several projected views of the model assembly, you will practice using the View Properties settings to further customize the view layouts.

Step 1 – Edit the isometric base view

1. Double-click anywhere inside the selection boundary of the isometric view to activate it.
2. Select the **Scale** ratio in the View Properties dialog box to change the scale.
3. Change the Scale to: **1:1**
4. Click **OK** to accept the drawing view changes.

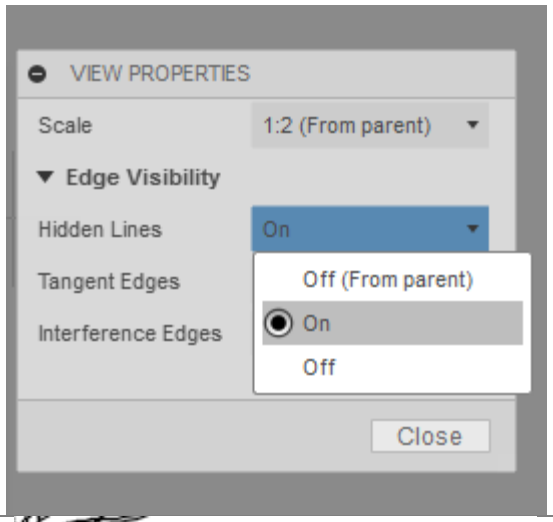
If the projected view properties are changed, they no longer inherit the settings of the base view.



Step 2 – Edit the right projected view

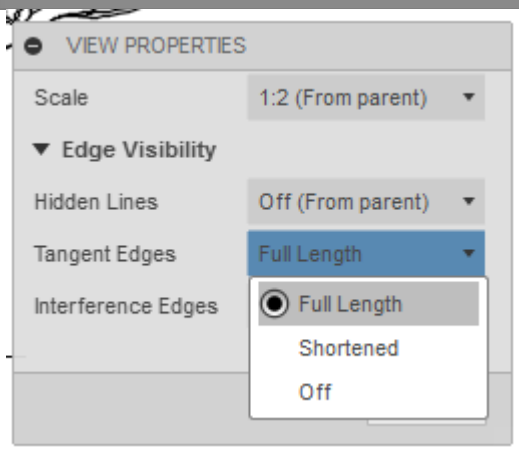
1. Double-click anywhere inside the selection boundary of the right projected view to activate it.
2. In the View Properties dialog box, click Hidden Lines and select **On**.
3. Click **Close** to accept the drawing view changes.

When the projected view properties are changed, they no longer inherit the settings of the base view.

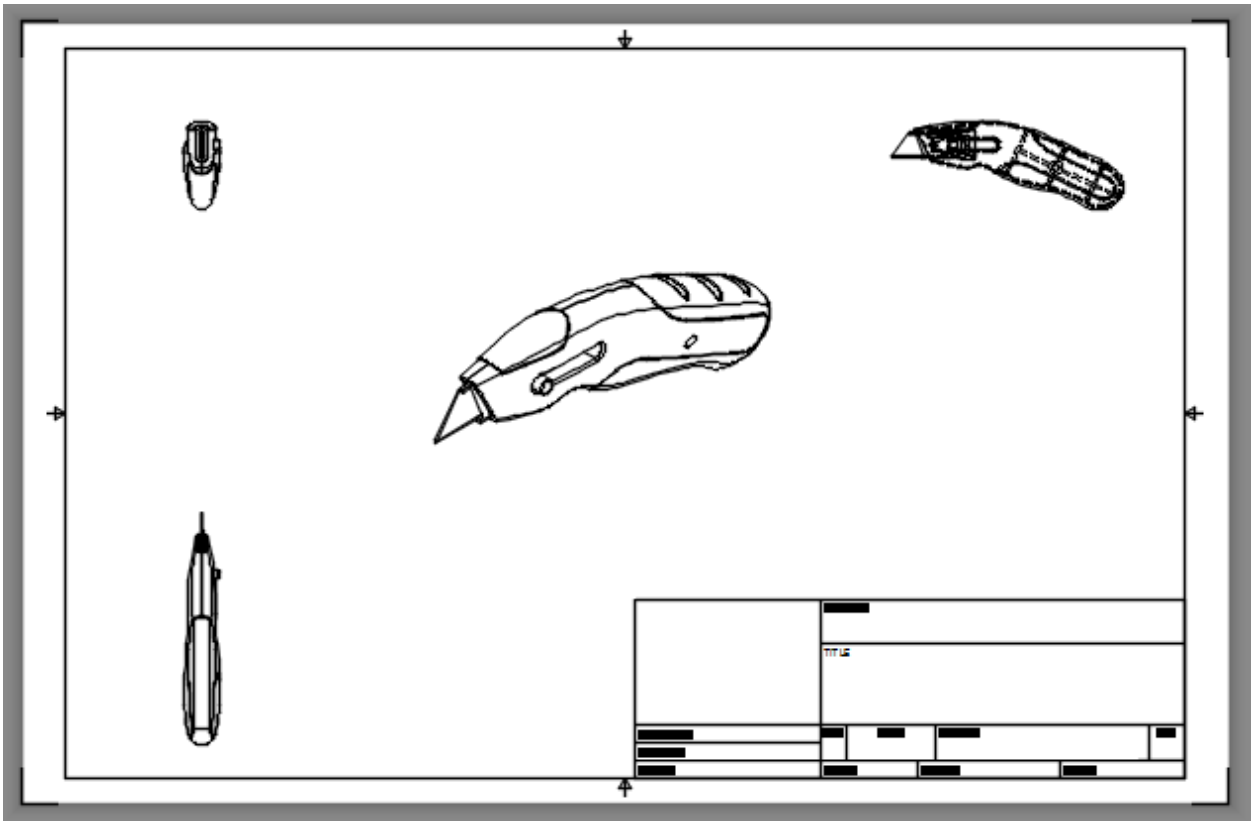


Step 3 – Edit the bottom projected view

1. Double-click anywhere inside the selection boundary of the bottom projected view to activate it.
2. In the View Properties dialog box, change Tangent Edges to **Full Length**.
3. Click **Close** to accept the drawing view changes.



By now, your drawing should look something like this:



Text and Annotations

At any point in the drawing creation, you can add a variety of text and annotations to the drawing and the views to document additional detailed information. The tools for adding annotation and text to a drawing include:

Dimension: Linear

Creates a horizontal or vertical dimension.

Dimension: Aligned

Creates a linear dimension that is aligned with the origin points of the extension lines.

Dimension: Angular

Measures the angle between selected geometric objects or 3 points.

Dimension: Diameter

Creates a diameter dimension for a circle or an arc.

Dimension: Radius

Creates a radius dimension for a circle or an arc.

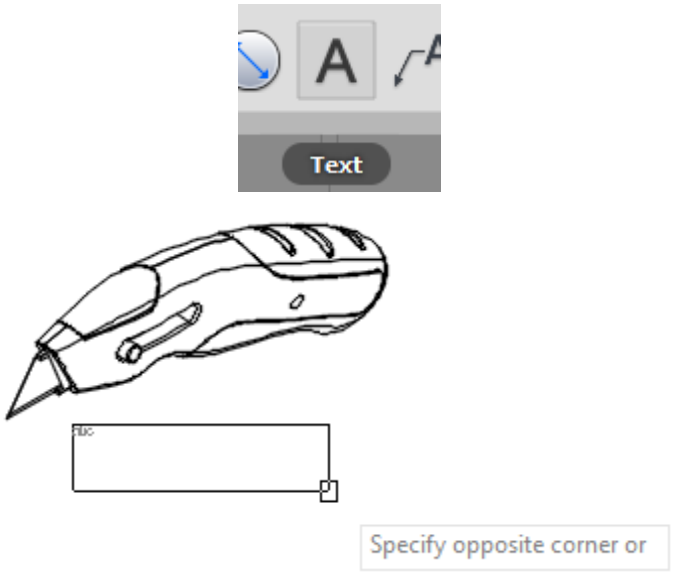
Text

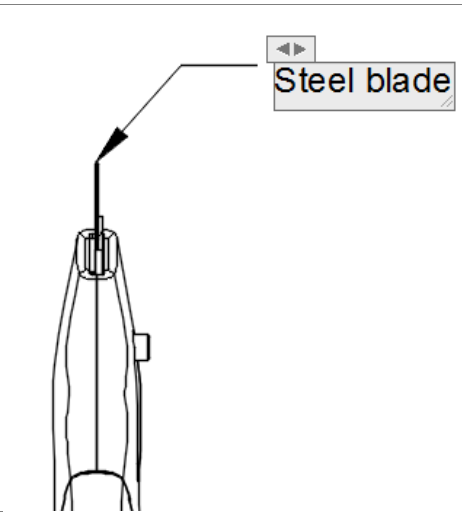
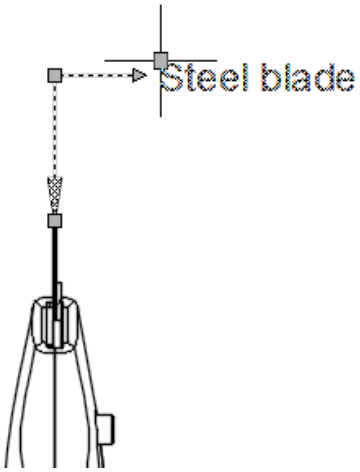
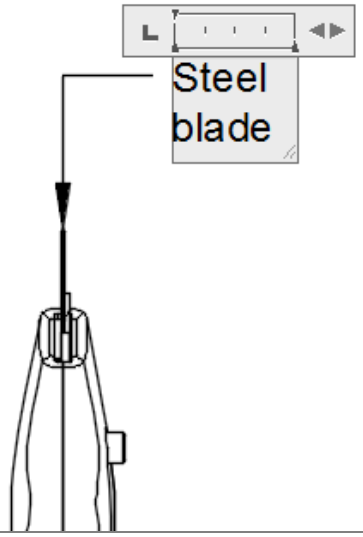
Creates single and multi-line text.

Leader

Creates a leader note.

Text and Leader Notes: In this section you will learn how to create, reposition and edit text and leader notes in the drawing.

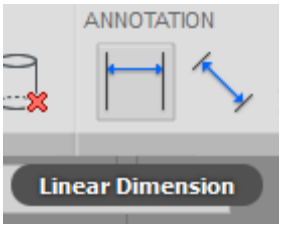
Step 1 – Create Text 1. Click **Annotation > Text**. 2. Select two corners below the Isometric Base View to create a text box. 3. Type the following text into the text box: **NE Isometric View** 4. Select anywhere outside of the text box to commit the action. 5. Repeat process to add text below the to the initial base view with the text: **Front View** 6. Repeat process to add text below the right projected view with the text: **Right View** 7. Repeat process to add text below the bottom projected view with the text: **Bottom View**	

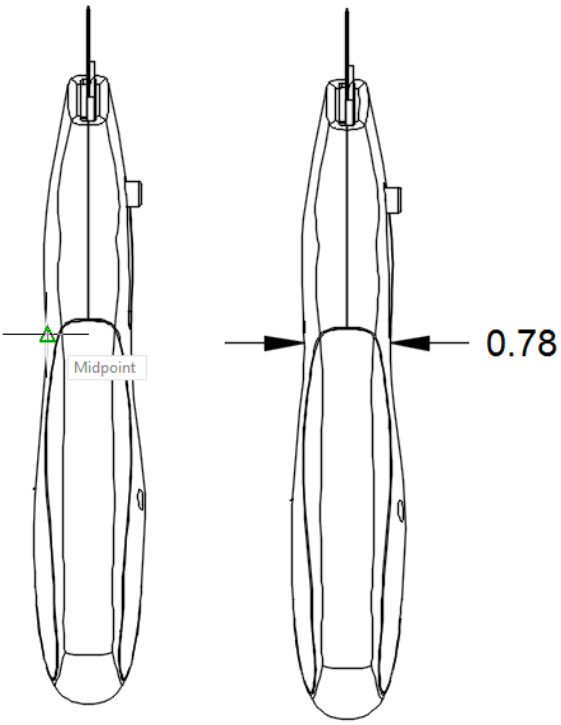
Step 2 – Create Leader Notes 1. Click **Annotation > Leader**. 2. Click near the blade of the bottom view to place the start of the leader. 3. Click outside the view to place the end of the leader. **4.** Type the following text: **Steel blade** 5. Select anywhere outside of the text box to commit the action.	
Step 3 – Reposition Leader Notes 1. Click on the leader note to activate. 2. Drag the leader note using the text grip to the left so that the leader is at a 90 degree elbow. 3. Press the **Esc** key to commit the changes and exit the command.	
Step 4 – Edit Text 1. Double-click on the leader note to activate the text editor. 2. Drag the < > to the right to format the text into 2 lines. 3. Select anywhere outside of the text box to commit the action.	

Dimensions: In this section you will learn how to create, reposition and edit a variety of dimension types in the drawing.

Step 1 – Create linear dimensions

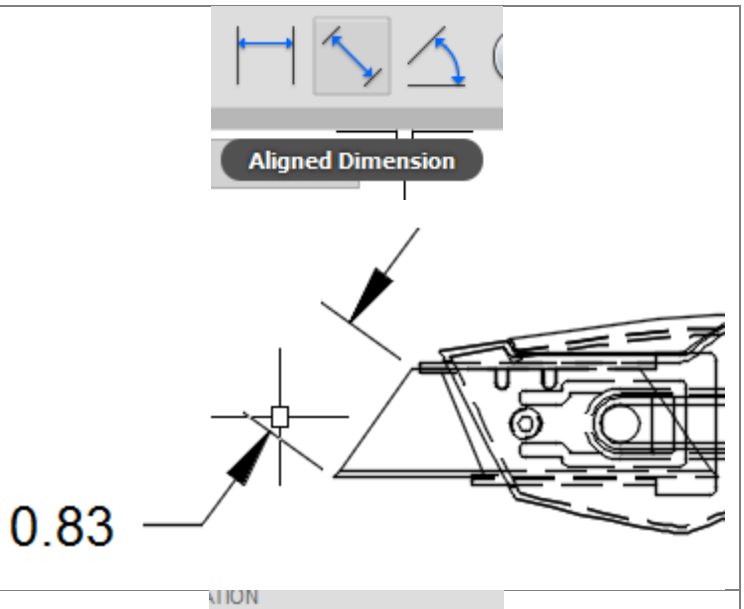
1. Click **Annotation > Linear Dimension**.
2. Click the two midpoints of the bottom view and a preview is displayed on your curser.
3. Click again to place the dimension.





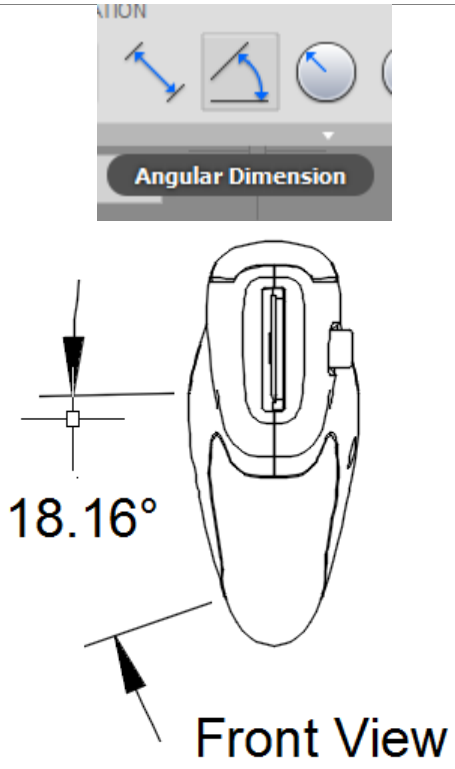
Step 2 – Create aligned dimensions

1. Click **Annotation > Aligned Dimension**.
2. Click the top edge and bottom point of the cutting blade of the right view.
3. Move the cursor out to the left and see a preview of the dimension.
4. Click again to place the dimension and finish the command.



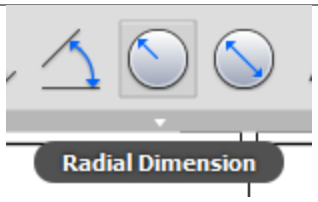
Step 3 – Create angular dimensions

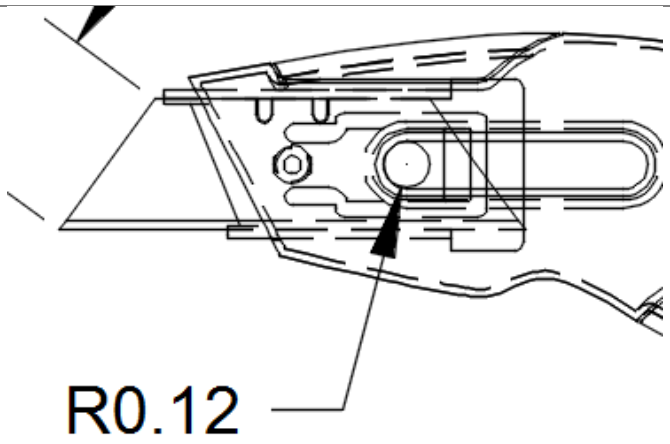
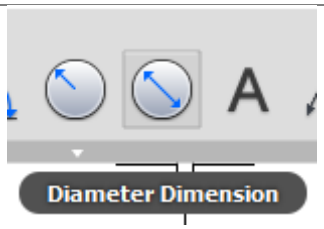
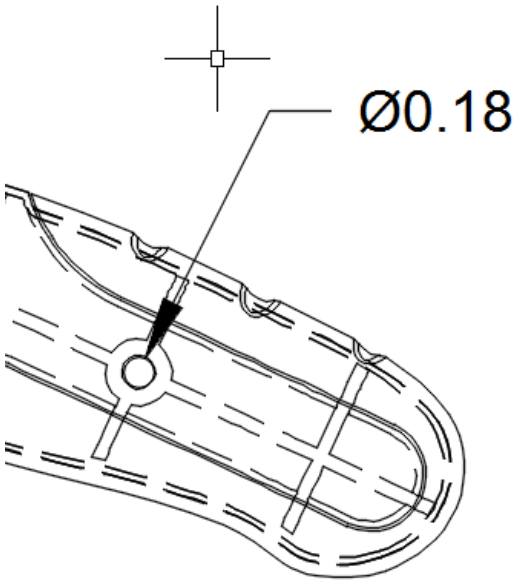
1. Click **Annotation > Angular Dimension**.
2. Select the left curved edge of the front view.
3. Move the cursor out to the left and see a preview of the dimension.
4. Click again to place the dimension and finish the command.

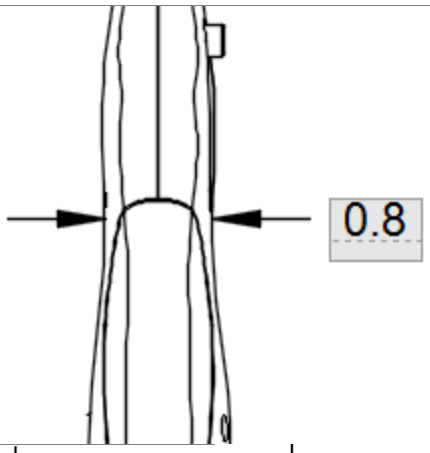
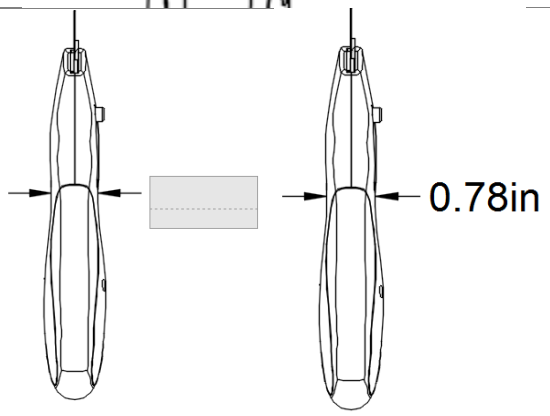


Step 4 – Create Radial Dimensions

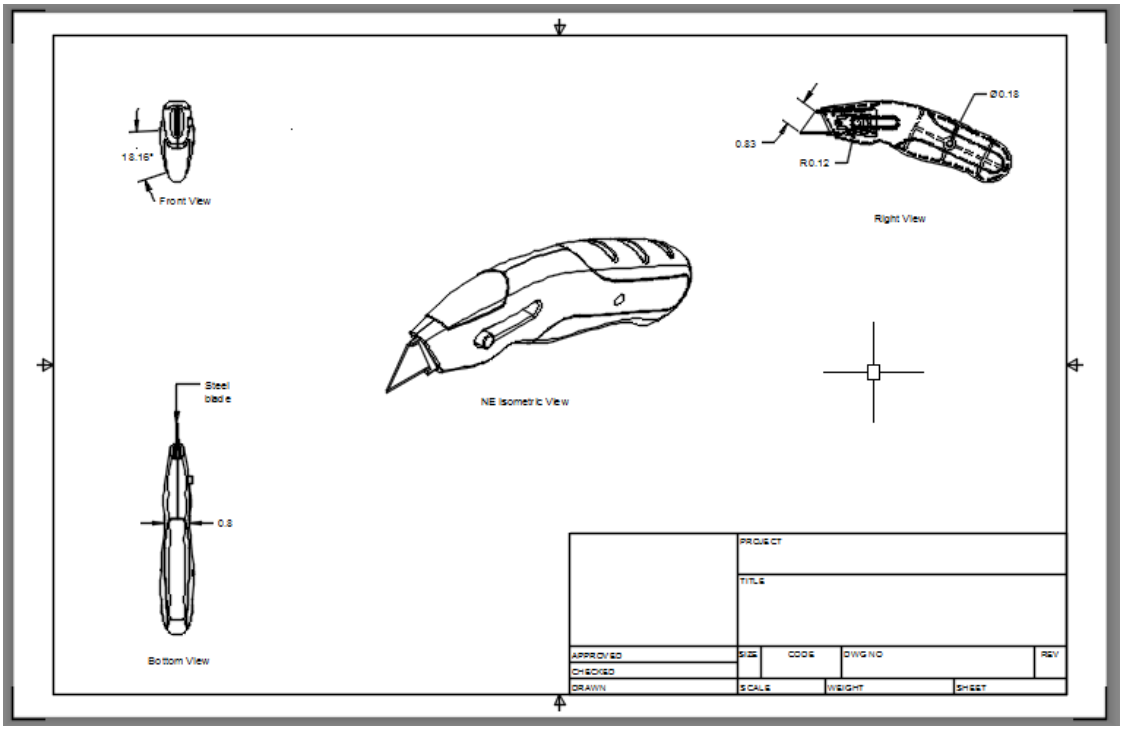
1. Click Annotation > Radial Dimension.
2. Select the blade slider of the right view as the circle to dimension.
3. Move the cursor down and to the left and see a preview of the dimension.



4. Click again to place the dimension and finish the command.	
Step 5 – Create diameter dimensions 1. Click **Annotation > Diameter Dimension**. 2. Select the circular boss in the webbing of the right view as the circle to dimension. 3. Move the cursor up and to the right and see a preview of the dimension. 4. Click again to place the dimension and finish the command.	 

Step 7 – Edit dimensions 1. Double-click the linear dimension on the bottom view to activate it. 4. Replace the current value with **0.80**. 5. Select anywhere outside of the text box to commit the action and see the override	
Step 8 – Return dimension to measured value 1. Double-click the linear dimension you just edited. 2. Highlight the text and delete the “0.80” text. 3. Select anywhere outside of the text box to commit the action and see the initial associated value (0.78 in).	

By now, your drawing should look something like this:



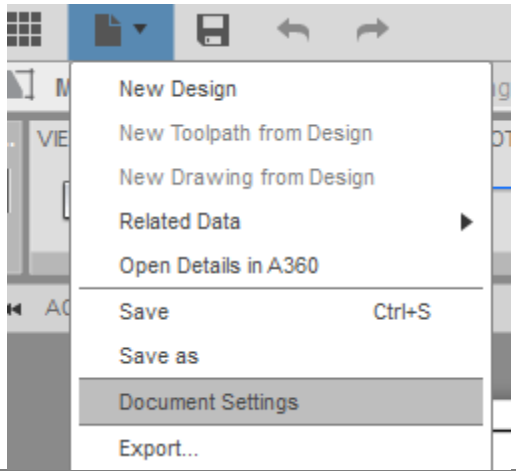
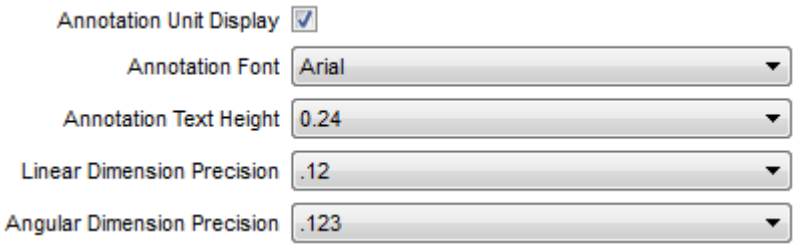
Drawing Settings & Preferences

You can change settings such as the default projection angle, sheet size, title block, annotation properties and dimension precision.

The default settings are applied whenever you create a new drawing. If required, you can override some of the settings once the drawing is created.

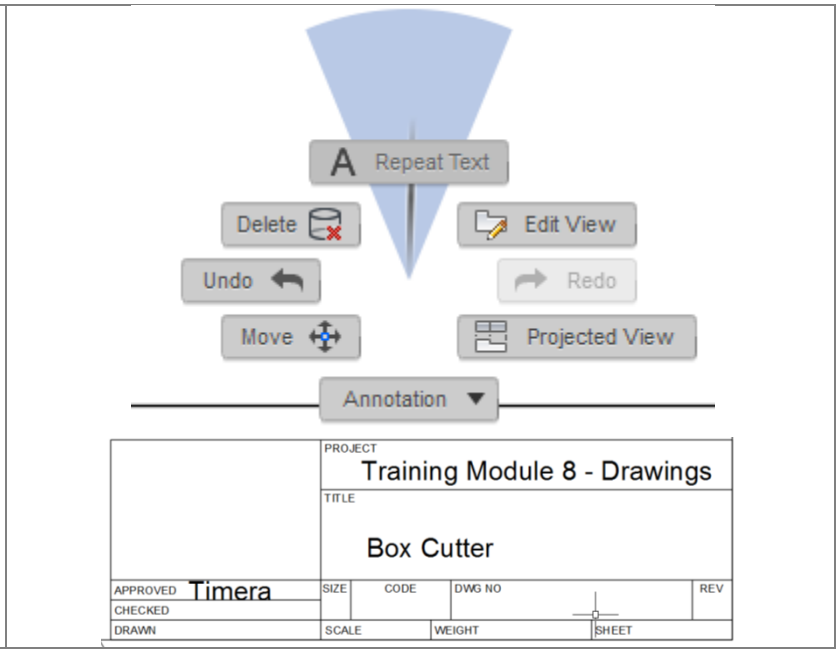
Change Drawing Settings: Making adjustments to the Drawing Settings are local and drawing-specific. These preferences will be used as the default override for the active drawing only.

Note: If you would like any of these settings to permeate as a default in future drawings, you can use the same workflow as below in the Preferences dialog. The Preferences dialog can be found in the application bar under Your Name:

Step 1 – Open Document Settings 1. On the application bar, click the file dropdown menu. 2. Select **Document Settings** to display the Document Settings dialog box.	
Step 2 – Edit annotation preferences 1. Check the box for **Annotation Unit Display**. 2. Change the Annotation Text Height to **0.24.** 3. Change the Angular Dimension Precision to **.123.** 4. Click **OK** to accept the changes	

Step 3 – Edit title block

1. Click **Annotation > Text**.
2. Add a description to the Project section of the title block.
3. Right-click and select **Repeat Text**.
4. Add a description to the Title section of the title block.
5. Right-click and select **Repeat Text**.
6. Add a description to the Approved section of the title block.



Associatively Update the Drawing

Any change you make to the model's geometry, the drawing views are immediately updated to reflect the changes.

The 3D models are likely to change even after drawing views are created and annotated.

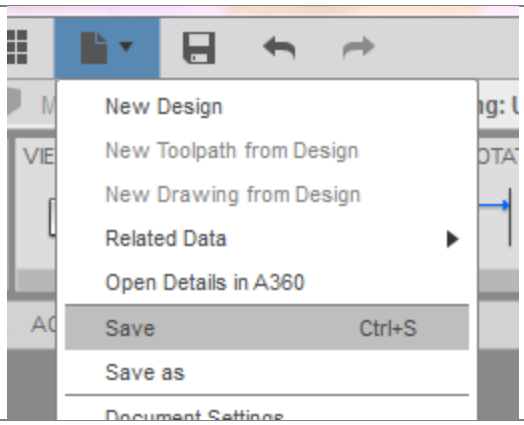
If any annotations associated with the drawing view geometry get disassociated because of the model change, badges are displayed on the screen. To delete or manually re-associate these badged annotations to the view geometry, you can snap to specify the points or select the objects you want the dimension to get re-associated.

Output the Drawing

When the drawing is completed, you have the ability to output the layout to either a PDF or DWG. Both of these options creates a copy of the drawing and prompts you to save it locally onto your machine.

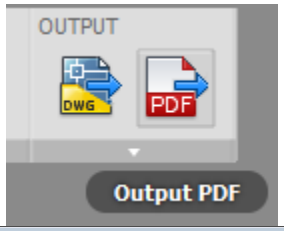
Step 1 – Save the drawing

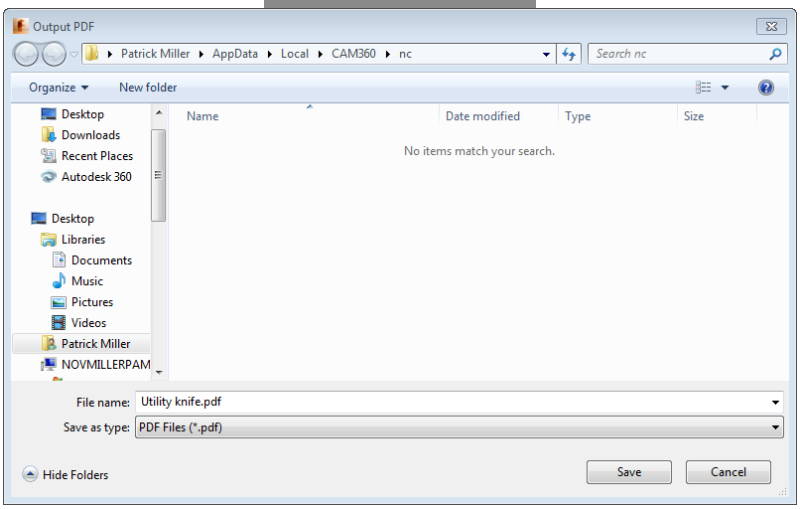
1. Click **File > Save**.
2. Enter **Utility Knife** in the Name field.
3. Click **OK**.



Step 1 – Output a PDF

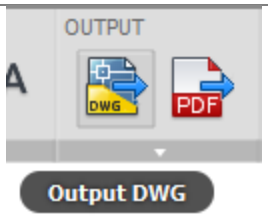
1. Click **Output > Output PDF**.
2. Navigate to a desired local location on your computer.
3. Click **Save**.

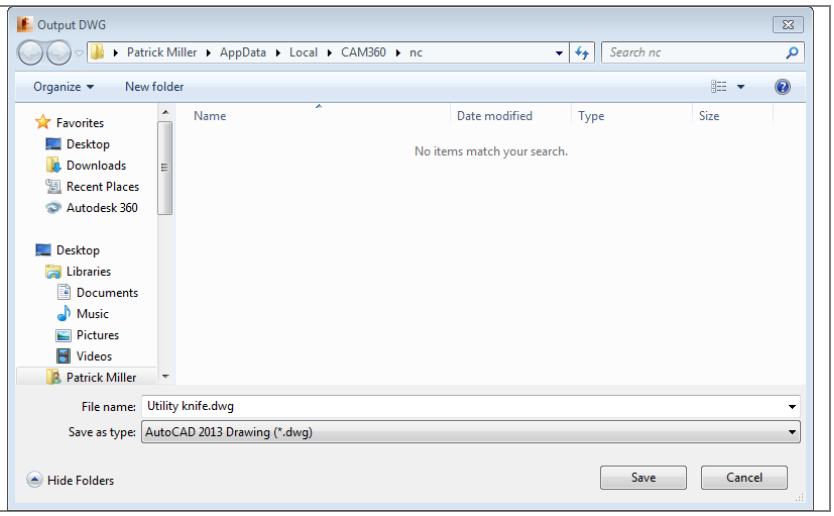




Step 2 – Output a DWG

1. Click **Output > Output DWG**.
2. Navigate to a desired local location on your computer.
3. Click **Save**.




Output DWG
Patrick Miller ▸ AppData ▸ Local ▸ CAM360 ▸ nc
Search nc
Organize
New folder
Favorites
Desktop
Downloads
Recent Places
Autodesk 360
Desktop
Libraries
Documents
Music
Pictures
Videos
Patrick Miller
Name
Date modified
Type
Size
No items match your search.
File name: Utility knife.dwg
Save as type: AutoCAD 2013 Drawing (*.dwg)
Hide Folders
Save
Cancel