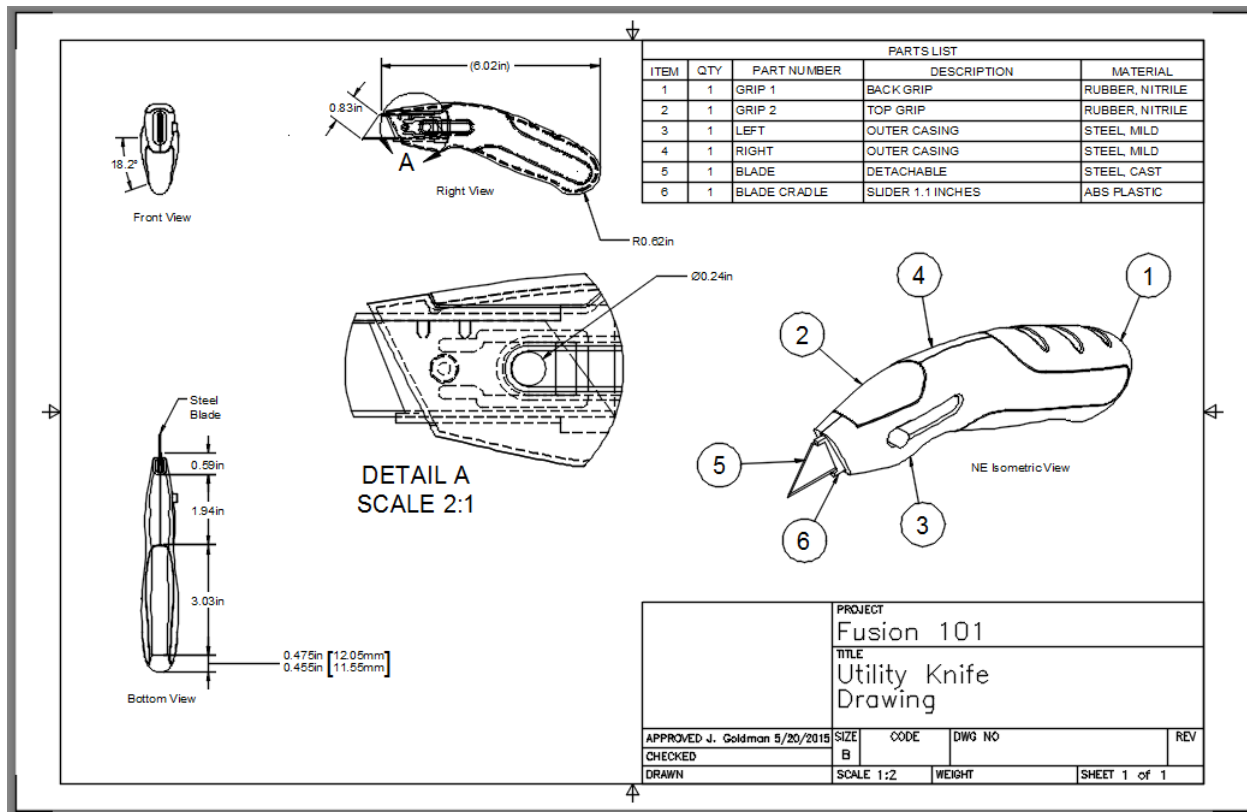


8.1: Drawings



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

Lesson 1: Introduction to Drawing Views

Learning Objectives

1. What are Drawing Views?
2. Initiate a New Drawing
3. Place a Base View

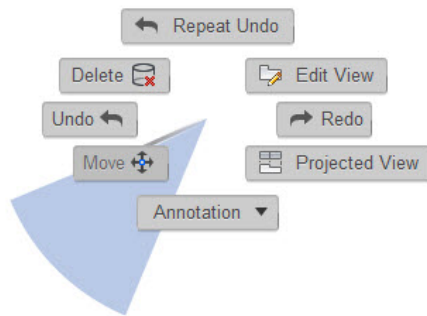
A drawing view is an object that contains a 2D projection of a 3D model.

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. A **projected view** takes the characteristics of a base view and projects it from a different angle.

About the Marking Menu

The **marking menu** is a radial display of the most frequently used commands. It also includes an overflow menu that provides quick access to all commands found in the toolbar.

Using the marking menu can be the fastest way to input a command in any workspace. You can access the marking menu by right-clicking anywhere within the drawing canvas.



As you move the cursor from the center of the marking menu towards a command, its wedge highlights. Clicking anywhere in the wedge launches the command.

Datasets Required

In Samples section of your Data Panel, browse to:

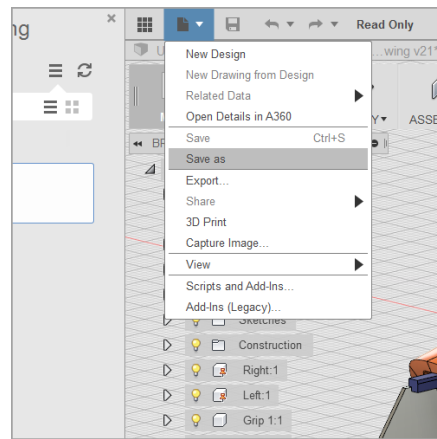
Fusion 101 Training > 08 – Drawing > **08_Drawings Utility Knife**

Open the design and follow the step-by-step guide below to get started with the lesson.

Create a Drawing

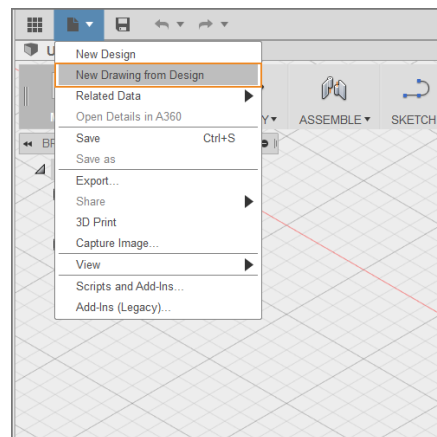
Step 1: – Save the model

1. If the model is “Read Only,” go to the File Menu and click **Save As**.
2. Give the model a location and a new name, then click **Save**.



Step 2: – Initiate a New Drawing – With the Utility Knife design open, do the following:

1. Click on the File dropdown menu from the top menu bar.
2. Select **New Drawing from Design** from the file menu dropdown.

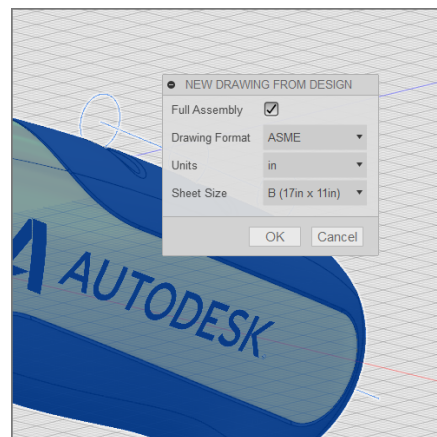


Step 3: – Choose Assembly

1. Select “Full Assembly” from the dialog and click **OK** to initiate the drawing.
2. Notice that a new file tab is automatically generated.

Note: If you un-check “Full Assembly” from the New Drawing dialog, you can select individual or multiple components to create a drawing of part of the assembly.

Tip: When selecting multiple components, it helps to use **Ctrl+click**.



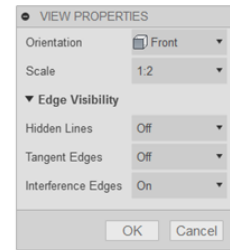
You can set the drawing format, units, and sheet size before you create a drawing.

However, Drawing Format and Units cannot be changed once you create a new drawing.

Step 4: – Commit a Base View

1. Move your cursor around the screen. Notice that 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.



Lesson 2: Projected Views and Detail Views

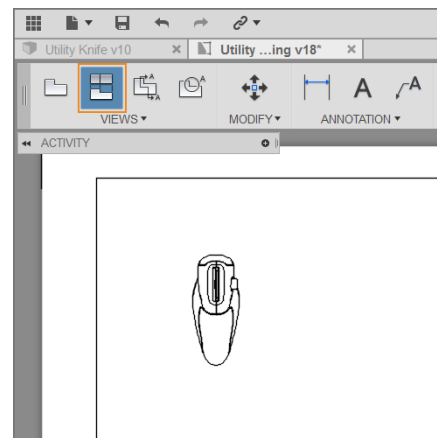
Learning Objectives

1. Create Projected Views
2. Edit View Properties
3. Create Detail Views

Add to the Layout

Step 1: – Initiate Projected View – Now that you we've created a base view of the model assembly, let's create projected views and edit their properties to create a complete drawing layout.

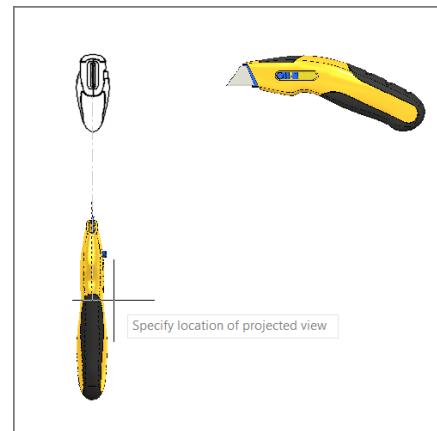
1. From the View Toolbar, select **Projected View**.
2. Click the base view to select it as the parent view from which the projected views will be created from and associated.



Step 2: – Place the views

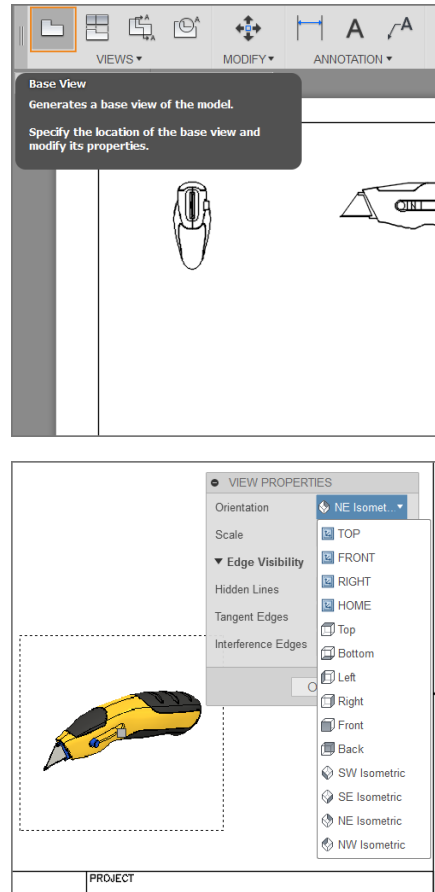
1. Drag the cursor to the right of the base view, and notice that the projected view is previewed based on this alignment.
2. Select to the right of the existing base view to place a projected view.
3. Select to the bottom of the existing base view to place a second projected view.
4. Press **Enter** to finish the task.

Note: A projected view inherits all its properties from the parent. When the properties of the parent view change, the corresponding properties on the projected view also change. You can change the properties of a projected view by double-clicking it.



Step 3: – Create an Isometric Base View

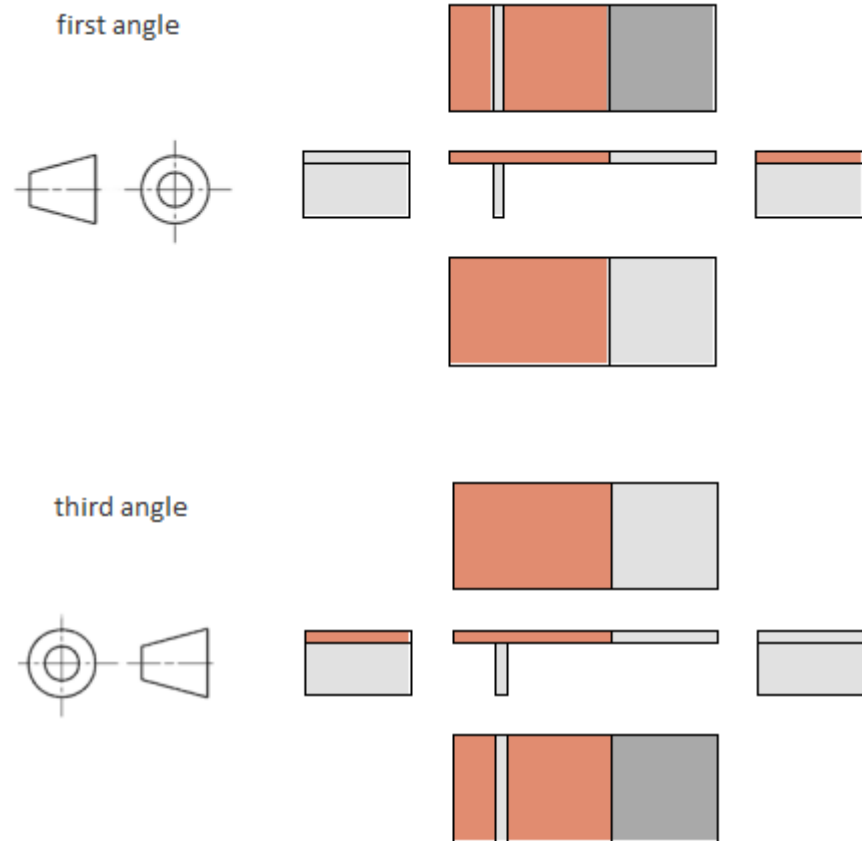
1. From the View Toolbar, select **Base View**.
2. Click to place the view in the lower right of the sheet layout, above the title block.
3. Set the Orientation to **NE Isometric**.
4. Press OK to commit the view.



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.



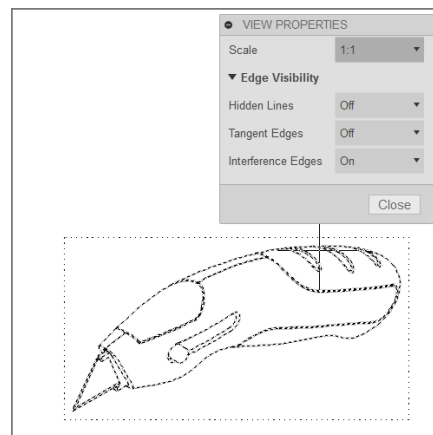
First Angle Projection – When you use first angle projection, projected views placed to the right of a base view depict the appearance when viewing it from the left. Projected views placed below the base view depict the appearance from above. The ISO drafting standard specifies that drawings use first angle projection. **By default, when the drawing format is set to ISO, the Drawings workspace will 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 ASME drafting standard specifies that drawings use third angle projection. **By default, when the drawing format is set to ASME, the Drawings workspace will use third angle projection.**

Edit the Layout Views

Step 1: – Edit the isometric base view - Now that you have created a base view and several projected views of the model, let's use the View Properties settings to further customize the view layouts.

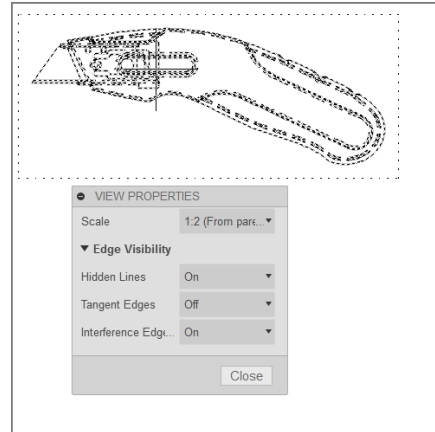
1. Double-click anywhere inside the selection boundary of the isometric view to activate it.
2. Select the Scale ratio in the View Properties dialog box to scale the base view.
3. Change the Scale to: **1:1**.
4. Click **Close** to accept the drawing view changes.



Step 2: – Edit the right projected view

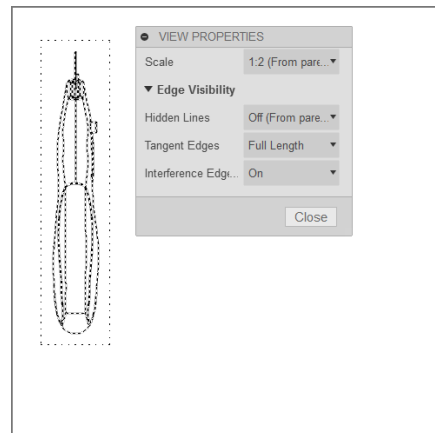
1. Double-click anywhere inside the selection boundary of the right projected view to activate it.
2. In the View Properties dialog box, edit the properties of the component.
3. For **Hidden Lines**, select **On** from the drop-down list..
4. Click **Close** to accept the drawing view changes.

Note: Once the projected view properties are changed, they no longer inherit the settings of the base view. If you change the properties back to “From parent,” they will once again inherit properties from the parent view.



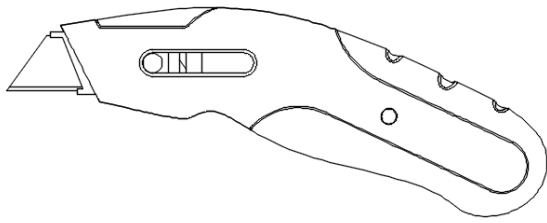
Step 3: – Edit the bottom projected view

1. Double-click anywhere inside the selection boundary of the bottom projected view to activate it.
2. In the View Properties dialog box, edit the properties of the component.
3. For **Tangent Edges**, select **Full Length** from the drop-down list.
4. Click **Close** to accept the drawing view changes.

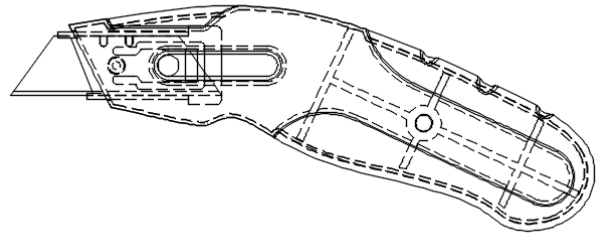


View Properties

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.

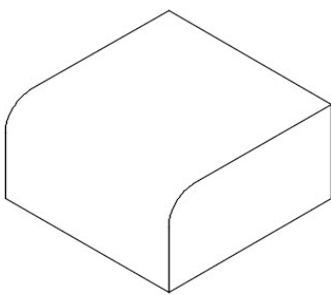


Hidden Lines OFF

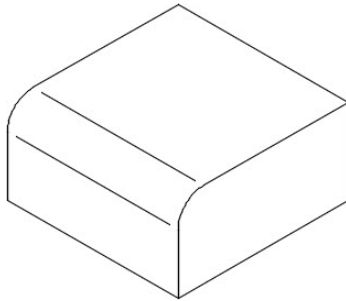


Hidden Lines ON

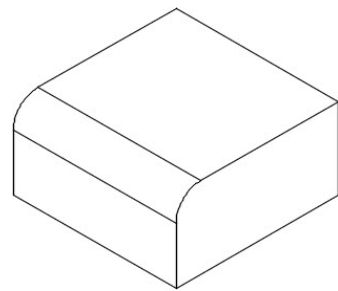
Tangent Edges – Select Full length, Shortened or Off from the drop-down list to display Tangent edges within the selected base view. Tangent edges mark the transition between a flat surface and a rounded edge, most commonly seen as filleted edges. Tangent edges can be set to Full Length, Shortened, or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



Tangent Edges Off

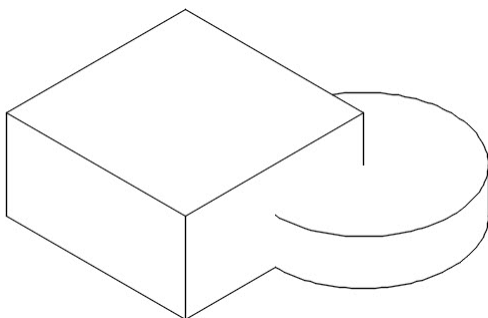


Tangent Edges Shortened

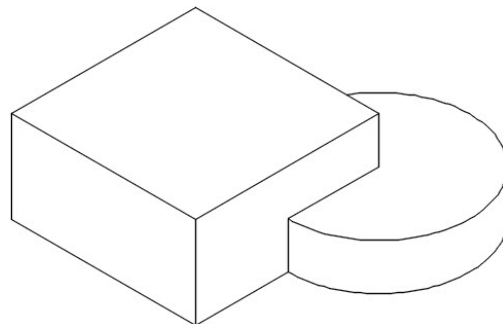


Tangent Edges Full Length

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



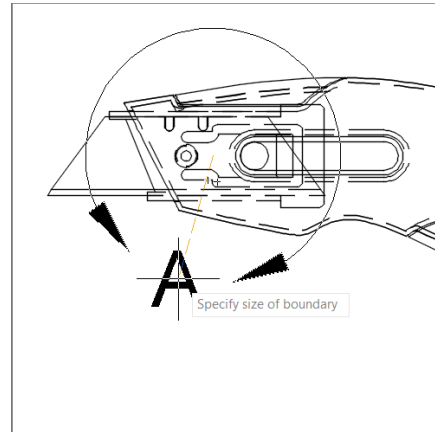
Interference Edges ON

Create a Detail View

A detail view is a projected view that shows a specific portion of a view at an enlarged scale.

Step 1: – Create a Detail View

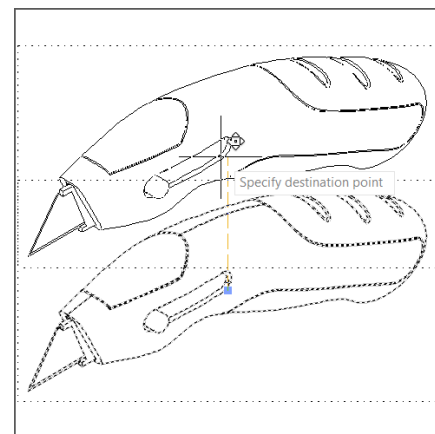
1. Select **Detail View** in the View Toolbar.
2. Select the right side view as a parent view.
3. Click the center point for the detail boundary.
4. Click again to specify the size and location of the detail boundary, and then click to place the detail view.
5. Change the Scale of the detail view to **2:1**, then press OK to generate the detail view.



Move Objects

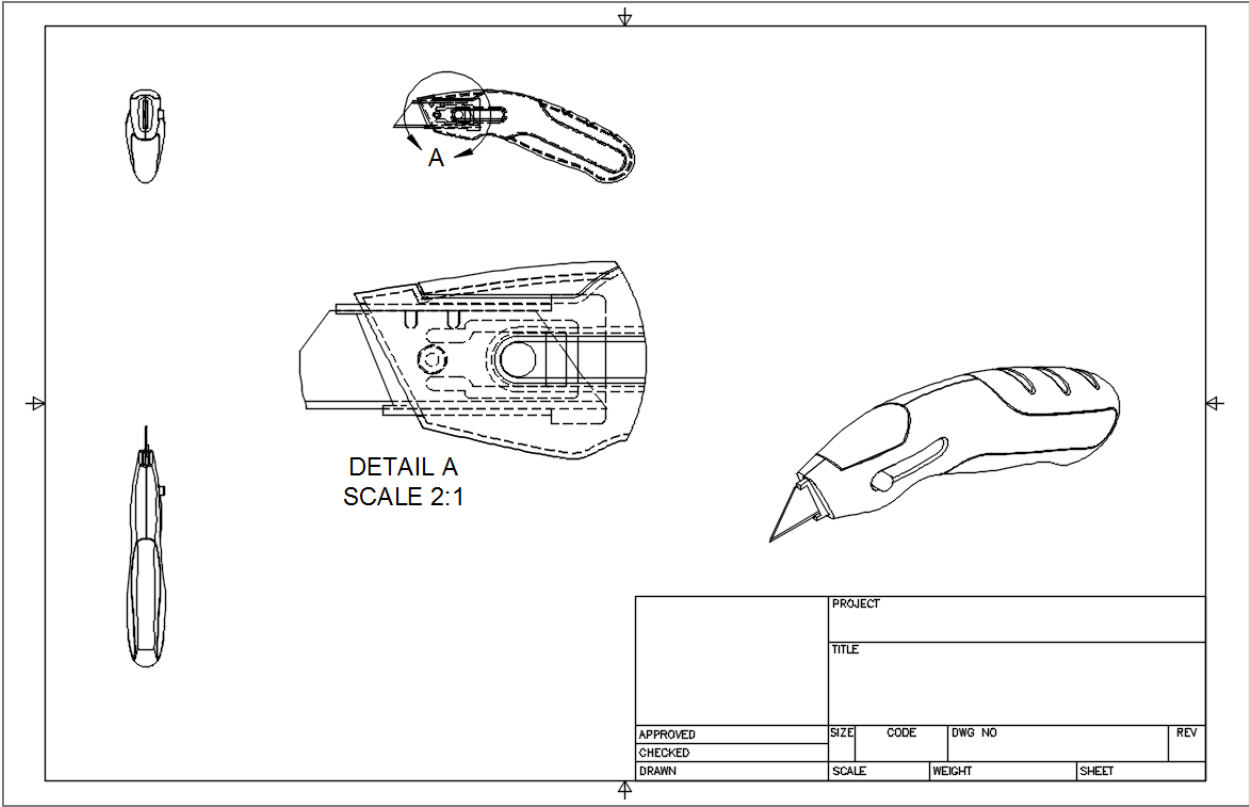
Click anywhere on an object to select it, click the gray grip in the center of the object, then click a new location for the object.

Note: The Move action works the same way for all views, text objects, dimensions, and balloons.



Move your objects around your drawing so they are nicely spaced, leaving the top right of the drawing space open (for a Parts List, to be added later).

By now, your drawing should look something like this:



Lesson 3: Text and Leader Notes

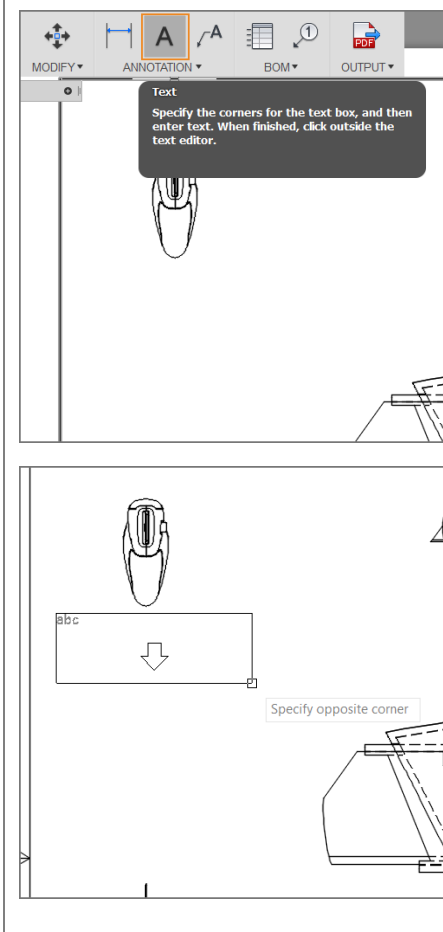
Learning Objectives

1. Create Text
2. Create Leader Notes
3. Reposition Annotation and Edit Text

Adding Text and Leaders

Step 1: – Create Text

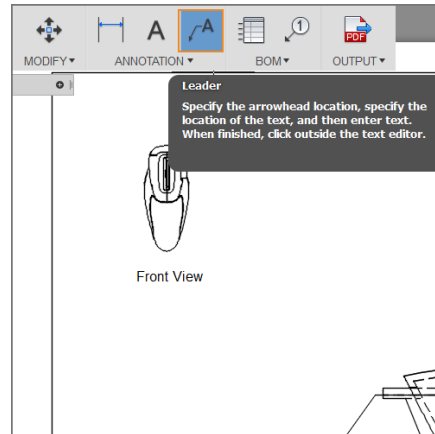
1. Initiate **Text** from the toolbar.
2. Select two corners below the Front View to create a text box.
3. Type the following text into the text box: “Front View”.
4. Select anywhere outside of the text box to commit the action.
5. Repeat the process to add text below the to the other views, naming them “Bottom View,” “Right View,” and “NE Isometric View”.



Step 2: – Create Leader Notes

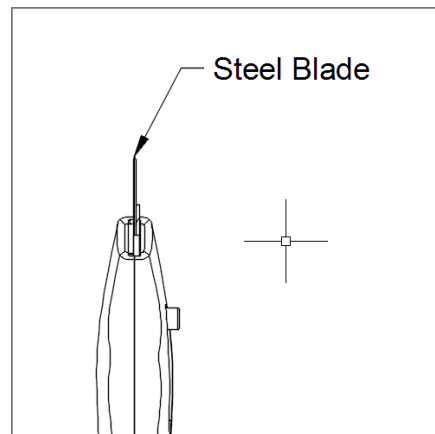
1. Initiate **Leader Note** from the toolbar.
2. Click the blade of the bottom view to place the start of the leader.
3. Click outside the view to place the end of the leader.
4. Type the following text: “Steel blade.”
5. Select anywhere outside of the text box to commit the action.

Tip: When selecting a part of an object, you'll see green shapes on the object. These are called **Object Snap points**, and they help you specify a precise point on the object, like an endpoint, a midpoint, or a center point.



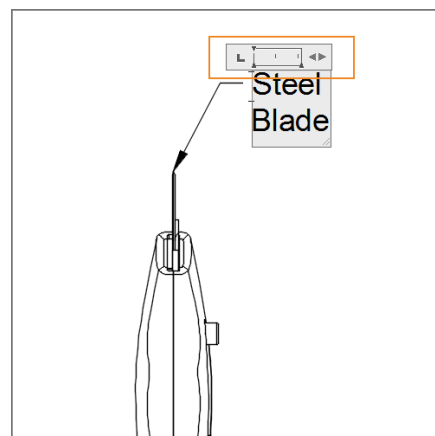
Step 3: – Reposition Leader Notes

1. Click on the leader note to activate it.
2. Click any of the gray grips to move the leader note to a new location.
3. Click anywhere outside the leader note to commit the action.



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. Click anywhere outside of the text box to commit the action.



Lesson 4: Dimensions

Learning Objectives

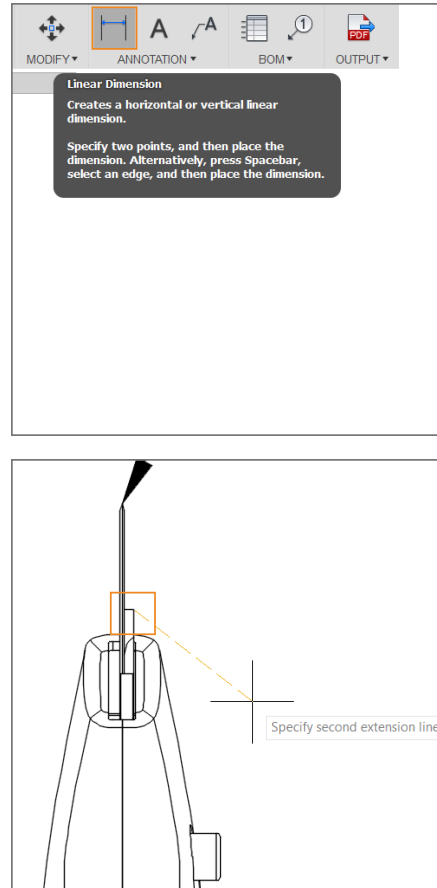
1. Create Linear, Aligned, Angular, Radial, and Diameter Dimensions
2. Reposition Dimensions
3. Use the Baseline and Chain Dimensioning tools

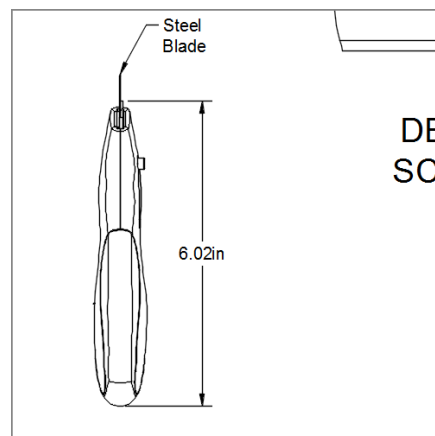
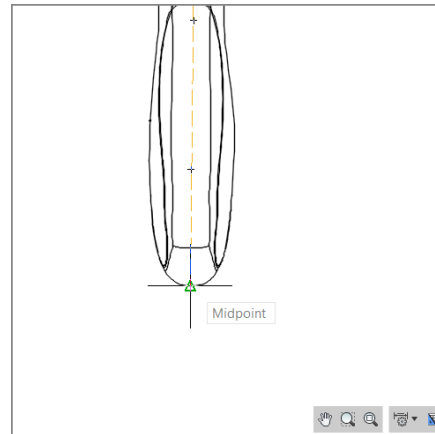
Create Dimensions

Step 1: – Create a Linear Dimension

1. Initiate **Linear Dimension**.
2. Use Object Snaps to click the two endpoints of the bottom view. A preview is displayed on your cursor.
3. Drag the cursor out to the right
4. Click again to place the dimension and complete the action.

Note: You can also press **Spacebar** and then select an edge to create a dimension with only one click.

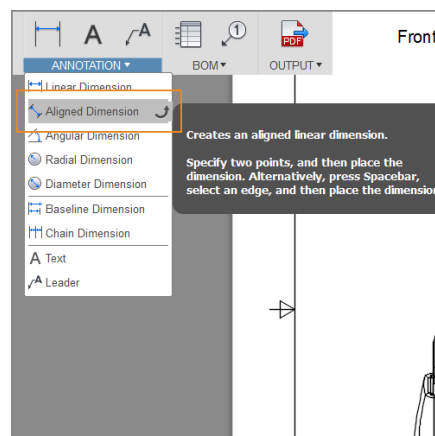


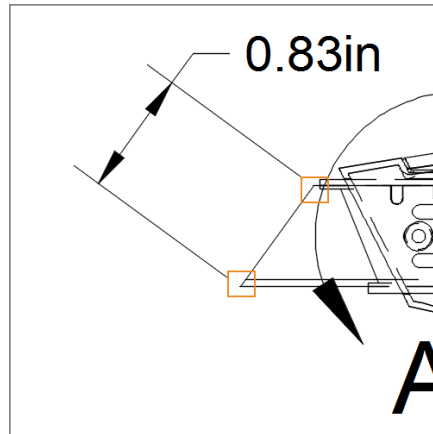


Step 2: – Create an Aligned Dimension

1. Initiate **Aligned Dimension**.
2. Using Object Snaps, click the top edge and bottom point of the cutting blade on the right view.
3. Drag the cursor out to the left to see a preview of the dimension.
4. Click again to place the dimension and finish the action.

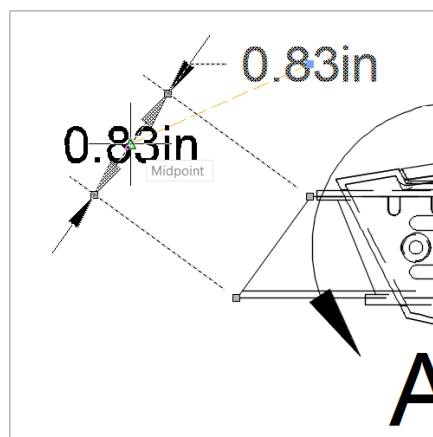
Note: Linear dimensions measure the horizontal or vertical distance between two points. Aligned dimensions measure the true distance between two points.





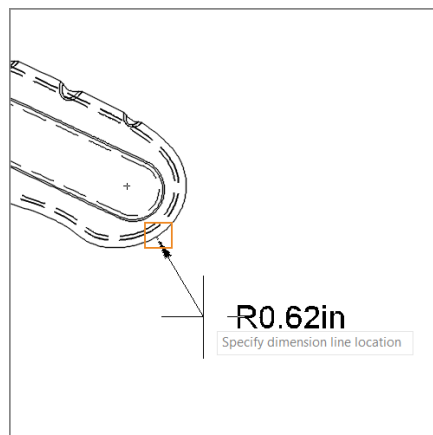
Step 3: – Reposition the Dimension

1. Click the aligned dimension.
2. Click the gray grip on the text.
3. Move the text so it is between the dimension arrows.
4. Click anywhere outside the dimension to complete the action.



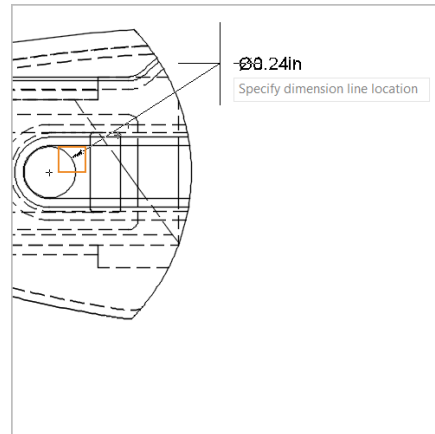
Step 4: – Create a Radial Dimension

1. Initiate **Radial Dimension**.
2. Select the curved end of the handle in the right side view.
3. Drag the cursor down and to the right to see a preview of the dimension.
4. Click again to place the dimension and complete the action.



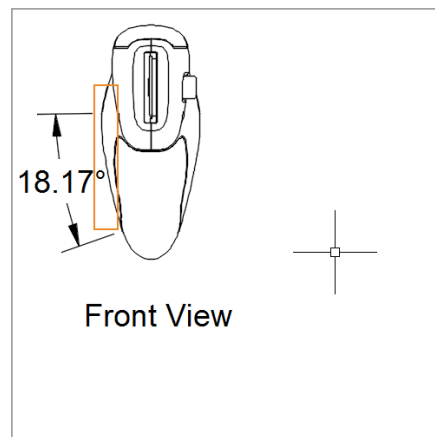
Step 5: – Create a Diameter Dimension

1. Initiate **Diameter Dimension**.
2. Select the blade slider on the detail view.
3. Drag the cursor up and to the right and see a preview of the dimension.
4. Click again to place the dimension and finish the action.



Step 6: – Create an Angular Dimension

1. Initiate **Angular Dimension**.
2. Click the side of the knife in the Front View.
3. Drag the cursor to the left to see a preview of the dimension.
4. Click again to place the dimension and finish the action.



Use Chain and Baseline Dimensions

The Baseline Dimension tool creates multiple linear dimensions measured from the same location. The Chain Dimension tool creates multiple linear dimensions placed end to end.

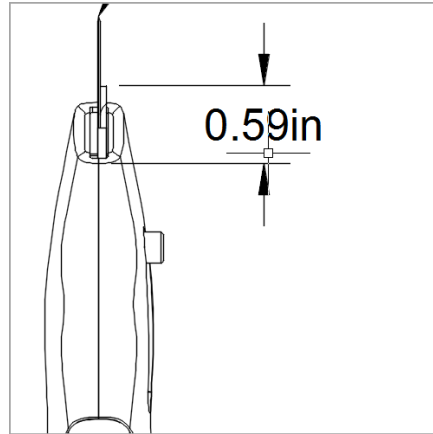
Baseline and chain dimensions derive from an original dimension. You must first create a linear dimension before you create baseline or chain dimensions.

Step 1: – Create a new Linear Dimension

1. Let's delete the Linear Dimension on the bottom view by selecting the dimension, then pressing the Delete

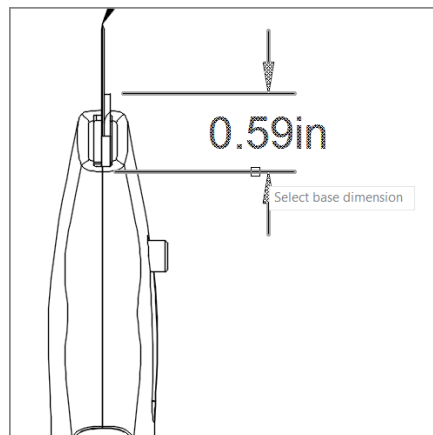
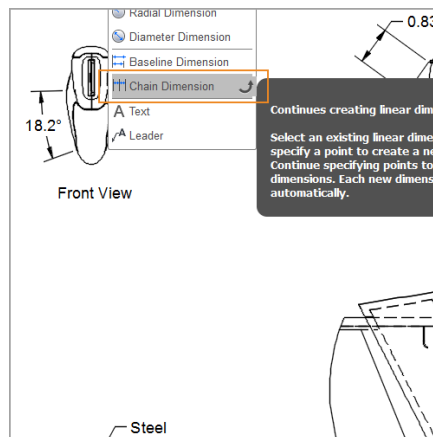
key, or by selecting Delete in the Marking Menu.

2. Create a new linear dimension at the end of the blade cradle.



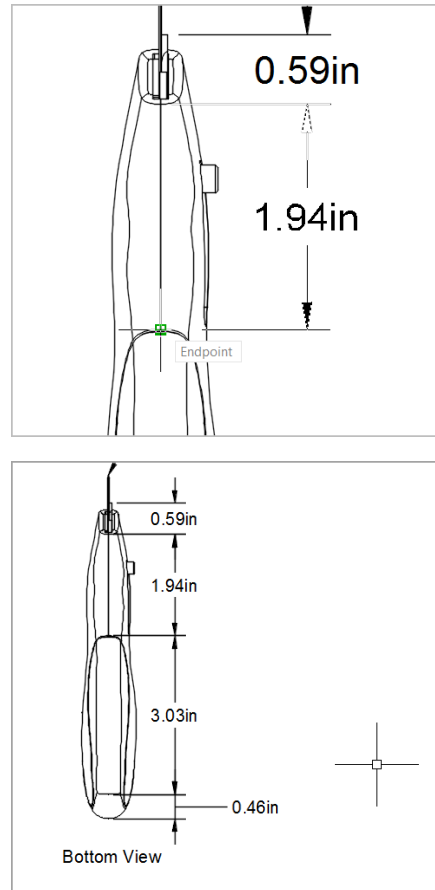
Step 2: – Place Chain Dimensions

1. Select Chain Dimension in the Annotation toolbar.
2. Select the lower extension line of the newly placed dimension.

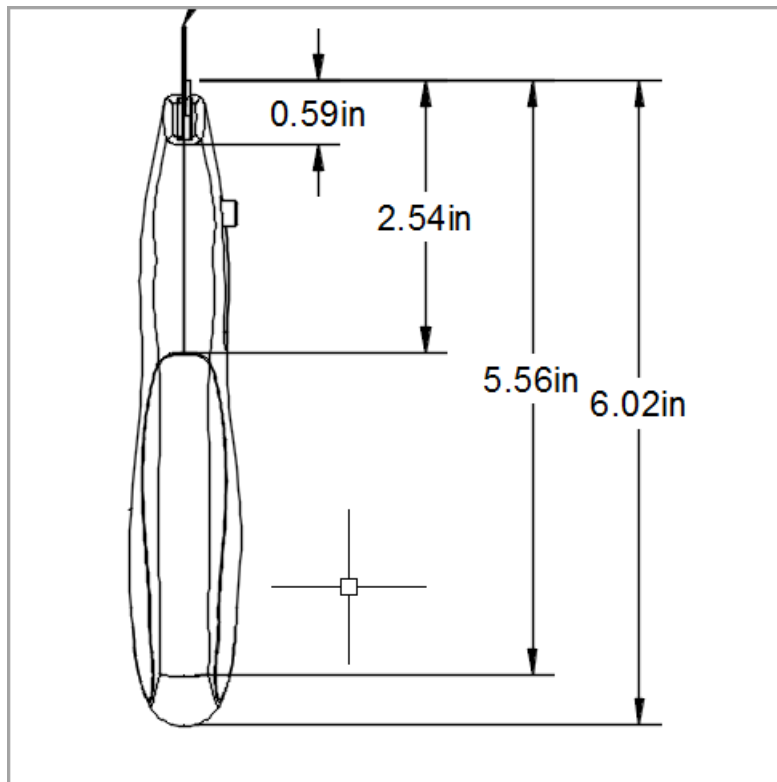


Step 3: – Complete the Chain Dimension

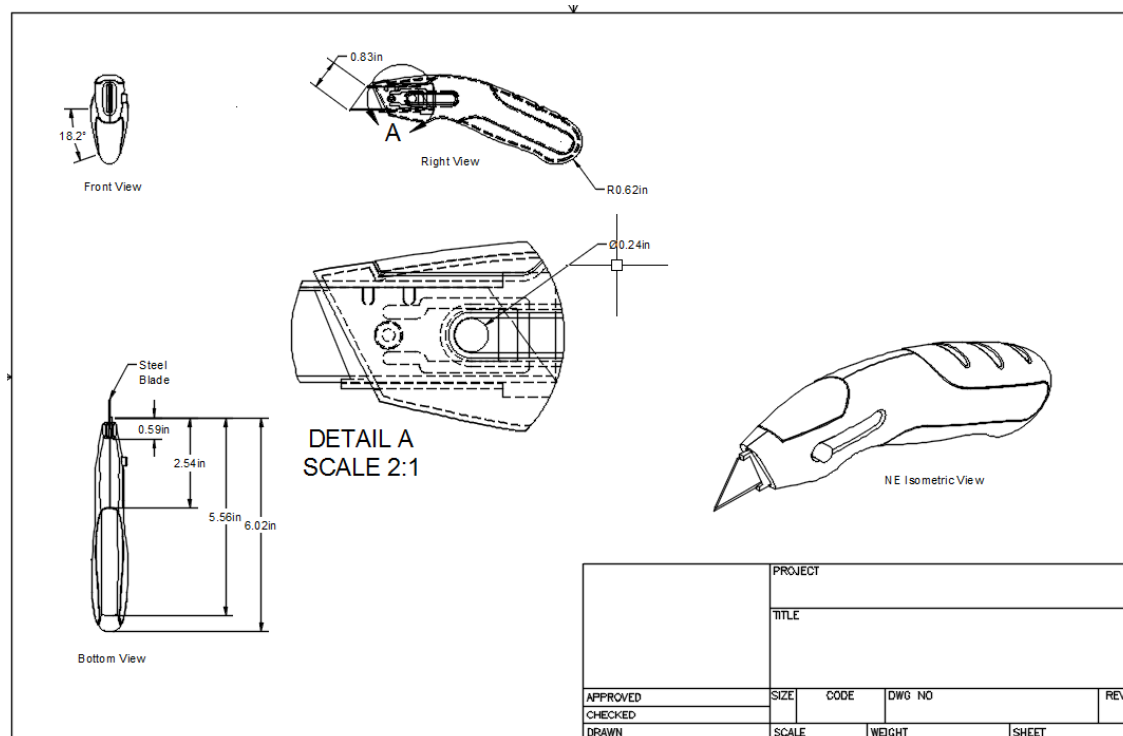
1. Click the end of the knife's grip to specify the endpoint of the first dimension.
2. Click again to specify another endpoint of a dimension.
3. Click once more at the very end of the bottom view.
4. To finish the action, press **Enter**.



Baseline Dimensions work the same way as Chain Dimensions, only they create multiple dimensions that all begin from the same location. If you were to use Baseline Dimensions here, it would look like this:



By now, your drawing should look something like this:

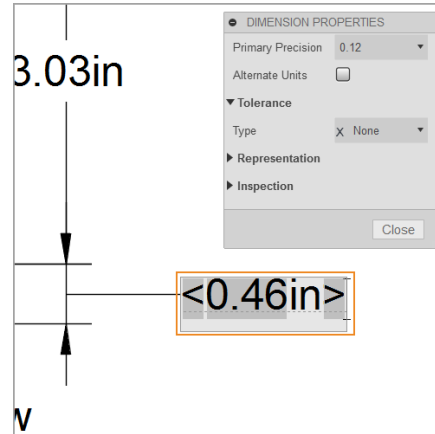


Modify the Properties of an Individual Dimension, and add Tolerance and Representation.

Step 1: – Access Dimension Properties

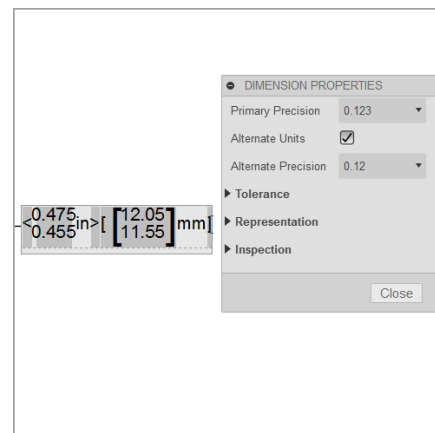
1. Double-click the dimension on the back grip of the bottom view.

The Dimension Properties dialog box opens.



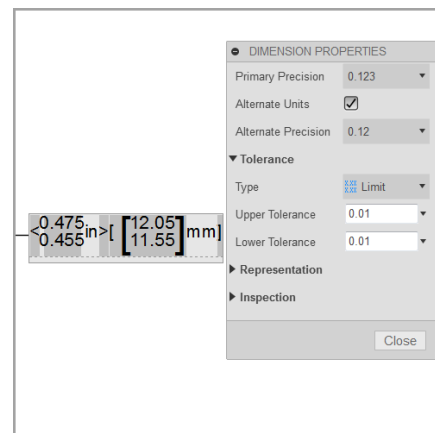
Step 2: – Modify the dimension's properties

1. Change the primary precision for this dimension to **0.123**.
2. Have the dimension display Alternate Units.



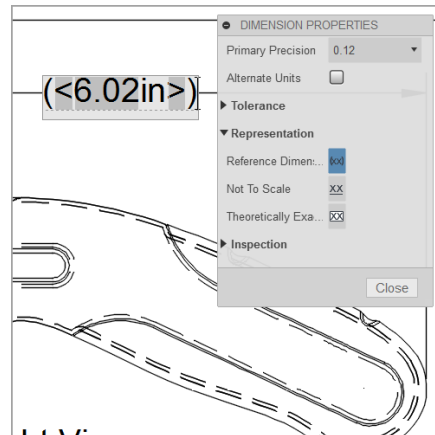
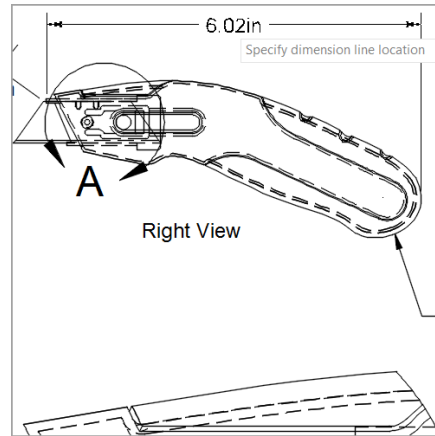
Step 3: – Add tolerance to the dimension

1. Under **Tolerance**, select Limit, and give it an Upper and Lower tolerance of **0.01**.
2. Press Close.



Step 4: – Create a reference dimension

1. Create a new linear dimension in the right view, using the corner of the blade cradle and the quadrant snap point on the end of the handle as snap points.
2. Under **Representation**, select Reference Dimension.



Lesson 5: Parts List, Balloons, and Title Block

Learning Objectives

1. Insert a Parts List
2. Create Annotative Balloons
3. Populate the Title Block

A parts list is a table that catalogs the components of the design. It itemizes all components in the drawing, and includes the item number, the quantity, the part number, the description, and the material.

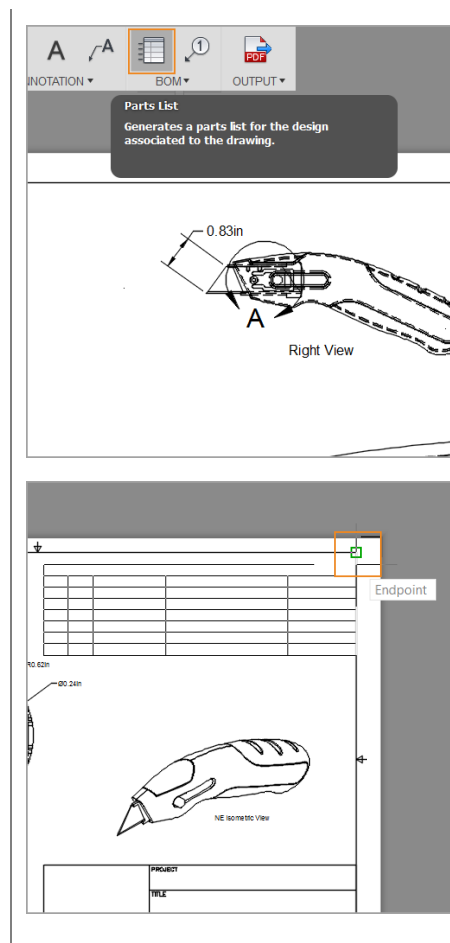
A balloon identifies a component included in the parts list by labeling it in the drawing.

Insert a Parts List and Add Balloons

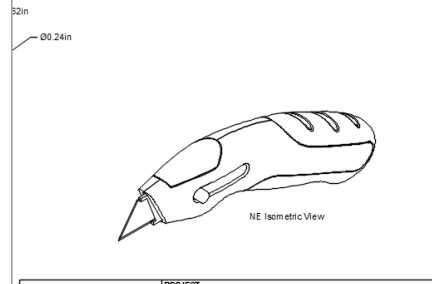
Step 1: – Insert a Parts List

1. Click **Parts List** and click the very top right corner to place the Parts List there (you may have to move some of your views over a bit, to make room).

The Parts List automatically populates, taking information from Model Space.



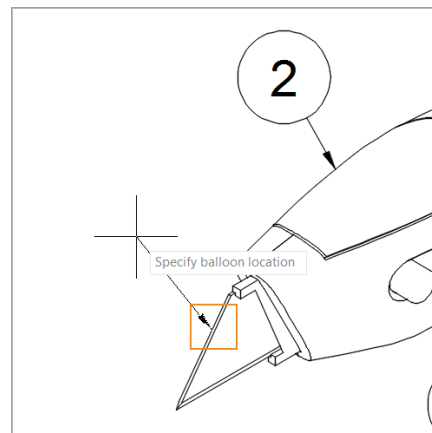
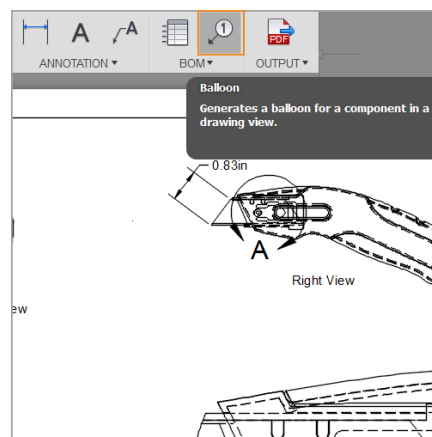
PARTS LIST				
ITEM	QTY	PART NUMBER	DESCRIPTION	MATERIAL
1	1	GRIP 1	BACK GRIP	RUBBER, NITRILE
2	1	GRIP 2	TOP GRIP	RUBBER, NITRILE
3	1	LEFT	OUTER CASING	STEEL, MILD
4	1	RIGHT	OUTER CASING	STEEL, MILD
5	1	BLADE	DETACHABLE	STEEL, CAST
6	1	BLADE CRADLE	SLIDES 1.1 INCHES	ABS PLASTIC



32in
Ø0.24in
NE Isometric View

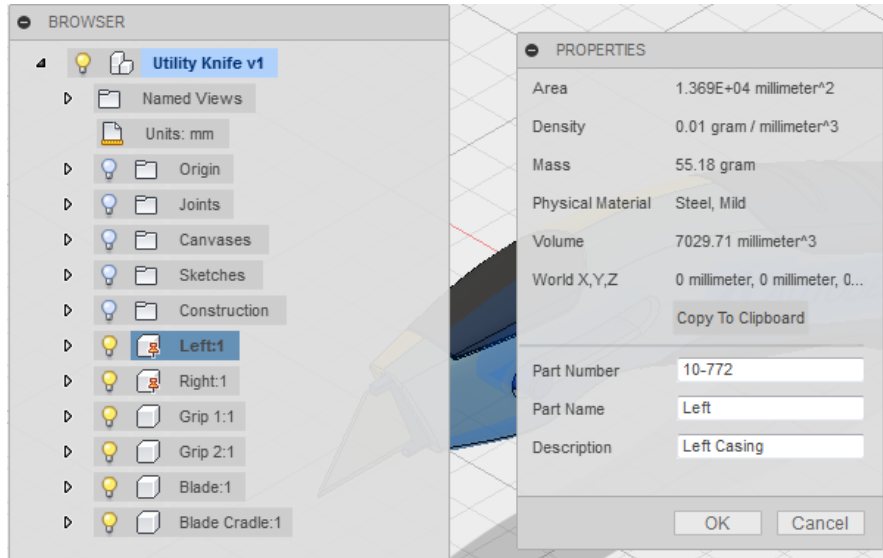
Step 2: – Create Annotative Balloons

1. Click **Balloons** in the BOM toolbar.
2. On the NE isometric view, select an edge of a component.
3. Click to place the balloon.
4. Continue selecting components until all items in the Parts List are numbered.
5. When finished, press **Enter**.



About the Parts List

A component's part number and description can be modified in the Properties Panel in the Model workspace:



If you change the name or properties of a component in the Model workspace, the changes will be applied to the drawing once you associate the drawing with the saved model (see [Associatively Update the Drawing](#)).

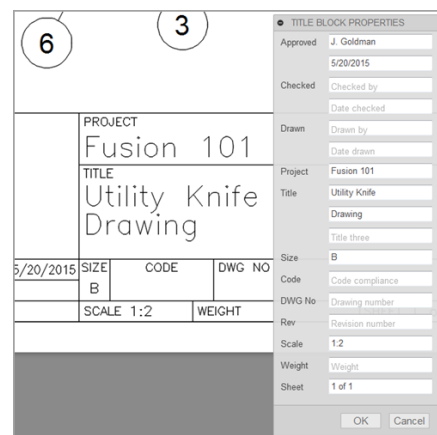
Note: The parts list table is not editable or customizable.

Populate the Title Block

To populate or edit the contents of the title block, simply double-click any boundary of the title block.

The Title Block Properties dialog box opens. Input your text, and click **OK** when finished.



Note: The title block is not customizable.



Lesson 6: Drawing Settings and Preferences

Learning Objectives

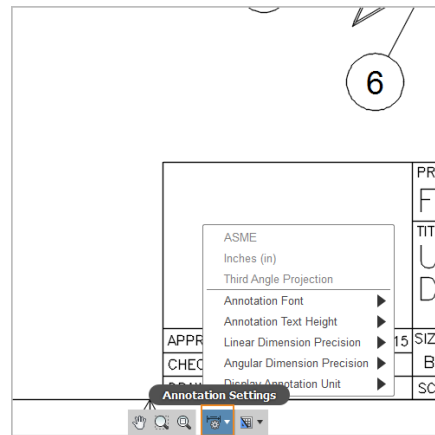
1. Modify the global annotation settings for the current drawing
2. Modify the properties of an individual dimension
3. Change the default settings for future drawings

You can modify annotation settings such as annotation font, annotation text height, dimension precision, and whether or not you want to display dimensions, by accessing **Annotation Settings**  in the bottom of the drawing area. You can change the sheet size by accessing **Sheet Settings**. 

Modify the Annotation Settings for the Current Drawing

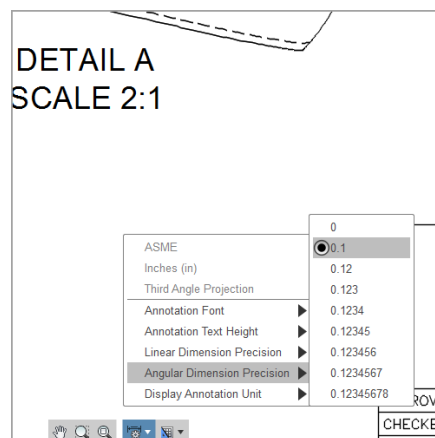
Step 1: – Access Annotation Settings

1. **Click Annotation Settings** in the bottom of the drawing area.



Step 2: – Edit the Annotation Preferences

1. Change “Display Annotation Unit” to **On** if it is not already set that way.
2. Change the Angular Dimension Precision to **0.1**.



The annotation settings apply to all dimensions and text in the drawing. However, you can change the properties of an individual dimension, which overrides any “global” settings you have changed.

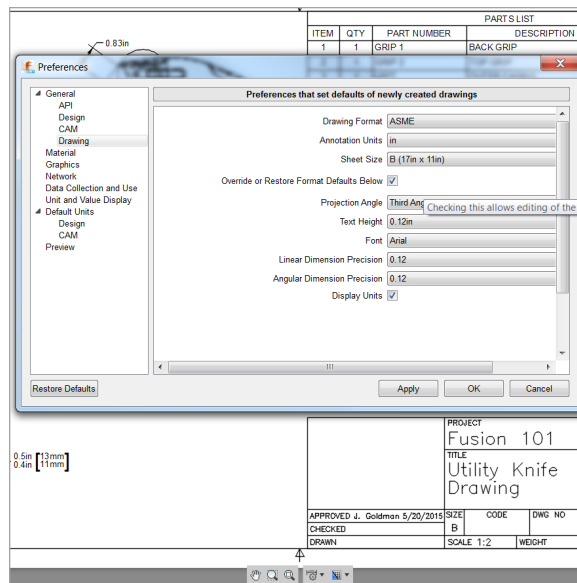
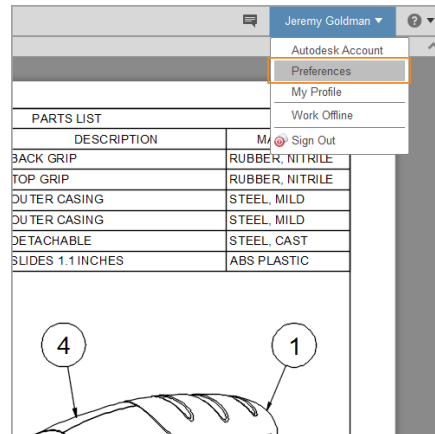
Modify the Default Settings

The default settings affect all future drawings.

Step 1: – Access Drawing Preferences

1. Click **Preferences** in the User Profile drop-down (your name).
2. Under General, click Drawing.

Here you will see the default settings that apply to any new drawings that you create.



Note: The default settings are applied whenever you create a **new** drawing. Some of the settings can be modified after you have created a drawing, but settings like Drawing Format, Units, and Projection Angle cannot be modified once a drawing is created. If you need to switch these, you will have to create a new drawing.

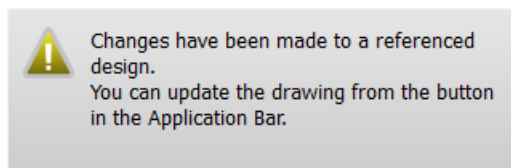
Lesson 7: Associatively Update the Drawing

Learning Objectives

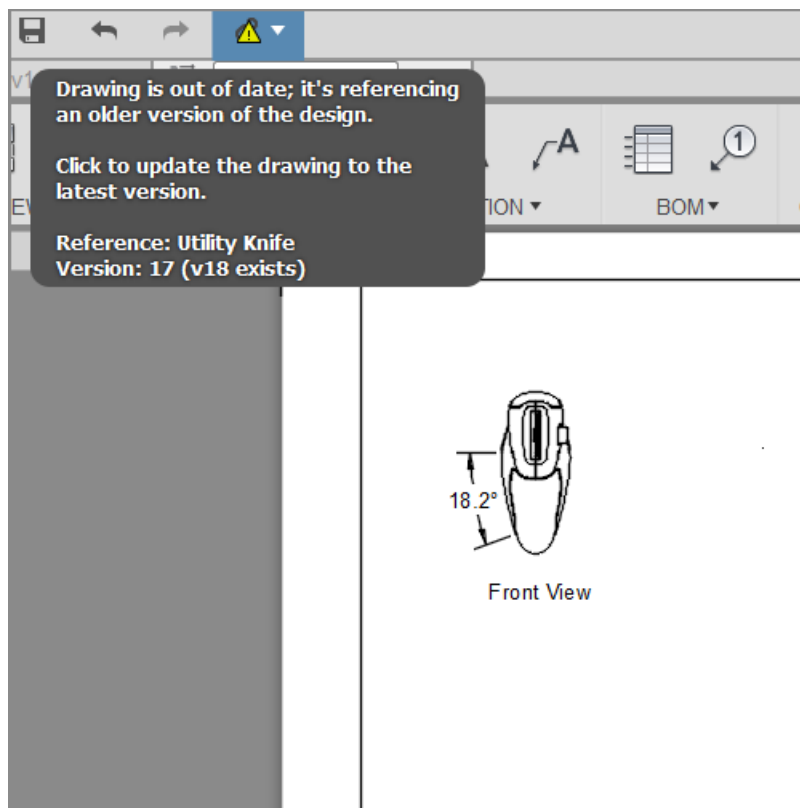
1. How to update a drawing so it reflects any changes made to the model

If you make any changes to the model's geometry and then save the model, you can update your drawing to reflect those changes by clicking “Get Latest” on the toolbar.

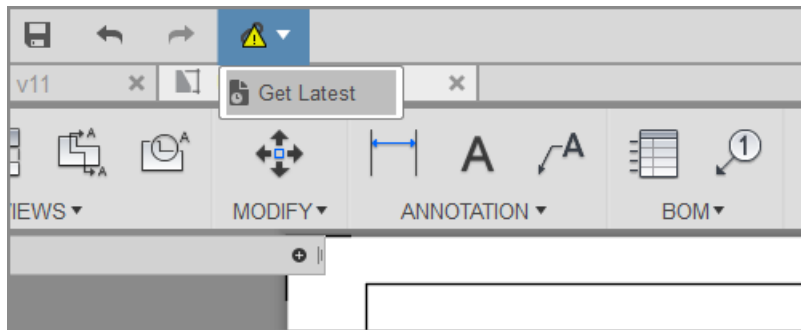
If there are saved changes to a model, you will see this message in the bottom right of your screen:



And you will see this message if you hover over the versions button:



If you click “Get Latest,” the drawing will update to reflect the new changes.



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.

Lesson 8: Output the Drawing

Learning Objectives

1. Output your Drawing as a PDF or DWG file

When the drawing is complete, you can output the layout as either a PDF or DWG file. Both of these options creates a copy of the drawing and prompts you to save it locally.

Output the Drawing

Step 1: – Save the drawing

1. Initiate the file menu dropdown and click the Save button.
2. Enter 'Utility Knife' in the Name field.
3. Click OK.

Step 2: – Output a PDF

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

