Drawing Views with Autodesk® AutoCAD® 2014: 2D Drawings from AutoCAD® 3D Models

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AC1633-L This intermediate-to-advanced hands-on lab offers Autodesk AutoCAD 3D veterans and 2D users a chance to explore 2D model documentation of 3D models from AutoCAD 2014 and Autodesk® Inventor® 2014 software. Create base views and projected views from 3D solid or surface models. Use the new Viewbase command to generate a base 2D view and the Viewproj command to create projected orthographic views. Learn to edit the 3D models and update the derived drawing views, and add dimensions to the drawing views and modify the 3D model to update the dimensions. Are commands such as Flatshot, Solview, Soldraw, and Solproj superseded? Find out! In this class, we also explore the Exportlayout command for exporting a layout containing drawing views to model space. If you have used AutoCAD 3D in the past or are new to 3D but have a good 2D background, attend this lab and get ready to be surprised!

Learning Objectives

At the end of this class, you will be able to:

- Use AutoCAD 2014 3D models to create multi-view projections

- Add dimensions to drawing views in AutoCAD 2014

- Edit AutoCAD 2014 3D models and the associative drawing views

- Export drawing views to create model space objects

About the Speaker

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For AutoCAD 2012, 2013 and 2014

Model Documentation

There are multiple techniques available to create drawing views from 3D models in AutoCAD. Using these techniques, orthographic views and other standard drawing views can be created in order to produce multiview drawings. The techniques available are discussed in this paper.

Drawing Views

You can create 2D drawing views from 3D solid or surface models using the **VIEWBASE** command. This command allows you to quickly create a multiview drawing layout made up of orthographic views. Drawing views created with the **VIEWBASE** command are created in paper space (layout space). After creating a layout of views, you can dimension them using AutoCAD dimensioning commands.

Drawing views created with the **VIEWBASE** command are _**associative**_. This means that they are linked to the model from which they are created and can be updated to reflect changes to the model geometry. This capability allows you to keep drawing views up-to-date when design changes are required.

When using the **VIEWBASE** command, you create a **base view** of the model and then have the option to create additional views that are projected from the base view without exiting the command. You can create both orthographic and isometric views. A base view created with the **VIEWBASE** command is defined as a _parent view_. Views projected from the base view inherit the properties of the base view, such as the drawing scale and display properties, and are placed in orthographic alignment with the base view. If the base view is moved, any projected views are moved with it to maintain the parent-child relationship.

Drawing View and Layout Setup

In order to create views with the **VIEWBASE** command, you must first activate paper space (layout space). Activate paper space by picking a layout tab. Then, before creating views, set up

the page layout as required. If a default viewport appears in the layout, delete the viewport. Then, right-click on the active layout tab and select **Page Setup Manager** to access the layout settings. Select the desired paper size, drawing orientation, and other settings. If you have a template drawing already created, you can create a new layout based on the template by right-clicking on a layout tab and selecting **From template…**.

By default, drawing views created with the **VIEWBASE** command are generated using third-angle projection. This is the ANSI standard. The default projection used for drawing views can be set using the **VIEWSTD** command. This command and other drawing view commands are accessed from the **Create View panel** on the **Layout tab** of the ribbon.
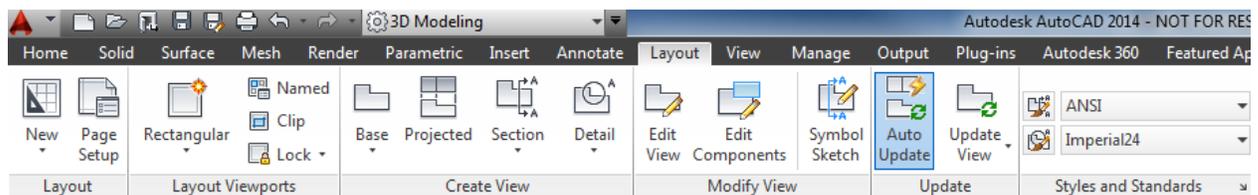
Pick the dialog box launcher button at the lower-right corner of the **Styles and Standards** panel to initiate the **VIEWSTD** command. This opens the **Drafting Standard** dialog box.
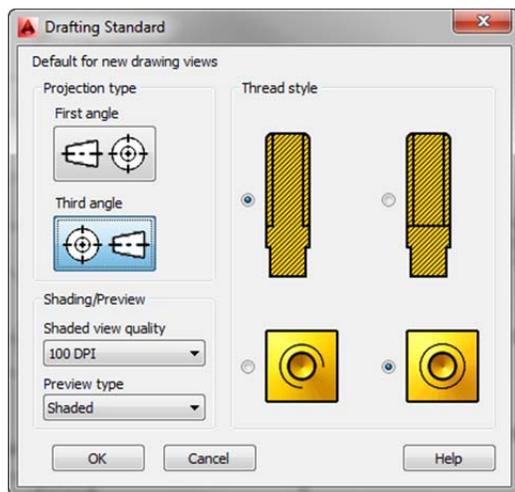

The options in the **Projection type** area determine the type of projection used for new drawing views. By default, the **Third angle** button is selected. If the **First angle** button is selected, views are created using first-angle projection. Select this option to create views in accordance with ISO standards. The options in the **Thread style** area determine the type of thread representation used for views showing threaded features. The thread representation

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options are used when creating views from Autodesk Inventor models or imported models containing threaded features. A full circle thread representation conforms to the ANSI standard. This is the default option. A partial circle thread representation conforms to the ISO standard.

The options in the **Shading/Preview** area of the **Drafting Standard** dialog box determine the type of preview that appears when placing a drawing view. When **Shaded** is selected in the **Preview type** drop-down list, a shaded view appears. This is the default option. This option provides a preview image of the view orientation before the view is placed. When the **Bounding box** option is selected, a preview box representing the extents of the view appears. However, no preview image is displayed. The **Shaded view quality** option is set to 100 dpi by default. A higher setting will provide better resolution quality when a shaded preview is displayed. However, for all practical purposes, a setting of 150 dpi or higher does not produce a dramatic difference in resolution quality.

Creating Drawing Views

After setting up a layout and making the appropriate drawing view settings, you are ready to place the base view. As previously discussed, drawing views can be created from a 3D solid or surface model. Drawing views created in the layout are generated from the model you have created in model space. If no model exists in the drawing and you initiate the **VIEWBASE** command. You can select a model created in AutoCAD or a model created in Autodesk Inventor. You can select **Base** from the **Create View panel, Layout tab**.
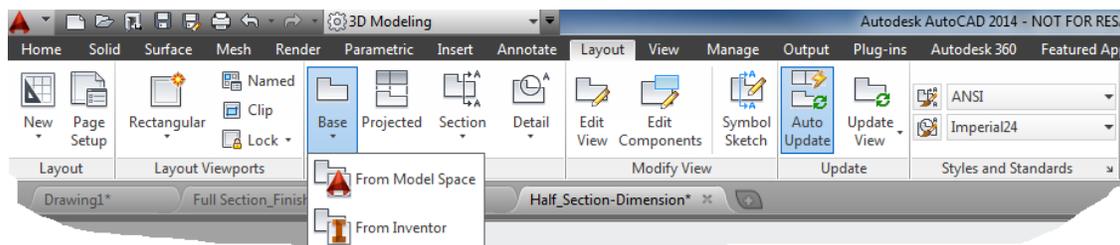
When you select the **VIEWBASE** command from the ribbon, a preview representing a scaled orthographic view of the model appears. This is the *base view* of the model. You can

then specify the location of the base view or select an option. To select an option, use dynamic input or make a selection from the **Drawing View Creation** contextual ribbon tab.
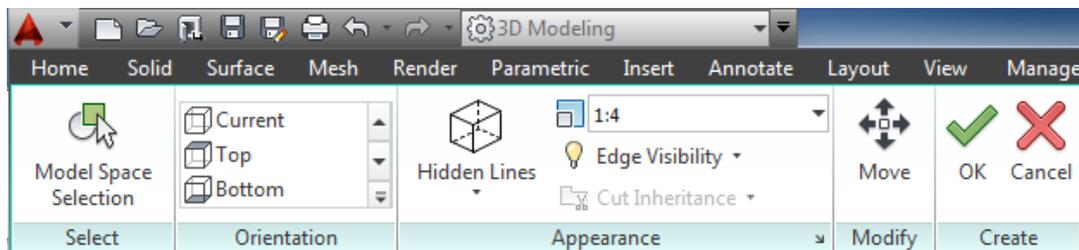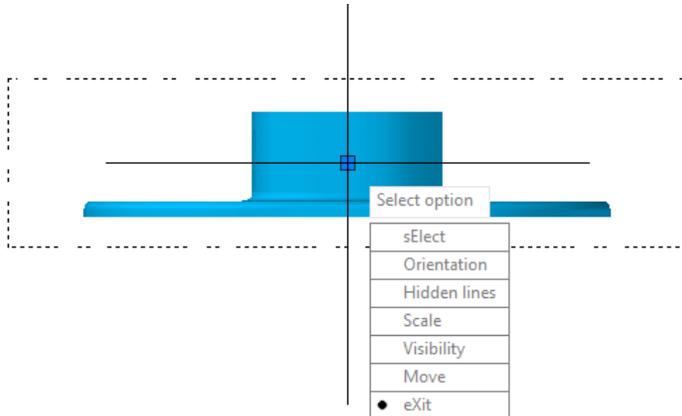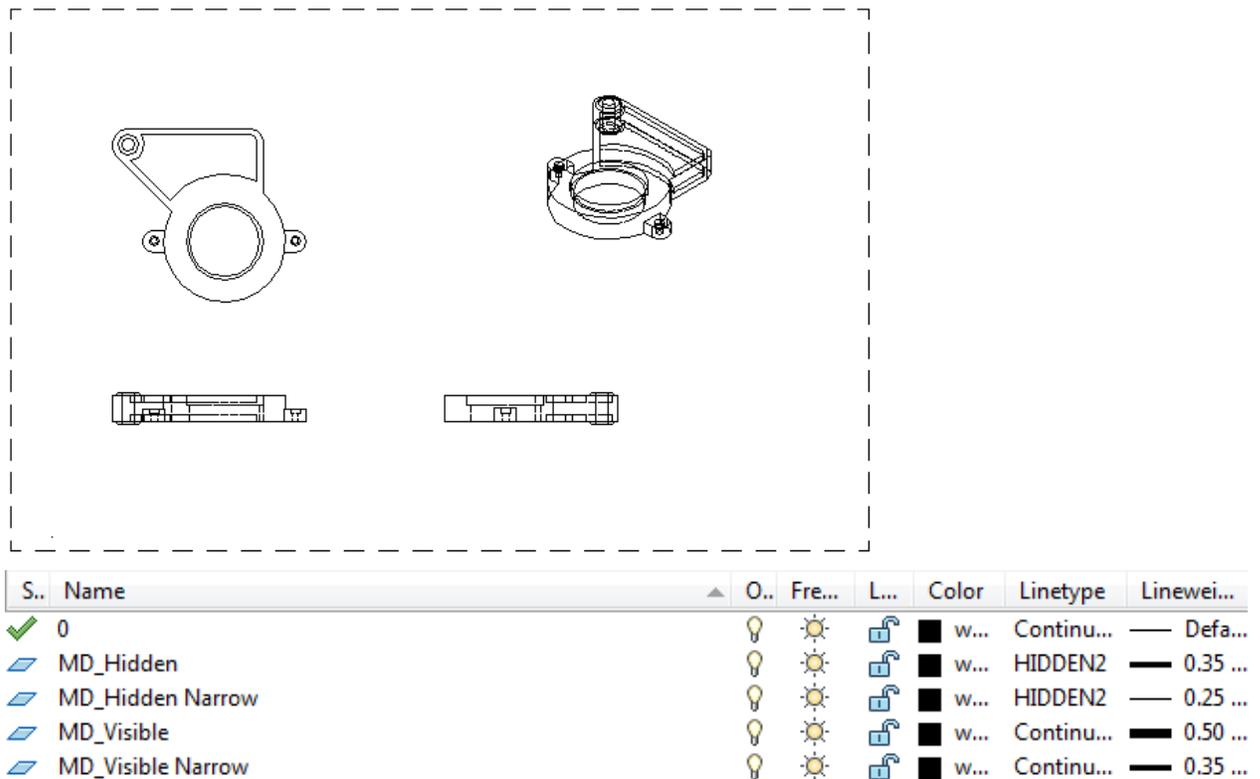

Using the VIEWBASE command from the command line you can use various command options.

```
Command: VIEWBASE
Specify model source [Model space/File] <Model space>:
Type = Base and Projected  Hidden Lines = Visible and hidden lines  Scale = 1:4
VIEWBASE Specify location of base view or [Type sElect Orientation Hidden lines Scale Visibility] <Type>:
```

The **Select** option switches you to model space and allows you to add objects to or remove objects from the base view. If necessary, use the appropriate option to select solids or surfaces to add or remove. Use the **Return to Layout** option to return to the layout after making changes.

S..	Name	▲	O..	Fre...	L...	Color		Linetype	Linewei...
✓	0		♀	☼	🔓	■	w...	Continu...	— Defa...
◿	MD_Hidden		♀	☼	🔓	■	w...	HIDDEN2	— 0.35 ...
◿	MD_Hidden Narrow		♀	☼	🔓	■	w...	HIDDEN2	— 0.25 ...
◿	MD_Visible		♀	☼	🔓	■	w...	Continu...	▬ 0.50 ...
◿	MD_Visible Narrow		♀	☼	🔓	■	w...	Continu...	▬ 0.35 ...

The layer properties can be modified to change the appearance of the drawing view geometry, but the geometry cannot be otherwise modified.

The **Orientation** option can be used to select a different view from the default base view. The **Orientation** options correspond to AutoCAD's six orthographic and four isometric preset views. These views are based on the WCS. By default, the front view is used as the base view by AutoCAD. Depending on the construction of your model, you may want to use a different view, such as the top view, to serve as the base view. In **Figure 14-8**, the top view of the model is selected as the base view. This view will establish the front view on the orthographic drawing. In most cases, the front orthographic view on the drawing describes the most critical contour of the model. You will typically use the front or top view of the 3D model for the front orthographic view on the drawing.

The **Type** option of the **VIEWBASE** command is available before picking the initial location of the base view. This option is used to specify whether projected views are placed after placing the base view. By default, this option is set to **Base and Projected**. With this option, you can

place additional, projected views while the **VIEWBASE** command is still active. Projected views that you place are *projected* from the base view and oriented in the proper orthographic alignment. As previously discussed, projected views have a parent-child relationship with the base view. The base view is the *parent* view and any projected views are *child* views. If the base view is moved, for example, the projected view is moved with it. Projected views are not created with the **VIEWBASE** command when the **Type** option is set to **Base only**. If you place a base view in this manner and later decide to create projected views from the base view, you can use the **VIEWPROJ** command. The **VIEWPROJ** command allows you to place projected views from any selected view.

After you pick the location for the base view, you can press [Enter] to place projected views (if the **Type** option is set to **Base and Projected**). Drag the cursor away from the base view and then pick to locate the projected view. The direction in which you move the cursor determines the orientation of the projected view. You can continue placing projected views, or you can press [Enter] to end the command. To end the command without placing projected views, press [Enter] twice.

The display style of the base view can be changed by using the **Hidden Lines** option. The four display style options available are **Visible Lines**, **Visible and Hidden Lines**, **Shaded with Visible Lines**, and **Shaded with Visible and Hidden Lines**. See **Figure 14-9**. These options can be selected using dynamic input or the drop-down list in the **Appearance** panel in the **Drawing View Creation** contextual ribbon tab.

The **Scale** option is used to set the scale of the base view. You can select a scale from the drop-down list in the **Appearance** panel in the **Drawing View Creation** contextual ribbon tab. You can also specify the scale by typing a value.

The **Move** option allows you to move the base view after picking the initial location on screen. When you select the **Move** option, the view is reattached to the cursor so that you can move it to another location. After you pick a new location, the **VIEWBASE** options are again made available.

The **Visibility** option allows you to control the display of drawing geometry in the view. The **Interference edges** option controls whether both object and hidden lines are displayed for interference edges. By default, this option is set to **No**. The **Tangent edges** option controls whether tangential edges are displayed to show the intersection of surfaces. By default, this option is set to **No**. If this option is set to **Yes**, you can specify whether tangent edges are

shortened to distinguish them from object lines that overlap. The **Bend extents** option is only available when working with a model that includes a view with sheet metal bends. The **Thread features** option controls thread displays on models with screw thread features. The **Presentation trails** option is used to control the display of trails in views created from presentation files.

Additional drawing view options can be accessed in the **Appearance** panel of the **Drawing View Creation** contextual ribbon tab. Selecting the dialog box launcher button at the bottom of the panel opens the **View Options** dialog box. The options in the **View Justification** drop-down list determine the justification of the view. The *justification* refers to how the view is "anchored." When changes are made to the model, such as a change in size, the view updates based on the justification setting. If the justification is set to **Fixed**, geometry in the view unaffected by the edit does not change from the original location. If the justification is set to **Center**, the geometry updates about the center point of the view.

NOTE

The **Representation** option of the **VIEWBASE** command is used to create a view based on a model representation and is only available when working with models created in Autodesk Inventor.

Updating Drawing Views

Drawing views maintain an associative relationship with the model from which they are created. However, it is important to note that this associative relationship is controlled by the model, not the drawing view. When making design changes, you make changes to the model geometry, *not* the drawing view.

During the design process, it is often necessary to make modifications. If you have created drawing views from a model, and then make modifications to the model, the derived base view and any views projected from the base view will require updating. When this occurs, drawing views are considered *out-of-date*. AutoCAD does not automatically update the views when the model is modified. However, you can quickly update drawing views when needed.

VIEWUPDATE

When you make changes to a model in model space and then switch to a layout with

drawing views, red markers appear to indicate that the views are out-of-date. In addition, a balloon notification appears on the status bar. You can pick the link in the balloon notification to update the views, or you can use one of the view update commands in the **Drawing Views** panel on the **Annotate** tab of the ribbon. In the example, all of the views are updated at once based on a change to the original part.

Example   Change the cylinders diameters to a larger diameter and then use VIEWUPDATE to update the drawing views. What happens?
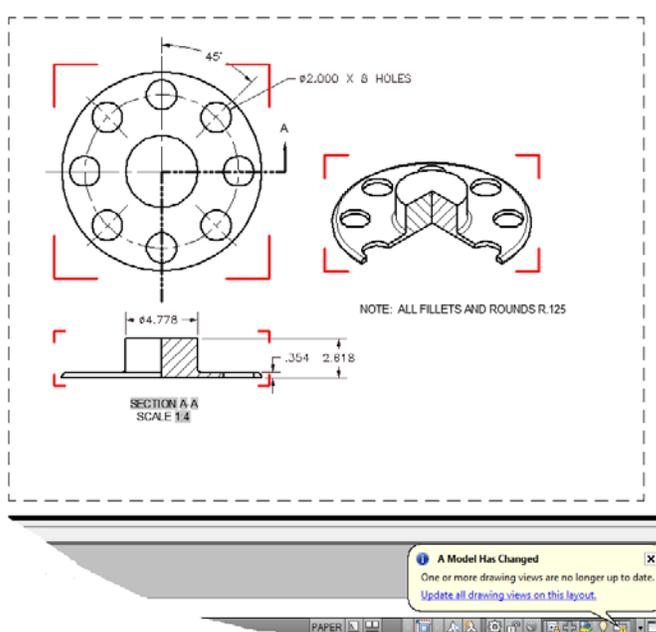
This is the preferred method for updating views. However, you can also update views individually by selecting **Update View** from the drop-down menu in the **Update** panel on the **Layout** tab of the ribbon. This requires you to select each view to update individually. Selecting **Update View** is the same as using the **VIEWUPDATE** command.

NOTE:

Drawing views are different from viewports. You cannot pick inside of the view and edit or modify the drawing geometry by any standard AutoCAD editing technique. Drawing views can only be updated by changing the 3D model geometry.

PROFESSIONAL TIP:

In order to modify solid primitive subobjects in a composite solid model, the solid history must be recorded.

1. Open the drawing **Flange_Base.dwg**
2. Select the ANSI B layout.
3. Create a 4 view drawing as shown below.
4. Turn Auto Update off.
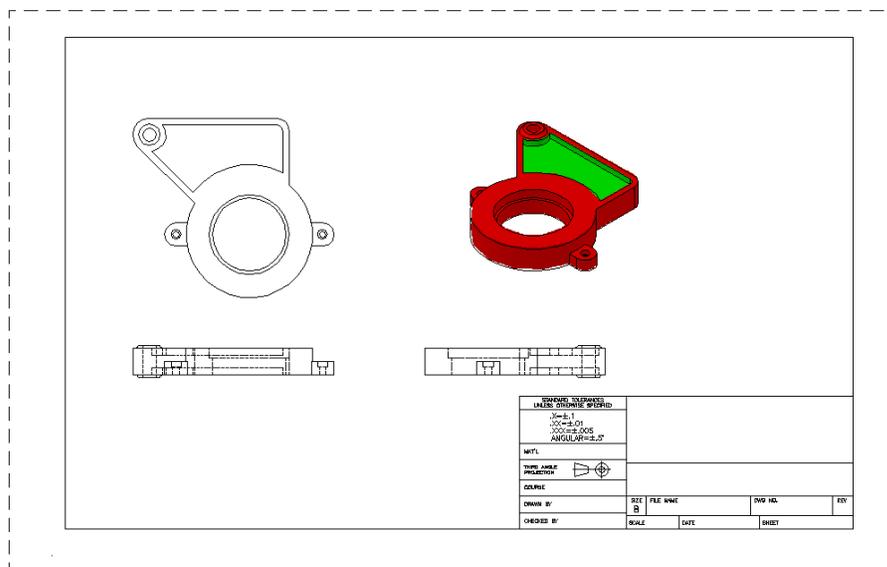
5. Set the orientation to Top and place the top view.
6. Project the front, right side and isometric views.
7. Double click on the isometirc view and change the visibility to Shaded with visible lines.
8. In the 3D model, use sub-object editing to change the diameter of the center bore to **2.00 (radius 1.00). Make changes in the properties palette.**
9. Change the diameter of the filleted boss hole to **.600 (radius 3.00).**
10. Update your drawing.
11. Save the drawing as **Flange_Base_Done.dwg**

Drawing views can be edited after being created. Like other AutoCAD objects, drawing views can be moved or rotated. In addition, certain properties of drawing views, such as the display style and scale, can be modified.

VIEWEDIT

The **VIEWEDIT** command can be used to edit the properties of a drawing view. You can quickly initiate this command by double-clicking on a view. The **Drawing View Editor** contextual ribbon tab appears. The editing options are similar to the options available when you create a base view.

Creating an Assembly drawing and adding parts to the assembly

1. Open the drawing **Fixture Assembly.dwg**
2. Create the base view (front) and the top, right side and isometric views.
3. Scale the isometric view to a larger scale.
4. Use the INSERT command to insert the plug at X=20, Y=8 and Z=7, centered on the top hole as shown. Update the drawing.
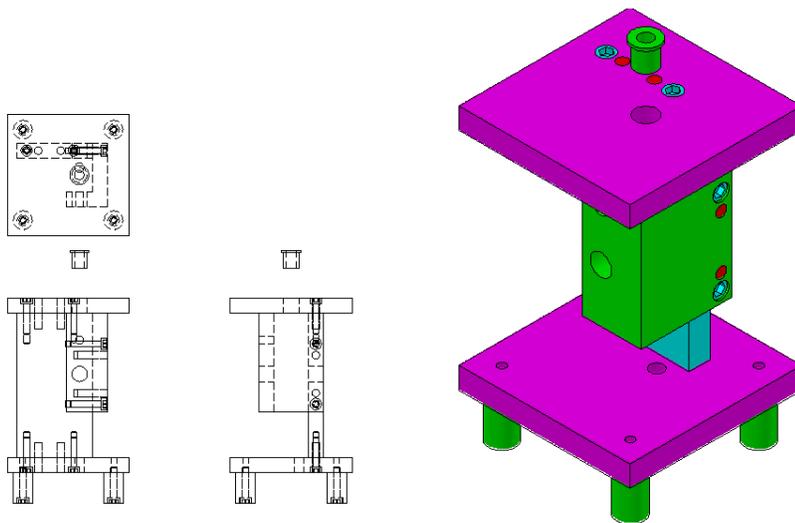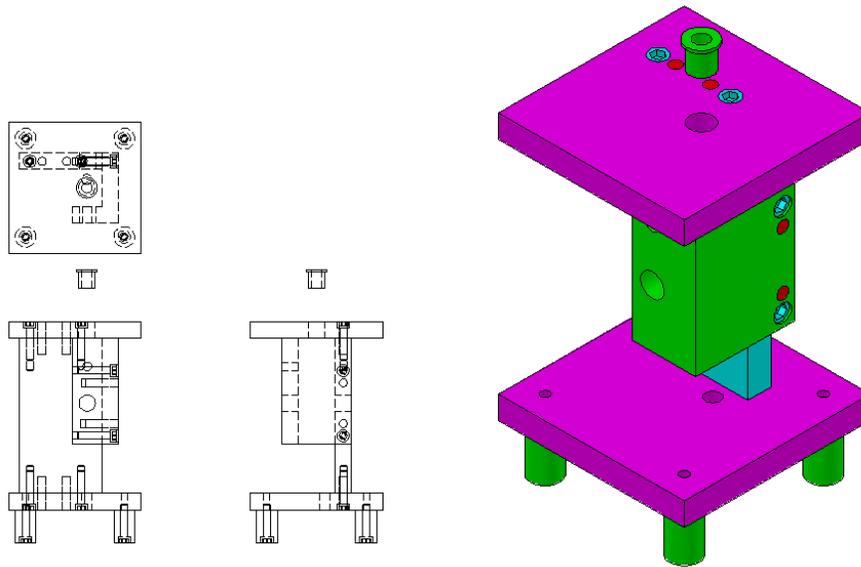
5. Move other parts as needed. Remember to update the drawing!

**Creating Drawing Views from Inventor part files (IPT's)**

Drawing views can be created from sources other than AutoCAD 3D models. You can create views from Autodesk Inventor® part (IPT), assembly (IAM), or presentation (IPN) files. You can also create drawing views from models imported with the **IMPORT** command. Drawing views can also be created from an assembly or subassembly to create an assembly drawing.

CLOSE all AutoCAD drawings.

OPEN: Blank Drawing.dwg.

**OPEN: Coupler3.ipt in Autodesk Inventor.**

1. In AutoCAD, turn off Auto Update.
2. Create the 4 view drawing of the **Coupler3.ipt** file starting with a Base view From Inventor inside AutoCAD.
3. Change the LTSCALE as needed.
4. Save the drawing as **Coupler3.dwg.**
5. Close the drawing.
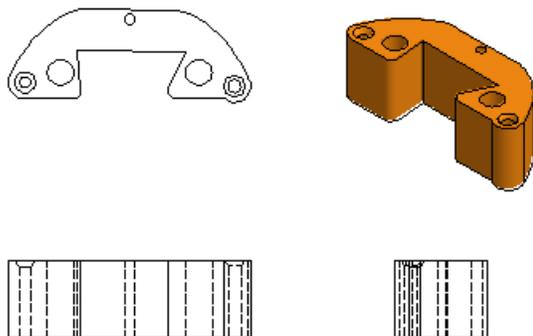
6. In Autodesk Inventor, **edit Holes 6 and 7** (the two larger holes and the one smaller hole) in Autodesk Inventor to a **.03** diameter.

7. Save the **Coupler3.ipt** file.

8. Open the AutoCAD drawing **Coupler3.dwg**. Update the views.

Dimensioning Drawing Views

1. Open the drawing **Flange_Base_Done.dwg or Flange_Base_Finished.dwg.**

2. Type **DIMASSOC** and set the variable to **2**.

3. Add the three dimensions as shown on the ANSI B layout.
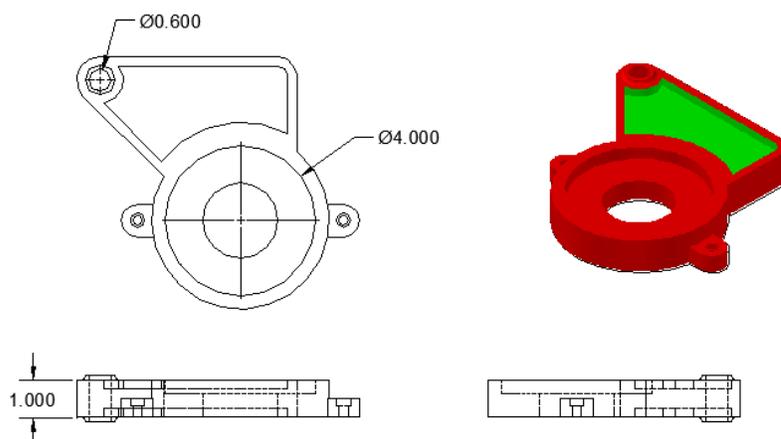
4. Edit the .4000 diameter hole to 3.000.

5. Edit the .600 diameter hole to .500.

6. Edit the 1.00 height to .750.

7. Update the drawing. What happens? Use **DIMREASSOSSOCIATE**.
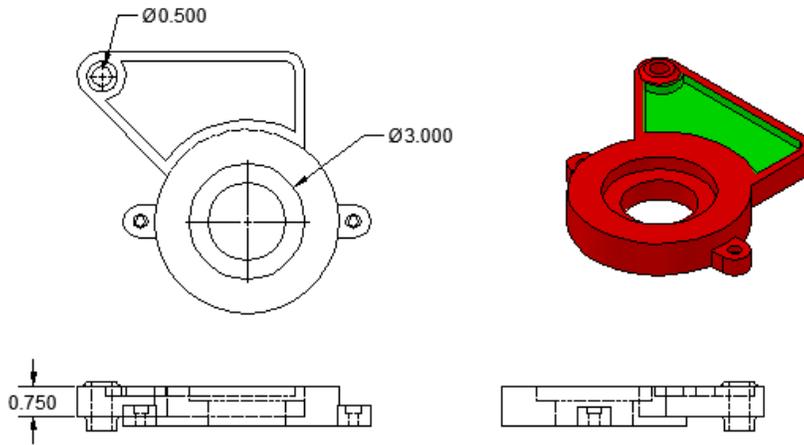
8. Do not save the drawing.

After placing drawing views on a paper space layout, you can dimension each view as needed. When dimensioning drawing views, make sure that the **DIMASSOC** system variable is set to 2 so that associative dimensions are created. Associative dimensions are associated to the dimensions of the model and update when the physical model changes.

If you dimension drawing views with associative dimensions, make changes to the model, and then update the views, the dimensions may become disassociated. In this case, AutoCAD will ask if you want to reassociate the dimensions. Selecting **Run DIMREASSOCIATE** will allow you to pick points in the view to reassociate the dimensions. For more information on the **DIMREASSOCIATE** command and associative dimensioning, refer to *AutoCAD and Its Applications—Basics*.

Using the EXPORTLAYOUT Command

You can use the **EXPORTLAYOUT** command to export a layout containing drawing views into a new drawing (DWG) file. However, this technique will break the associativity between the 3D model and the drawing views. The advantage to using this technique is that the drawing geometry can be edited in the same way as any other AutoCAD 2D geometry in model space. The disadvantage is that the 2D geometry has lost any associativity back to the 3D model.

When using the **EXPORTLAYOUT** command, you save the exported layout as a DWG file. The drawing views become blocks in the new drawing file.

1. Open the drawing **flange.dwg.**
2. Type **EXPORTLAYOUT**.
3. Save the default name to create the new DWG. Open the new DWG when prompted.
4. What are the 4 new AutoCAD objects? Make modifications to the drawing as needed. Dimension in Model Space as needed.
5. Do not save the drawing.