

Autodesk Inventor Tips & Tricks - Placing Features On Cylindrical Shapes

The adoption of parametric, feature-based solid modeling will present many challenges to the new user as they are confronted for the first time with “thinking in 3D.” Some feature creation techniques will come naturally to the users; others will not be so intuitive. One of the first hurdles a user may face, and one of the most common questions in training classes, is that of locating sketched features properly on a curving or cylindrical surface. This whitepaper will address the conceptual thought and techniques necessary to properly place features as desired in these modeling scenarios. Specifically, we will look at placing Holes, sketched Embossed or Engraved features, and Decals.



Work Plane Creation

The first step necessary in creating certain features placed on a cylinder is to create a Work Plane tangent to that cylinder. We will look at two scenarios here.

As a general reminder, recall that all parts, by default, will have 3 pre-defined XY, XZ, and YZ Work Planes accessible under the Origin folder in the browser. Although it is not required, the user may wish to turn on the visibility of any or all of these for reference. This is also true for the default X, Y, and Z axes. This can be accomplished by toggling the Visibility right-click option through the browser (Figure 1)...

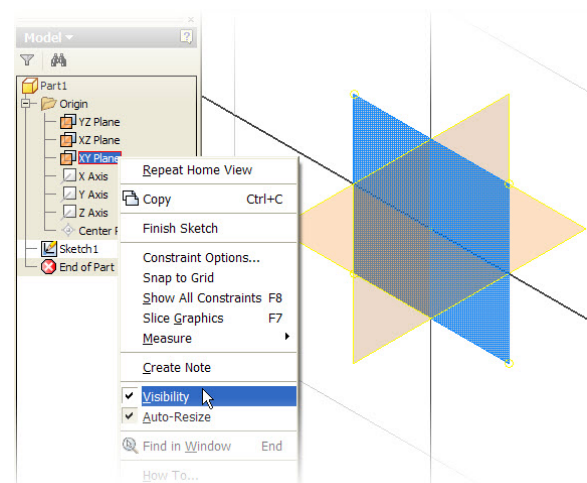


Figure 1

...or by globally enabling Origin Plane visibility through the Object Visibility selection tool on the View tab (Figure 2). (Notice the corresponding “Ctrl-]” keyboard shortcut.)

TIP: This is also a convenient mechanism for globally controlling the visibility of several object types in Inventor.

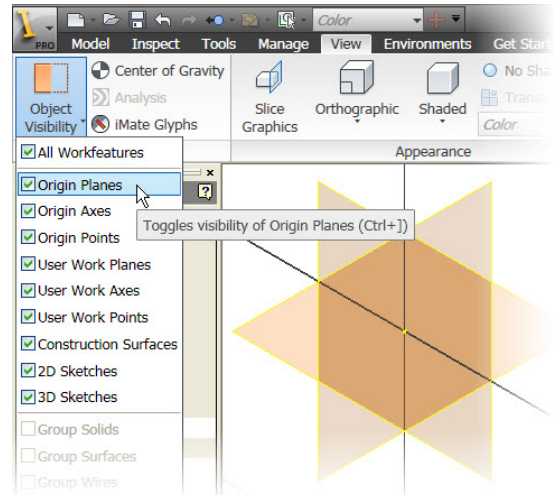




Figure 2

SCENARIO 1

Our first scenario will utilize these default work features. In this example it is highly recommended to start our feature on a projected Center Point so we are assured the default planes bisect our part in a known location, and we remain coaxial with the corresponding axis. (**TIP:** To accomplish this, right-click anywhere in the sketch background, choose the  Project Geometry tool and then select the Center Point either in the graphic screen or the  Center Point browser node.) Our resultant feature will appear as in Figure 3.

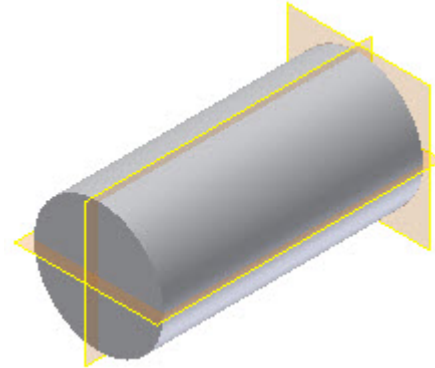


Figure 3

To create the tangent Work Plane: From the Work Features pane on the Model tab, select the Plane tool as shown in Figure 4. Begin this operation by selecting the existing Work Plane, in the graphic screen or in the browser, and then hover the mouse near the appropriate side of the cylinder (Figure 5). When the preview appears to your satisfaction, click the cylinder and the new Work Plane is created (Figure 6). Given the parametric nature of work features, this plane will remain tangent to the cylinder even if the diameter is changed.

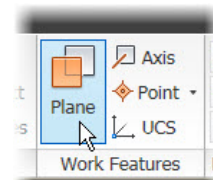


Figure 4

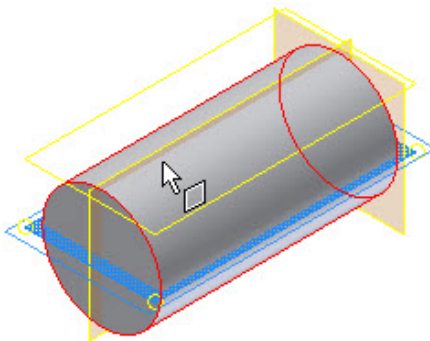


Figure 5

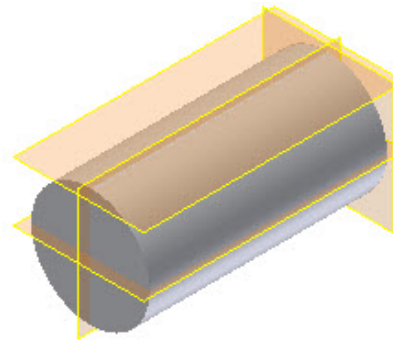


Figure 6

SCENARIO 2

In the first scenario we employed the Origin planes that exist by default in all Inventor part files. There may be times you need to locate a Work Plane elsewhere in the model. If that Work Plane is to be, and is to remain, parallel to one of the default Origin planes, the method illustrated in Scenario 1 may also be proper and viable. However, an alternative method may be necessary, and it has the benefit of increased flexibility in that the Work Plane (and therefore, its dependent features) may later be rotated about the cylinder axis.

In this case a Work Axis is placed by initiating the Axis tool (refer to Figure 4) and selecting the cylindrical object as shown in Figure 7. Then start the Plane command and select the axis created in the previous step. The assumption is made that the new plane will be created through that axis and at an angle relative to an existing planar object. At this point select a model face or Work Plane and enter the desired angle.

TIP: Even if there are seemingly no appropriate planar faces or user-defined Work Planes to select at this point, remember that the Origin planes can be utilized. See Figure 8.

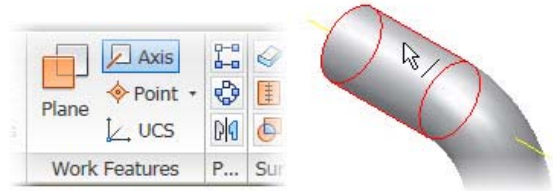


Figure 7

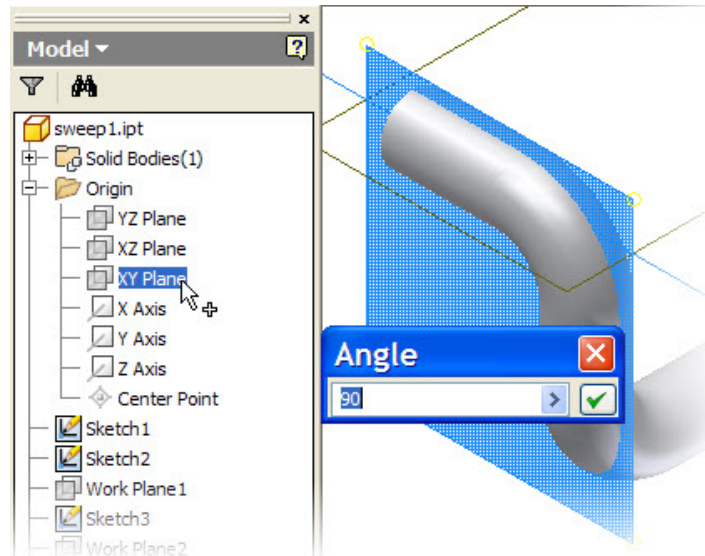



Figure 8

Now that this Work Plane is placed, another can be defined parallel to it and tangent to the cylinder using the same process as in Scenario 1. By basing this Work Plane on a relative angle, it (and any dependent planes and features) can now be rotated – simply double-click on the Work Plane, change the angle accordingly (see Figure 9), and then select the Update button  on the Quick Access toolbar. Therefore, there are times that the creation of another Work Plane in this fashion can be useful in the first scenario cited as well.

TIP: Work Planes, like any part feature, can be renamed in the browser by slowly double-clicking and entering a more descriptive name.

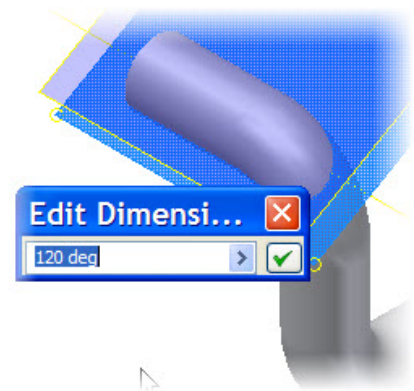
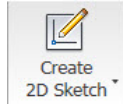


Figure 9

Placing Features On Cylindrical Shapes - Holes

With our new Work Plane defined, we'll start by activating it as a sketch plane.



As before, it may be desirable to use the Project Geometry function to provide an accurate positioning reference, as shown in Figure 10. In this case an axis was projected but conceivably a vertical plane could be used if positioned correctly as in Figure 3.

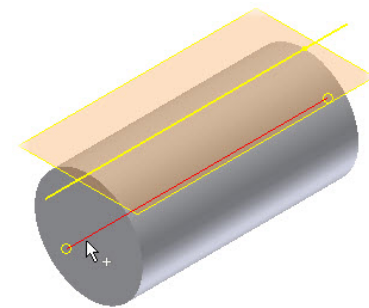


Figure 10

Select the **Point** command from the Draw panel in the Sketch environment to place the desired hole center(s). Ensure that the points snap to the projected line, if that is the intent, thereby inferring a Coincident constraint (Figure 11). The hole centers may be further dimensioned and constrained as desired (Figure 12).

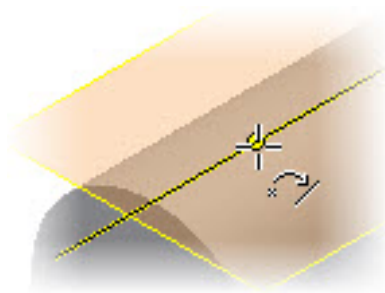


Figure 11

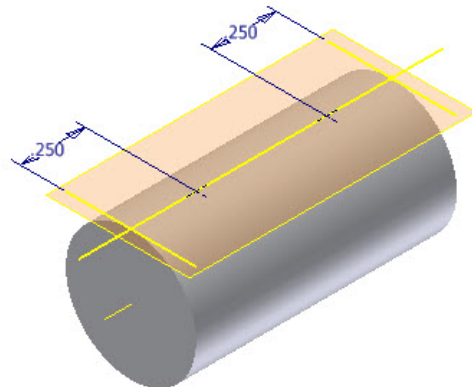


Figure 12

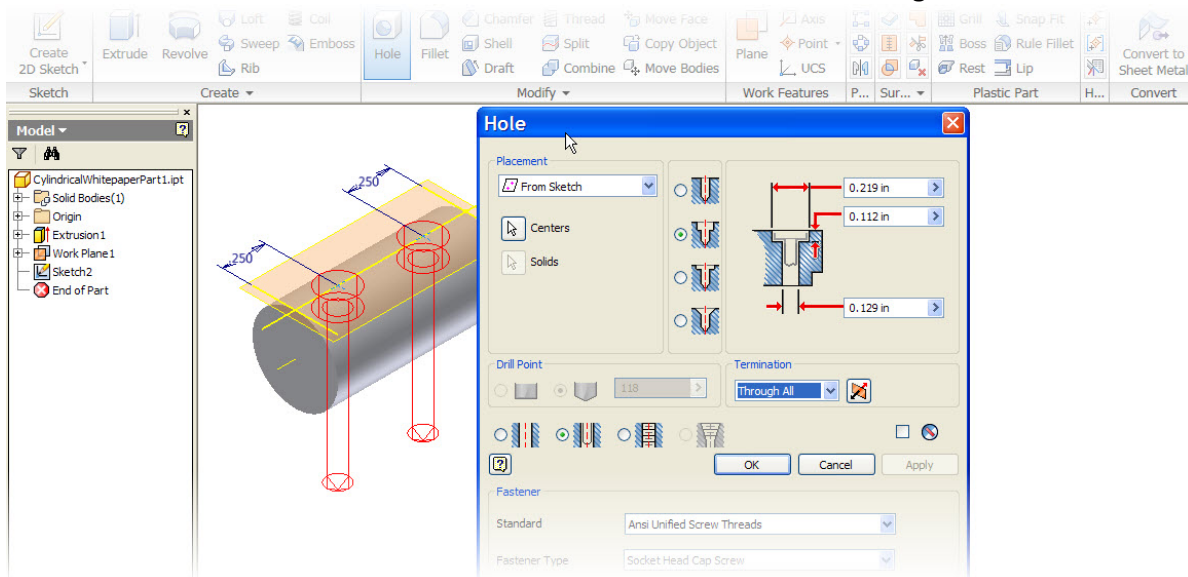


Figure 13

When the Hole command is invoked (Figure 13), Inventor will assume that the unconsumed Point objects are to serve as hole centers and will default to the “From Sketch” placement method. Enter the desired Hole options and select OK to complete the command. The final result is shown in Figure 14.

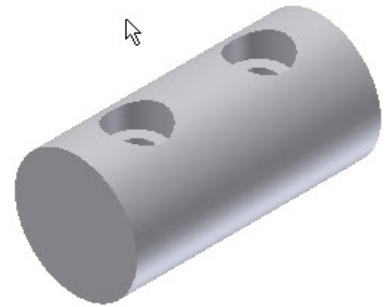


Figure 14

Placing Features On Cylindrical Shapes – Engraving & Embossing

Inventor has the ability to utilize sketch geometry to create Engraved and Embossed features. While not unique to cylindrical shapes, both allow the user to, in essence, “wrap” a planar sketched feature onto a curved surface.

ENGRAVE

This process begins as before, with the creation and activation of an appropriate sketch plane. To be sure, it is **not** mandatory to create a tangent Work Plane for this purpose, as any in the proper orientation will suffice. However, there may be times where it is preferable for location and visualization purposes. Figure 15 illustrates the sketch being used in this example. For clarity, Inventor’s “Look At” command was used to obtain a view normal to the sketch.

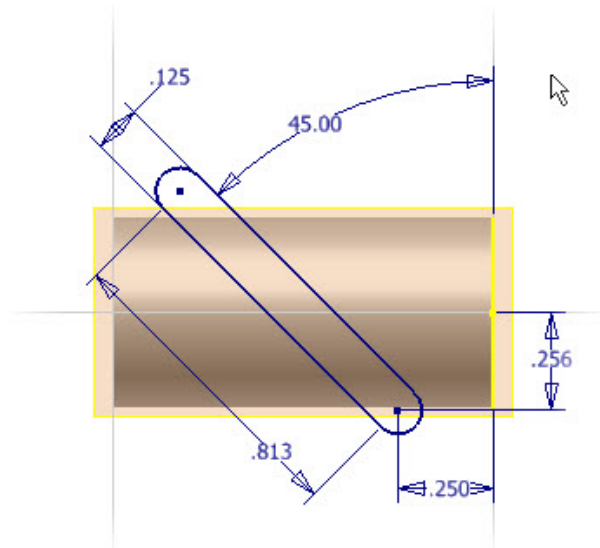



Figure 15

To create the feature we select the  **Emboss** tool from the Create panel on Inventor’s Model tab (Figure 16). Within this dialog we have a choice of Emboss From Face, Engrave From Face (currently selected), and Emboss/Engrave From Plane. Direction can be controlled and the feature Depth can be entered as well.

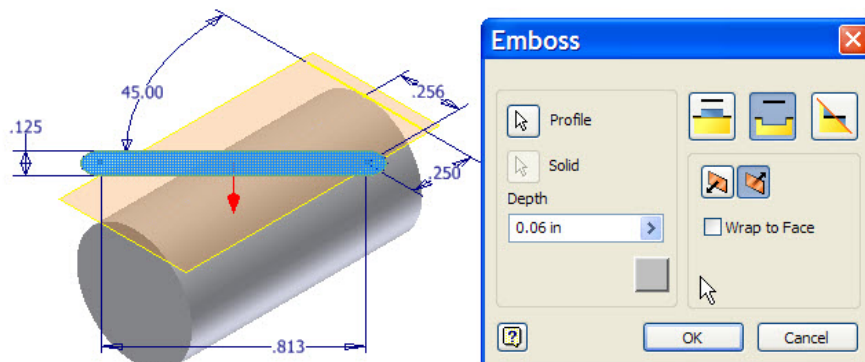


Figure 16

In addition, a color may be selected for the feature by selecting the color swatch below the Depth field. Lastly, the “Wrap to Face” option may be selected to choose between planar or cylindrical mapping (Figure 17) of the feature onto the surface; if selected, the user will be prompted to select a surface.

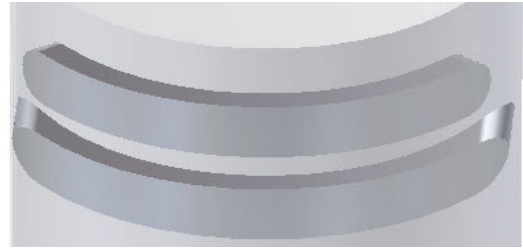


Figure 17 – Cylindrical (Top) vs. Planar Mapping with Identical Sketches

The completed example may be seen in Figure 18.

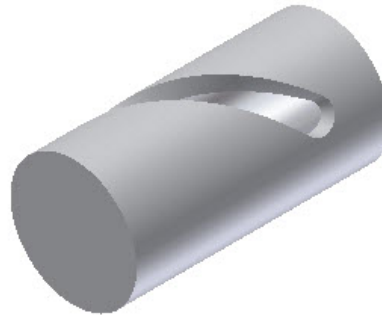


Figure 18

EMBOSS

As mentioned, the Emboss command is capable of creating raised features as well. In this example we will also show that text may be embossed onto a part. We start by creating a sketch as per the previous example. Select the **A Text** tool from the Draw panel in the Sketch environment and place the text in the appropriate position.

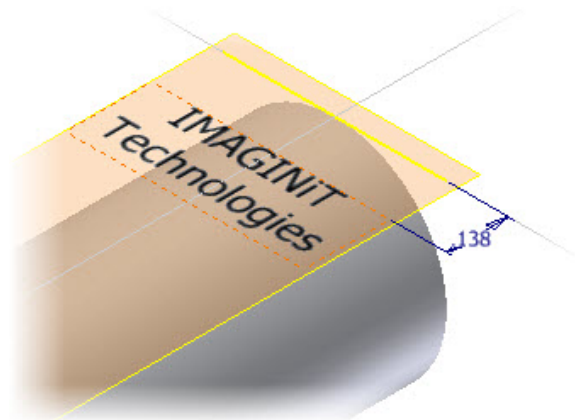


Figure 19

(**TIP:** If the text comes in for some reason on the “wrong side” of the Work Plane, simply select the plane, right-click, and select the “Flip Normal” option). Once placed, text behaves as any other sketch object in that it can be moved, rotated, and dimensioned as necessary. See Figure 19.

Once the text is positioned, and the sketch completed, start the Emboss command. This time we will select the “Emboss from Face” method, enter our height and choose a feature color if desired. Unless planar mapping is desired, make sure to check the “Wrap to Face” option and then select the cylindrical face. The dialog box is shown in Figure 20.

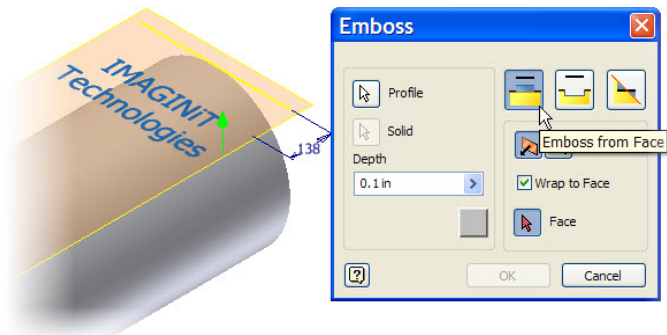


Figure 20

The completed Embossed Text feature is illustrated in Figure 21. As a sketch-based object, both the feature and sketch can be independently edited after creation if needed.

TIP: Embossed and Engraved features can add a fair bit of complexity to an otherwise simple part. It may be beneficial, as with other complex features in general, to add them as late in the design as possible.



Figure 21

Placing Features On Cylindrical Shapes - Decals

Occasionally we may desire to place an image on our model to represent a label, decal, or texture. We will discover that we have the ability to place .bmp (or other bitmap graphic formats), Microsoft Word .doc, and Microsoft Excel .xls files as images onto our model.


This process also starts in a similar fashion to the Emboss command in that a Work Plane is created or selected but it need not be tangent to the cylinder. Once we have activated it as a sketch plane, we select the  Image tool from the Insert panel in the Sketch environment, and browse to the desired image file. We may dimension and constrain this image as any sketch object (Figure 22). We may also resize the image while keeping the original aspect ratio.



Figure 22

TIP: If the image selected does not show properly (Figure 23) or generates a warning message (Figure 24), either take the steps necessary to associate the file format with the Microsoft Paint application or choose another graphic format



Figure 23

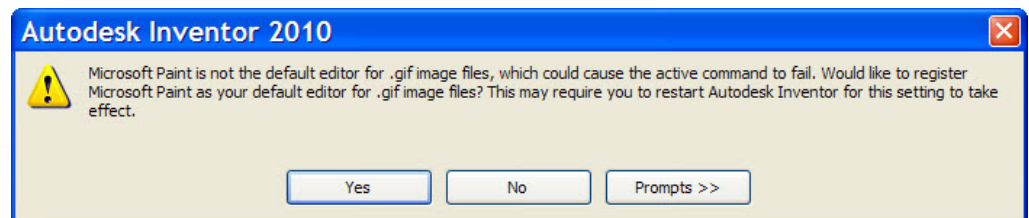



Figure 24

Once the image is placed, and the sketch completed, start the  Decal command by expanding the Create panel on the Model tab (see Figure 25). The user will be prompted to select the placed image, and the face onto which it will be projected.

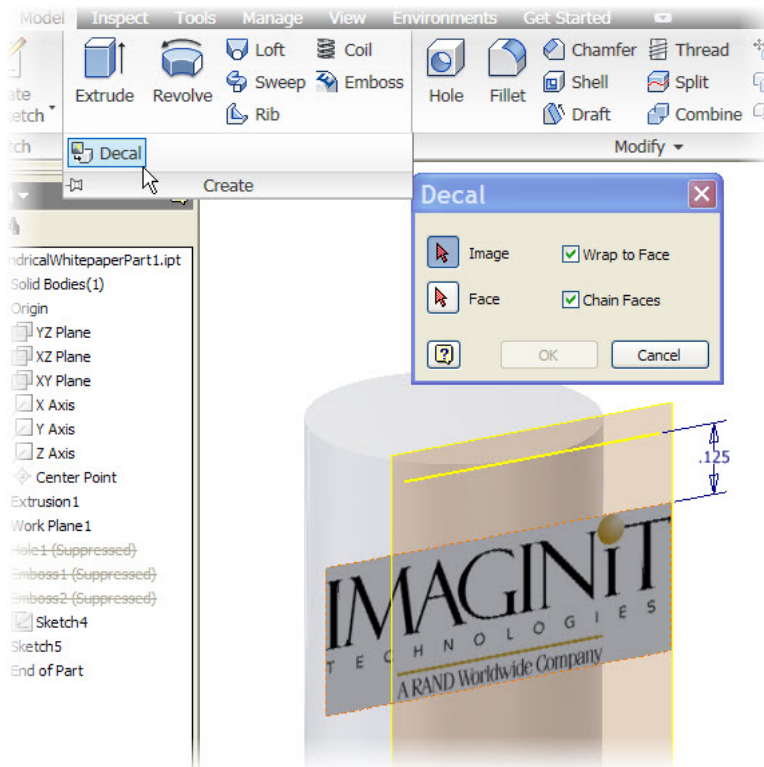


Figure 25

Wrap to Face – This option controls whether the image wraps (cylindrical mapping) to the curved face or is projected in a linear fashion (refer to example shown in Figure 17).

Chain Faces – This will serve to apply the decal to adjacent faces as well as the one selected.

The completed decal is shown in Figure 28. **TIP:** Decals will also show in any shaded drawing view of the model.



Figure 26 and Figure 27 - with and without “Chain Faces” selected.



Figure 28