

REPORT NO:

REVISION:

A

---

DRAWING, DRAFTING, AND MODELING STANDARDS AND PRACTICES - SHEET METAL

REV A

Engineering Manager \_\_\_\_\_ Date: 5/13/10

REVISION SHEET

<b>Revision Letter</b>	<b>Description</b>	<b>Approval Initials / Date</b>
A	INITIAL RELEASE	

**TABLE OF CONTENTS**

---

<b>REPORT NUMBER 10-0006</b> .....	<b>ERROR! BOOKMARK NOT DEFINED.</b>
<b>1. INTRODUCTION</b> .....	<b>4</b>
<b>2. PURPOSE</b> .....	<b>4</b>
<b>3. REFERENCE &amp; RELATED PROCEDURES</b> .....	<b>4</b>
<b>4. PART MODELING</b> .....	<b>5</b>
4.1 OPENING A NEW SHEET METAL PART.....	5
4.2 SETTING UP SHEET METAL STYLES .....	5
4.3 PART SKETCHING .....	6
4.4 CREATING FEATURES.....	7
<b>5. REQUIREMENTS FOR SHEET METAL PART DRAWINGS</b> .....	<b>9</b>
5.1 REQUIRED VIEWS .....	10
5.2 DIMENSIONING SHEET METAL PARTS .....	14
5.3 HOLE DIMENSIONS.....	16
5.4 CREATING A HOLE TABLE.....	19

## 1. INTRODUCTION

This document defines standard procedures and practices for modeling and drafting sheet metal parts. It is a supplement to document 10-0005 to provide further guidelines with respect to sheet metal modeling and drafting.

## 2. PURPOSE

Blah recognizes that the quality and consistency of 3D models and related documentation can directly impact the cost and quality of the finished product. Current drafting methods defined by the American Society of Mechanical Engineers (ASME) can be utilized to improve product quality through the reduction of ambiguity. By following standardized practices, requirements are fully defined without room for personal interpretation.

## 3. REFERENCE & RELATED PROCEDURES

Document Number	Title
ASME Y14.3M-1994	Multiview and Sectional View Drawings
ASME Y14.5-2009	Dimension and Tolerancing
EP-001	Medical Mechanical Drafting & Drawing Standards
10-0005	Drawing & Drafting Standards and Practices
10-0047	Bonding Point Procedure
10-1006	EPS-121 Implementation in a Drawing

## Table of Figures

Figure 4-1: New File Dialogue .....	5
Figure 4-2: Sheet Metal Styles .....	6
Figure 4-3: Good and Bad Constraints .....	6
Figure 4-4: Fully Constrained Sketch .....	7
Figure 4-5: Adaptive Feature and Sketch.....	7
Figure 4-6: Types of Features.....	8
Figure 4-7: Feature Browser Examples.....	9
Figure 5-1: Sheet Metal Box.....	10
Figure 5-2: Required Isometric View .....	11
Figure 5-3: Detail Views .....	12
Figure 5-4: Section Views.....	12
Figure 5-5: Auxiliary Views.....	13
Figure 5-6: Rotated Auxiliary Views.....	13
Figure 5-7: View Cross Reference .....	14
Figure 5-8: Dimension from Hard Edge .....	15

Figure 5-9: Dimension from Outermost Edge..... 15

Figure 5-10: Dimension Using Detail View..... 16

Figure 5-11: Hole to Hole Dimension on Machined Part..... 16

Figure 5-12: Non-preferred Hole Dimensioning for Sheet Metal..... 17

Figure 5-13: Preparing for Tolerance Stack Up..... 18

Figure 5-14: Accurate Hole Dimensioning ..... 18

Figure 5-15: Holes on Different Planes ..... 19

Figure 5-16: Ordinate Dimensions Used for Many Holes ..... 20

Figure 5-17: Hole Table..... 21

Figure 5-18 Create Hole Table ..... 22

Figure 5-19: Placing Origin Point ..... 22

Figure 5-20: Setting Up Hole Table ..... 23

Figure 5-21: Splitting the Table..... 24

**4. PART MODELING**

4.1 OPENING A NEW SHEET METAL PART

Click the “new” icon, or go to the file menu and select new. You will be presented with the Blah template dialogue (see Figure 4-1). Select the “BLAH\_SHEET METAL.ipt” option :



Figure 4-1: New File Dialogue

4.2 SETTING UP SHEET METAL STYLES

Although the new part opens in the sketch environment, it is advisable to close the sketch and set the part thickness to the appropriate gauge (see 10-0005 for the thicknesses commonly used by Blah), and the material that the part will be made from. All other options can be left in their default state as they are already dictated by the template.

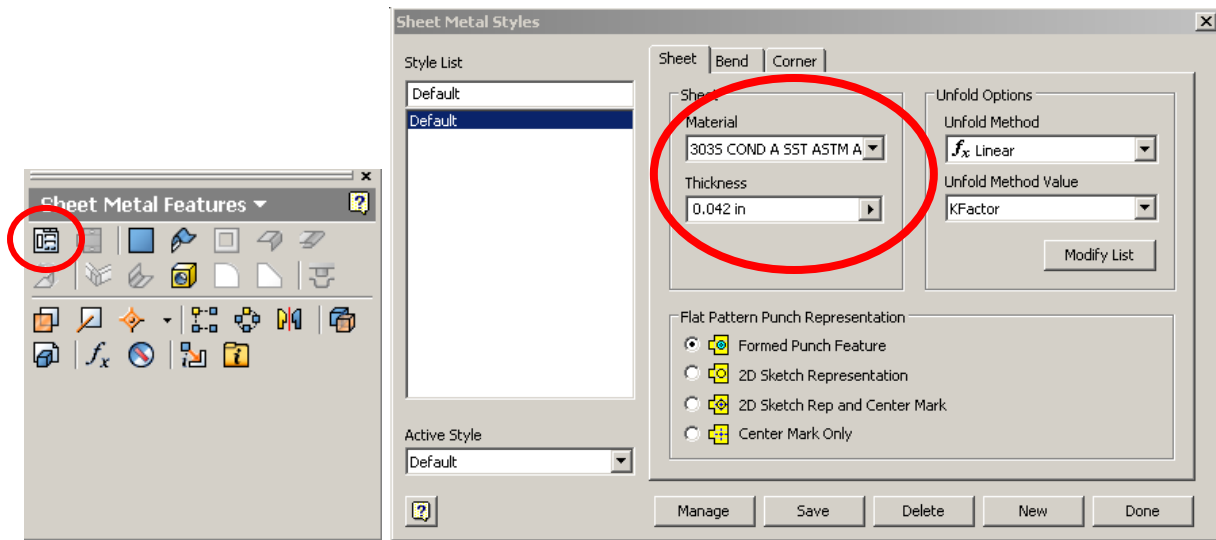


Figure 4-2: Sheet Metal Styles

#### 4.3 PART SKETCHING

Part sketches should be dimensioned and constrained in a manner that allows them to be easily modified without deforming the part and causing rebuild errors. Design intent should be considered when creating part sketches.

- The critical features of a part, such as bolt holes, should be dimensioned or constrained so that they will not move when another feature dimension is changed. Example: Depending on the design intent, you may want to constrain the holes to the centerlines of a part, rather than placing a linear dimension from the edge. That way, if the height or width of the part changes, the holes will remain centered, as intended.

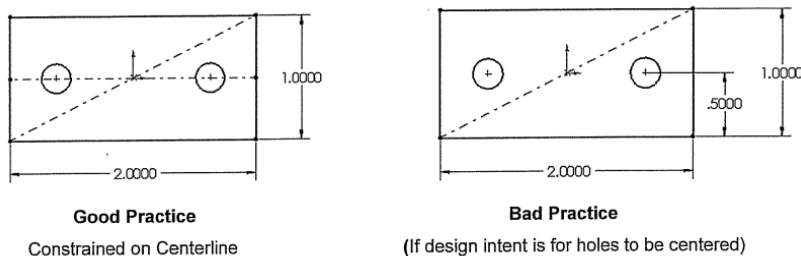


Figure 4-3: Good and Bad Constraints

- All completed sketches should be fully defined and the base feature sketch must be constrained to the model's origin. A fully defined sketch will be notated in the bottom right of the screen when in the sketch environment.

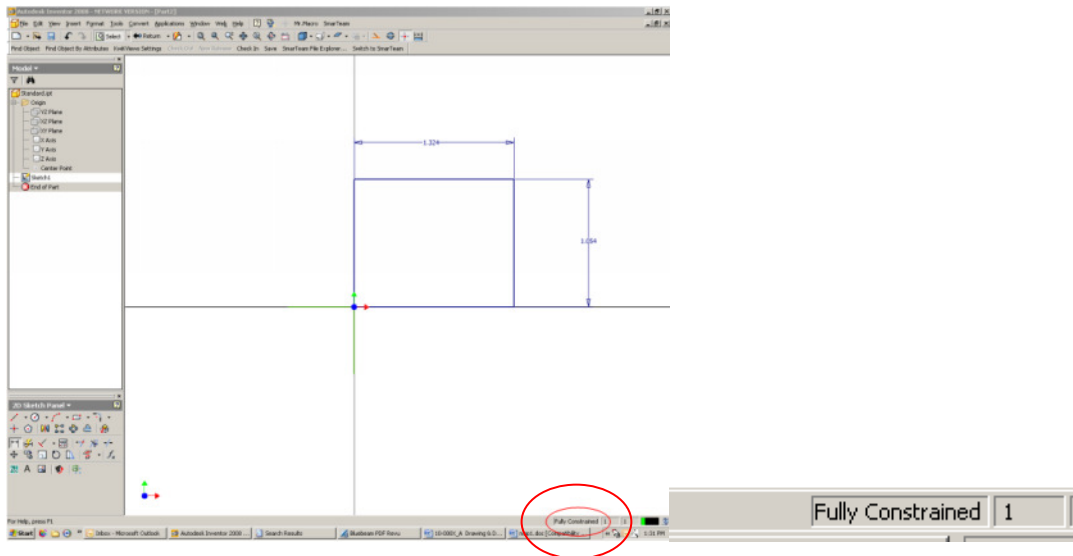


Figure 4-4: Fully Constrained Sketch

- If a sketch is created in the context of an assembly, this will cause the part to be “adaptive” (see Figure 4-5.) You must then go back into the part, turn off the adaptivity of the sketch, delete the constraints to the assembly, and dimension the sketch as usual. When a sketch remains tied to an assembly, the part can change unintentionally when the components in the assembly move.

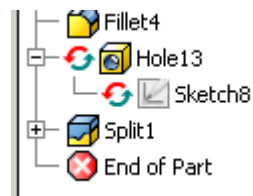


Figure 4-5: Adaptive Feature and Sketch

#### 4.4 CREATING FEATURES

Once a sketch is complete, you will be able to select the features. There are two types to choose from, sheet metal and part features:

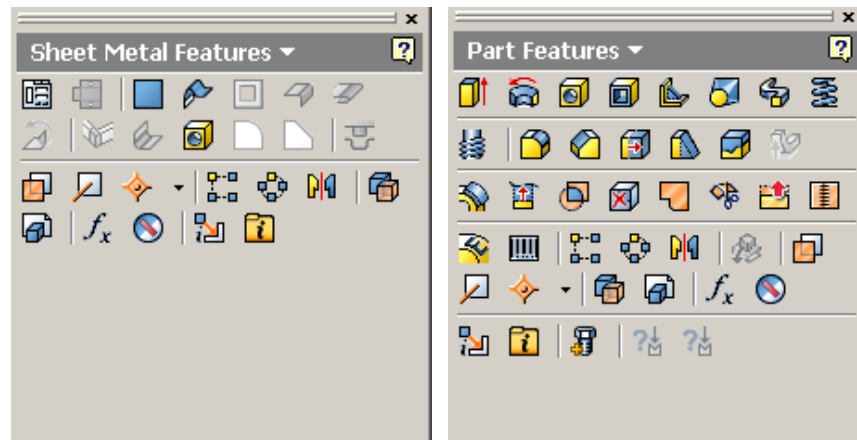
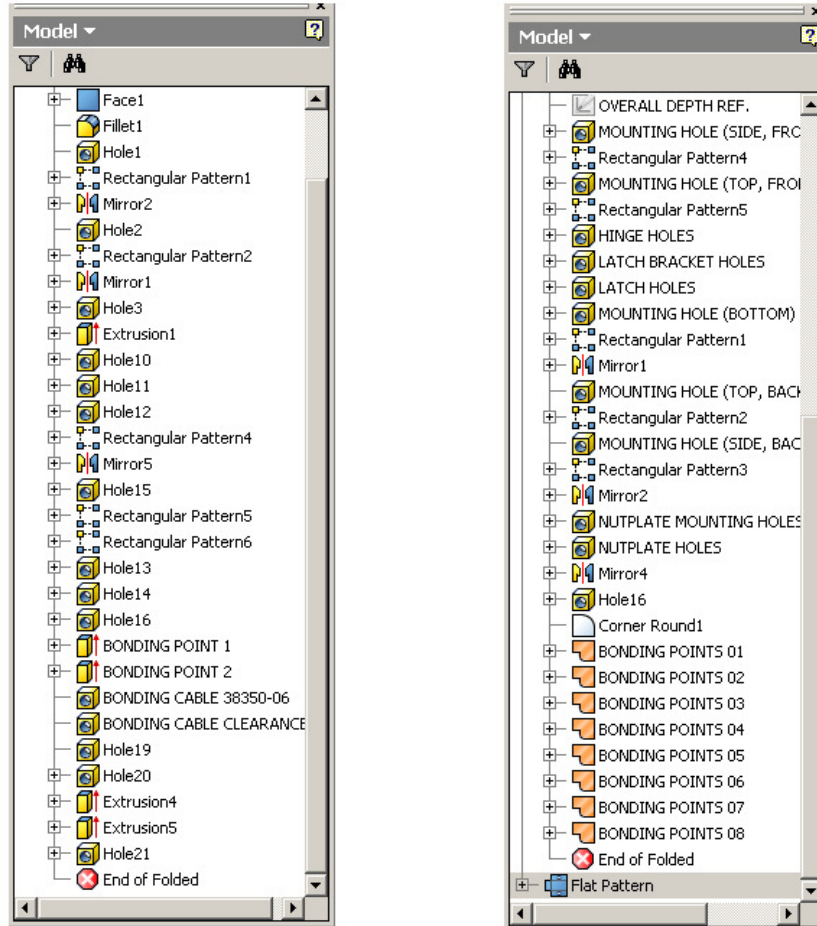


Figure 4-6: Types of Features

Although many features between the two are similar, it is preferred that you use all sheet metal features if possible. The only feature that you should need from the “parts features” list is the boundary patch, which you will use for bonding points (see 10-0047 for bonding point procedure). When placing features, it is advisable to make them in logical order.

All bonding point features should be placed at the end of the part, and not intermittently between manufacturing features. As you can see in the “bad practice” browser (see Figure 4-7), some of the bonding point features are placed before hole features. This is unacceptable because, if the hole features created after the bonding point feature are dimensioned to the holes that contain the bonding points, then they are constrained to the bonding point and not the hole. If the bonding point gets deleted or moved, the dependent feature now loses its constraint creating a bad sketch that may cause issues if the part needs to be updated or changed.





Bad Practice

Good Practice

Figure 4-7: Feature Browser Examples

## 5. REQUIREMENTS FOR SHEET METAL PART DRAWINGS

Sheet metal parts are typically created from material less than .250 inches thick, and are formed by folding the material at specific locations with a controlled bend radius. See document 10-0005, *Blah Drawing & Drafting Standards and Practices*, for the proper setup of a sheet metal drawing. For door panels, front panels, interior panels, and racks see document 10-1006 for information on adding a note regarding the implementation EPS-121.

**NOTE: Although notes on each drawing will differ, the first note on ALL sheet metal drawings that contain bends should be as follows: MINIMUM BEND RELIEFS ALLOWED**

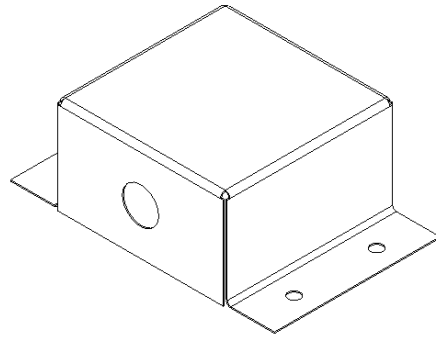


Figure 5-1: Sheet Metal Box

### 5.1 REQUIRED VIEWS

Formed sheet metal parts must be fully defined as a finished good in the completed (folded) state per section; however, sheet metal parts also require specific data to be shown on the drawing to ensure proper manufacturing and inspection.

All part drawings require the minimum number of views required to fully define the part; however, never less than two orthographic views should be shown, unless it is a flat part, in which case a second view is not required since the material thickness is called out in the title block.

All part drawings will show an isometric view on sheet one to help visualize the part. Isometric views shall be shown with tangent edges visible unless doing so creates too much clutter when the drawing is printed. It is important that the isometric view is relative to one of the views on the first page of the drawing, preferably the view which all other views are derived from, or the view with the most detail. Vendors use this view to verify what the part should look like, and so it should also be in the correct orientation. If necessary, to show details of a complex part, it is acceptable to add more iso views, but there should ALWAYS be an isometric view that is derived directly from a view on the drawing (see Figure 5-2).



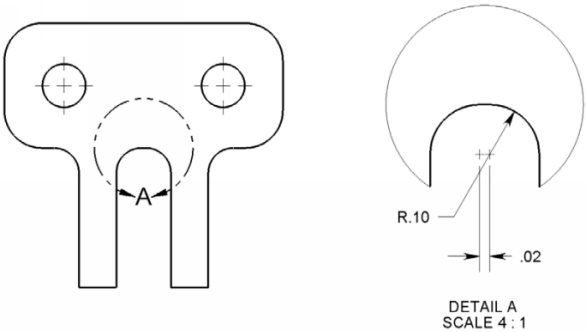


Figure 5-3: Detail Views

Section views shall be located with a phantom line, and identified with an alphabetical identifier at each end of the section line. The section view itself shall be identified by its label and the view scale if different than that of the page scale. The section view shall be projected correctly, *i.e.*, “behind the arrows”. See Figure 5-4 as an example.

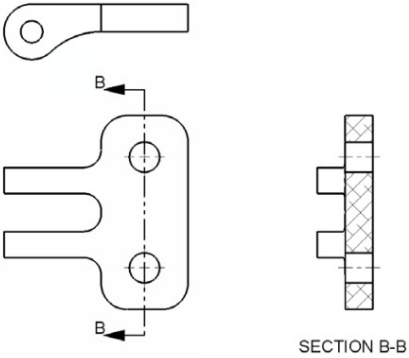


Figure 5-4: Section Views

Auxiliary views shall be treated in the same manner as section views; however, auxiliary views may have their alignment broken if space is limited or clarity of the drawing is enhanced.

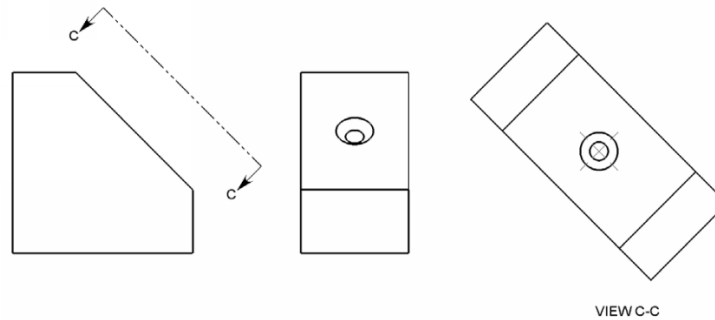


Figure 5-5: Auxiliary Views

If space is limited or if clarity is greatly enhanced, it is also permissible to rotate an auxiliary view; however, the angle and direction of rotation must be noted with the view label as shown in Figure 5-6. The use of the abbreviations “CW” and “CCW” (for clockwise and counter clockwise, respectively) may be used to identify direction of rotation.

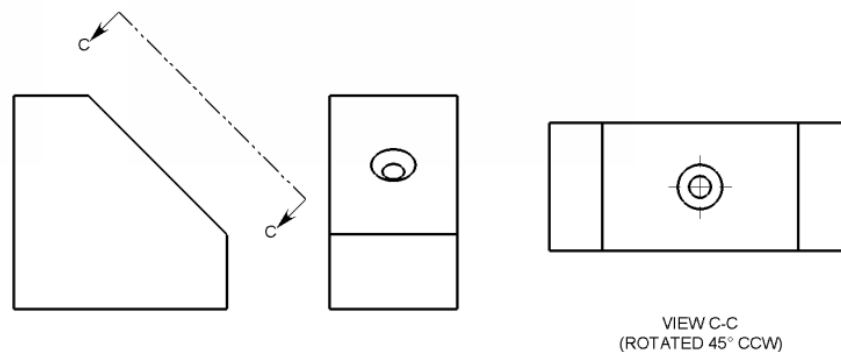


Figure 5-6: Rotated Auxiliary Views

**NOTE: All detail, section, and auxiliary views shall have a unique alphabetical identifier. The letters “I”, “O”, “Q”, “S”, and “X” shall not be used as view identifiers.**

If space is limited, a detail, section, or auxiliary view may be moved to another sheet. In this instance, a cross reference shall be provided to guide the user of the drawing between the views. This cross reference shall indicate on the original view where the resultant view is located by sheet and zone, and the resultant view shall indicate where the view was created by sheet and zone as shown in Figure 5-7.

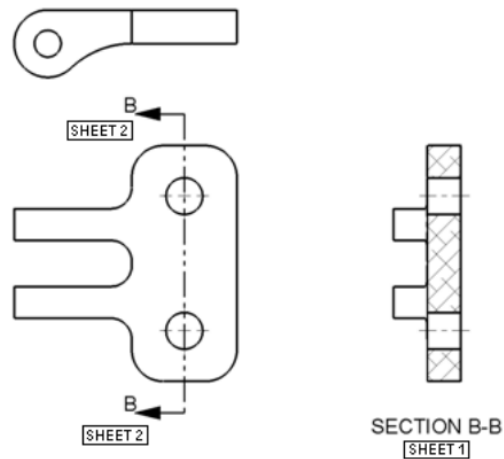


Figure 5-7: View Cross Reference

## 5.2 DIMENSIONING SHEET METAL PARTS

Unlike most machined parts, which are solid blocks that are machined into their required shapes, sheet metal starts as the name implies, as a sheet of metal that will need to be bent to achieve the desired shape. To that end, extra consideration must be taken into account when dimensioning a sheet metal part. Since most sheet metal features are cut before the part is actually folded, it is important to consider this when calling out crucial features such as hole locations.

When dimensioning features on sheet metal, try to keep the following three rules in mind (in order of importance): dimension from (1) a hard edge (edge that has not been bent or will not be bent), (2) the outermost edge, and (3) in a detail view.

The first rule of dimensioning from a hard edge is the most efficient rule because no matter what form of the part someone is looking at, be it folded or flat, the dimension will be exactly the same, allowing you to use tighter dimensions for crucial feature locations. As you can see in Figure 5-8, the hole is dimensioned from the edge of the flange, so even if the part is unfolded, the dimension will still be valid.

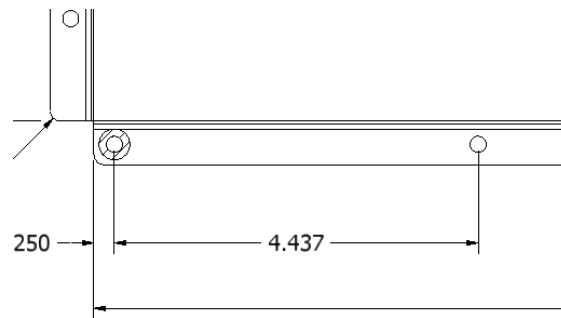


Figure 5-8: Dimension from Hard Edge

The second rule of dimensioning from the outer most edge is often used because many times there is not a hard edge that would make sense to use for a point of origin available. Although there are times when you have no choice but to use a three place dimension callout from the outer most edge, keep in mind that many times there may be more than one bend involved. You now must take into account, not only the tolerance of the flange size itself, but also the bend angle. Therefore, it is recommended to use a looser (two place) dimension if possible. Once the initial feature location is defined, then you can tighten the tolerance between features to three places as seen in Figure 5-9.

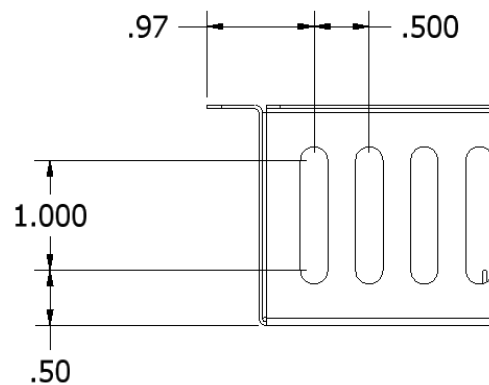


Figure 5-9: Dimension from Outermost Edge

The last rule is creating a detail view to show the initial dimension of a feature. It should be reserved for drawings where there is plenty of free space to add the extra views. Although it is allowable to add detail views on different pages of the drawing, it may end up causing unnecessary confusion, so if you are unable to add a detail view on the same page that the detail is taken from, then one of the first two rules should be used. However, if there is room, a

detail view is a great way to get a dimension from a hard edge that may not be visible in a regular orthographic view due to line weights, see Figure 5-10.

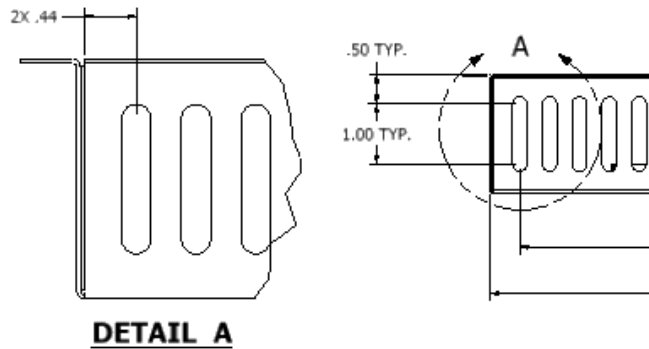


Figure 5-10: Dimension Using Detail View

### 5.3 HOLE DIMENSIONS

Dimensioning from hole to hole on sheet metal is different than dimensioning holes on a machined part. Dimensioning holes on the face of a machined part is easier to dimension from hole center to hole center since it is all one plane on a solid part that is easily held in a vice. However, the same is not true with sheet metal. When looking at a model or drawing, the holes appear to be on the same plane. In reality, each different flange becomes a separate plane, and therefore should be dimensioned accordingly.

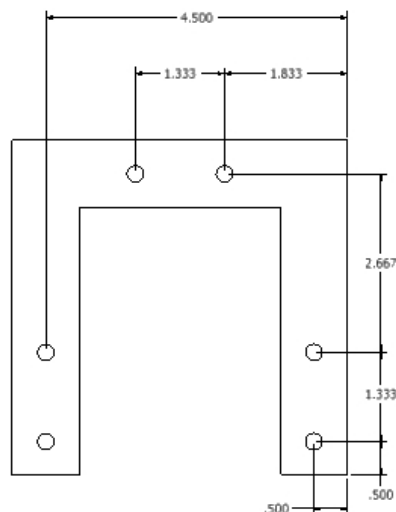


Figure 5-11: Hole to Hole Dimension on Machined Part



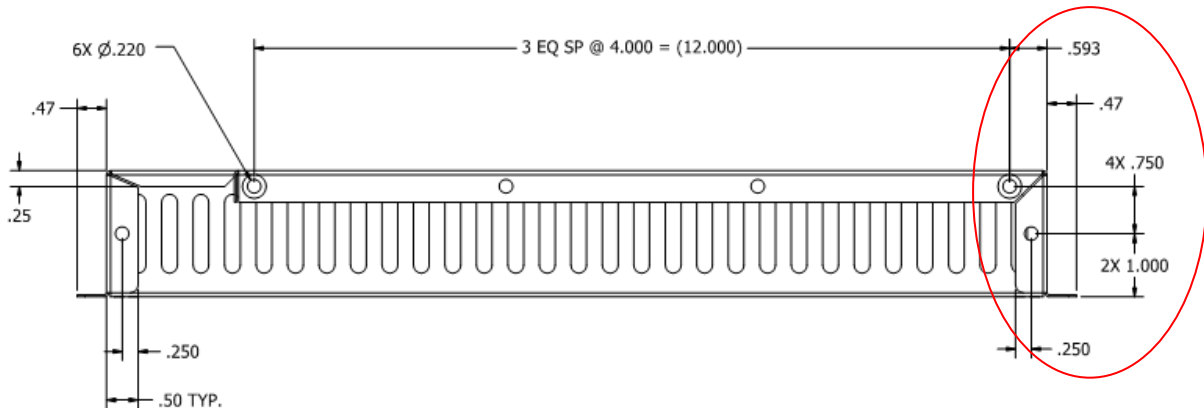


Figure 5-12: Non-preferred Hole Dimensioning for Sheet Metal

As seen in Figure 5-12, two of the dimensions, although possible to make, and may be necessary since these holes will likely go to a mating part, will be much harder to achieve on a sheet metal part. The first reason is that the “2X 1.000” dimension is coming from an edge that has been bent twice, which will be a large amount of tolerance to take into consideration. Secondly, the “4X .750” dimension is between holes that are on two different flanges which also may not be exactly in line after they are bent.

The best way to dimension holes that will mate with holes on another part will be to allow room for the least material condition, meaning that the part that the fastener will be going through should have a larger diameter hole than the part which the fastener will be securing to. As shown in Figure 5-13, the part on the left will be fastened to the part on the right; therefore, the thru holes are larger in diameter ( $\phi.120$ ) in order to account for the  $\pm$  tolerance that can occur. The holes that are on the part on the right will be smaller ( $\phi.096$ ) so the rivet will fit properly to secure the two parts together.

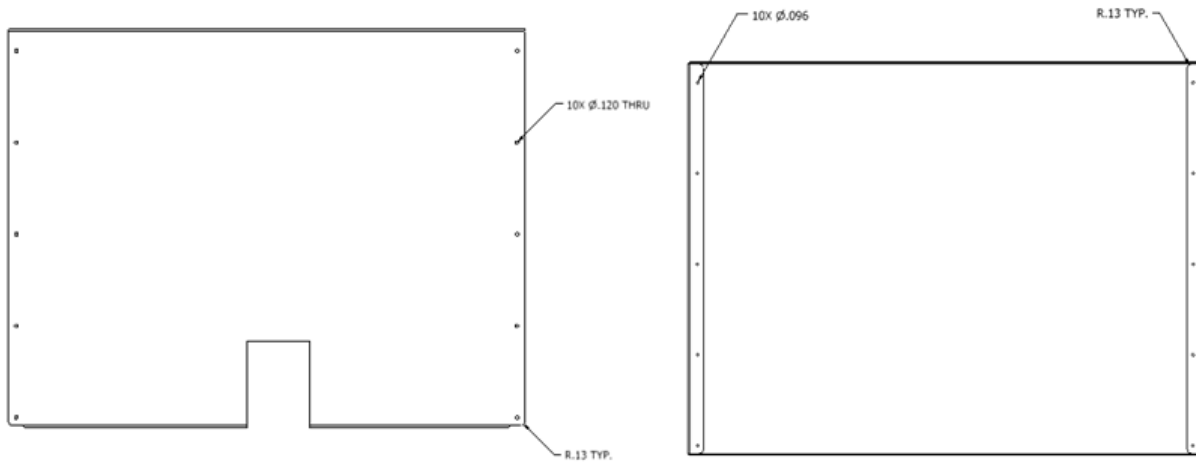


Figure 5-13: Preparing for Tolerance Stack Up

Since the larger hole diameter accounts for the tolerance, as opposed to dimensioning from hole center to hole center, the most accurate way to dimension holes on separate flanges would be to use the least amount of bent edges as possible, and not use hole to hole dimensions. As shown in Figure 5-14, all of the holes are dimensioned “.250” from a hard edge of the flange that they are located on, which will be the most accurate dimension. Next, the holes on the vertical flanges are dimensioned from the top, outermost edge as opposed to being dimensioned from the holes that are in the horizontal flange (as shown in Figure 5-12).

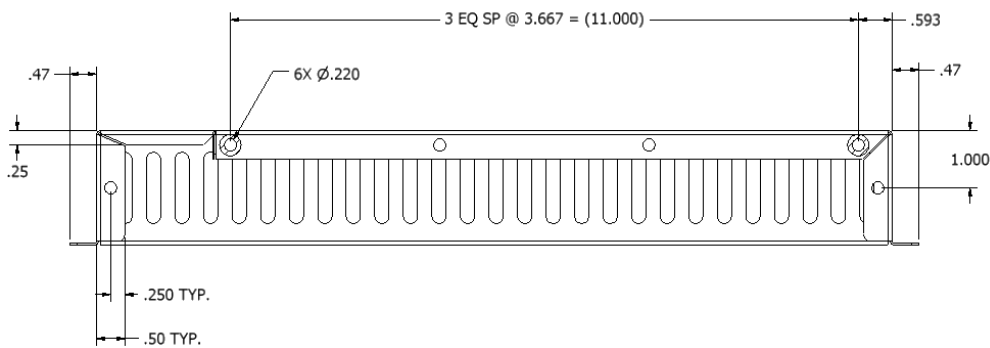


Figure 5-14: Accurate Hole Dimensioning

The advantage of dimensioning from the outermost edge is to ensure that the holes are as close as possible in relative location, even if they are located on completely different planes since they will be matched up with a mating part (see Figure 5-15).

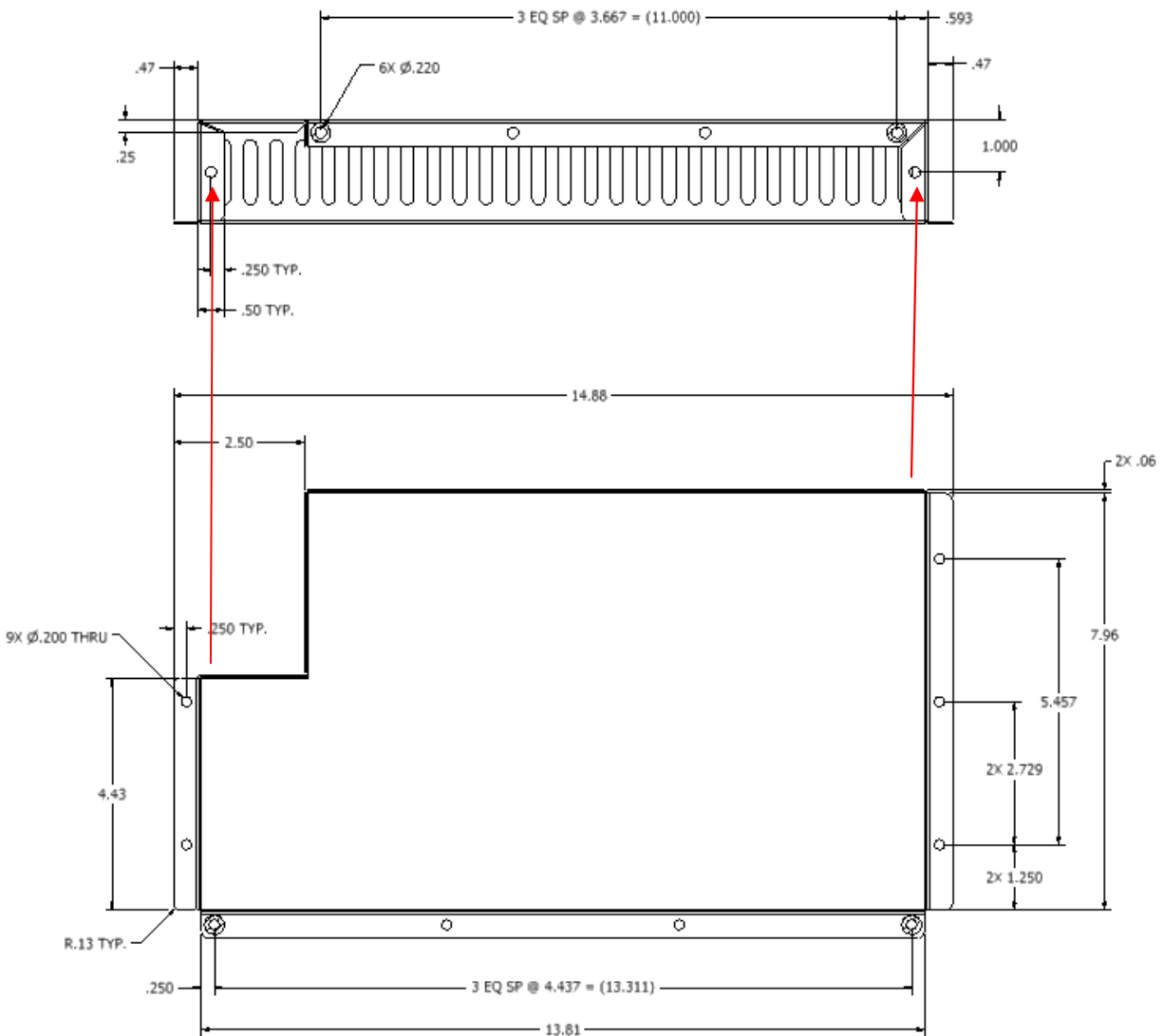


Figure 5-15: Holes on Different Planes

#### 5.4 CREATING A HOLE TABLE

For parts that have many holes, it is recommended that you use a hole table. Originally we used coordinate dimensions; however, when a part has many holes, there is a possibility that some of the holes could be very close in size and location. This could make it very difficult to distinguish which size holes to use at the correct locations, and is time consuming to both create and read (see Figure 5-16).

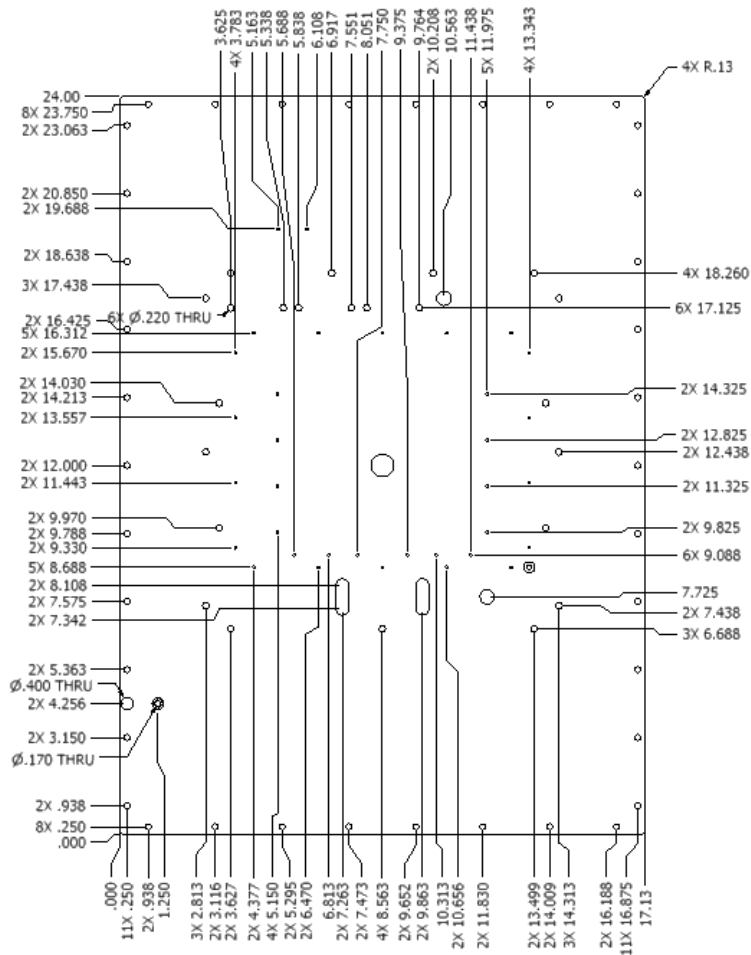


Figure 5-16: Ordinate Dimensions Used for Many Holes

When you create a hole table you can group by hole size and give every size a letter designation which will allow a vendor to program the hole locations more quickly and more efficiently, thus reducing time and mistakes (see Figure 5-17).

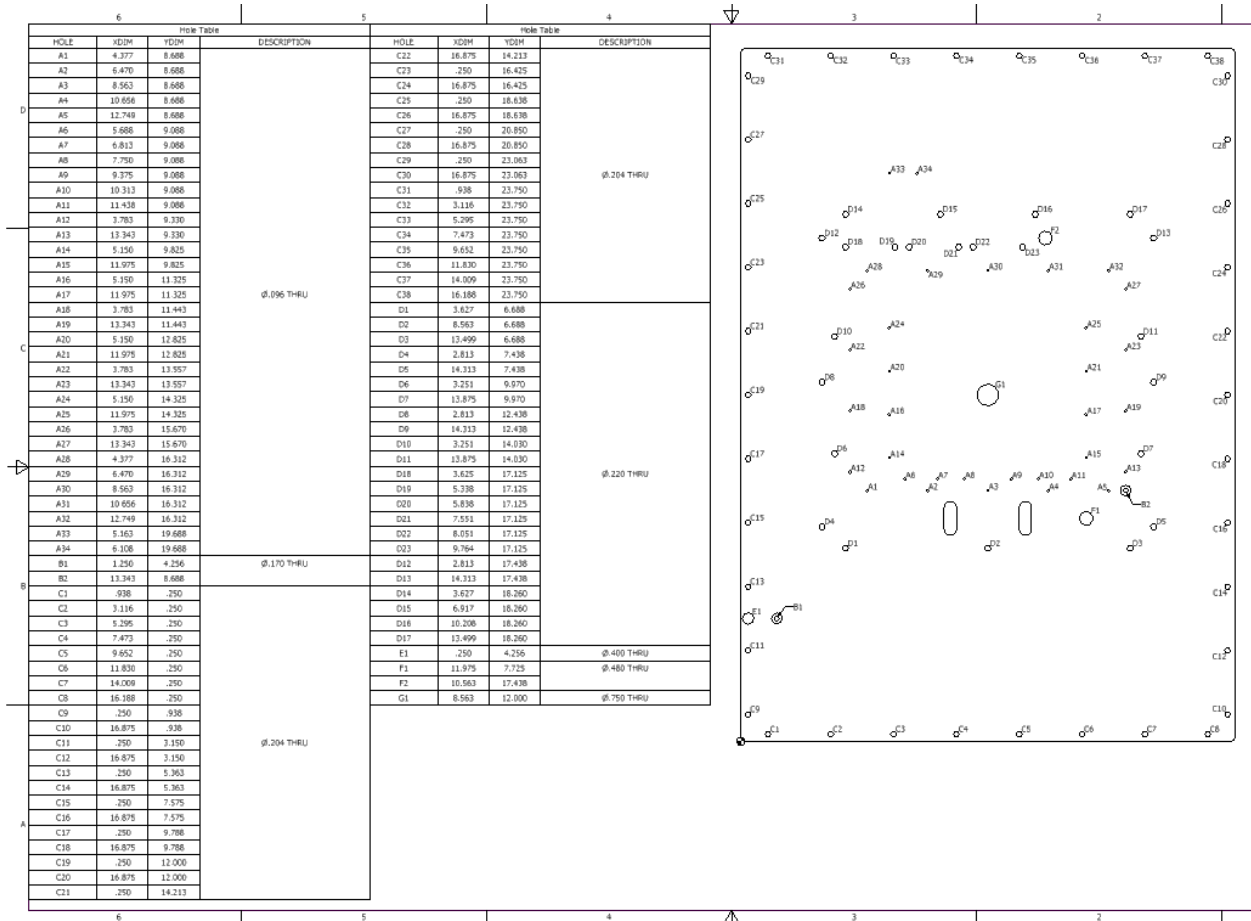


Figure 5-17: Hole Table

In many cases, since there are enough holes to create a need for a hole table, you may need to increase the sheet size of the drawing from an A size to a B size sheet because it is likely that the hole table will be larger than the drawing limits. To create a hole table, place a front or back view of the part, preferably the same view used on sheet one. But, if there are counter sink holes, they should be on the face that is shown.

Click on the hole table icon in the “Drawing Annotation Panel”, select “Hole Table – View” (shown in Figure 5-18).

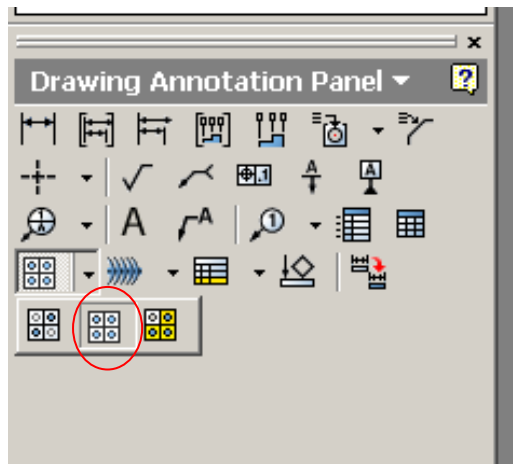


Figure 5-18 Create Hole Table

You will then be prompted to place an origin point, which should be in the lower left hand corner (Figure 5-19).

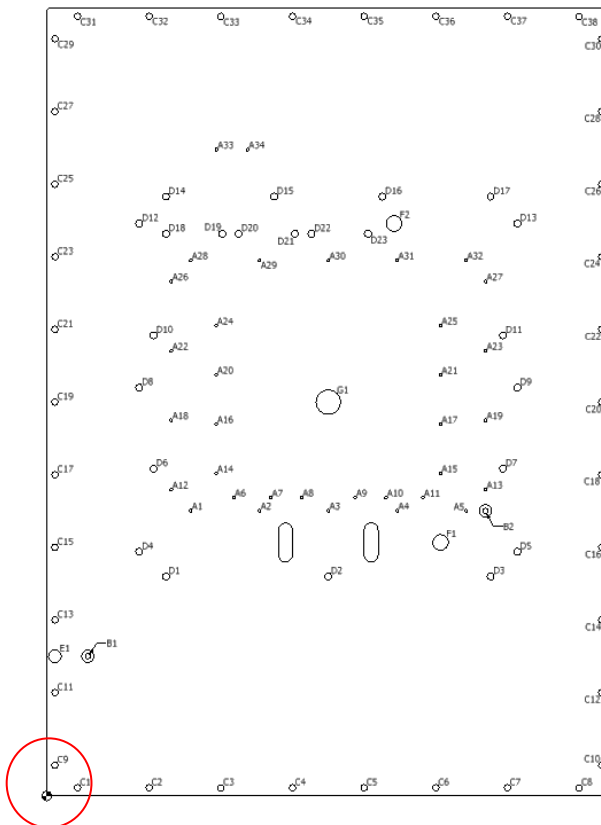


Figure 5-19: Placing Origin Point

You can now place the hole table on the sheet, usually placed in the top left corner.

To group the table by hole size, right click the hole table and select “Edit Hole Table”. Then, select “Arrange by Size”, turn on “Combine Notes”, and select “Reformat Table on Custom Hole Match” (Figure 5-20).

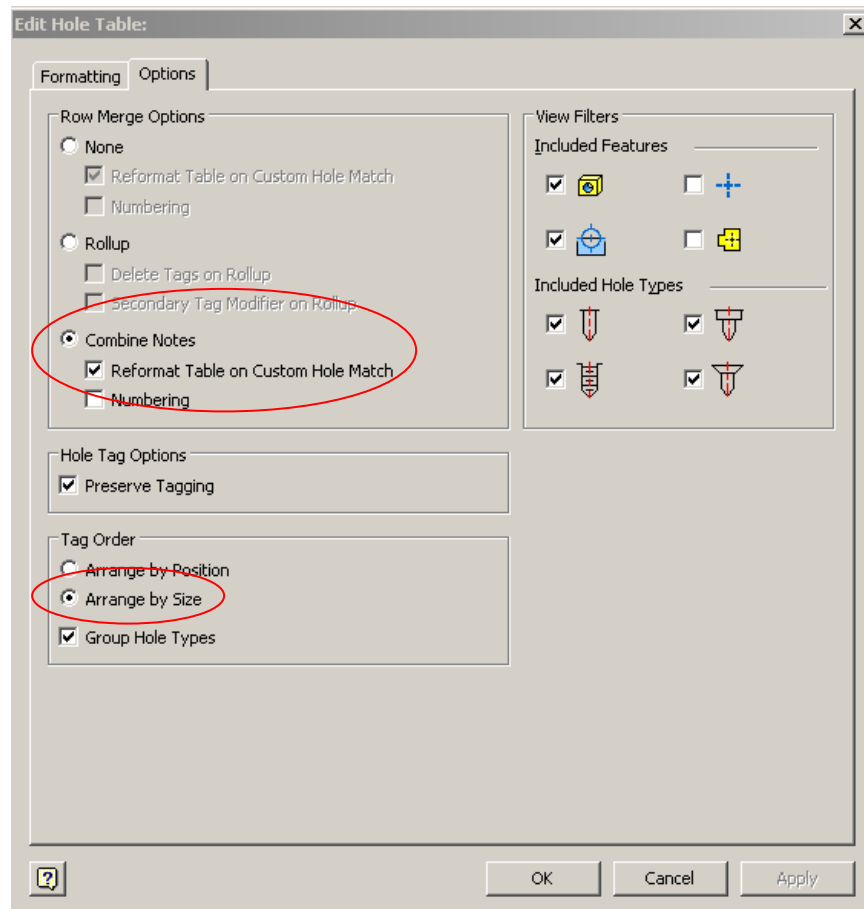


Figure 5-20: Setting Up Hole Table

If the table is longer than the sheet, find a value where the table begins to overlap the border, right click on the value, place pointer over “Table”, and then choose “Split Table” (see Figure 5-21), and the table will now be split at that value. You can split the table as many times as needed, but make sure the holes remain grouped properly by size.

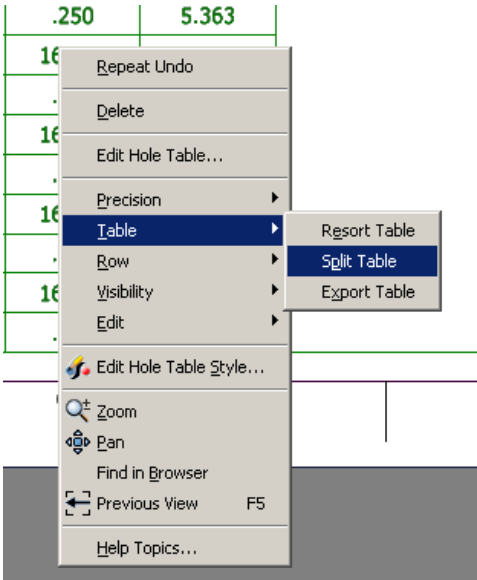


Figure 5-21: Splitting the Table