GTRI MACHINE SERVICES DRAFTING STANDARDS

Prepared by:
Machine Services Department
Georgia Tech Research Institute

December 16, 2011

Georgia Tech Research Institute

Problem. Solved.

TABLE OF CONTENTS

TABLE OF CONTENTS

Paragraph | **Page**

TABLE OF CONTENTS

Paragraph **Page**

TABLE OF CONTENTS

Paragraph **Page**

LIST OF FIGURES

FOREWORD

For many years Machine Services has supplied design and fabrication support for GTRI, GT, and other universities. Seeing many drawings from many sources we have a good understanding of how drafting problems cause parts to be made incorrectly and research dollars to be wasted. Our goal with these compiled drafting standards is to help clarify the need for accurate and complete drawings. This information gives direction for some of the best ways the draftsman can convey to the machinist exactly what is needed. The drawing is the main communication tool to the machinist. It should not be left up to the shop or machinist to guess what a part should look like or how an assembly of parts should function together.

With the introduction of computers and design software much of the art and good drafting skills have been lost. Design software is very helpful, but the designer must fill in the blanks as to how precision parts should fit together. Which parts have close tolerance holes or surfaces? Which parts are slip fit and which are press fit? How much press? What type of surface finish is needed? There are many more questions the designer should ask while he is generating the drawing. The software cannot be relied on to do all the work. At times the shop will receive just a solid model assembly of something to estimate or build. Without the drawings to go with the model, there is no way to determine the intent of the designer. The engineering is only half done.

This manual references established standards recognized by the ASME. Many documents and printed information were reviewed to build this manual. Most of the information references ASME Y14.100. While we have not addressed all aspects of a drawing, we have touched on enough to help the researcher or drafting person to eliminate many problem areas. We have also put in guidance and information regarding welding, welding symbols, and some types of fasteners.

Much of the information in these standards can be referenced back to the "Drafting Requirements Manual" 11th Edition, by Jerome H. Lieblich in association with Bryan R. Fischer. Where there are parenthetical references in a heading, they refer to this manual.

1.0 VIEW LAYOUT AND PROJECTION

The proper view orientation should be used on each drawing. This will reduce the risk of confusion and misinterpretation, which in turn reduces the chances of machining errors.

1.1. Projection Method (3.4.3)

The preferred method in the United States is the Third Angle Projection Method. The third angle method is the method of 'rolling up' the views, or you can think of this as putting the part in a bowl and sliding it up the sides. This is the method typically used by GTRI Machine Services. Because third angle is the assumed projection method, drawings submitted with any other projection method run the risk of being misinterpreted. This can result in costly machining errors. If First Angle is used, a note must be present on the drawing with this information. Mixing First and Third angle projections in the same drawing should not be done.

1.2. Inclusion of Views on First Sheet

All drawing views containing features should be included on the first sheet of a document. This displays important orientation details of the part. It is acceptable to give basic dimensions on these views, and more detailed dimensions of the same view on separate sheets. When one or more views containing features are left out of the first sheet, the machinist is forced to guess the orientation of a view relative to the part. This costs both time and money, and can result in parts with incorrectly placed features.

1.3. Detail Assembly Drawings (Pg. 4-40)

Detail assembly drawings should be included when applicable, to show the orientation of separate parts that are to be assembled. Including assembly drawings aids part fit up and helps correctly tolerance a set of associated parts.

2.0 DRAWING FORMAT

The correct drawing format should be adhered to whenever possible. Keeping a consistent format will help the machinist correctly interpret the drawing faster, which will reduce the time and money spent on each part.

2.1. Notes (9.7)

Notes should be located in the upper left hand corner of a sheet in a numbered column as shown in Figure 1. The first line should be headed with NOTES on the left side of the column. Placing the notes column in a standard location reduces clutter and eases the reading of a drawing.

2.2. Revision History Block

All revised drawings should include a revision history block as shown in Figure 1.

2.2.1. ***Revision History Block Location (6.10.11)***The revision history block shall be located in the upper right hand corner of a sheet as shown in Figure 1.

2.2.2. *Revision History Included Information*

A revision history block with completed dates helps avoid confusion relating to how current a drawing is. If a revision history block is not included, it is possible that the wrong part could be machined, due to confusion about the revision level. A revision history block should include the following information:

Revision
Description of the changes made (if any)
Date of approval
Approval signature

2.3. Title Block (2.6.13)

The drawing shall include a completed title block. When the title block is used instead of a list of notes for this information, the drawing is cleaner and easier to read. The title block should include:

Title
Basic Unit of Measure (English or SI)
Drawing Number and Part Number (if applicable)
Material
Quantity
General Tolerances
Date Checked
Date of Approval
Approval Signature
Drawing Revision
Sheet set information (SHEET 1 OF 3)

The drawing must always have the name of the drafting person plus those that have checked and approved the drawing. The drawing checking and approval process should always be followed to catch drawing errors before the drawing is used for fabrication.

2.3.1. *Title Block Location*

The title block should be located in the lower right hand corner of the sheet as shown in Figure 1. This provides a significant amount of the required information on the drawing.

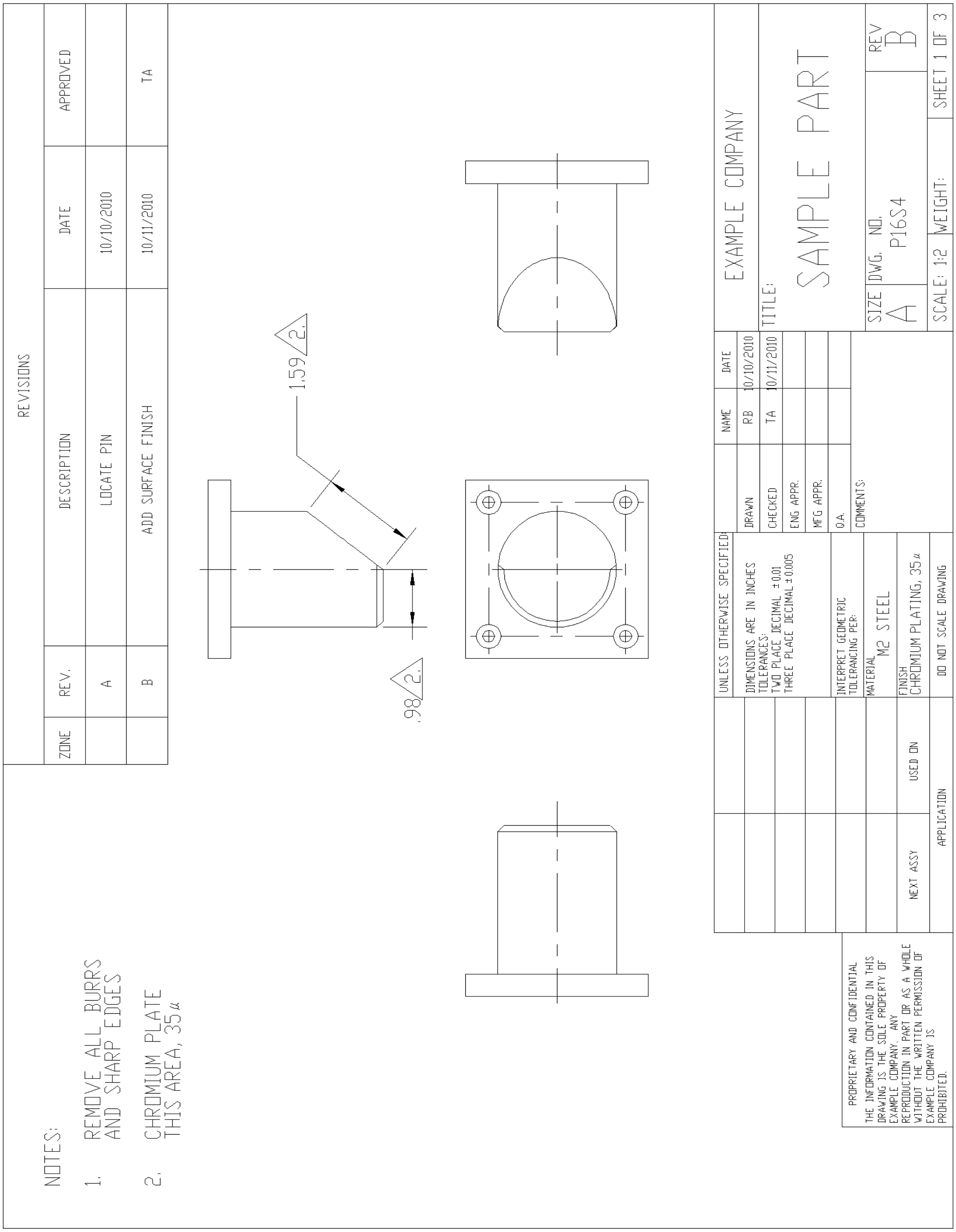


Figure 1. Shows the correct placement of the title block, revision block, and notes.

3.0 DIMENSIONING AND TOLERANCES

Correctly and clearly dimensioning each feature on a drawing is imperative. Failure to do so could result in incorrectly machined parts, increased lead time, and increased costs.

3.1. Dimensions

Drawings shall be dimensioned in accordance with ASME Y14.5-2009.

3.1.1. *Mandatory Dimensioning Rules (5.5)*

a. Dimensioning and tolerancing shall clearly define the design intent. Dimensioning and tolerancing shall clearly define the functional requirements of the design.
b. Each dimension shall have a tolerance, except dimensions for minimum, maximum, reference, or stock size. This tolerance can be applied directly to the dimension or indirectly through a note or title block.
c. Dimensioning and tolerancing shall be complete so there is full understanding of the characteristics of each feature. Neither scaling nor assumption of a distance is permitted.
d. Dimension, extension, and leader lines should not cross unless absolutely necessary. A dimension line should never be broken, but when necessary, it is acceptable to break an extension line.
e. Dimensions are to be shown in the view that most clearly represents the form of the feature.
f. Sufficient dimensions shall be shown to clearly and completely define the form, size, orientation, and location of each feature.
g. A feature shall not be located by more than one toleranced dimension in any one direction.
h. Dimensions are shown outside the outline of the part unless absolutely necessary.
i. Avoid incremental dimensioning to avoid tolerance accumulation between features.
j. Each dimension shall be expressed so that it can be interpreted in only one way.
k. The drawing shall define a part without specifying manufacturing methods.
l. Only the end product dimensions and data will be shown on drawings.
m. Center lines, object lines, or extension lines should not be used as dimension lines.
n. Do not dimension to hidden lines.
o. The word TYPICAL or abbreviation TYP should not be used. Instead, indicate the number of places the dimension applies.
p. Materials manufactured to gage numbers should be dimensioned with linear dimensions indicating the thickness or diameter. Gage numbers should be included in parenthesis following the linear dimensions.
q. Do not mix English and SI units in a drawing, except when using dual dimensioning.
r. Include outside dimensions of the part.
s. When possible, avoid letter dimensioning.

3.2. Datum Features and Origins (5.10.5)

Datum features and origins shall be specified on a drawing when they are used as shown in Figure 2. If a feature is to be used as a datum feature, it must be identified with a datum feature symbol.

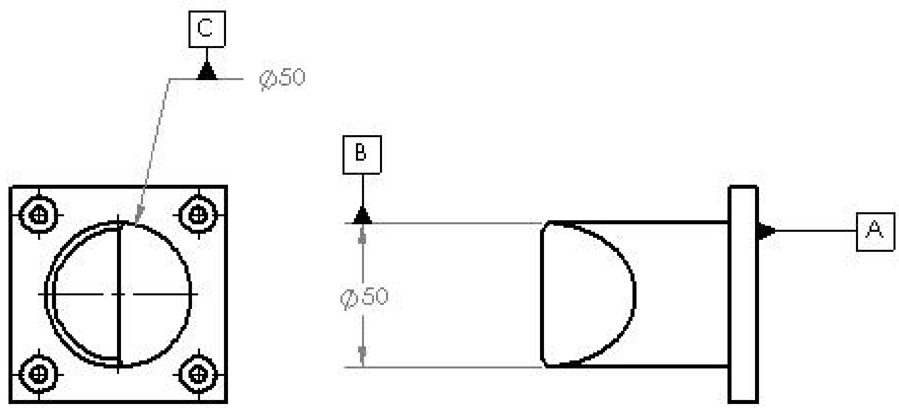


Figure 2. Shows the correct usage of feature datum symbols.

3.3. Correct Decimal Place Usage (5.4.2.1)

It is important to use the correct number of decimal places in a drawing.

3.3.1. *Decimal Place in Design*

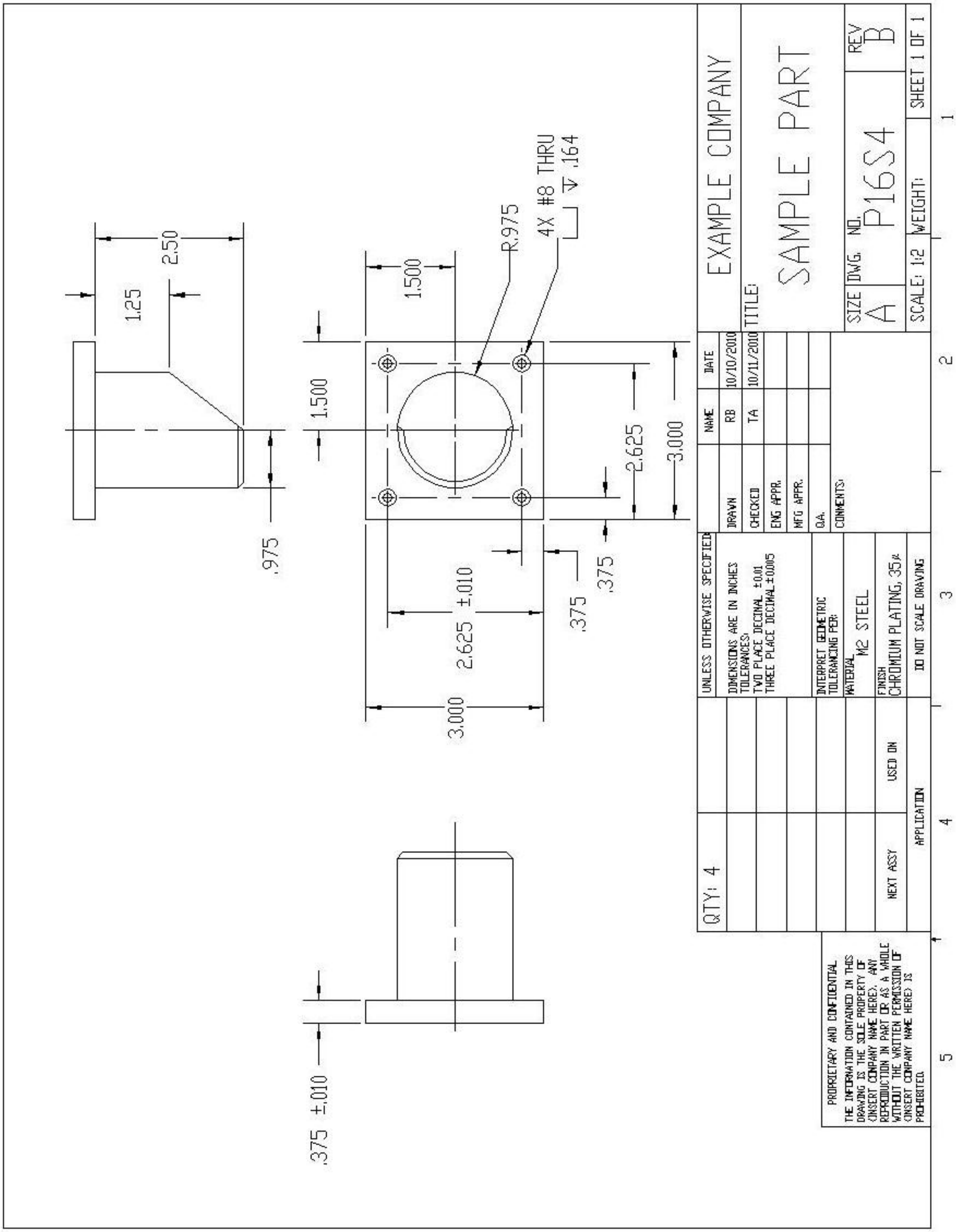
The correct number of decimal places is the minimum number needed to ensure design functionality. For example, if a dimension should only be accurate to .01", it is not necessary to use more than two decimal places in the dimensioning of this feature. In fact, calling out more decimal places than needed can increase the cost and lead time of a part. This is because more accuracy can require more machining time.

3.3.2. *Decimal Place and Tolerance (5.4.2.1)*

The dimension and its associated tolerance shall have the same number of decimal places when the dimension is toleranced directly. When a dimension is toleranced through a note or block, then the number of decimal places on the dimension dictates the tolerance, such as 2 place or 3 place decimal tolerances.

3.3.3. *Decimals in Drawings Using English Units (5.4.2.1)*

Refer to



EXAMPLE COMPANY
TITLE:
SAMPLE PART
SIZE A
DWG. NO. P16S4
REV B
SCALE: 1:2
WEIGHT:
SHEET 1 OF 1
QTY: 4
UNLESS OTHERWISE SPECIFIED
DIMENSIONS ARE IN INCHES
TOLERANCES:
TWO PLACE DECIMAL ±0.01
THREE PLACE DECIMAL ±0.005
INTERPRET GEOMETRIC TOLERANCING PER:
MATERIAL
M2 STEEL
FINISH
CHROMIUM PLATING, 35µ
DO NOT SCALE DRAWING
NEXT ASSY
USED ON
APPLICATION
DRAWN RB 10/10/2010
CHECKED TA 10/11/2010
ENG APPR.
MFG APPR.
Q.A.
COMMENTS:
NAME
DATE
PROPRIETARY AND CONFIDENTIAL
THE INFORMATION CONTAINED IN THIS DRAWING IS THE SOLE PROPERTY OF <INSERT COMPANY NAME HERE>. ANY REPRODUCTION IN PART OR AS A WHOLE WITHOUT THE WRITTEN PERMISSION OF <INSERT COMPANY NAME HERE> IS PROHIBITED.
4X #8 THRU
R.975
2.50
1.25
1.500
2.625
3.000
.975
.375
2.625 ±.010
.375 ±.010

Figure 3 for illustration.

3.3.3.1. <u>Whole Number Decimal Placement</u>

The minimum number of decimal places in drawings using English units is one. If the desired dimension is 8”, the correct dimension is 8.0, not 8.

3.3.3.2. <u>Decimal Placement in Dimensions Less Than One Inch</u>

For dimensions less than one inch, no zero precedes the decimal point.

3.3.3.3. <u>Decimal Placement in Dimensions Exceeding a Whole Number</u>

For dimensions exceeding a whole number, a dimension shall have the same number of decimal places as its tolerance, and zeros should be added as necessary.

3.3.4. *Decimals in Drawings Using SI Units (5.4.2.1)*

Refer to

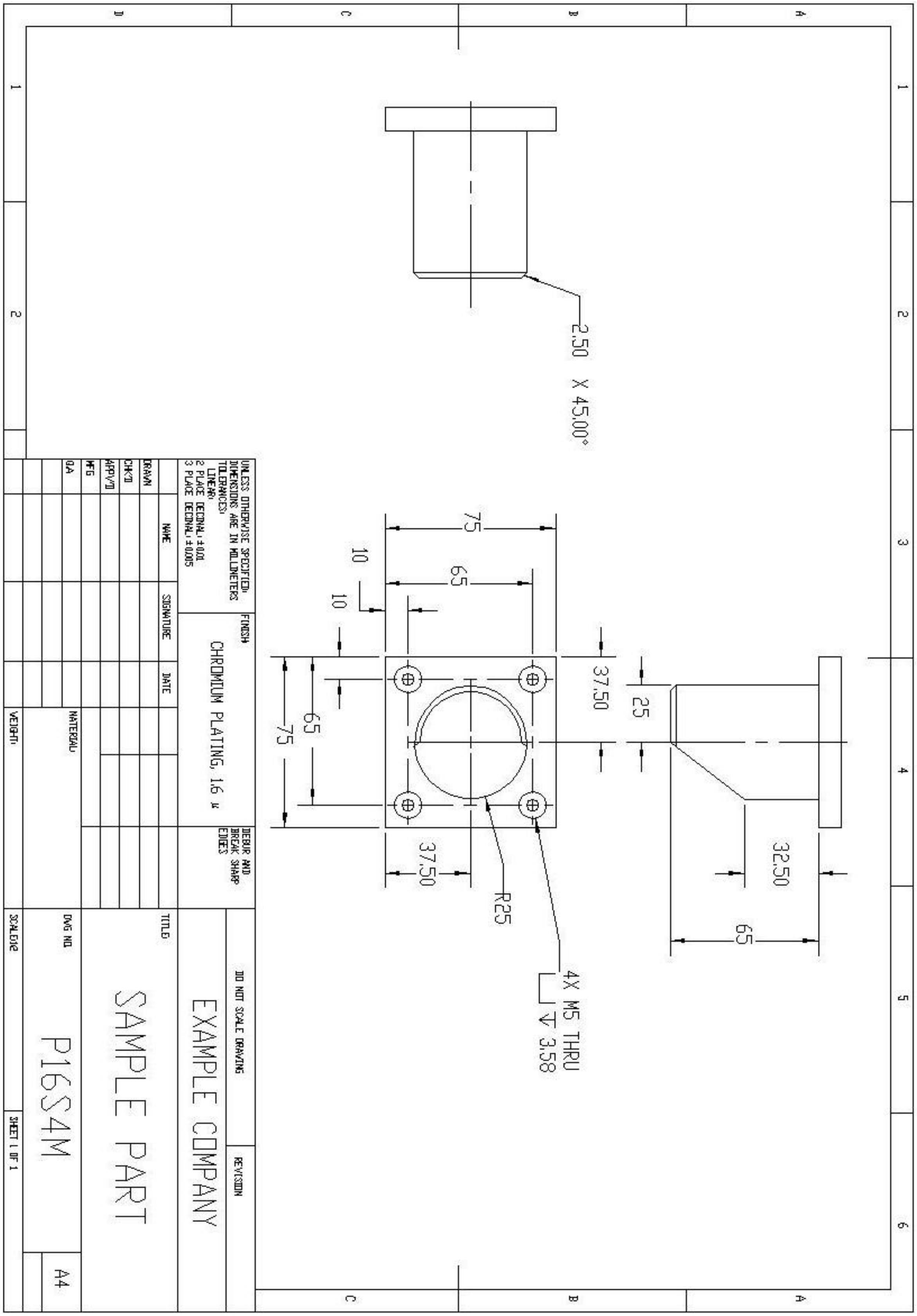

2.50 X 45.00°
75
65
10
10
37.50
25
65
75
37.50
R25
32.50
65
4X M5 THRU
⌴ ⊽ 3.58
UNLESS OTHERWISE SPECIFIED:
DIMENSIONS ARE IN MILLIMETERS
TOLERANCES:
LINEAR:
2 PLACE DECIMAL ±0.01
3 PLACE DECIMAL ±0.005
FINISH
CHROMIUM PLATING, 1.6 μ
DEBUR AND BREAK SHARP EDGES
NAME
SIGNATURE
DATE
DRAWN
CHK'D
APPV'D
MFG
Q.A
MATERIAL:
WEIGHT:
DO NOT SCALE DRAWING
REVISION
EXAMPLE COMPANY
TITLE:
SAMPLE PART
DWG NO.
P16S4M
A4
SCALE:1:2
SHEET 1 OF 1

Figure 4 for an illustration.

3.3.4.1. Whole Number Decimal Placement

The correct way to denote a whole number of millimeters in the SI system uses no decimal places. For example if the desired dimension is 5mm, the correct dimension is 5, not 5.0.

3.3.4.2. Decimal Placement in Dimensions Less Than One Millimeter

For dimensions less than one millimeter, a zero precedes the decimal point.

3.3.4.3. Decimal Placement in Dimensions Exceeding a Whole Number

For dimensions which are not a whole number, the last digit to the right of the decimal point is not followed by a zero. The tolerance and dimension do not need to have the same number of decimal places.

3.4. Dimensioning Counterbores and Countersinks (5.6.2.4, 5.6.2.6)

When dimensioning couterbores, it is necessary to give both the diameter and depth dimensions. For a countersink the included angle and diameter at the surface must be provided. Typically countersinks are 82 degrees or 90 degrees for screws and 100 degrees for rivets. When the counterbored or countersunk hole is for use with a fastener, it is better to simply state the fastener size and depth of the hole and the depth of the couterbore or countersink. This makes the machinists' job easier and can cut down on extensive set up costs. See

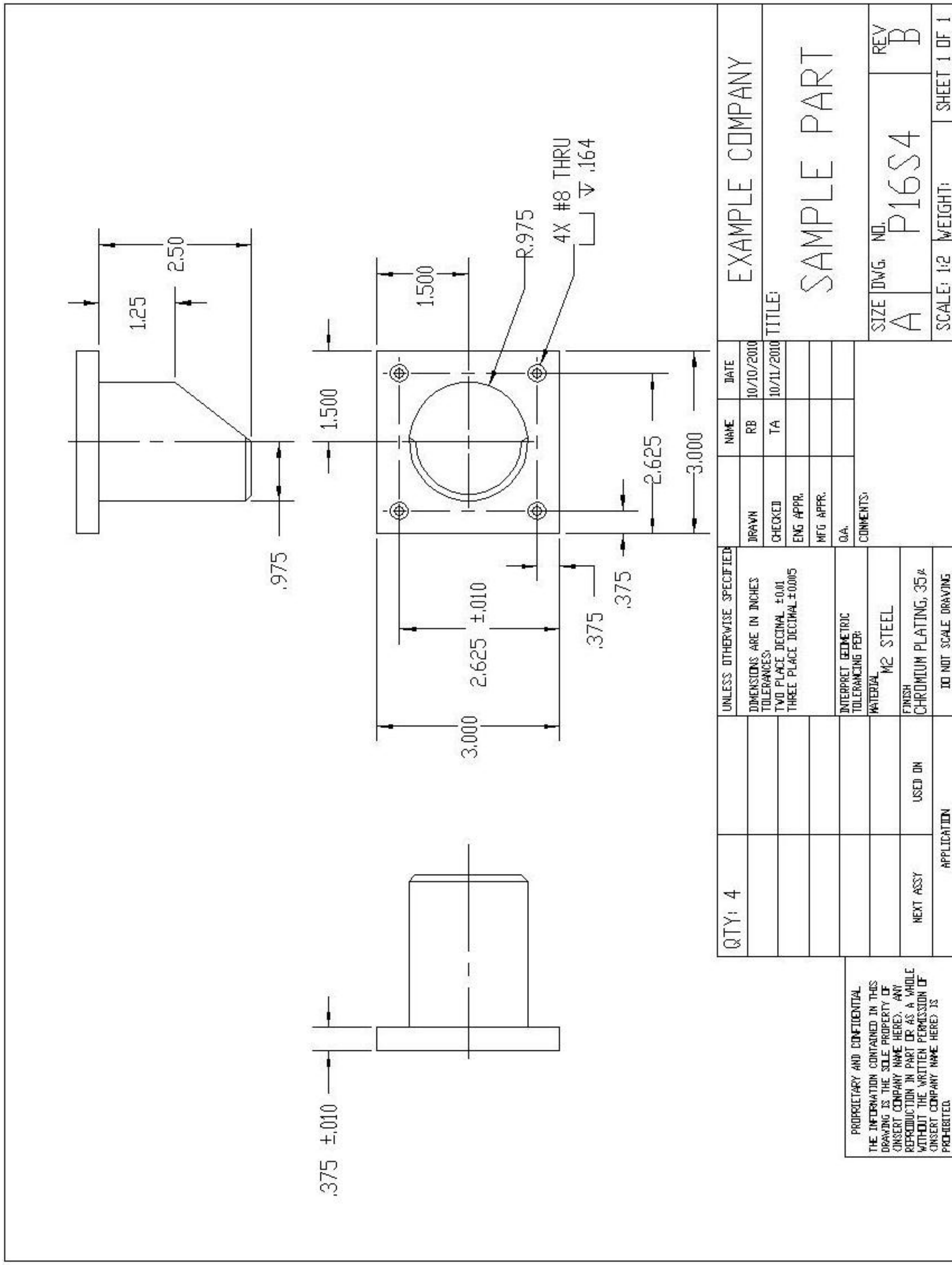


Figure 3,

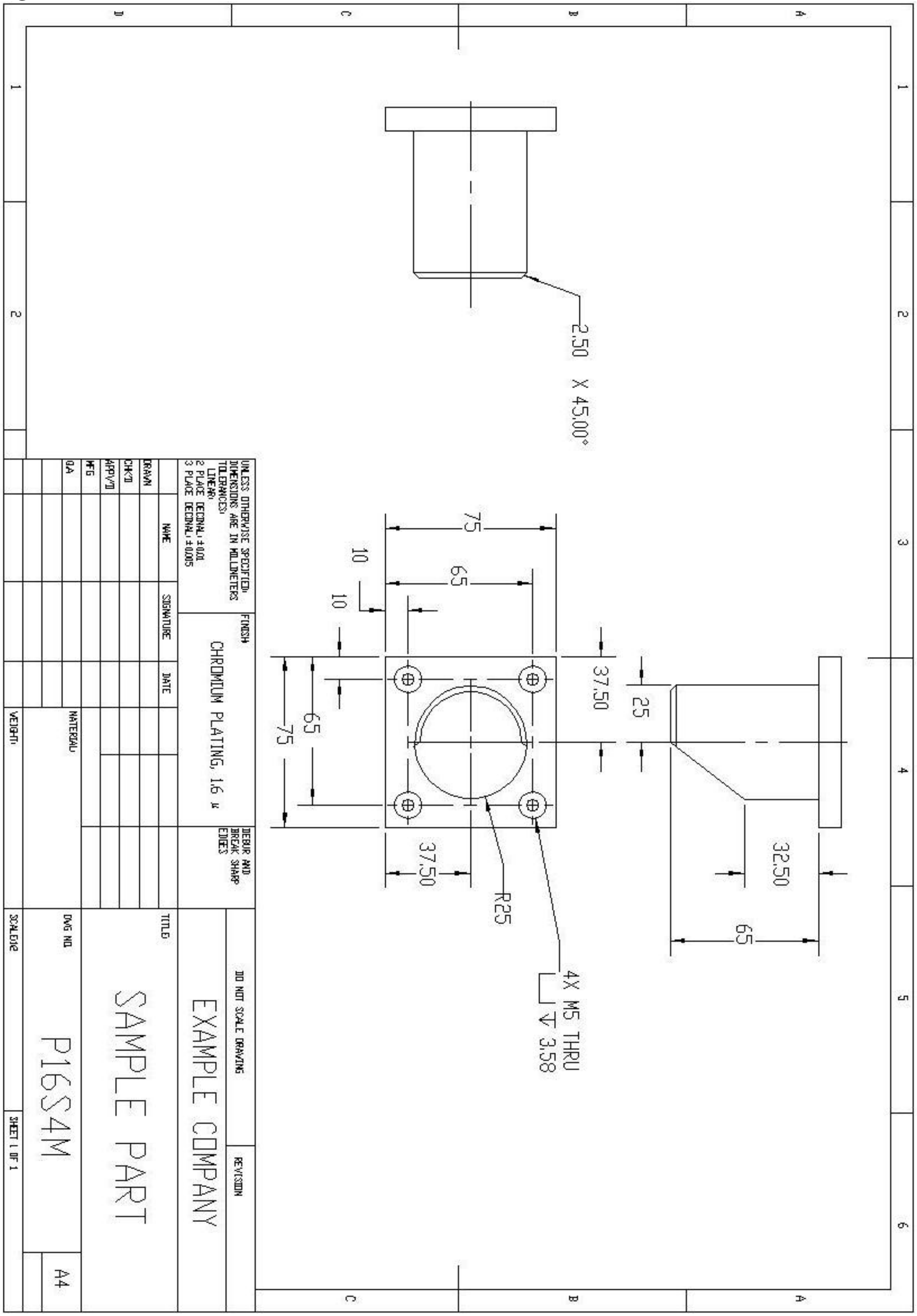

2.50 X 45.00°
75
65
10
10
37.50
25
75
65
37.50
R25
32.50
65
4X M5 THRU
⊔ ⊽ 3.58
UNLESS OTHERWISE SPECIFIED:
DIMENSIONS ARE IN MILLIMETERS
TOLERANCES:
LINEAR:
2 PLACE DECIMAL ±0.01
3 PLACE DECIMAL ±0.005
FINISH
CHROMIUM PLATING, 1.6 μ
DEBUR AND BREAK SHARP EDGES
NAME
SIGNATURE
DATE
DRAWN
CHK'D
APPV'D
MFG
Q.A
MATERIAL:
WEIGHT:
DO NOT SCALE DRAWING
REVISION
EXAMPLE COMPANY
TITLE:
SAMPLE PART
DWG NO.
P16S4M
A4
SCALE:1:2
SHEET 1 OF 1

Figure 4, and Figure 5 for examples.

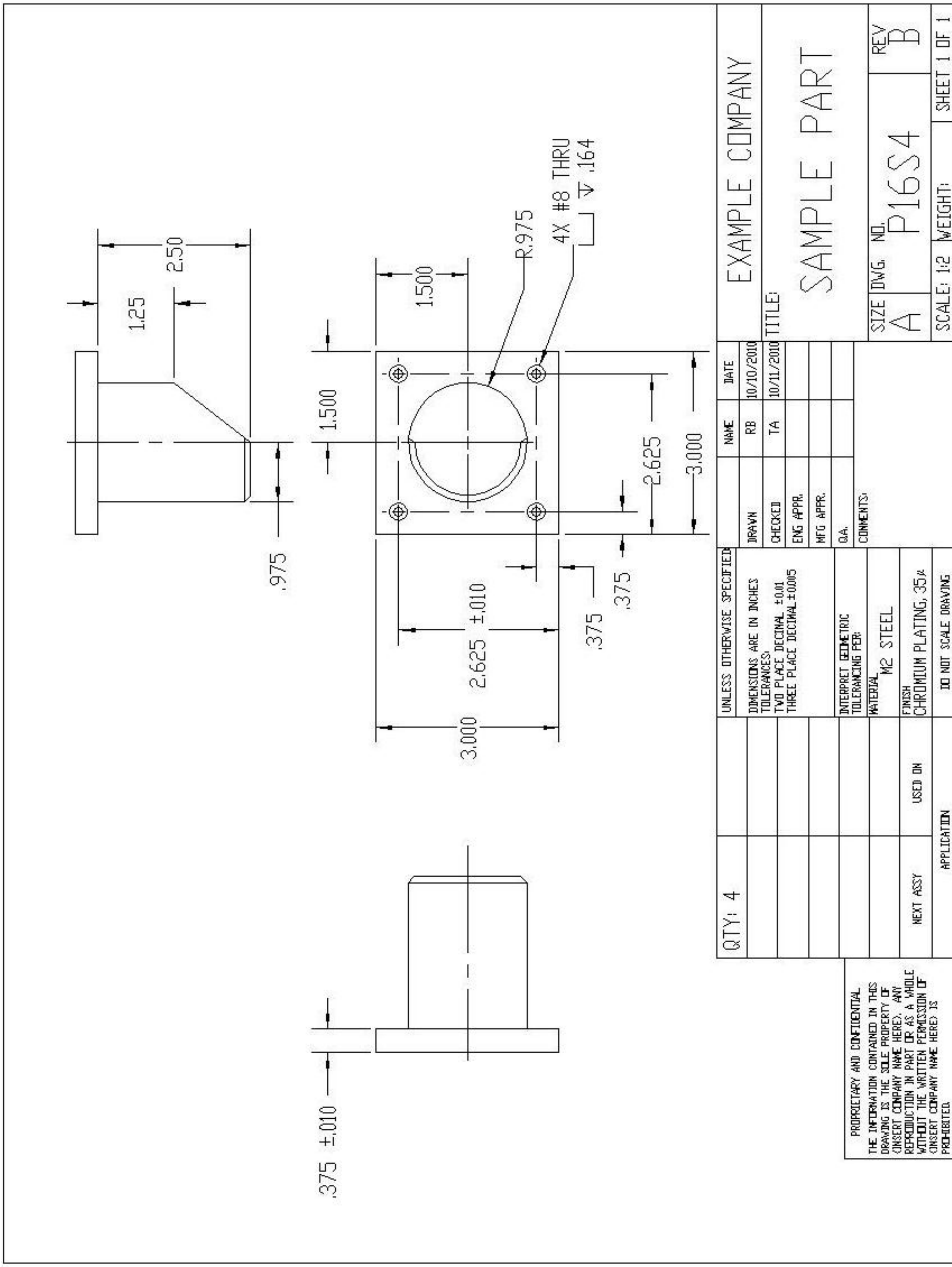


Figure 3. Shows the correct number of decimal places when dimensioning with English units.

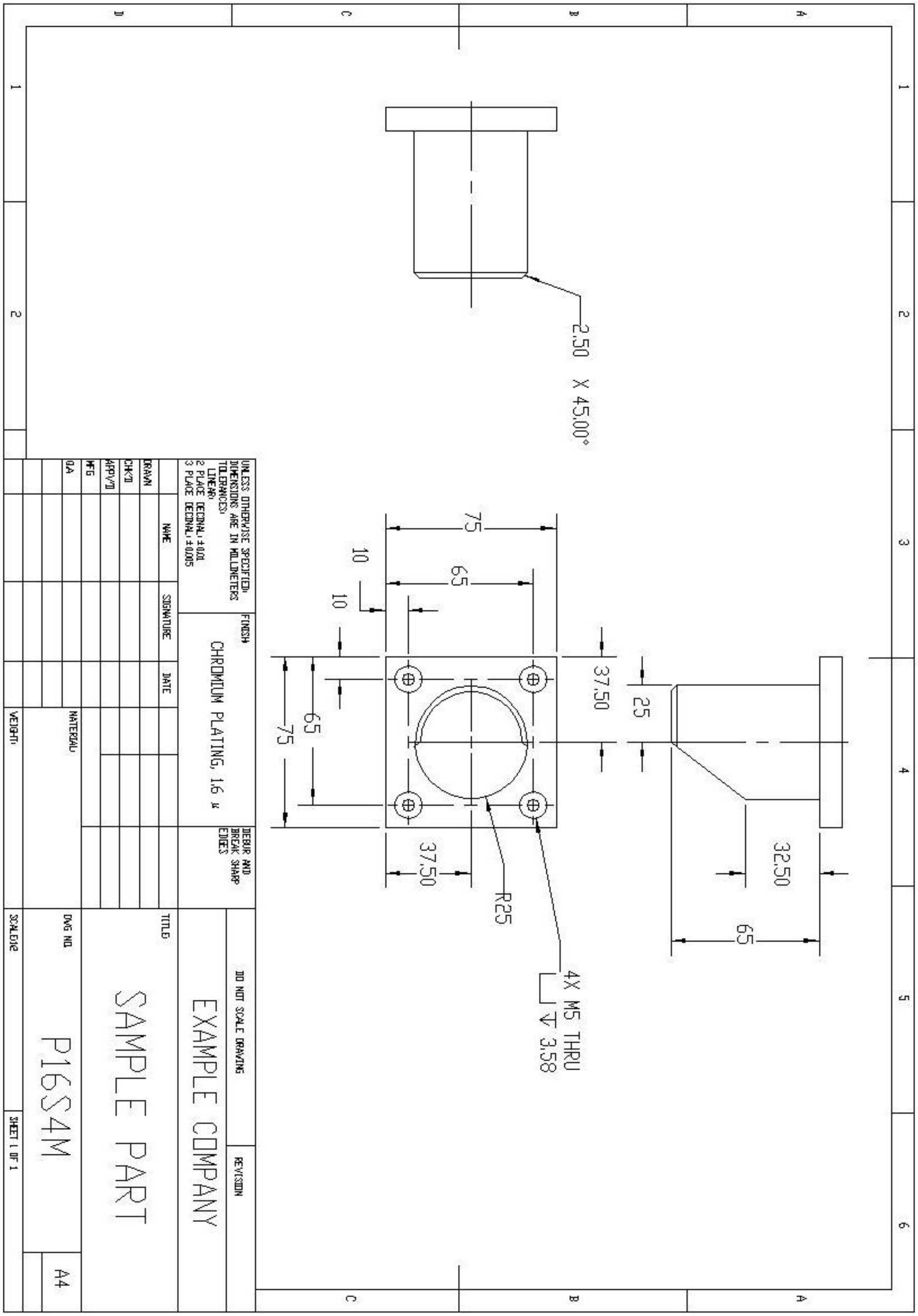


Figure 4. Shows the correct number of decimal places when dimensioning with SI units.

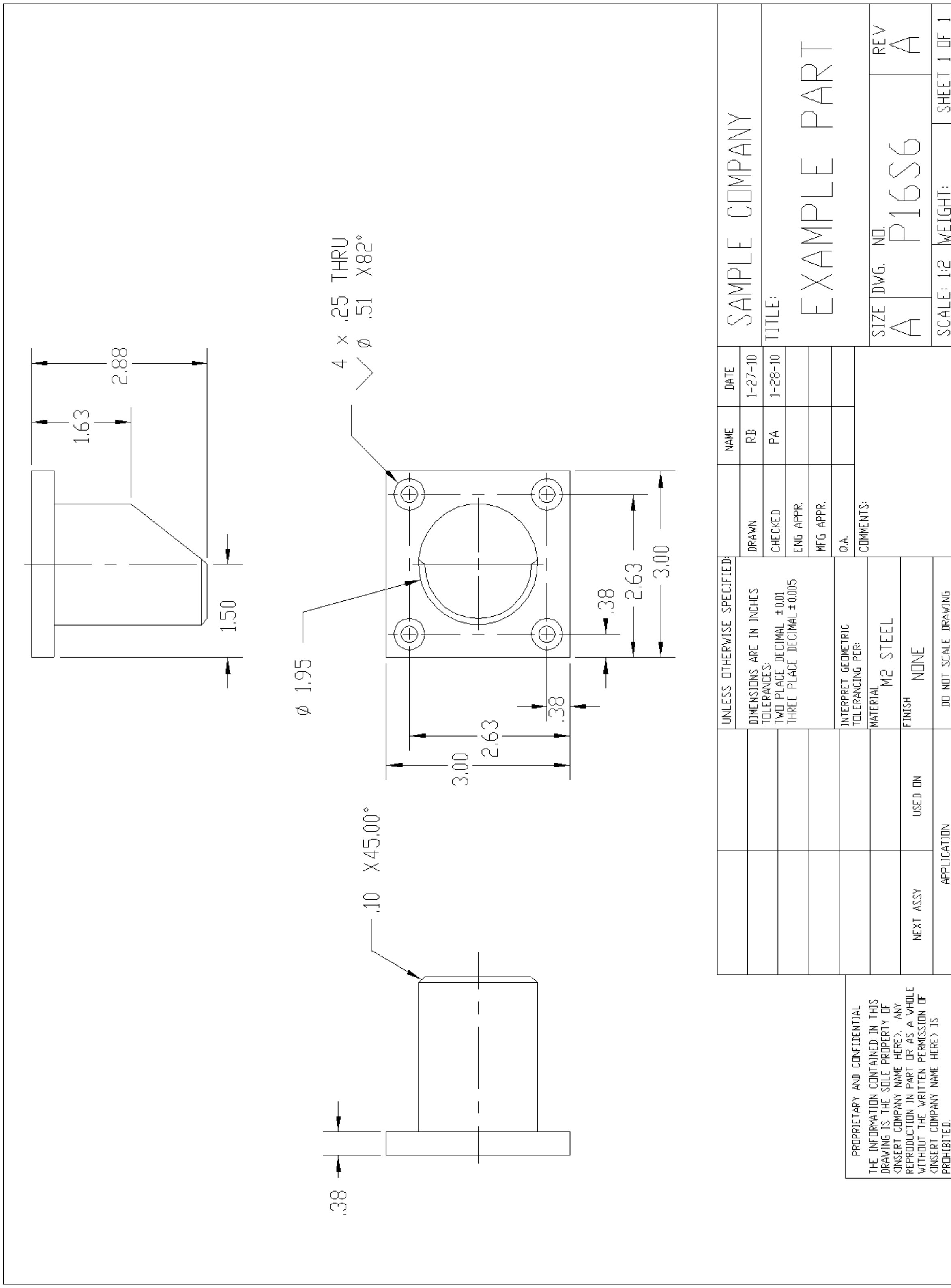


Figure 5. Shows how to correctly dimension a countersunk hole.

3.5. Ordinate Dimensioning

Ordinate dimensioning is a dimensioning system that uses an origin and then dimensions features based on this origin as shown in Figure 6. Using ordinate dimensioning is not mandatory, but can be very useful in CNC machining because CNC programs use an origin of their own. So, providing this can help speed up machine setup time.

3.6. Chamfer and Radius Dimensioning (5.6.5)

Figure 7 and Figure 8 show several ways to dimension chamfers and radii. For chamfers, it is necessary to call out the depth, and the angle of the chamfer, or the depth and width. Or, for chamfers on round parts, only the angle and a diameter are needed. For edge radii tangent to two faces, it is only necessary to call out the radius, and it will be assumed tangent. If it is not tangent, then start and end points should be provided.

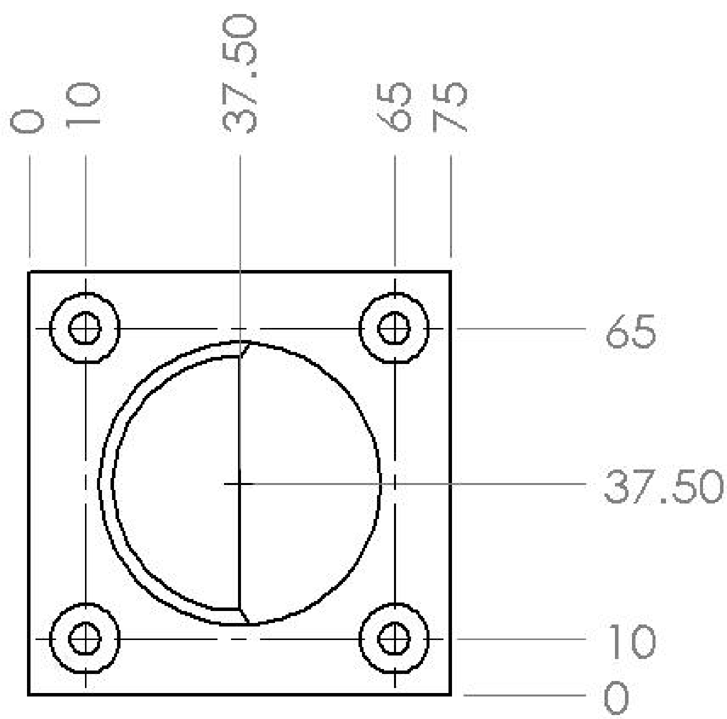


Figure 6. Shows the correct way to use ordinate dimensioning.

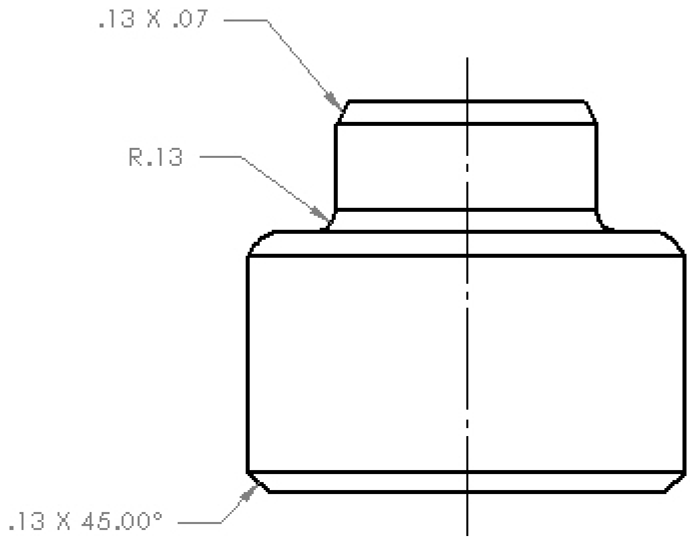


Figure 7. Shows several ways to dimension chamfers and radii.

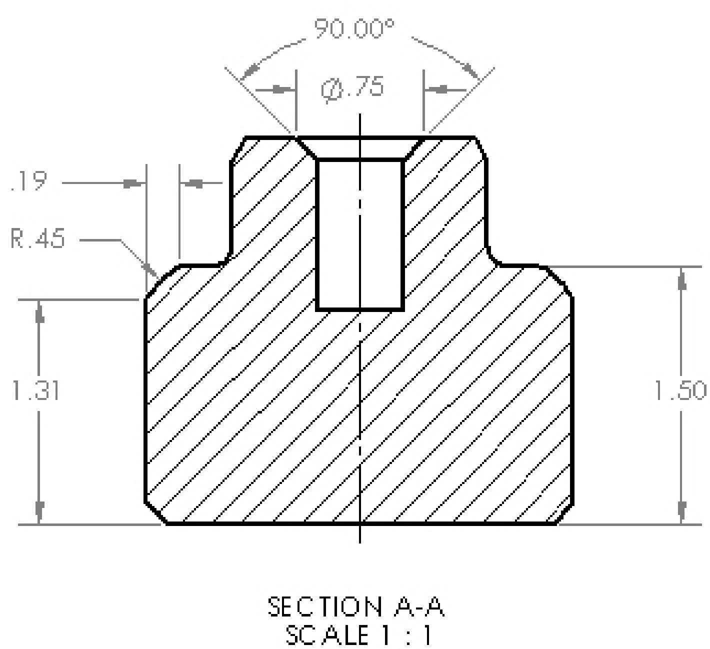


Figure 8. Shows how to dimension radii with nontangent edges as well as countersunk holes.

4.0 SURFACE FINISH AND SURFACE TEXTURE

It is important to describe the required surface finish or texture. Of course, failing to do so would result in an incorrect surface finish or texture or no surface finish or texture applied.

4.1. Surface Finish (15)

Where the required surface finish is different from the surface finish produced by the forming of the part, it should be called out as such, and denoted on the drawing.

4.1.1. *Plated or Coated Surface Callouts (15.8)*

Plated or coated surfaces shall be called out using a flag note along with the dimension of the surface to be coated as seen in Figure 9.

4.1.2. *Types of Coatings*

4.1.2.1. <u>Chemical Coating (15.3.1)</u>

A layer of a metallic compound produced by a chemical or electrochemical treatment of the surface, such as oxide or anodize films.

4.1.2.2. <u>Lubricant Coating (15.3.3)</u>

A fluid or dry film used to reduce surface friction, such as graphite or oil.

4.1.2.3. <u>Metallic Coating (15.3.4)</u>

A film of metal or metal alloy deposited by chemical, electrochemical or other processes, such as electroplate or hot dip.

4.1.2.4. <u>Organic Coating (15.3.5)</u>

A film of organic material such as primer or paint.

4.1.2.5. <u>Permanent Protective Coating (15.3.7)</u>

A protective coating used to preserve an item from corrosion, erosion or other forms of deterioration during its designed service life.

4.1.3. *Plated or Coated Surface Texture (13.5.5)*

Unless otherwise specified, when a surface has a surface finish callout as well as a surface texture callout, the texture applies only to the finished surface. This is done to keep from grinding or otherwise finishing a surface texture twice.

4.2. Surface Texture (13)

Where the required surface texture is different from the surface texture produced by the forming of the part, it should be called out as such, and denoted on the drawing.

4.2.1. *Definitions*

4.2.1.1. **Roughness Height Average (Ra) (13.4.8.1)**

The average of the absolute values of the measured profile height deviations when compared to the graphical surface. Basically, this is the average measured distance from the graphical surface. This average is taken over some specified sampling length.

4.2.1.2. **Waviness Height (W) (13.4.8.2)**

The peak to valley height of the measured profile. This measurement is taken normal to the graphical surface.

4.2.1.3. **Roughness Width Cutoff (13.4.6.1)**

The sampling length used to determine the roughness height average. This width is measured in millimeters.

4.2.1.4. **Roughness Width (13.4.5.1)**

The average spacing between adjacent peaks or valleys within the roughness sampling length.

4.2.1.5. **Waviness Width (13.4.5.2)**

The average spacing between adjacent peaks or valleys of the measured profile within the waviness sampling length.

4.2.1.6. **Lay (13.3.2.3)**

The direction of the predominant surface pattern, which is usually determined by the production method used.

4.2.2. *Surface Texture Callouts*

The surface texture should be called out on the drawing or in the general notes as shown in Figure 10. The surface texture symbol is a check mark with a horizontal top extension. The long leg and the horizontal top extension are to be on the right as the drawing is read. The surface texture limits are placed inside the check, the lower limit to the bottom and the upper limit to the top. The limits are in units of microinches or micrometers. Only the necessary limits need be shown.

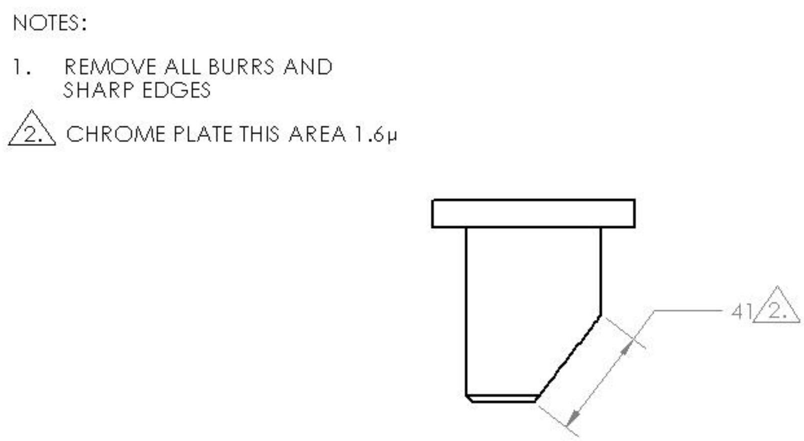


Figure 9. Shows the correct way to display faces to be coated, if the entire piece does not need to be coated.

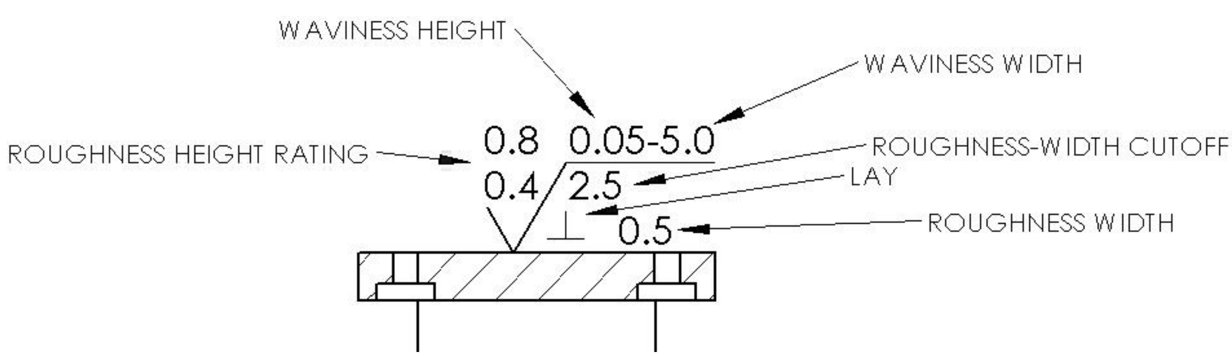


Figure 10. Shows all of the options available with a surface roughness callout.

5.0 DESIGN CONSIDERATIONS

It is important to have an understanding of some of the processes available at a machine shop to be able to more accurately determine what type of tolerance could be expected from these processes. This will help to ensure that a design can be machined to the customer's specifications while staying within their budget.

5.1. Dimensional Capabilities of Machining Methods

Different machining methods will have different levels of expected tolerance and it is important to look at this when designing a part.

5.1.1. *Drilled Holes*

Drilled holes have a variety of different expected tolerances, depending on the drill size to be used. This can be anywhere from +.004/-.001" to +.012/-.001" depending on the diameter of the drill. This tolerance range is caused by a number of factors, such as drill 'walk' or drill bit wear. See appendix.

5.1.2. *Reamed Holes*

Reamed holes are of much higher precision and have a much better surface finish than drilled holes. This makes them very useful for reaming holes for pins and slip fits. The tolerance for reamers runs from ±.0003" to ±.005" for reamers of size .000" to 4.5", again depending on the size of the reamer.

5.1.3. *Boring and Turning*

Boring and turning have a higher tolerance than milling or waterjetting, but a lower tolerance than reaming. Boring or turning have a tolerance range of ±.001" to ±.010" in the size range of .01" to 7.5".

5.1.4. *Milling*

Milling is a standard process used in every machine shop. Typically, it will produce tolerances in the range of ±.002 to ±.010, depending on the size of end mill or cutter used. With great care and added expense, milling can be held to ±.0005".

5.1.5. *Waterjet Cutting*

Waterjet machines are very quick and well suited to cutting two dimensional shapes from sheets or plates. The tolerance range on a waterjet machine is from ±.005 to ±.025, for material of thickness .025 to 2.5" thick. On thicker material, a waterjet machine can also produce a tail, or a widening of the water column as it goes through the material. This is one of the primary reasons for the poorer tolerances typical to a waterjet machine.

5.1.6. *Threading*

5.1.6.1. Inch Threads (12.4.6)

To thread a hole or a bar, a tap or die is used as shown in Figure 11. These come in different thread classes, with class two being the most common. If no other notation is provided, it is assumed that class two will work. If closer tolerances are needed, then a class three should be specified, and if a looser tolerance is required, then a class one should be called out. The suffix A denotes an external thread and the suffix B denotes an internal thread. So, a call out of ¼-20 UNF-2B will produce a hole suitable for a machine screw of a nominal diameter of .25", with 20 threads per inch, of a standard tolerance and external threads. A tap will be used for this process. Other information that must be provided is the pitch.

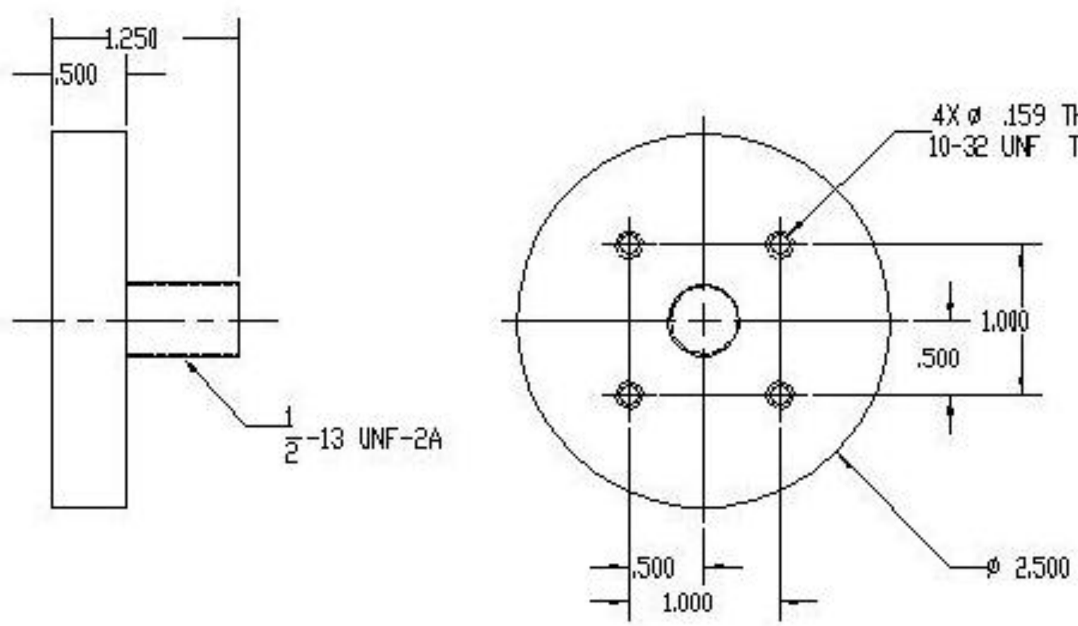


Figure 11. Displays the correct way to dimension both internal and external threads.

5.1.6.2. Metric Threads

Metric threads are designated by an M followed by the nominal diameter in mm followed by an X, then thread pitch in mm. Metric threads also have a tolerance designation, class 6 is roughly similar to inch class 2A/2B, and is the default thread class if no other note is made. Classes 1-5 are of tighter tolerances than class 6 and classes higher than 6 are of looser tolerances. For metric threads, e, g, or h designates external threads, and G or H designates internal threads. G also shows a small tolerance allowance while H shows no tolerance allowance. Similarly, e shows a large allowance, g shows a small allowance, and h shows no allowance. So, a M4 X 1-6H denotes an internal thread with a 4mm nominal diameter, a thread pitch of 1mm, a standard tolerance, and no tolerance allowance.

5.1.7. *Other Thread Types*

5.1.7.1. **<u>ACME Screw Threads (12.3.1)</u>**

ACME screw threads are used primarily for power transmission and transferring motion along the axis of the screw.

5.1.7.2. **<u>Buttress Threads (12.3.5)</u>**

Buttress threads are designed for transferring motion and power, in one direction only. They are built to handle extremely high stresses, and are set up to either push or pull along the axis of the threads.

5.2. Fits

Different types of fits will require different levels of tolerance to ensure that the correct fit type is achieved. If the two parts that are to be mated in a fit are incorrectly toleranced, then this could lead to trouble upon assembly

5.2.1. *Shafts and Bores*

When bores and shafts are to be mated, there are a variety of different fits that can be used, depending on how freely the bore and shaft need to move in relation to each other. On inch drawings, the fit designation should not be used, instead use the fit dimensional limits. However, the fit designation is ok to use on metric drawings.

5.2.2. *Clearance Fit (5.3.16.1)*

This is the most common type of fit. It will have limits such that the mating parts always have some clearance when assembled. An example of this would be a fit between a clearance hole for a bolt and the bolt itself. The hole will always be larger than the shaft in this case and the difference between the two parts with a clearance fit is typically .005” to .015”.

5.2.3. *Interference Fit (5.3.16.2)*

An interference fit is a fit designed such that the mating components will have some interference when assembled. This is typically used for a pin connection in which the pin is permanently installed. The shaft will always be larger than the hole in this case, typically having a difference of dimension of around .0005”to .005” depending on the base size. The two components to be mated will have to be pressed together in this case. The pin should have a small chamfer to lead the pin into the hole and help prevent galling during assembly.

5.2.4. *Line Fit (5.3.16.3)*

A line fit is designed so that surface contact or clearance between two surfaces may result when the mating parts are assembled. This might be used in a part between the top of a bolt head and a pocket surrounding the bolt head, where the most important aspect of the fit is that there is no interference.

5.2.5. *Slip Fit*

A slip fit is a fit that should hold two parts together securely and prevent twisting or sliding against each other. Typically the tolerance for a hole to be used in a slip fit is +.005/-.000”. This allows the shaft to be inserted and removed without causing interference, but it will still be a tight fit. A slip fit is usually used for alignment between two parts with a pin or shaft used for the alignment.

5.3. Geometry Considerations

It is especially important to understand the types of machining processes and how they relate to the geometry of a design. This will help to make sure that the design can be machined for a reasonable cost.

5.3.1. *Pockets or Slots*

For pockets or slots to be machined into a face, it is very important to specify a minimum radius for the corners. If the corners are specified as square, then it becomes very difficult and time consuming to machine the pocket or slot as specified and so cost will go up significantly. Instead, it is good to leave the largest radius that will work with the application on the corners of the slot to aid machinability and reduce cost. For example, the drawing shown below in Figure 12 will be much more costly to produce because of the sharp internal corners, while the part shown in Figure 13 will be much less costly because the radius is the same as a standard end mill so this part can be machined at one time on a standard mill.

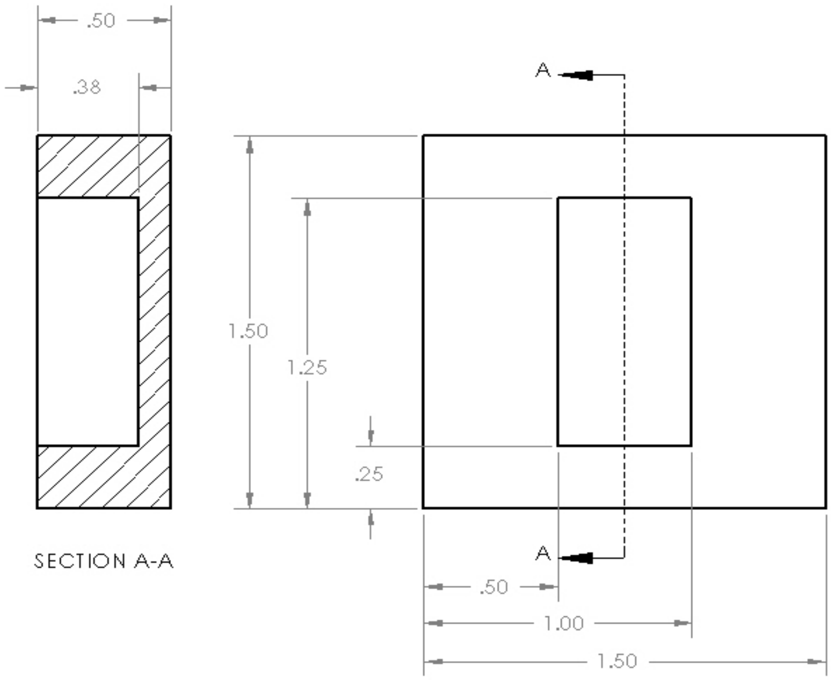


Figure 12. Example of a drawing that will be very costly despite its simple geometry. This is because the internal corners are difficult to machine.

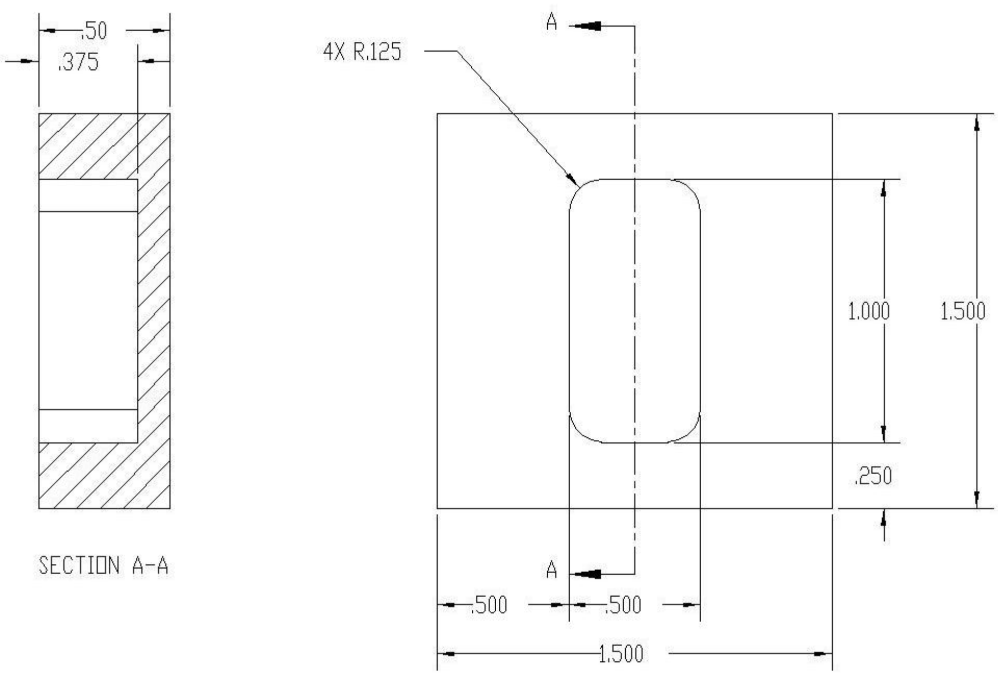


Figure 13. Example of a drawing that will be considerably easier and much less costly to machine than the one shown in Figure 12. This is because the internal corners have a radius, so they will be much easier to machine.

5.3.2. *Keyways (5.6.7)*

Figure 14 shows how keyways are used in shafts to connect a gear, pulley, or some other component to the shaft and secure it such that both the shaft and the component will always have the same rotational speeds. To dimension a keyway, the key size and location are needed. It is best to dimension keys from a standard key size as they typically correspond to standard tool sizes. Of course, because a keyway is considered a pocket, it is also good to keep the corners from being square. Both of these things will reduce machining time and cost.

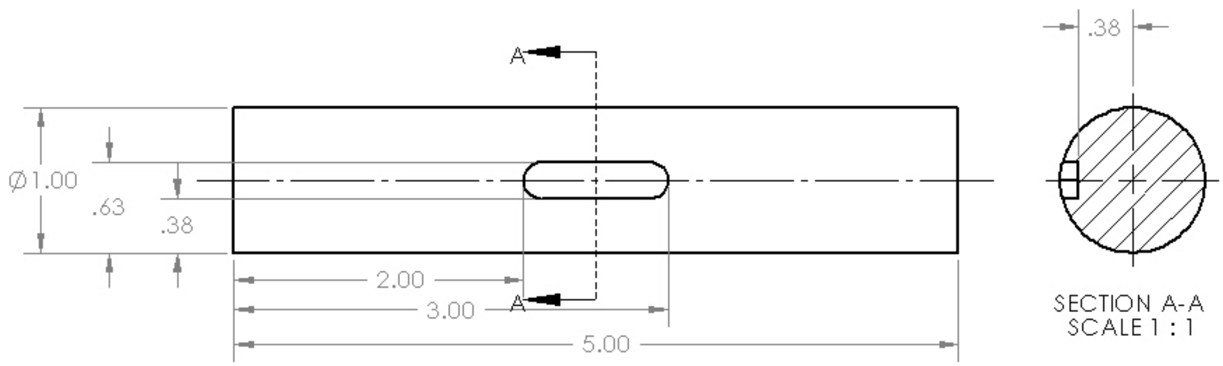


Figure 14. Shows the correct way to dimension keyways in a shaft.

5.3.3. *Engraving Spec.* 15-0000-900

Most engraving is usually performed with a 60 degree single flute cutter. The tip has a small flat ground on the end to create relief for the point to cut without loading up with cut material. Below are suggested engraving sizes and depths. This information was obtained from collaboration between GTRI-ATAS and MSD. One must remember the depths and character size are difficult to maintain on wavy materials. The best materials to engrave are brass, aluminum, and some plastics. Hard materials cannot be engraved with conventional methods. Engraving in very soft materials is not a good practice and will not come out clean. If the part will be painted after engraving, the character size may need to be increased slightly to accept the paint without filling in totally.

height	**stroke width**	**depth**
0.093	.020-.022	.008"
0.12	.021-.024	.010"
0.15	.022-.026	.012"
0.18	.024-.027	.013"
0.25	.028-.031	.015"
0.31	.032-.035	.016"
0.38	.036-.039	.018"

6.0 WELD SYMBOLS

For welded parts, it is important to include the type of weld as well as any other important information concerning the weld. Doing this will help avoid confusion and ensure that the assembly meets the customer specifications.

6.1. Weld Symbol Notation (14.5)

All weld symbols shall have the correct ASME layout as illustrated in Figure 15 and Figure 17. The symbol showing the type of weld can be on top of, beneath, or on top of and beneath the weld symbol, depending on where the weld is to be located. Only the important information needs to be included. For example, if the size of the weld bead is not important, the size constraint can be left out. In Figure 15 below, any variables that are necessary in the drawing should be replaced with numbers. For example, if the depth of bevel constraint is .25, then the S shown below should be replaced with .25.

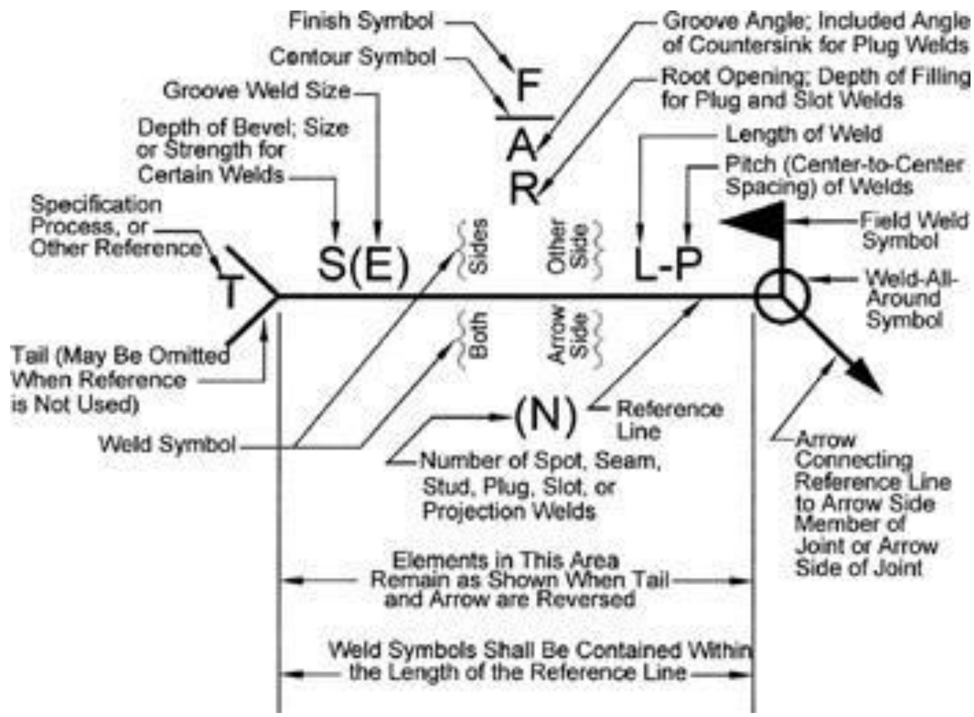
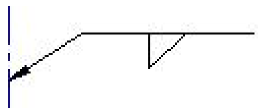


Figure 15. Shows the proper locations for important weld information that can be included on a weld symbol.

6.1.1. *Types of Weld Joints*

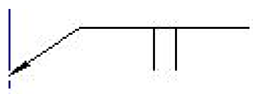
6.1.1.1. <u>Fillet Weld (14.9.1.3)</u>

Figure 16 shows how this weld is formed between two pieces of metal placed roughly at right angles to each other. The filler will melt into both pieces of material, but through neither.



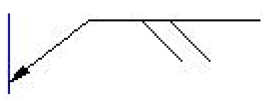
6.1.1.2. <u>Square Weld (14.9.1.1)</u>

A weld formed in a square groove between two pieces of metal that are placed end to end. These pieces of metal should have a right angle on the end to be welded. The filler will be placed in between the two pieces of metal and melt into both of them as shown in Figure 16.



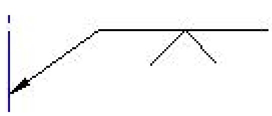
6.1.1.3. <u>Scarf Weld. (14.9.1.1)</u>

A weld between two sides of a scarf joint, which is a type of butt joint made by cutting the opposing pieces at an angle to one another such that they overlap. This type of weld is very similar to a square weld, but instead of the pieces having square ends, the angle should be less than 90 degrees.



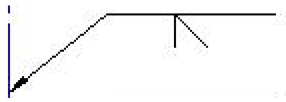
6.1.1.4. <u>V-Groove Weld (14.9.1.1)</u>

A weld made in a V shaped groove that is cut into two pieces of metal that are placed end to end as shown in Figure 16. Each piece should have a bevel cut into the face to be welded so that when they are put together, the two bevels form a V. The two pieces should touch at bottom of the V, and the filler will be placed in the groove formed there.



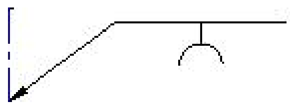
6.1.1.5. Bevel Weld (14.9.1.1)

A weld made between two pieces of material that are placed end to end, in which one of the faces is beveled and the other is square as shown in Figure 16. This is a combination of a square weld and a V-groove weld.



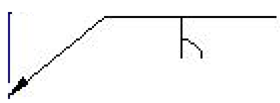
6.1.1.6. U-Groove Weld (14.9.1.1)

Figure 16 shows a weld made in a U shaped groove between two pieces of material which are to be laid end to end. A U-groove weld is very similar to a V-groove weld, except instead of two angles, the two pieces of metal should each have a concave radius.



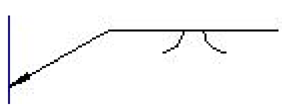
6.1.1.7. J-Groove Weld (14.9.1.1)

A weld between two pieces of butted material in which one edge is square and the other has a concave radius as shown in Figure 16. This is like half of a U-groove weld.



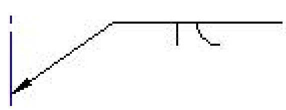
6.1.1.8. Flare V-Groove Weld (14.9.1.1)

Figure 16 shows a weld in a groove between two convex radial faces or a radial face and a flat face. This is like a V-groove weld, but instead of an angle, there is a radius cut into the face of at least one piece of metal.



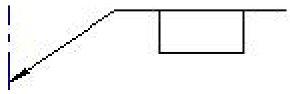
6.1.1.9. Flare Bevel Groove Weld (14.9.1.1)

A weld in a groove between two faces, one of which is square and the other of which is convex radial. This is a combination of a flare V-groove weld and a square weld.



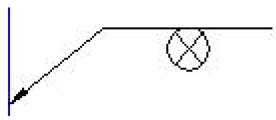
6.1.1.10. Plug or Slot Weld (14.9.1.4)

A weld between two overlapping pieces of material, one of which has holes (or slots) and the other of which has no holes or slots. Weld bead is put into the holes or slots and fused with the material of the other piece. The filler is melted into both surfaces.



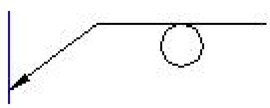
6.1.1.11. Stud Weld (14.9.2.2)

A welding procedure that involves welding a nut or bolt onto another surface. This is typically done with a stud welding gun, and specially made welding studs or nuts.



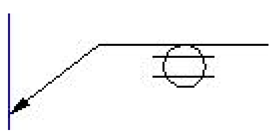
6.1.1.12. Spot Weld (14.9.1.5)

A weld in which two overlapping pieces of material are joined by heating and fusing them together at a point. This does not use filler.



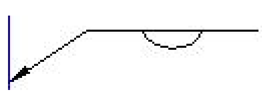
6.1.1.13. Seam Weld (14.9.1.6)

A continuous weld between or on two overlapping pieces of material. This will typically be some sort of groove weld.



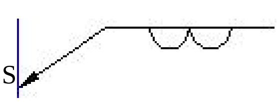
6.1.1.14. Back Weld (14.9.1.8)

A weld along the back of a groove weld. This is done to ensure complete penetration.



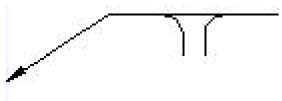
6.1.1.15. Surfacing Weld (14.9.1.7)

A weld made of one or more beads on an unbroken surface to obtain the desired dimensions.



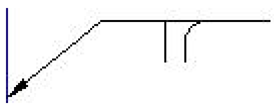
6.1.1.16. Flange Weld (14.9.1.2)

A weld in the groove formed between two flanges of metal; typically these two pieces are sheet metal as illustrated in Figure 16.



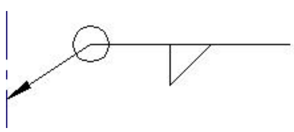
6.1.1.17. Flange Edge Weld (14.9.1.2)

A weld between two pieces of material, one of which is flanged and the other of which is square. The flanged piece will usually be sheet metal while the other piece can either be sheet metal or not.



6.1.1.18. Weld All Around (14.9.2.1)

Welds extending completely around a joint.



6.1.1.19. Double Groove Weld (14.9.2.1)

Figure 15 and Figure 16 shows a double groove weld when a groove is cut into both sides of a piece of material. This can be used with any type of groove weld, and is done to ensure penetration.

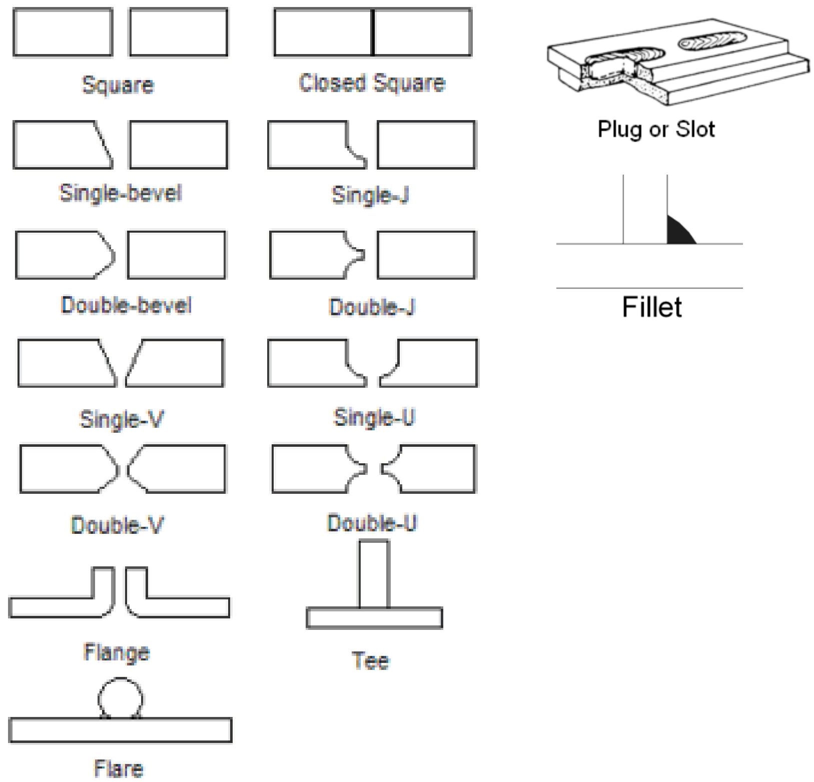


Figure 16. Shows a cross sectional view of how different types of welds are constructed.

6.1.2. *Weld Symbol Arrow Side/Other Side Layout (14.5.1)*

If a weld has components on two sides of a joint, the notation beneath the line is to be on the arrow side, and the notation above the line is to be on the other side as shown in Figure 17. The symbol shown is for a double V-groove weld.

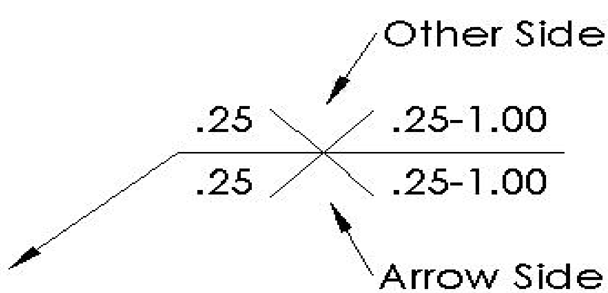


Figure 17. Shows which side of the weld symbol should have the information based on where the weld is located.

6.1.3. *Weld Symbols and Default Weld Type*

Figure 18 shows how weld symbols should be directed to the joint to be welded. If no weld type is called out, then typically a fillet weld or a square weld will be used, due to their ease and strength.

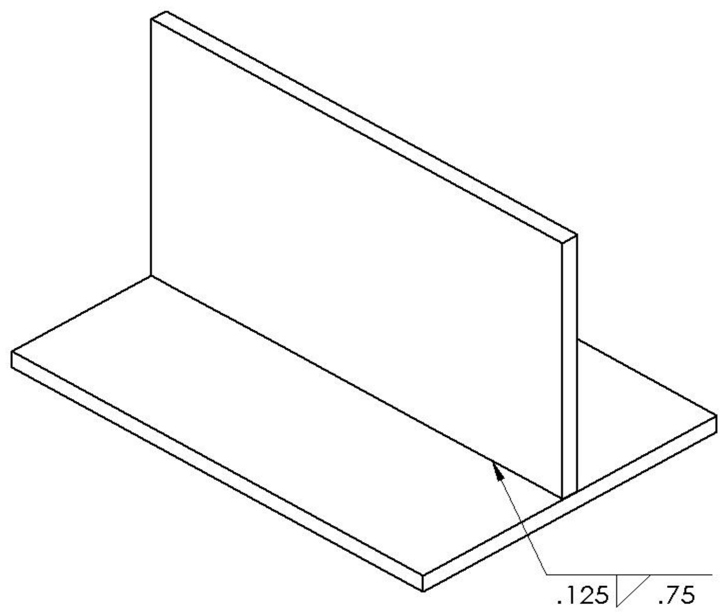


Figure 18. Shows how to correctly denote a weld. In this case, the weld is called out with a bead size of .125 and a length of .75.

7.0 INSERTS

Having an understanding of the different types of inserts available will help in designing components that better meet the needs of the customer. Below are some descriptions and directions for callouts for the insert types we commonly deal with. **See appendix for hole sizes and insert charts.**

7.1. Types of Inserts

7.1.1. *PEM Nuts and PEM Studs*

PEM nuts and studs are an effective way to put a threaded hole or a stud into a sheet metal part. Essentially, they are pressed into a piece of sheet metal, and a clinch ring keeps the insert from turning, and an underflow region secures the insert to the sheet metal. Most are designed to transfer the load through the base material. PEM nuts come in steel (S type), stainless steel, (CLS and CLSS type), and aluminum (CLA type). PEM studs come in steel (FH type) and stainless steel (FHS type). More information can be found at www.pemnet.com.

7.1.2. *Keenserts*

Keenserts are a threaded insert similar to a helicoil or other thread repair inserts, with the exception that they are keyed. They have an inner thread and an outer thread and are threaded into a tapped hole. After they are inserted, the keys are engaged and prevent the insert from backing out. Keenserts are also very useful in situations with small threads in a softer material such as a plastic to help distribute the load over a larger area and prevent failure. Keenserts come in miniature, lightweight, thinwall, heavy duty, and extra heavy duty assortments. More information about keenserts can be found at http://www.alcoa.com/fastening_systems/aerospace/en/product_category.asp?cat_id=679.

7.1.3. *Helicoils*

Helicoils are a wire type insert that is threaded into a tapped hole. This wire provides the threads for a bolt or screw. Helicoils have specific taps that are designated STI taps. Holes for Helicoils are drilled and tapper oversize to fit the Helicoil.

7.1.4. *Press Fit Inserts*

There are several types of press fit inserts, and all of them are fairly similar. The only major difference between them is how they are secured in the part, and their pullout resistance.

7.1.5. *Self-Tapping Inserts*

These inserts have an internal thread and an external self-tapping thread. They are typically used for softer materials to spread the load over a larger area, like a keensert. However, they are not keyed, and typically use chips from the tapping action to prevent them from unscrewing. They do not need a tapped hole for installation, just a drilled hole.

7.2. Insert Callouts

The correct callout for an insert is very important. This should include the insert type, hole size, and any other pertinent information. It is very important that the geometry of the hole be correct in the drawing for the insert. Many times the CNC equipment will cut the hole to the supplied geometry. For PEM nuts, this should include the PEM part number, and for threaded inserts, it should include the internal thread pitch, the external thread pitch, and the insert length as shown in Figure 19.

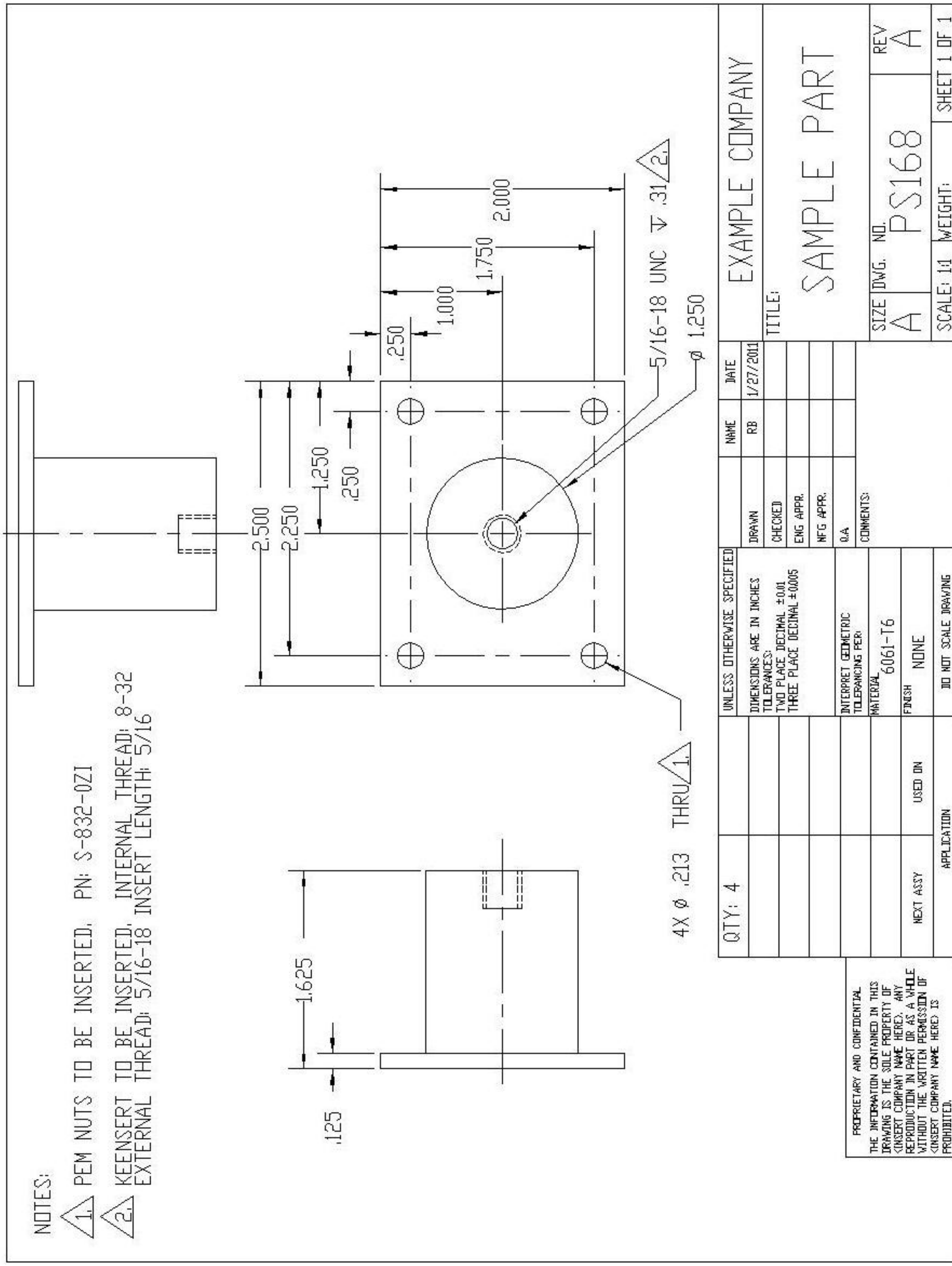


Figure 19. Shows how to correctly call out for inserts in a part.

8.0 MATERIALS AND THEIR PROPERTIES

Different materials have very different properties depending on many factors such as their base material, alloy content, level of heat treatment and how they are produced. It is important to understand some of the basic material properties when selecting a material for a design.

8.1. Aluminum Alloys

8.1.1. *2024*

2024 alloys have average machinability and good workability in an annealed state, due to their elongation properties. However, 2024 has poor corrosion resistance after welding, so these alloys are not weldable. 2024 alloy tempers can be heat treated and/or alclad. Alclad is a process of plating the material with another aluminum alloy to enhance the corrosion resistance.

8.1.1.1. 2024-0

2024-0 is the basic annealed 2024 temper. It is not heat treated, and is typically bare. The yield strength is fairly low on this temper, only 14ksi (96MPa). The elongation is 10-15%

8.1.1.2. 2024-T3

2024-T3 is heat treated and cold worked, and can be bare or alclad. The heat treatment drastically increases yield strength for this temper, to 40ksi (276MPa). The elongation is 10-15%

8.1.1.3. 2024-T351

2024-T351 is also heat treated and cold worked, but is also stress relieved. 2024-T351 can be bare or alclad. The yield strength is slightly higher than T3, at 42ksi (289MPa), but the elongation is also increased slightly to 19%. Forming a tight radius in this state is not an option.

8.1.2. *6061*

6061 alloys have average machinability, good weldability. However, the area around the weld will typically lose strength in heat treated alloys. 6061 alloys also have good corrosion resistance. Some tempers are very good for forming or bending applications due to their ductility.

8.1.2.1. 6061-0

6061-0 is the basic annealed temper of the 6061 alloy. It is very ductile and good for forming and bending applications, and is also good for welding applications, but its machinability is poor. The yield strength is 8ksi (55MPa).

8.1.2.2. <u>**6061-T1**</u>

6061-T1 is heated and allowed to cool naturally, and naturally aged. Its machinability is fair, and it is also fair for forming and bending applications. Corrosion resistance is good, and weldability is also good. The yield strength is 14ksi (96MPa).

8.1.2.3. <u>**6061-T4**</u>

6061-T4 is heat treated and naturally aged. Its properties are very similar to 6061-T1. The yield strength is higher, at 16ksi (110MPa).

8.1.2.4. <u>**6061-T6**</u>

6061-T6 is heat treated and artificially aged. It has fair machinability, good corrosion resistance, and good weldability. 6061-T6 is not good for forming and bending applications. Its yield strength is 35ksi (241MPa).

8.1.3. *6063*

6063 alloys have good corrosion resistance, good forming and bending properties, good weldability, but have poor machinability. 6063 alloys have particularly good surface finish properties though.

8.1.3.1. <u>**6063-0**</u>

6063-0 is the basic annealed 6063 temper. It is very good in forming or bending applications, and has good corrosion resistance and weldability. But its' machinability is poor and the ultimate tensile strength is 19ksi (131MPa).

8.1.3.2. <u>**6063-T1**</u>

6063-T1 is heated, naturally cooled and naturally aged. It has good corrosion resistance, good weldability and is good for bending or forming applications. The machinability is very poor, and the yield strength is 9ksi (62MPa).

8.1.3.3. <u>**6063-T4**</u>

6063-T4 is heat treated, and naturally aged. It has very similar properties to 6063-T1, but the yield strength is slightly higher, 10ksi (68MPa).

8.1.3.4. <u>**6063-T6**</u>

6063-T6 is heat treated and artificially aged. It has very good weldability and corrosion resistance, and is fair for machining and bending or forming applications. The yield strength is 25ksi (172MPa).

8.1.4. *7075*

7075 alloys have fair corrosion resistance, a very high strength to weight ration, and are good for machining. However, they are very poor in forming or bending applications, and are poor in welding applications.

8.1.4.1. <u>7075-0</u>

7075-0 is the basic annealed temper of 7075. 7075-0 has fair corrosion resistance and good machinability, but poor weldability and is poor in bending or forming applications. Its yield strength is 21ksi (145MPa).

8.1.4.2. <u>7075-T6</u>

7075-T6 is heat treated and artificially aged. It has poor weldability and bending or forming characteristics, but has good machinability and fair corrosion resistance. Its yield strength is 66ksi (455MPa).

8.1.5. *5052*

5052 is a nonheattreatable alloy with high weldability and corrosion resistance as well as good forming or bending characteristics. However, it has poor machining characteristics.

8.1.5.1. <u>5052-0</u>

5052-0 is the basic annealed temper of 5052. It has good corrosion resistance, weldability, and has good forming or bending properties. It does not have good machining characteristics though. Its yield strength is 28ksi (196MPa).

8.1.5.2. <u>5052-H32</u>

5052-H32 is a temper that is hardened through cold working. Its properties are very similar to 5052-0, but the yield strength is 33ksi (231MPa).

8.1.5.3. <u>5052-H34</u>

5052-H34 is also hardened through cold working, but is harder than 5052-H32. The other properties are very similar though, except yield strength, which is 38ksi (266MPa).

8.2. Steels

8.2.1. *Carbon Steels*

Carbon steels are steels with carbon as the main or only alloying agent. Their properties fluctuate with steel content and heat treating. Lower carbon steels are generally better for forming or bending applications and have good weldability, but lower yield strength. Higher carbon steels have higher strength but poor weldability and do not work well in forming or bending applications. Depending on carbon content and heat treatment, carbon steels have a yield strength between 27ksi – 110ksi (186-758MPa). The naming conventions for carbon steels are as follows: 1xxx, the 1 shows that it is a carbon steel, the first x shows the type of carbon steel (plain, resulpherized, rephosphorized, or plain with manganese) and the last two xx show the carbon content as 0.xx%. For instance 1050 steel is a plain carbon steel with 0.50% carbon content by weight.

8.2.1.1. <u>1018 Steel</u>

1018 steel is a very commonly used carbon alloy. It has good weldability, forming and bending characteristics and machinability. It can also be heat treated in thin sections or case hardened in thicker sections. The yield strength can be 56ksi (386MPa), depending on heat treat condition.

8.2.1.2. <u>1045 Steel</u>

1045 steel is another common carbon alloy. It has good machinability, but poor weldability and forming or bending characteristics. However, it does respond well to heat treatment. The yield strength can be 73ksi (505MPa), depending on heat treat condition.

8.2.2. *Other Steel Alloys*

Alloy steels are steels with carbon as an alloying agent, but significant quantities of other alloying agents as well. There are hundreds of types of alloy steels, and their properties change depending on their composition and heat treatment. The yield strength for alloy steels can be between 53-260ksi (366-1793MPa).

8.2.2.1. <u>Tool Steels</u>

Tool steels are alloy steels with especially high hardness levels. As the name suggests they are typically used in tool making. They also have very good wear resistance, but poor machinability.

8.2.3. *Stainless Steels*

Stainless steels have a chromium content of 10.5-11% by weight and are stain and rust resistant. Like alloy steels, there are many types of stainless steels that vary by the type and amount of alloying agent. Their properties also vary greatly with the type and amount of alloying agent. The yield strength of stainless steels can be between 30-80ksi (207-552MPa).

8.2.3.1. <u>303 Stainless Steel</u>

303 stainless steel has a yield strength of 35ksi (240MPa). It is alloyed to produce good machining characteristics, but has poor welding characteristics.

8.2.3.2. <u>304 Stainless Steel</u>

304L (low carbon) stainless steel has a yield strength of 35ksi (240MPa), and type 304 has a yield strength of 42ksi (290MPa). Both have good formability. The welding characteristics of 304 and 304L are decent, but special considerations must be made.

8.2.3.3. <u>316 Stainless Steel</u>

316 stainless steel has a yield strength of 42ksi (290MPa). It shows good formability, and but special considerations must be taken for welding.

8.2.3.4. <u>17-4 PH Stainless Steel</u>

17-4 PH Stainless Steel has a yield strength from 130kpsi (896MPa) to 200kpsi (1379MPa), depending on its heat treatment. Formability is poor, but special considerations must be taken for good weld strength.

8.3. Preferred Materials

Aluminums and low carbon steels can be more easily and cost effectively worked with than many other materials. This is due to their ease of machinability and their relatively low cost.

9.0 TIPS FOR SOLIDWORKS

Because SolidWorks is the most widely used design software package at GTRI, below are some tips on how to use it more effectively.

9.1. Formatting Drawings

SolidWorks has many different options for how a sheet can be formatted. Unfortunately, these options can cause confusion, so the formatting settings shown below should be applied.

9.1.1. *Projection*

The 3rd angle of projection should be used. SolidWorks defaults to 1st angle for each new document, so the projection angle must be changed for each new document. To do that, right click on the sheet, and select 'properties'. In sheet properties, select 3rd angle in the box in the upper middle of the sheet properties window.

9.1.2. *Detailing*

The drawing should be shown in the ANSI dimensioning standard. In order to do this, select the 'tools' toolbar, then select 'options' at the bottom of the bar. In the window that comes up, select 'document properties'. In the outline tree to the left, select 'detailing'. In the drop down bar in the upper middle, select 'ANSI' for dimensioning standard.

9.1.3. *Units*

The correct unit system and decimal placing should be used for your drawing. To change these options, select the 'tools' toolbar, then select 'options' at the bottom of the bar. In the window that comes up, select 'document properties'. In the outline tree to the left, select 'units'. In the unit system box, select the appropriate unit system. In each of the following boxes, adjust the necessary number of decimals according to the necessary level of tolerance needed in your drawing. In this section, you can also adjust primary and secondary dual units.

9.1.4. *Editing Sheet Properties*

To edit the sheet properties, right click any blank place on a sheet. Select 'edit sheet format'. This allows the editing of the title block, and when in the sheet edit mode, notes can be snapped to the sides of the sheet.

9.1.5. *Dimension Types*

To use the different types of dimensions, simply click on the 'tools' tab in the main tool bar. In the menu that drops down, you will see dimensions. When this is highlighted, another menu will drop down displaying all of the dimension types. Select the correct type for your application.

9.1.5.1. **Smart Dimension**

This is typically the best dimension type for most applications. It will automatically insert the necessary symbols such as dimension or radius, and will also snap to horizontal, vertical or normal dimensions based on where it is dragged.

9.1.5.2. **Horizontal and Vertical Dimension**

Horizontal and vertical dimensioning simply dimensions features in the horizontal or vertical axes.

9.1.5.3. **Baseline Dimension**

Baseline dimensioning is the same as horizontal and vertical dimensioning with the exception that it only requires a baseline and which features to dimension. It will also automatically drag dimensions to appropriate place.

9.1.5.4. **Ordinate Dimension**

Ordinate dimensioning is similar to baseline dimensioning, with the exception that it does not use arrows. Instead it uses leader lines, and a specified origin. Ordinate dimensioning is especially useful on parts that must be cnc machined because cnc programs require an origin, and specifying one makes the machinists' job easier.

9.1.5.5. **Chamfer Dimension**

Chamfer dimensioning provides a quick and easy way to dimension chamfers.

9.1.6. *Toolbars*

SolidWorks allows a great deal of customization to the drawing workspace. In order to add or remove toolbars, click on the 'view' tab in the main toolbar, and in the dropdown menu, select 'toolbars'. Clicking the various buttons that appear will add or remove the corresponding toolbar from the workspace. If there is not a tool included in a toolbar included that you would like to use, open the toolbar menu as before, but select 'customize' at the bottom of the menu. The customize menu contains five tabs that allow you to make several adjustments as shown below.

9.1.6.1. **Toolbars**

This tab allows you to adjust which toolbars are shown as well as their size, text, and buttons.

9.1.6.2. **Commands**

The commands tab shows all the available buttons and gives a description of each. It also allows you to drag and drop these buttons to any tab in the workspace.

9.1.6.3. **Menu**

The menu tab shows all the dropdown menus and offers descriptions of what each item in the menu does. It will also allow you to change what items are in each menu as well as their position in the menu.

9.1.6.4. <u>Keyboard</u>

The keyboard tab allows you to change or add hotkeys for any command in SolidWorks.

9.1.6.5. <u>Options</u>

The options tab allows you to reset customizations that you have done.

9.2. Formatting Parts

The correct formatting of parts can ease the design process. This includes things like unit systems, and part information.

9.2.1. *Units*

To change the units in a part document, select ‘tools’, then select ‘options’ at the bottom of the tool toolbar. Select the ‘document properties’ tab, and then select ‘units’ in the outline tree.

9.2.2. *Hole Wizard*

The hole wizard is a very useful feature in SolidWorks. It allows the user to quickly and easily add holes, tapped holes, pipe tapped holes, or helicoil tapped holes. In addition to this, any manner of counterbore or countersink can be added to the hole. To add tapped holes, select the straight tapped hole type. This will display a standard drop down menu. From this menu, select the appropriate standard. Next, select the appropriate end condition. SolidWorks will select the correct drill size and all of this information will be displayed on a drawing by using the Hole Callout command.

9.2.3. *Part Information*

Editing part information is an easy way to streamline the drawing process after you are done with the 3D modeling. In order to change the part information, select the ‘file’ toolbar, and then ‘properties’ near the bottom. This will bring up a window with three tabs. In the center tab, custom, select the first box under the property name heading. This will display a drop down menu. Select the desired property from this menu. Complete the rest of the property values. Some of the properties are linked to other commands. For example, if you select material as the property, then also select material as the value, SolidWorks will link the material property to the material specified in the design tree to the left. If a property that you desire is not in the list, select edit list in the upper right corner. Here you can add or remove properties as you see fit. When you create a drawing from this part, you will see that any of the properties that you have filled in will be displayed in the title block. For instance, if you fill in part description, the title on the drawing will change to reflect that.

9.3. General Practices

9.3.1. *Updating Files*

Always update the solid model and drawing files to be the same. That is, do not override a dimension in a drawing without first changing that dimension in the solid model. This can cause problems, because some machines are set up to take dimensions directly from the solid model, and will use the dimensions that are included in the solid model instead of what is printed from the drawing file. Naturally, this can lead to machining errors.

INDEX

APPENDIX: Hole Making, Threading and Inserts

Table 1 PEM Nut Sizes (Note: Chart Is Valid For Steel and Stainless Steel Nuts Only)

PEM Nut Thread Size	Part Numbers	Hole Size	Drill Size
2-56	S-256-X, CLSS-256-X	0.166	#19
3-48	S-348-X, CLSS-348-X	0.166	#19
4-40	S-440-X, CLSS-440-X	0.166	#19
6-32	S-632-X, CLSS-632-X	0.1875	3/16
8-32	S-832-X, CLSS-832-X	0.213	#3
10-24	S-024-X, CLSS-024-X	0.250	1/4
10-32	S-032-X, CLSS-032-X	0.250	1/4
12-24	S-1224-X, CLSS-1224-X	0.277	J
1/4-20	S-0420-X, CLSS-0420-X	0.3437	11/32
1/4-28	S-0428-X, CLSS-0428-X	0.3437	11/32
5/16-18	S-0518-X, CLSS-0518-X	0.413	Z
5/16-24	S-0524-X, CLSS-0524-X	0.413	Z
3/8-16	S-0616-X, CLSS-0616-X	0.500	1/2
3/8-24	S-0624-X, CLSS-0624-X	0.500	1/2
1/2-13	S-0813-X	0.6562	21/32
1/2-20	S-0820-X	0.6562	21/32

Table 2 PEM Stud Sizes

PEM Stud Thread Size	Part Numbers	Hole Size	Drill	Minimum Mat. Thickness
2-56	FH-256, FHS-256	0.086	44	0.040
4-40	FH-440, FHS-440	0.111	34	0.040
6-32	FH-632, FHS-632	0.136	29	0.040
8-32	FH-832, FHS-832	0.166	19	0.040
10-24	FH-024, FHS-024	0.189	12	0.040
10-32	FH-032, FHS-032	0.189	12	0.040
1/4-20	FH-0420, FHS-0420	0.250	1/4	0.062
5/16-18	FH-0518, FHS-0518	0.313	5/16	0.093

Table 3 Pem Nut Dash Number Size Convention

Thread Range	Dash Number	Minimum Mat. Thickness
2-56 - 12-24	0	0.030
	1	0.040
	2	0.056
	3	0.091
1/4-20 5/16-24	1	0.056
	2	0.091
	3	0.125
3/8-16 1/2-20	1	0.091
	2	0.125
	3	0.250

Table 4 Miniature and Lightweight Keensert Thread Sizes and Depths

Internal Thread Size	External Thread Size	Minimum Depth
0-80	6-40	0.13
2-56	6-40	0.13
2-56	8-32	0.14
4-40	10-32	0.16
6-32	12-28	0.16
8-32	1/4-28	0.21
10-32	5/16-18	0.37
10-24	5/16-18	0.37
1/4-28	3/8-16	0.43
1/4-20	3/8-16	0.43
5/16-24	7/16-14	0.5
5/16-18	7/16-14	0.5
3/8-24	1/2-13	0.56
3/8-16	1/2-13	0.56
7/16-20	9/16-12	0.62
7/16-14	9/16-12	0.62
1/2-20	5/8-11	0.68
1/2-13	5/8-11	0.68

Table 5 Heavy Duty Keensert Thread Sizes and Depths

Internal Thread Size	External Thread Size	Minimum Depth
8-32	5/16-18	0.37
10-32	3/8-16	0.37
10-24	3/8-16	0.37
1/4-28	7/16-14	0.43
1/4-20	7/16-14	0.43
5/16-24	1/2-13	0.50
5/16-18	1/2-13	0.50
3/8-24	9/16-12	0.56
3/8-16	9/16-12	0.56
7/16-20	5/8-11	0.68
7/16-14	5/8-11	0.68
1/2-20	11/16-11	0.75
1/2-13	11/16-11	0.75
9/16-18	13/16-16	0.94
9/16-12	13/16-16	0.94
5/8-18	7/8-14	1.00
5/8-11	7/8-14	1.00
3/4-16	1-1/8-12	1.44
3/4-10	1-1/8-12	1.44
7/8-14	1-1/4-12	1.56
7/8-9	1-1/4-12	1.56
1-12	1-3/8-12	1.68
1-8	1-3/8-12	1.68

Table 6 Extra Heavy Duty Thread Sizes and Depths

Internal Thread Size	External Thread Size	Minimum Depth
6-32	5/16-18	0.31
8-32	3/8-16	0.37
10-32	7/16-14	0.37
10-24	7/16-14	0.37
1/4-28	1/2-13	0.44
1/4-20	1/2-13	0.44
5/16-24	9/16-12	0.50
5/16-18	9/16-12	0.50
3/8-24	5/8-11	0.56
3/8-16	5/8-11	0.56
7/16-20	11/16-11	0.68
7/16-14	11/16-11	0.68
1/2-20	13/16-16	0.75
1/2-13	13/16-16	0.75
9/16-18	7/8-14	0.94
9/16-12	7/8-14	0.94
5/8-18	1-12	1.00
5/8-11	1-12	1.00
3/8-16	1-1/4-12	1.44
3/8-10	1-1/4-12	1.44
7/8-14	1-3/8-12	1.56
7/8-9	1-3/8-12	1.56
1-12	1-1/2-12	1.68
1-8	1-1/2-12	1.68

Table 7 Expected Tolerances of Drilled Holes

Drilled Hole Size	Tolerance
.0135 - .125	+.004/-.001
.1260 - .2500	+.005/-.001
.2510 - .5000	+.006/-.001
.5010 - .7500	+.008/-.001
.7510 - 1.000	+.010/-.001
1.001 - 2.000	+.012/-.001

Table 8 Drill and Tap Drill Sizes

Drill Size	Decimal Inch	75% Tap Drill	Drill Size	Decimal Inch	75% Tap Drill	Drill Size	Decimal Inch	75% Tap Drill	Drill Size	Decimal Inch	75% Tap Drill
80	0.0135		42	0.0935	4-48	13/64	0.2031	10-24 STI	13/32	0.4062	
79	0.0145		3/32	0.0937	2-56 STI	6	0.2040		Z	0.4130	
1/64	0.0156		41	0.0960		5	0.2055		27/64	0.4219	1/2-13
78	0.0160		40	0.0980		4	0.2090		7/16	0.4375	
77	0.0180		39	0.0995		3	0.2130	1/4-28	29/64	0.4531	1/2-20, 7/16-14, 20 STI
76	0.0200		38	0.1015	5-40	2	0.2210	12-24 STI	15/32	0.4687	
75	0.0210		37	0.1040	5-44	1	0.2280		31/64	0.4844	9/16-12
74	0.0225		36	0.1065	6-32	A	0.2340		1/2	0.5000	
73	0.0240		7/64	0.1093		15/64	0.2344		33/64	0.5156	9/16-18, 1/2-13, 20 STI
72	0.0250		35	0.1100		B	0.2380		17/32	0.5312	5/8-18
71	0.0260		34	0.1110		C	0.2420		35/64	0.5469	
70	0.0280		33	0.1130	6-40, 3-48 STI	D	0.2460		9/16	0.5625	
69	0.0292		32	0.1160		E	0.2500		37/64	0.5781	
68	0.0310		31	0.1200	4-40 STI	1/4	0.2500		19/32	0.5937	
1/32	0.0313		1/8	0.1250		F	0.2570	5/16-18	39/64	0.6094	
67	0.0320		30	0.1285	4-48, 5-40 STI	G	0.2610	1/4-28 STI	5/8	0.6250	
66	0.0330		29	0.1360	8-32, 8-36	17/64	0.2656	1/4-20 STI	41/64	0.6406	
65	0.0350		28	0.1405		H	0.2660		21/32	0.6562	3/4-10
64	0.0360		9/64	0.1406		I	0.2720	5/16-24	43/64	0.6719	
63	0.0370		27	0.1440		J	0.2770		11/16	0.6875	3/4-16
62	0.0380		26	0.1470	6-40 STI	K	0.2811		45/64	0.7031	
61	0.0390		25	0.1495	10-24, 6-32 STI	9/32	0.2812		23/32	0.7187	
60	0.0400		24	0.1520		L	0.2900		47/64	0.7344	
59	0.0410		23	0.1540		M	0.2950		3/4	0.7500	
58	0.0420		5/32	0.1562		19/64	0.2968		49/64	0.7656	7/8-9
57	0.0430		22	0.1570		N	0.3020		25/31	0.7812	
56	0.0465		21	0.1590	10-32	5/16	0.3125	3/8-16	51/64	0.7969	
3/64	0.0469	0-80	20	0.1610		O	0.3160		13/16	0.8125	7/8-14
55	0.0520		19	0.1660		P	0.3230		53/64	0.8281	
54	0.0550		18	0.1695		21/64	0.3281	5/16-24 STI	27/32	0.8437	
53	0.0595	1-64, 1-72	11/64	0.1719		Q	0.3320	3/8-24, 5/16-18 STI	55/64	0.8594	
1/16	0.0625		17	0.1730	8-32 STI	R	0.3390		7/8	0.8750	1-8
52	0.0635		16	0.1770	12-24	11/32	0.3437		57/64	0.8906	
51	0.0670		15	0.1800		S	0.3480		29/32	0.9062	
50	0.0700	2-56, 2-64	14	0.1820	12-28	T	0.3580		59/64	0.9219	
49	0.0730		13	0.1850		23/64	0.3594		15/16	0.9375	
48	0.0760		3/16	0.1875		U	0.3680	7/16-14	61/64	0.9531	
5/64	0.0781		12	0.1890		3/8	0.3750		31/32	0.9587	
47	0.0785	3-48	11	0.1910		V	0.3770		63/64	0.9844	
46	0.0810		10	0.1935		W	0.3860		1	1.0000	
45	0.0820	3-56	9	0.1960		25/64	0.3906	7/16-20, 3/8-24 STI			
44	0.0860		8	0.1990		X	0.3970	3/8-16 STI			
43	0.0890	4-40	7	0.2010	1/4-20, 10-32 STI	Y	0.4040				