PROTOTYPE DESIGN AND MANUFACTURING MANUAL

University of Victoria

DEPARTMENT OF MECHANICAL ENGINEERING

RODNEY KATZ   *Rev. 0*

TABLE OF CONTENTS

INTRODUCTION

- The purpose of this manual is to present best practices for the design and manufacturing of prototype devices, low volume machined parts and assemblies.

- It is important to note that many of the methods and techniques outlined in the manual are specific to the UVic Mech. Machining Facility.
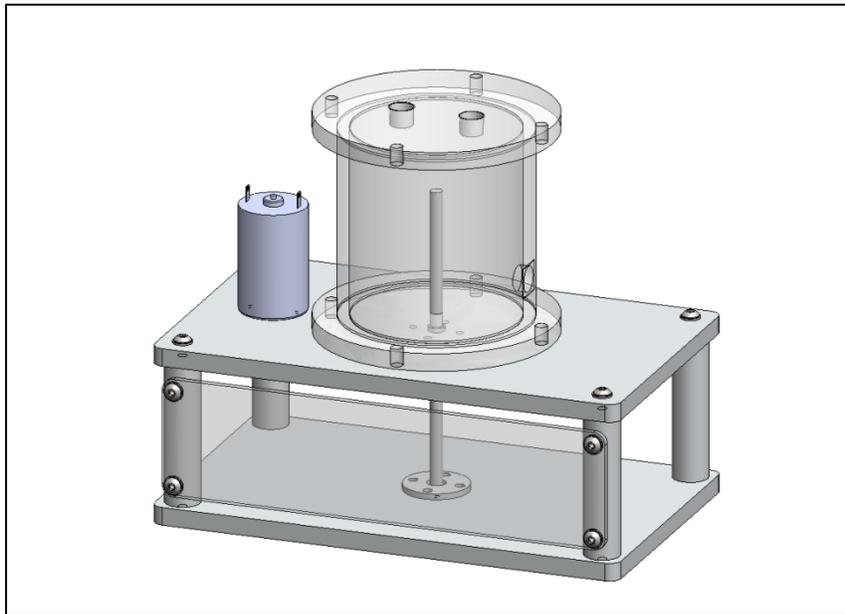

Figure 1.1 – Mixer device

- The manual is structured around the design and manufacturing of a single device (shown above in Figure 1.1) which encompasses many typical characteristics of mechanical features found in the Dept. Mechanical Engineering research apparatus.

- This design is very efficient to machine, incorporates inherent alignment, uses stock material readily available in the machine shop and is familiar with the shop machinists.

- The manual will also include comparison examples of less effective designs of the same device. These methods are often initially thought to be easier to fabricate but do not lend the same advantages of the recommended sample design shown above

- Design methodology is a very broad subject; this manual only covers the basics.

- Extended topics like high volume manufacturing, rapid prototyping and techniques used outside the Mech. Eng. Shop are not covered in this manual.

PART I - DESIGN

1 FIRST STEP TO DESIGN

- Design of a device should always begin with opening communication with the specific shop or facility where parts will be manufactured.
- The capabilities of the shop or facility need to be carefully considered in order to design economically feasible devices or equipment:
  - Machinery and shop specific expertise, CNC milling, lathe capability, sheet metal capabilities, welding capabilities, heavy machining, precision grinding, waterjet cutting, EDM, rapid prototyping, etc.)
  - Shop preferred manufacturing methods and materials.
  - Machine tool (lathe, mill) capacity. What is the largest or smallest size workpieces and cutting tools the shop's machines can comfortably accommodate.
  - Time line required to deliver parts.
- While in the design phase, continually communicating with shop personnel will always lead to designs which are less expensive and more efficient to build.
- Whenever possible, design parts in **inches**. The rationale being the machine shop's cutting tool and material inventory are in the imperial (inch) system. Designing in metric then converting to inches leads to odd dimensions thus complicating the design and machining process, resulting in a more costly and time consuming part
- Always keep in mind **stock material sizes** when designing parts. This is paramount to achieving an economically feasible design. Often the stock material size will be sufficient to accommodate your design with little additional machining required. In Canada we still work with stock material sizes in the inch system as most of our material comes from U.S. suppliers.

2 PROTOTYPING

- Generally prototype parts should incorporate the following:
  - o **Inherent Alignment**. Parts automatically fit into place. This eliminates the need for post machining and hand tool modification in order to fit parts together. Bosses and other features should be incorporated in the design to assure alignment of final assemblies.
  - o Ability to be adjusted with little rework. This could mean adding extra holes or elongating holes for unforeseen additions or when you may feel adjustability could be required.
  - o The use of easily machinable material i.e aluminum, acetyl (Delrin), PVC (plastic) where ever possible. Even though these materials may be twice or three times the cost of steel, their fabricating and machining costs will be drastically lower resulting in a less expensive part.
  - o Reduce the number of parts with tight tolerances (to lower cost and increase machining efficiency).
  - o Size parts with a consideration for material yield. E.g. aluminum sheet comes in 48" X 96" sheets. If you require four pieces 12 ½" X 12 ½" this will result in much waste material. Re-consider the design to work with 12" X 12" pieces. Sheet metal does not consume material for cuts where as plate material (3/16" and thicker) will consume approximately 3/16" per cut due to the sawing and clean-up process. Plate material is also supplied in 12" increments. Therefore if four 12" X 12" plates are required this will produce a large amount of excess material. Try to work the design with part sizes of 11.75" X 11.75"
  - o Tapped holes #6-32 and larger should be drilled all the way through in materials up to ¾" thick if possible as opposed to holes drilled to specific depths. It is always easier to tap a through hole opposed to a blind hole (hole that does not penetrate). The hole may only be tapped partially through.
  - o When designing a hole for a pressfit dowel pin, incorporate a smaller diameter through hole which can be used to push or knock the dowel pin out if required.

This will often be the case. (See Fig. 2.1 below). If no through hole is present it will be very difficult to remove the dowel pin and removal methods will result in damaging the pin and the maybe the hole.
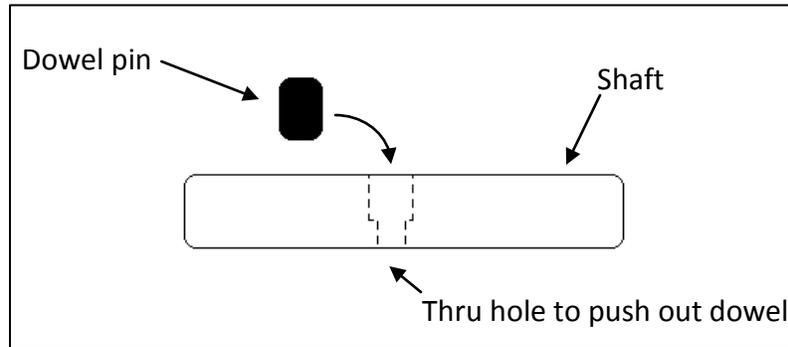

Figure 2.1 – Thru hole in shaft to allow removal of dowel pin

o   Tabs or marks for realigning parts if they are going to be placed back in a machine for post machining operations. This is often the case when the part needs holes aligned to each other from both ends. Another reason for alignment marks would be or if the part needs to be oriented in a specific rotational angle at assembly time. (See Figure 2.2).
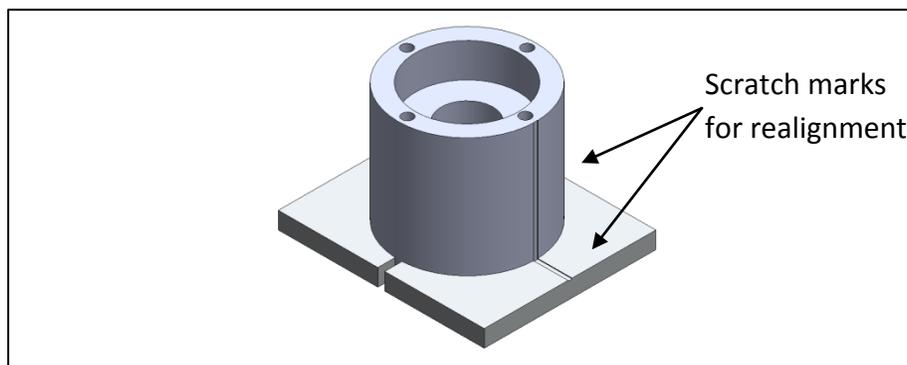

Figure 2.2 – Circular part and fixture with grooves for realignment

o   Flats on round parts. This allows for the part to be replaced into a vise with a known orientation for post machining or modifications at a later date from (See Figure 2.3)
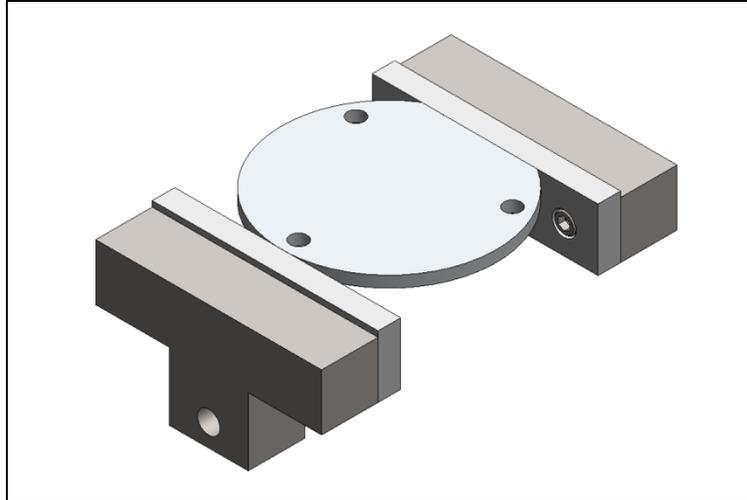
Figure 2.3 – Round part with flat self aligns in vise

- Avoid welding parts together. Welding does not allow for major changes if required. Separating parts can be very labour intensive and sometimes close to impossible. It is also very difficult to accurately align welded parts.

- Design the assembly of sufficient size to avoid working in confined spaces. Also avoid the use of tiny screws i.e. #2-56 and smaller.

- Try to keep the fastener thread type selection to a minimum. #10-32 is a very practical size for much of the metal work performed in the shop and is easily tapped. (use a   #10-24 for plastics)

- Utilise as many off-the-shelf items as possible. These will save many hours of design and machining work, paying for themselves many times over. Often off-the-shelf components may not meet all the requirements but can be machined to accommodate the design requirements.

2.1 EXAMPLE PROTOTYPE DEVICE

- The mixer device shown in Figure 2.4 is the example prototype assembly.

- Key components, which will be discussed in the manual, are noted on the diagram.
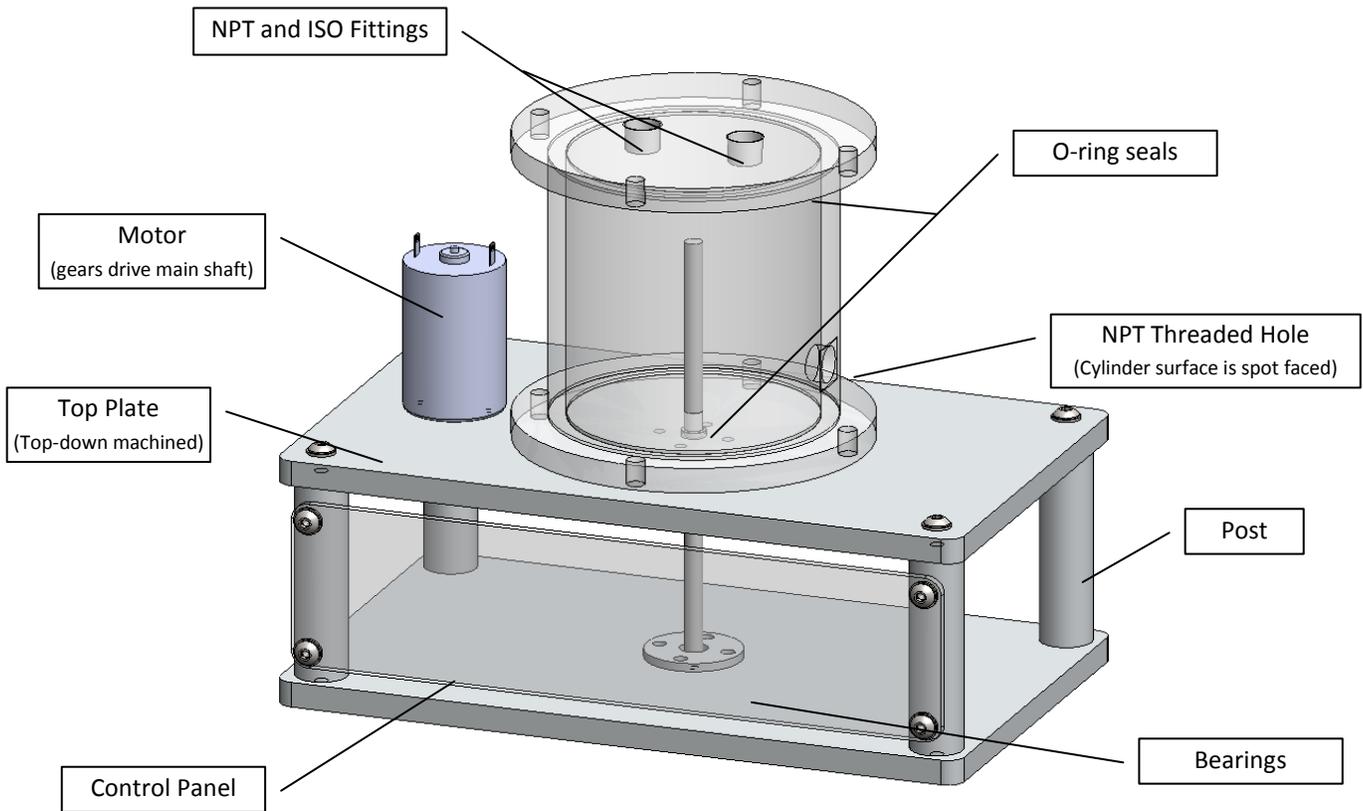
Figure 2.4 – Mixer Device

3 TOP DOWN MACHINING

- Flat parts (Top and Bottom plate) machined using a manual or CNC mill should be designed in order that they are machined on one or two sides only. This is called **Top Down** . Try to avoid having features machined in the sides of plates.

  o The actual process of cutting material is relatively quick. It is the setup and positioning of material and parts that takes majority of the time.

  o If a part can be setup once and then cut without rotating or repositioning the part machining efficiency will increase exponentially.

  o Figure 3.1 shows an example of a part which requires multiple setups to machine. This part will need to be repositioned in the vise four to five times to drill all the required holes in the sides consuming much time and effort. The more often a part is repositioned (clamped) in the machine the greater the possibility of misalignment of the features.

  o Also the long length of this part creates two possible problems:

    ▪ The milling machine being used may not have the required travel in the Z-axis needed to drill the end holes (shown in the third setup of Figure 3.1). Remember the drill chuck is extends approximately 3 inches, the ¼" dia. drill extends 4" and the part on end extends approximately 6" high, resulting in a consumption of 13" of the milling machines Z axis travel.

    ▪ If the part is machined while on end the stability of the part may not be enough to prevent material flex and result in inaccurate hole positioning.

    ▪ The part clamped on end (Fig 3.1 C) will also accentuate the deviation of the hole positioning (0.1 degree off perpendicular on a 6" protruding clamped part in the vise will cause a positioning error of the drilled hole by 0.010") Trying to ensuring the part is clamped perpendicular by this amount or better is very time consuming) .

Figure 3.1 – Multiple setups required for a part that cannot be machined TOP DOWN

4 ALIGNMENT!

- Alignment of parts and features is one of the most important aspects of mechanical design.

- If parts are designed with alignment in mind at all times it will prevent much frustration and added cost in the final assembly.
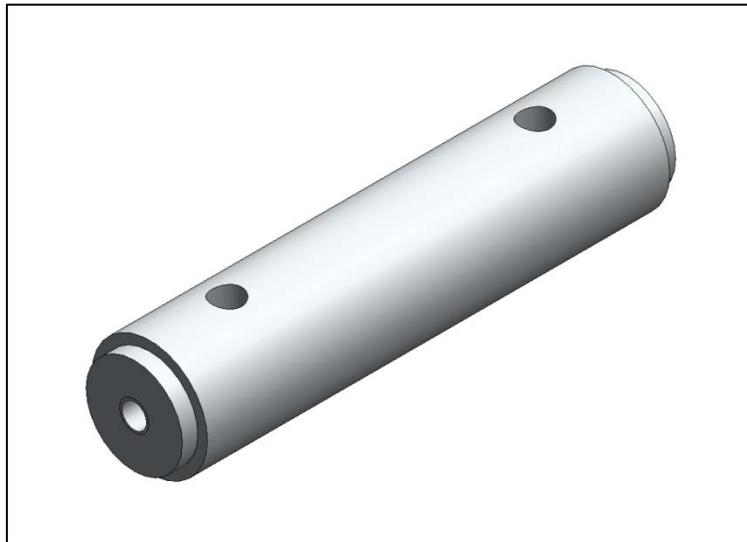

Figure 4.1 – Post with end bosses and extra holes in OD

- One basic method to align parts is to use posts with bosses on both ends. Figure 4.1 shows an example of a post which incorporates bosses to aid alignment.

- Incorporating a boss on each side of the post has the added advantage of allowing the post to be held in the lathe chuck so the critical length can be measured off the front face of the chuck jaws (the "Z" datum).

- The procedure used in the shop for accurately machining this type of post design is very efficient.

- Figure 4.2 shows how posts with integral bosses are used in the assembly.
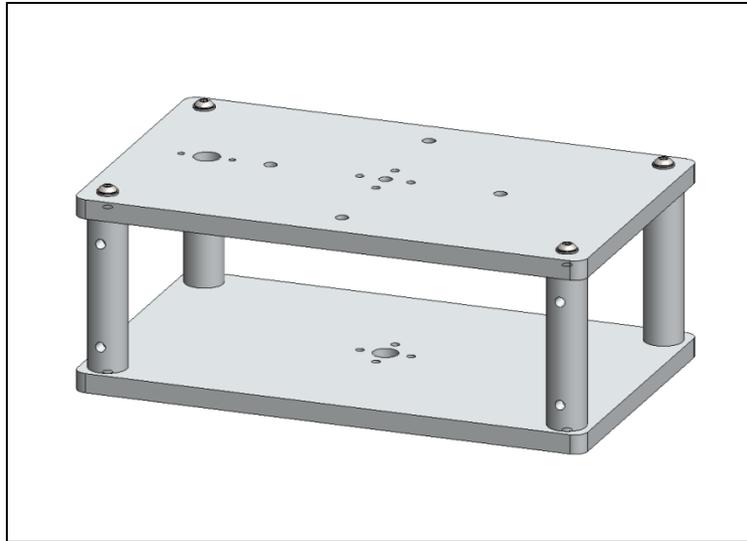
Figure 4.2 – Aligned plates and posts assembly

- Once the parts are assembled the two plates will be aligned sufficiently in order that no fussing or fiddling will be required to attain the desired result .

- This method allows the plates to be efficiently milled and drilled **Top Down** with **one** setup. No repositioning is required.

- This method can be altered to suit many different applications while still maintaining alignment of plate features (eg. centre holes, shaft holes, bearing bores etc.):
  - Plates can be different shapes if required (Fig. 4.3)
  - 2 or 3 posts can be used instead of 4 (Figure 4.3)
  - Not all posts need to be the same diameter
  - Plate thicknesses can be different

The machining process for posts is in Part II - Manufacturing, Section 1 (Page 39).
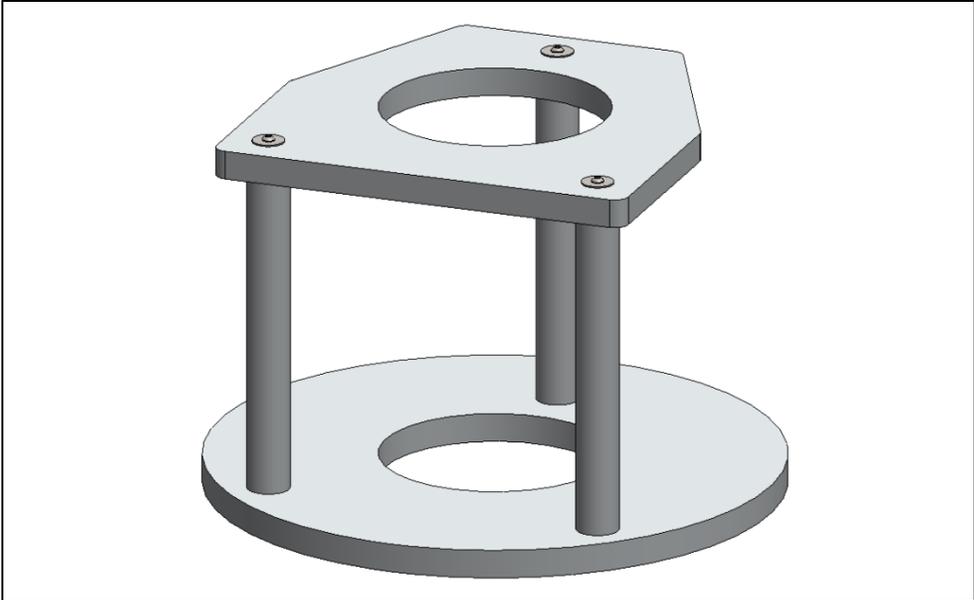
Figure 4.3 – 3 post design with difference shaped plates

Car chassis	Tapping Machine
Table	Submersible Chassis (fits in tube)
Winch	3 post design
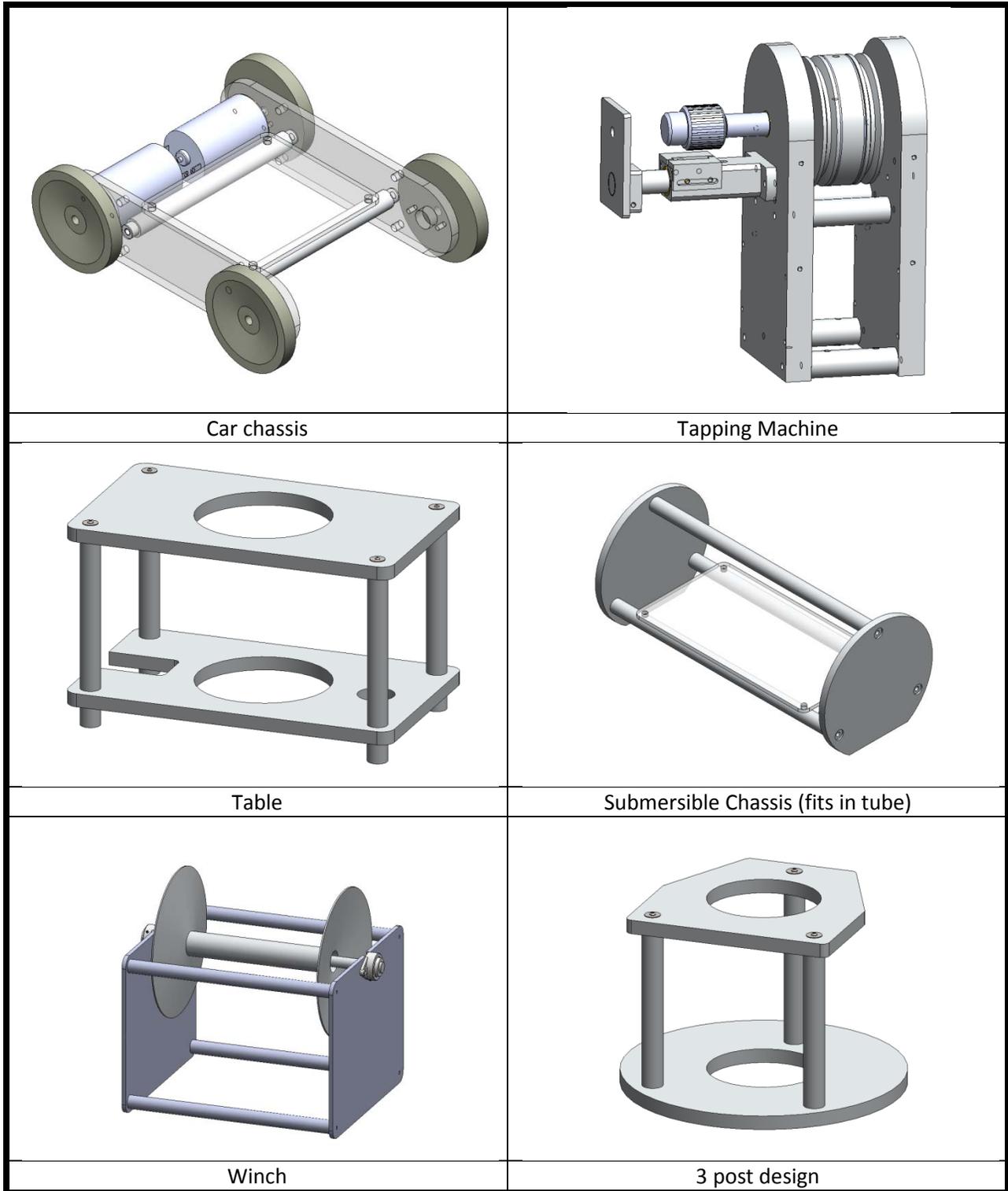
Figure 4.4 – Examples of how posts can be used

- Figure 4.4 shows examples of how posts can be used. (eg. Car chassis, table, ROV, etc.)

4.1 ALTERNATE METHODS

- This section presents some of the alternatives to using round posts with bosses. The alternatives may appear easier to fabricate but are often much more time consuming to machine and assemble and they do not incorporate inherent alignment features.

- Figure 4.5 shows a design similar to the recommended method
    - o The square posts are difficult to machine.
   - o Multiple setups are required to machine the posts.
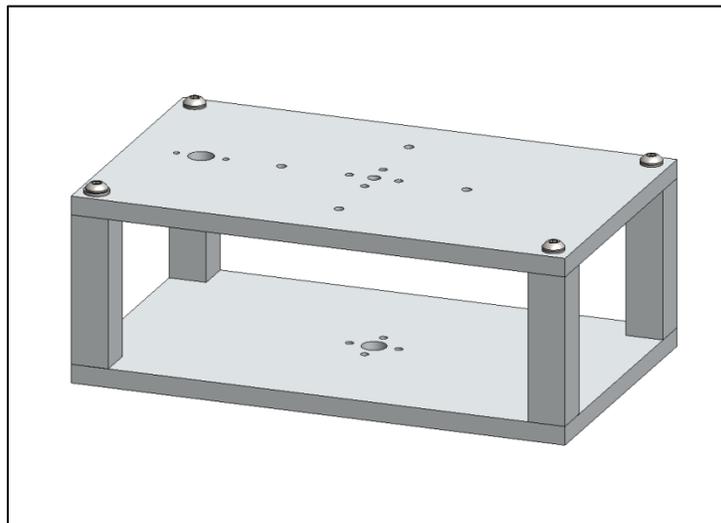
Figure 4.5 – Plate assembly using square posts

- Figure 4.6 and Figure 4.7 show the two plate assembly with angle brackets

- Angle brackets are appealing because they seem simple to machine (just cut to length and drill). **A note of caution!** Often what appears simple to fabricate results in being rather difficult and cumbersome. The method in Figure 4.6 is very time consuming to machine as the top and bottom plates cannot be machined **top down**. A total of 64 holes must be drilled. Half of the holes also need to be tapped in the sides of the plates.
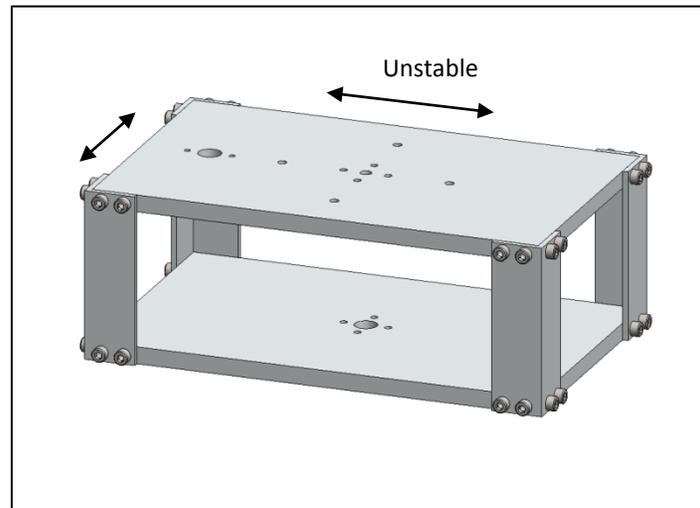
Figure 4.6 – Plate assembly using angle brackets

- The second angle bracket method uses welds.

  o This assembly requires 16 welds . Considerable time is required to setup and weld. Proper alignment of the plates will be virtually impossible often requiring some post machining or hand work.

  o In terms of prototyping welding does not allow parts to be easily modified.
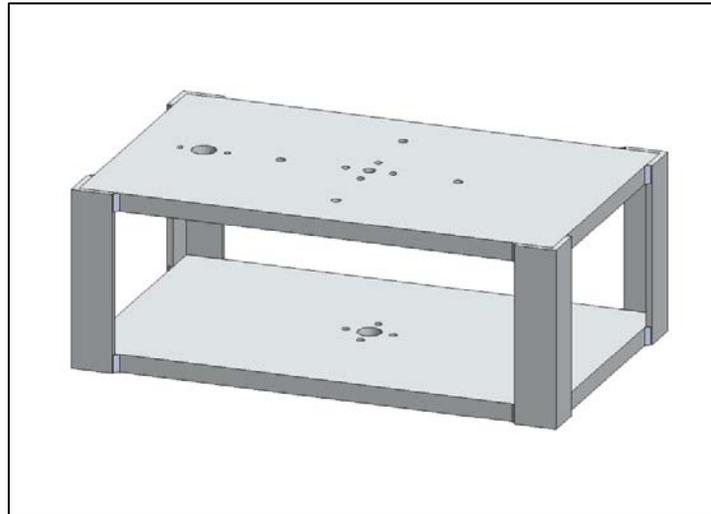

Figure 4.7 – Plate assembly using welded angle brackets

- Figure 4.8 shows a welded version of a winch frame. The winch in Fig 4.8 would take a considerable amount of time and effort to construct due of the number of welds and the requirement that the frame be accurately aligned for bearings blocks.
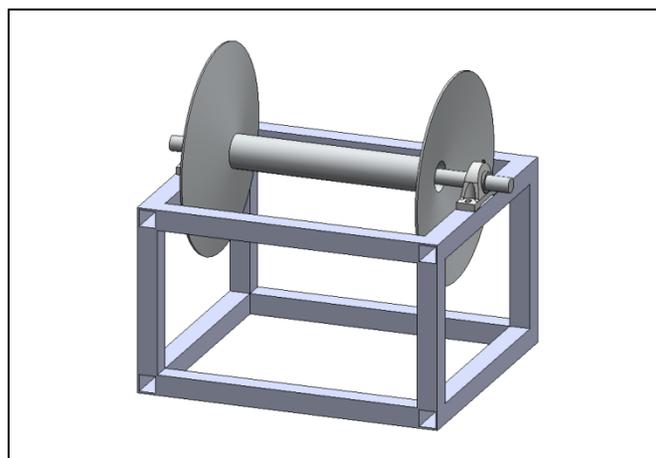

Figure 4.8 – Welded Winch Frame

NOTE: There is a time and place for welding but in most assemblies used in the Mech. Eng. Dept. it is not appropriate. Welding of small aluminum parts is very difficult and should be avoided.

- Figure 4.9 below shows the two plate assembly using C-channel. This method is appears relatively quick to machine but has significant alignment limitations:
    - C-channel comes in standard sizes therefore spacing of the plates has limited options. Spacers must be used to adjust plate spacing.
    - The size of extruded c-channel is not always accurate and the C-channel sides are often not perpendicular.
    - Ensuring the holes are drilled directly opposite one another in the C-channel is very difficult.
   - Securely holding the C-channel parts in the vice for drilling is cumbersome.
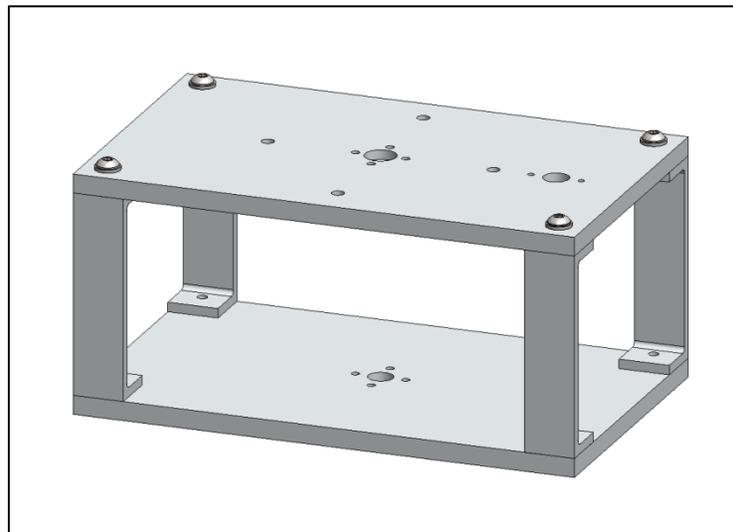
Figure 4.9 – Plate assembly using c-channel for posts

5 BEARINGS

- Bearings of many varieties are often used in mechanical design and are of utmost importance to the functionality of the assembly.

- Machined features that accept bearings must usually be made to tighter tolerances than other features.

- If a bore hole for a bearing is too small requiring excess force to seat the bearing it will often cause the bearing to run rough and lead to premature failure.

- If a bore hole is too loose the bearing will slop around and reduce the alignment of the shaft .

- Always include tolerances in drawings for dimensions related to bearings.

- If possible have the bearings available at time of machining. This will greatly help the machinist correctly size the bearing bore hole.

6 MOTOR MOUNTING

- Most small and midsize motors incorporate a boss and bolt holes on their face (Figure 1.1).

- The motor's boss assures concentric alignment of the motorshaft to the mounting surface.

- Correct tolerancing of the boss receiving bore is very important. If possible have the motor available to the machinist to check the bore size and fit at time of machining. This will often avoid time consuming rework of parts.

•
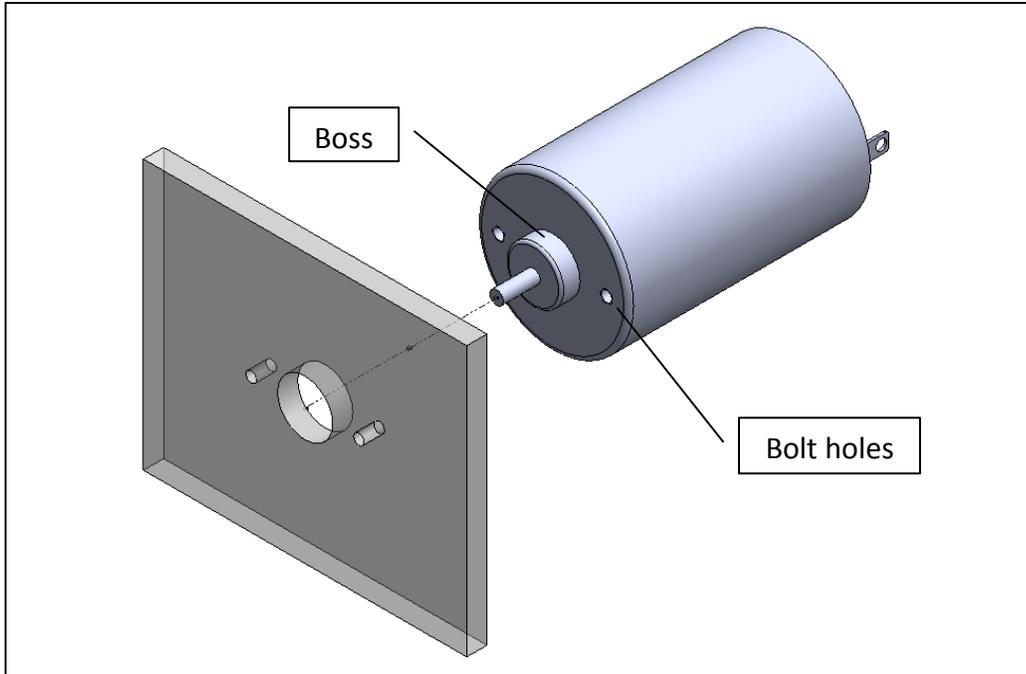
Figure 6.1 – Motor with boss and bolt holes

7 O-RINGS

- O-rings are one of the most common types of seals used. They are extremely efficient and very inexpensive. The machining processes required to accommodate O-rings is often simple if the part is designed with consideration.
- Sealing using O-rings allows for easy disassembly of the apparatus if modifications or cleaning is required.
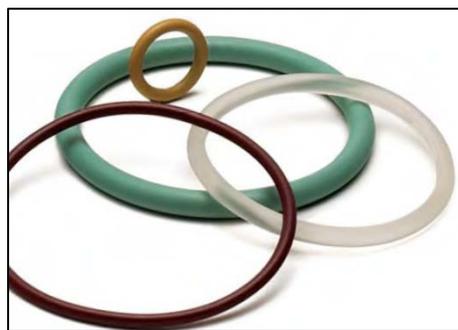
Figure 7.1 – O-rings

- Types of O-Ring Sealing Configurations

    o **Axial**: o-ring seal is on the face of a part.

    o **Radial**: o-ring seal is on the outer or inner wall of a part. Radial seals on small shafts can significantly decrease the strength of the shaft. In this case it is better to groove the inner diameter of the part the shaft will fit into.

    o Avoid the use of Radial O-ring configurations if possible, they are more difficult to machine and require much tighter tolerances in all respects. Note that the Plexiglas O-ring container configuration in the sample Mixer device uses an axial seal. There are several reasons for this choice:

    o It is easier to machine vs. a radial seal. If a radial seal were used, the Plexiglas container I.D. would have to be accurate and concentric. This is often not the case in stock tubular material. In order to attain an accurate and concentric I.D. on a tube it must be bored on a lathe which is very time consuming. For an axial seal only the ends of the container need to be machined which is a quick process

    o A radial seal is often makes it difficult to remove the end caps especially in oceanographic instruments.
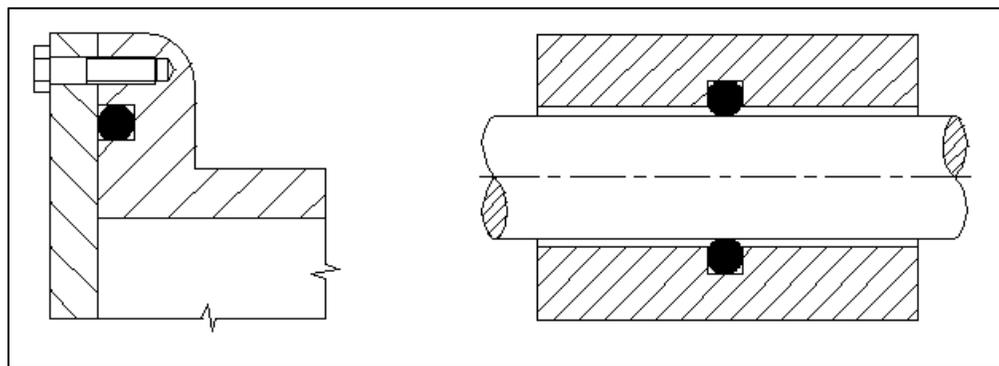

Figure 7.2   Axial O-Ring seal   **Radial O-Ring seal**
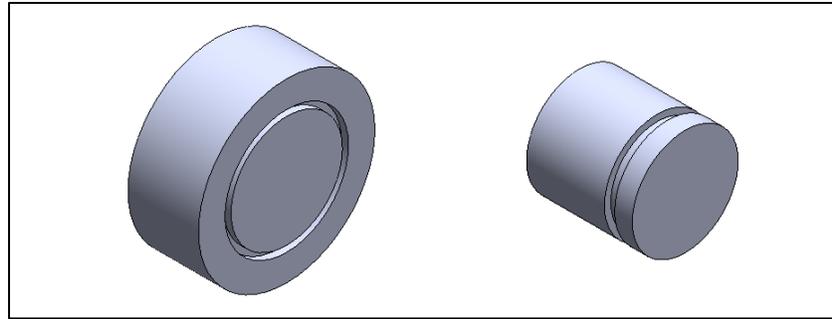
Figure 7.3   Axial gland   Radial gland

- The groove in which the O-Ring sits is called the "**gland**" (fig 7.3)

- Appendix V and VI show tables for gland sizes.

- The machining process for O-ring glands is in Part II - Manufacturing, Section 3

  (Page 48).

8 PIPES AND TUBES

- **Pipes**: Pipe sizes are referred to based on their nominal inner diameters, not their outer diameters. i.e. a ¾" schedule 40 pipe will have an O.D. of 1.050"

    o *Note:* The nominal inner diameter of pipe usually does not match the physical inner diameter. This is due to the different wall thicknesses (referred to as schedule sizes i.e sched. 40 or sched. 80). Different pipe materials have different O.D. sizes i.e. ¾" copper pipe has an O.D. of 0.875" vs. ¾" steel and aluminum pipe having an O.D. of 1.050".

- **Tubes**: Tube sizes are referred to based on their actual physical O.D. i.e. 1" tube is physically 1.00" O.D.

- **Pipe** is often less expensive than tube therefore it is used when larger quantities are required in the design or when fluid transport is required. **Tube** is used when a specific diameter is required.

8.1 PIPE AND FLUID FITTINGS

The following are the most common fluid fittings used in the Mech. Eng. Dept. research apparatus

- **NPT** (National Pipe Thread) is a standard for pipe fittings. NPT fittings are tapered to produce an effective friction seal when screwed into the mating fitting.

    Note: A ¼" NPT fitting has thread O.D. of approximately 1/2".

    NPT fittings are not used in thin material or sheet metal. A minimum of four threads must be engaged to produce a reliable seal.

- **ISO fittings** have straight threads and incorporate an O-ring in the base to create a seal. The mating surface for these o-rings must always be spot-faced to produce a reliable seal. See the offset hole in the top cap of the sample device. The spot face is required to produce a surface with machining marks concentric with the O-ring seal. If this is not done the striations or any scratches in the plastic plate would allow a leak. Whenever O-rings are used any machining marks or scratches must be concentric with the O-ring.

9 MATERIALS

- This section lists some of the more common materials used in the UVic Mech. Eng. Machine Shop. Consult **McMaster-Carr (www.mcmaster.com)** for more detailed information on the material sizes, characteristics and cost.

9.1 STOCK MATERIALS

- When designing keep in mind the standard sizes of stock materials.
  - In most cases stock materials cannot be depended on for accurate sizes or geometric tolerances (perpendicular sides, parallelism, etc.)
  - An example exception is precision ground steel rod which is ground to tight tolerances.
  - If high precision alignment is required the stock material must often be machined on the significant mating surfaces. Aluminum plate has good surface flatness as opposed to aluminum extrusion and is approximately twice the price. Use of aluminum plate can often reduce the amount of machining required paying for itself in the final result.

9.2 ALUMINUM

Aluminum is most often the material of choice in the Mech. Eng. Machining Facility.

- **Aluminum** Characteristics and Advantages
  - Lightweight
  - Large size selection
  - Good strength
  - Good corrosion resistance
  - Easy to machine
  - Clean to work
  - Good esthetics

- o High heat conductivity

- o Easily recyclable

- o Readily available

- o Difficult to weld properly

- o Can be anodized to achieve a very highly corrosion resistant surface and coloured surface.

9.3 STEEL

- **Mild Steel** Characteristics

  - o High strength

  - o Heavy

  - o Corrodes easily (rust)

  - o   Easy to Weld

  - o Certain types of steel prove difficult to machine

  - o Low cost

  - o   Considerably more costly to machine than aluminum

- Mild steel is not used extensively in the Mech. Eng. Machining facility. It is generally used for shafts, heavy welded structures, axels etc.

- **Stainless Steel** characteristics

  - o Very good corrosion resistance (Oceanographic applications)

  - o High temperature

  - o Very high strength

  - o Very heavy

  - o Expensive

  - o Welds easily

  - o Costly to machine, use only where absolutely necessary

9.4 PVC PLASTIC

- **PVC** characteristics:

  o Good corrosion resistance

  o Easy to machine

  o Lightweight

  o Can be bonded easily. Note: A mechanical interconnectivity of the two parts is always required when bonding PVC. It is not advisable to butt joint PVC.

  o Not recommended for wear applications due to high friction coefficient.

9.5 DELRIN (ACETAL)

- **Delrin (Acetal)** is often used as a replacement for aluminum parts. Due to its excellent machinability try to implement it into designs where possible. One major advantage being that it does not need any coating to be corrosion proof. It also produces very esthetically pleasing parts due its sheen and colour.

- **Delrin (Acetal)** has the following characteristics:

  o Excellent machinability

  o Corrosion proof

  o Good relative strength

  o Lightweight

  o Tough and wear resistant

  o Very low friction coefficient

  o Solvent and fuel resistant

  o White or black in colour

  o Commonly available in round rod/bar, sheet, and plate

- Some applications of Delrin:

  o Bushings for low speed applications

  o Wear pads

  o Fluid fittings

  o Gears , pulleys, idlers

- Important Notes:

O **Delrin** cannot easily be bonded to itself or other materials.

9.6 PLEXIGLAS

- Plexiglas characteristics:

  o Lightweight

  o Optically clear

  o Brittle

  o Scratches easily

  o Low operating temperature band

  o Commonly available in sheet, tube and solid round

  o Bonds to itself very easily

- Plexiglas can be bent using heat.

  o Bending Plexiglas is not recommended for prototype design because changes and adjustments cannot be made.

  o Accurately heat bending Plexiglas is can prove to be difficult.

Plexiglas can be easily bonded together with a butt joint resulting in relatively strong bond (Figure 9.1).

  o Surfaces must be smooth and flat for bonding Plexiglas i.e. routed or milled.
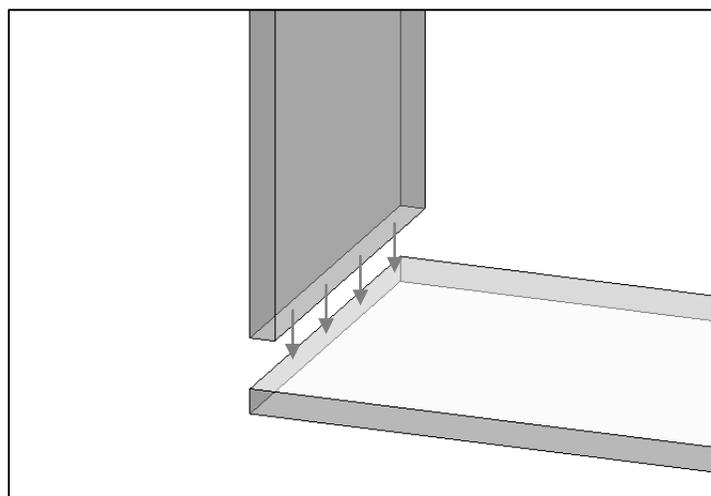
Figure 9.1 – Plexiglas bonded butt joint

10   DRAWINGS

- Try to always include an assembly view with a set of submitted drawings.
    - This will help the shop understand the purpose of the parts and where extra care should be taken when machining. If tolerances or dimensions were omitted in a drawing it will often allow the shop to make an informed judgment if they are not able to contact you.
- Make sure the drawing scale is noted so if dimensions need to be checked or are missing they can be measured off the drawing.
- When prototyping, printing drawings with a 1:1 scale can be useful to see the affects of changes on the part.
- General information should be included in the drawing title block. An example title block is shown in Figure 10.1.
    - Details to include: scale, quantity, material, part #, contact info, tolerances

Comments:	Company Name		
Dimensions in inches unless noted.	Project:		
Tolerances:	Part:		Qty. 1
.xxx ±.003 .xx ±.010 .x ±.015	Date: 7/22/2009	Drawn by	REV. A
unless otherwise stated	SCALE: 1:1	MATERIAL:	SHEET 1 OF 1

Figure 10.1 – Example Title Block

10.1   COMMON DRAWING MISTAKES

- Too many decimals places

- Dimensions referenced from wrong side of part edge

- Font to big or too small (should be 12 pt. → 14 pt.)

- Arrow heads too big

- Too many hidden lines shown making drawing difficult to interpret (use a section view if needed to show hidden details)

- Too much information on one drawing sheet (use more than one drawing sheet to show part more clearly if this is the case)

10.2   DIMENSIONING

- Use ordinate dimensioning to make drawings easier to read.

- Mark the origin of a part based on the origin which will be used when the part is machined.

- Draw ordinate lines on the side of the part which is closest to the detail they are showing the position of.

- The following pages show examples of drawings which are cluttered and difficult to understand followed by a drawing which uses best practices for dimensioning drawings.

- Dimension drawing features according to the shop preferred method or specific CNC mill that will be used.
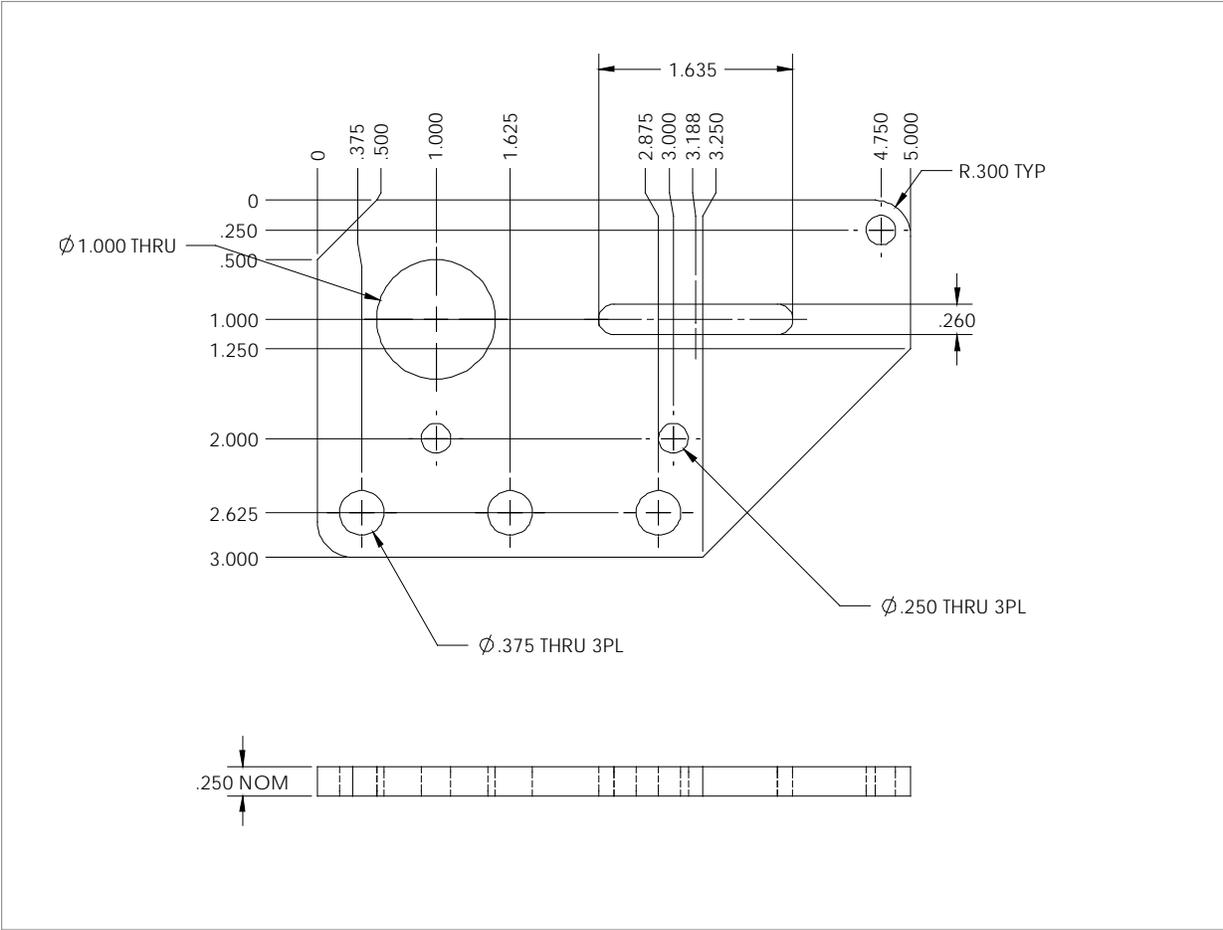
Figure 10.2 – Ordinate Dimensioning
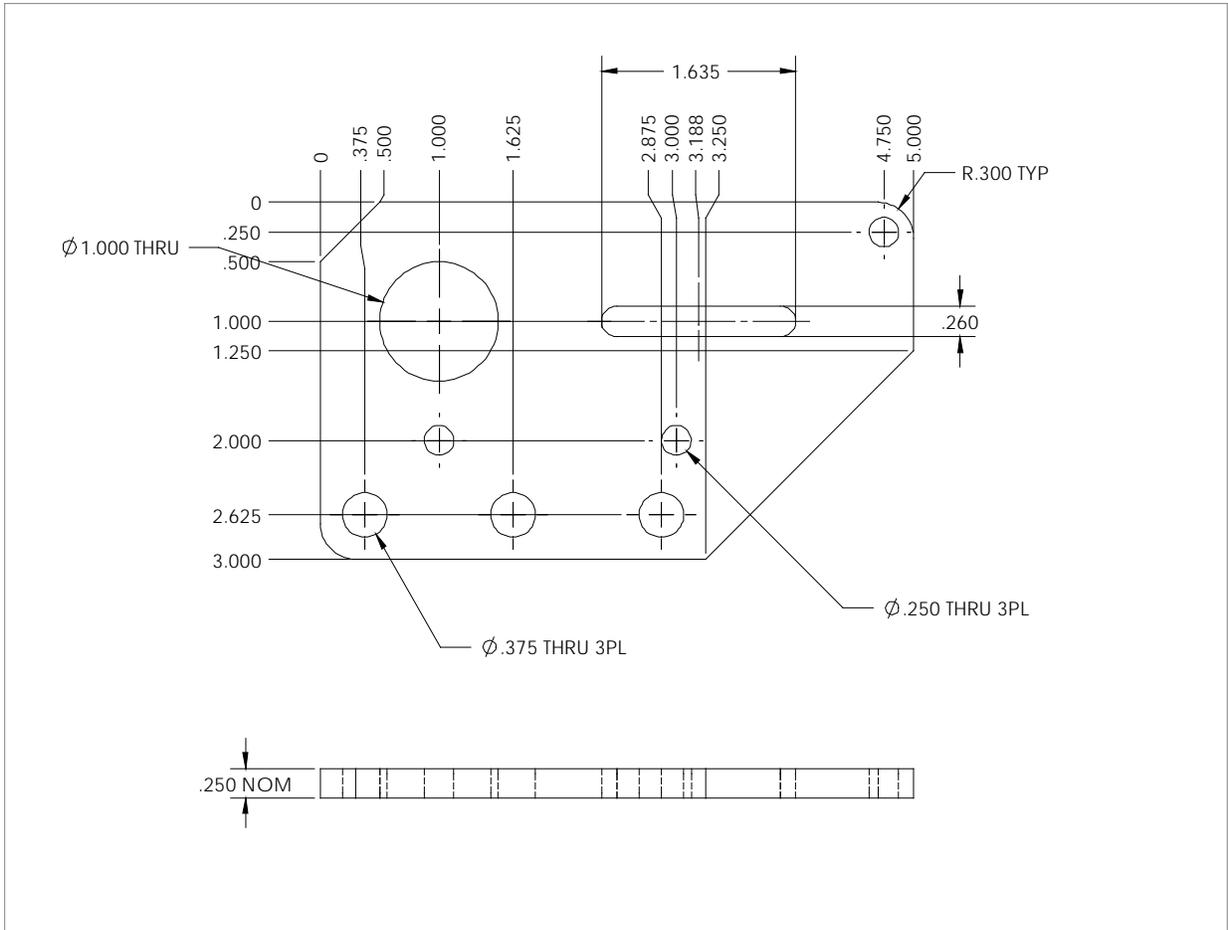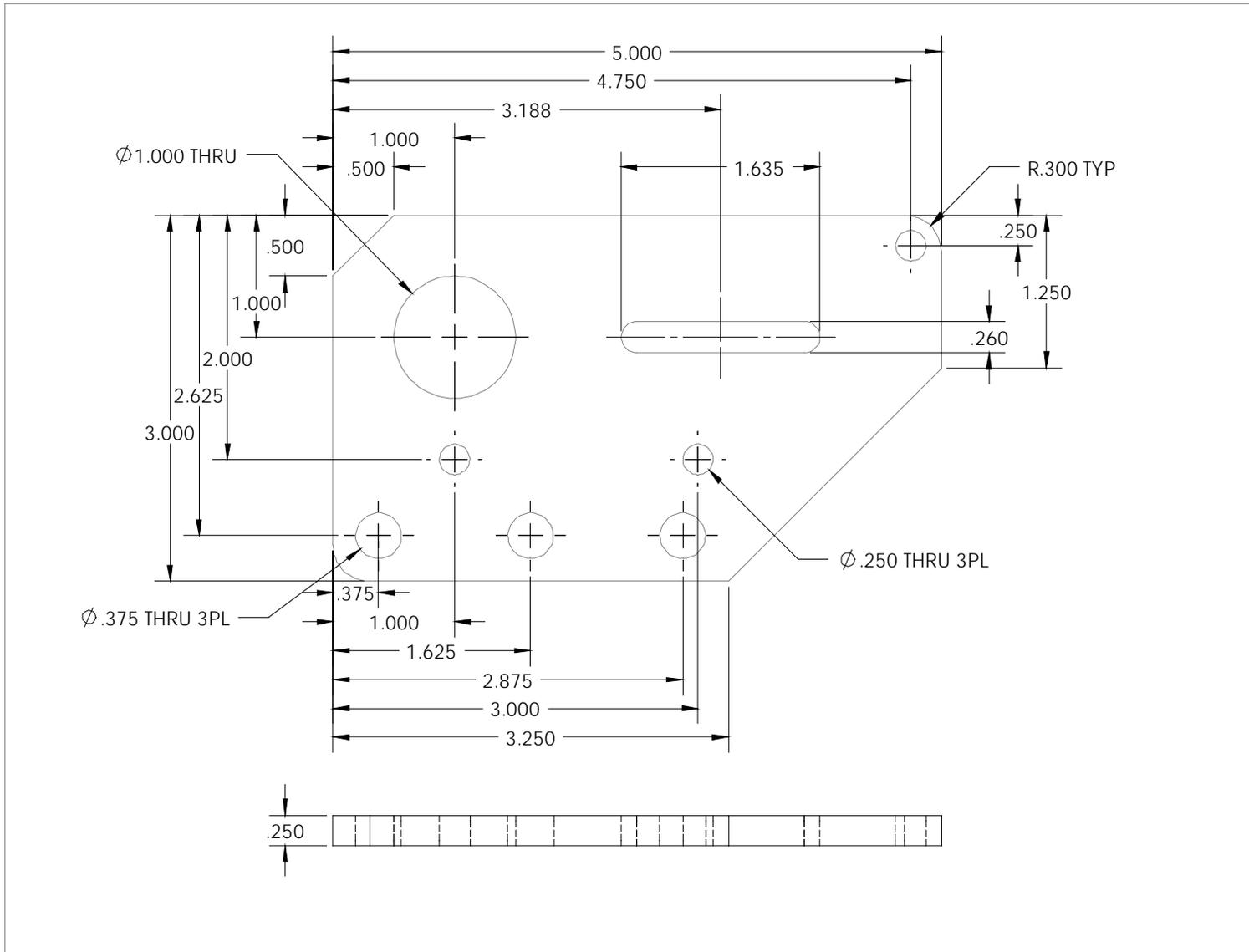
Figure 10.3 – Example of baseline dimensioning

Notes about drawing (Figure 10.3):

- Use of baseline dimensions makes drawing too cluttered and hard to read

- Elevation view shown with hidden lines is confusing. Should have hidden lines removed.

- Slot length dimension should be done to the total length of slot (as shown below).

Undesirable slot dimensioning

Preferred slot dimensioning style
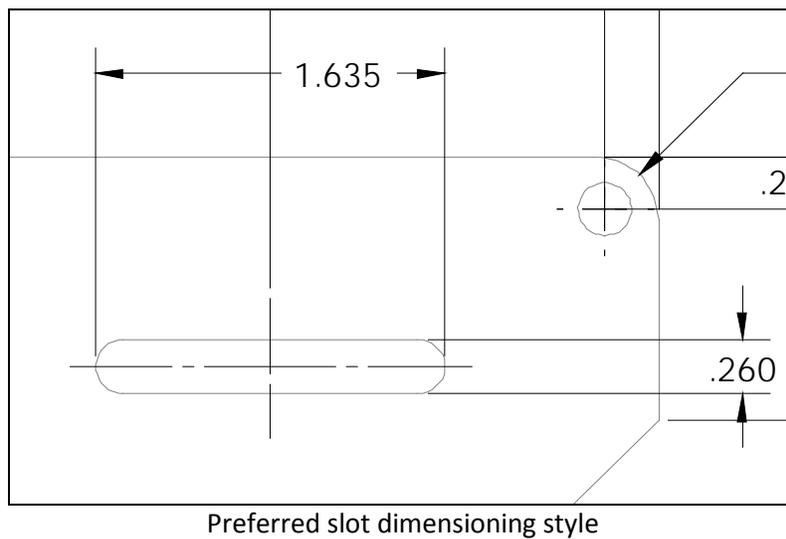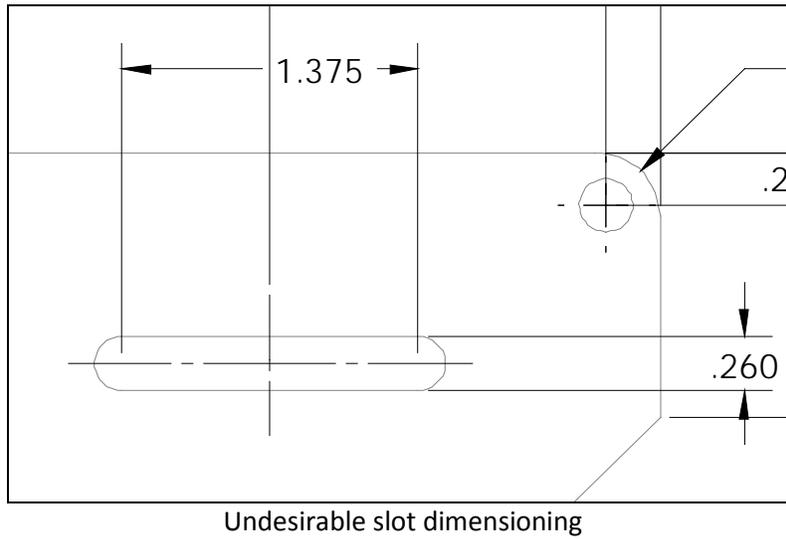
Figure 10.4 – Dimensioning of a slot length

Notes about drawing:

- Dimension lines are difficult to follow. Ordinate dimensions should be on the side of the part closest to the feature they are defining (shown in Figure 10.7).

- Elevation view of part should be shown without hidden lines (Figure 10.5).
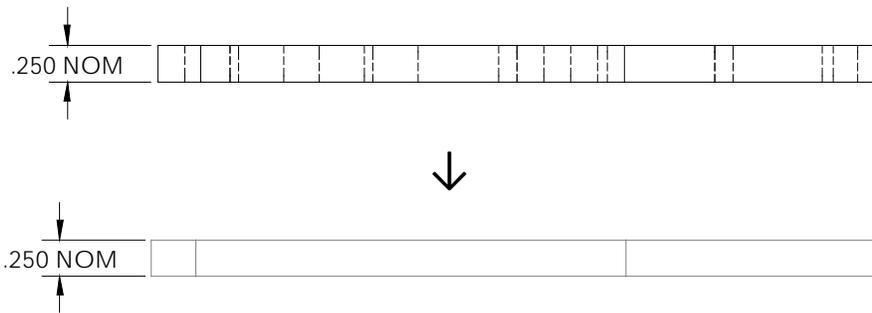
Figure 10.5 – Elevation view with and without hidden lines

- Dimension lines should not touch the part.
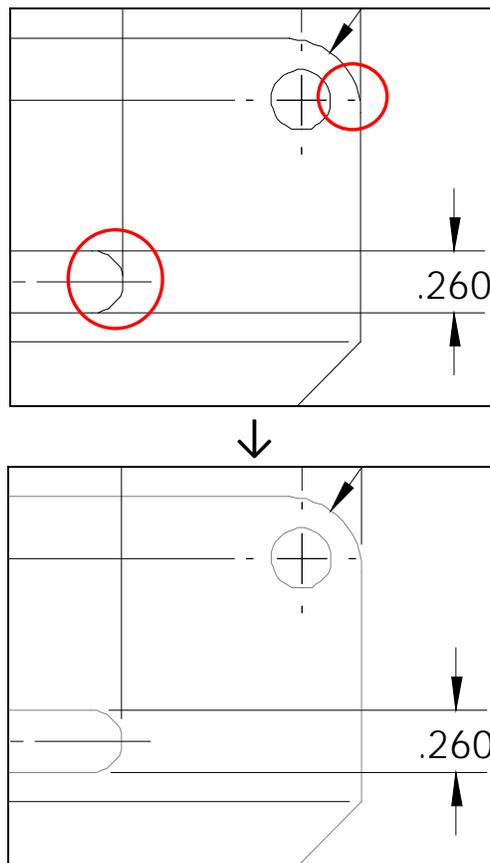
Figure 10.6 – Dimension line spacing
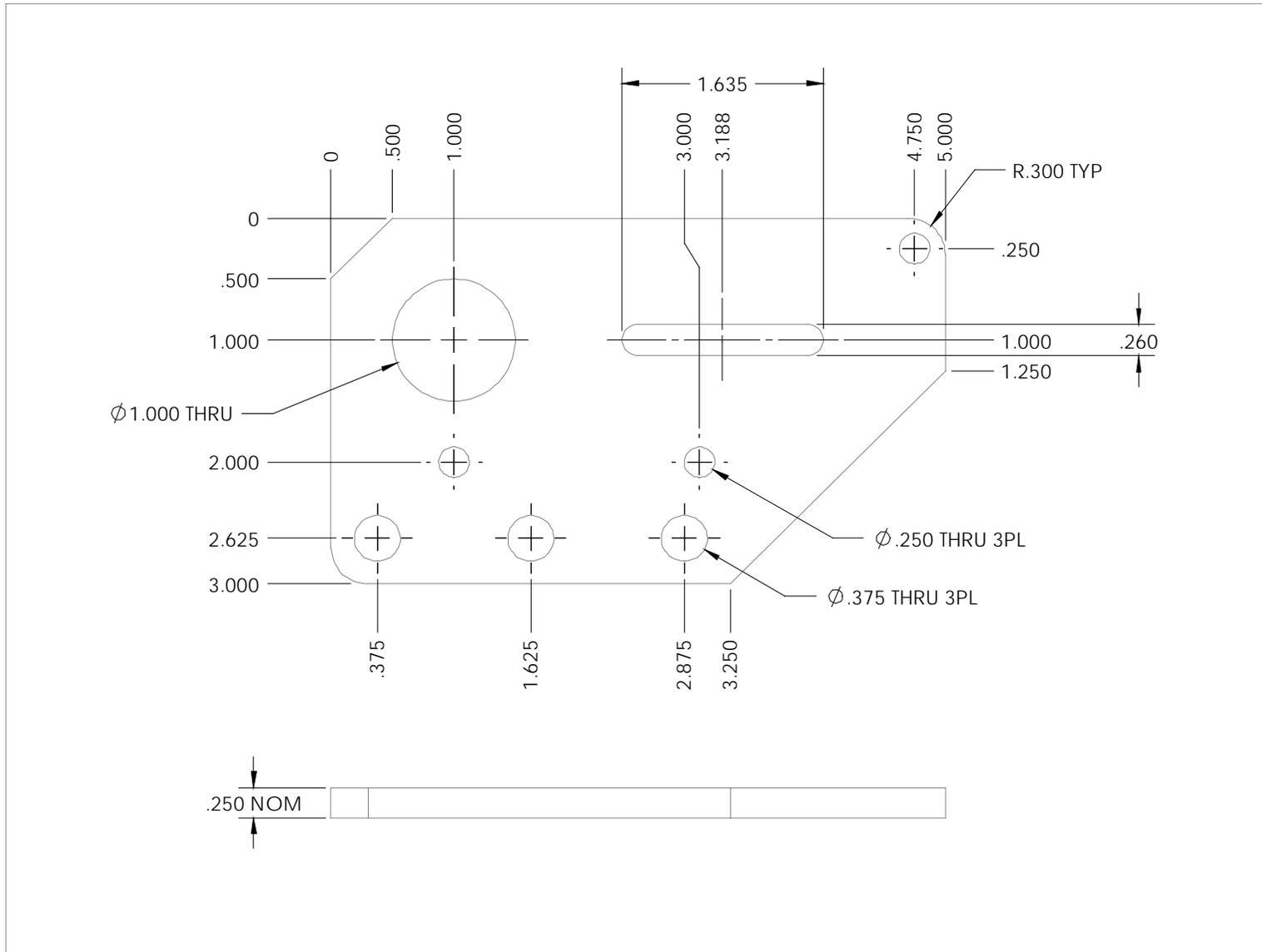
Figure 10.7 – Easy to read drawing using ordinate dimensions

Desirable drawing characteristics:

- The drawing above uses the preferred style of dimensioning.

- Dimension lines are spaced apart to avoid cluttering the drawing.

- Dimensions are close to the feature they describe.

- The slot's centre position is dimensioned and the width and **total** length are shown.

10.3 TOLERANCES

- Tolerances are a very important aspect of drawings which are often overlooked and are not given the attention they require.

All dimensions should have an associated tolerance. An efficient tolerancing method includes a section to the drawing sheet's title block which defines the standard tolerances based on number of decimal places (shown in

- Figure 10.8).

```
Tolerances
    .xxx        ±0.003
    .xx         ±0.01
    .x          ±0.015
Unless otherwise noted
```

Figure 10.8 – Drawing Block for Standard Tolerances

- If certain dimensions can accommodate wider tolerance then remove the appropriate decimals. When specifically needed add tolerances to individual dimensions to highlight the precision required.

10.4 ANILAM CNC MILL

- The CNC Milling machine used extensively in the Mech. Eng. Machine Shop utilises an Anilam control system.
- The following figures show the control system's different input screens (conversational programming) for various canned cycles.
  - A canned cycle is a set of machine operations initiated by a single line of code. It uses a "fill in the blank" type interface.
- **Use the screens shown below to determine the appropriate method to dimension part features which will be CNC milled (diagrams are specific to Anilam control system).**
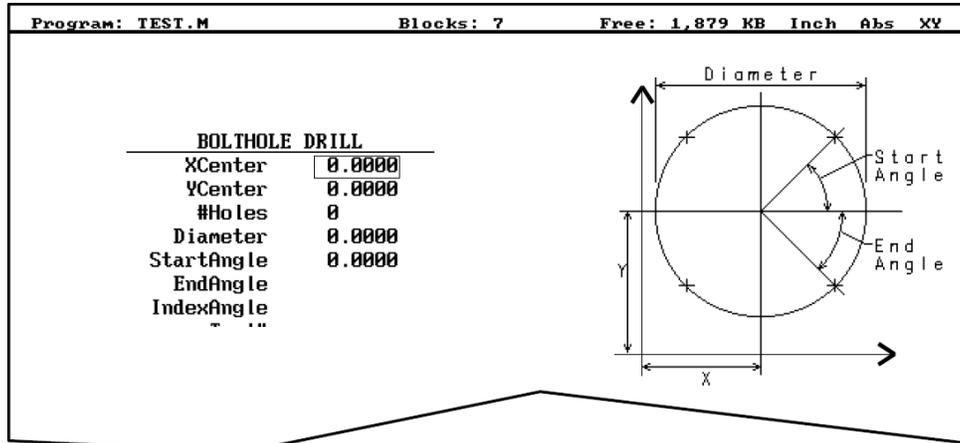
Figure 10.9 – Input screen for bolt hole circle
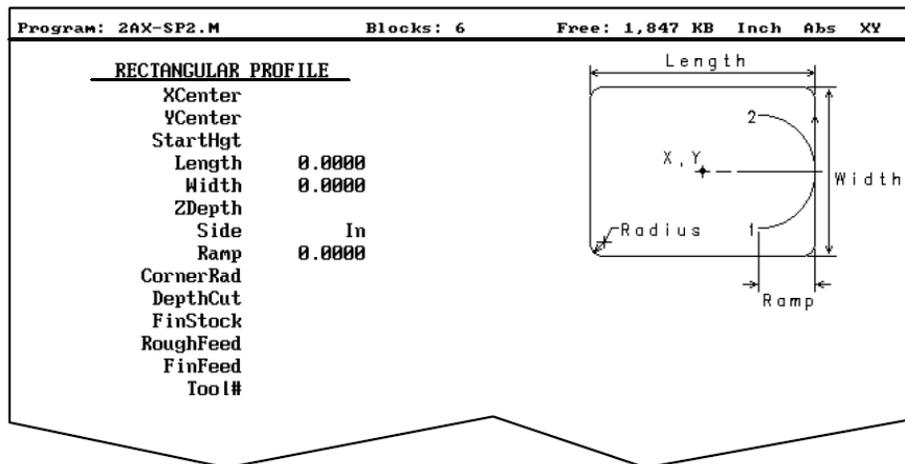

Figure 10.10 – Input screen for rectangular profile
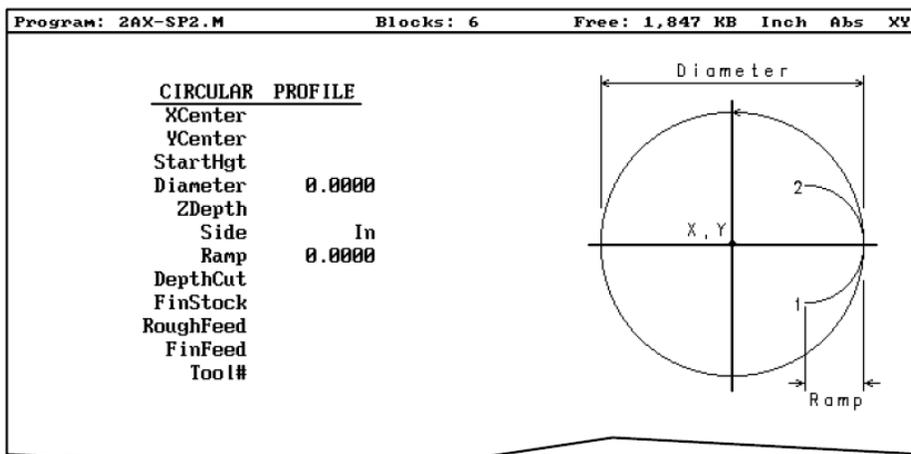
**Figure 10.11 – Input screen for circular profile**
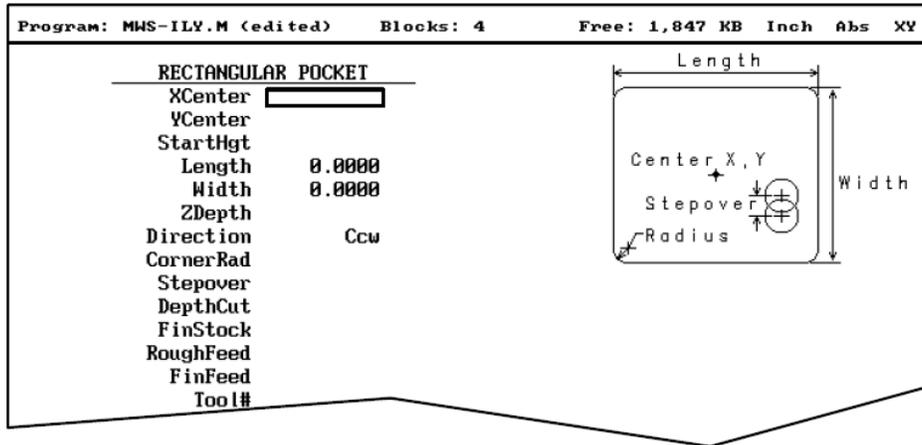
Figure 10.12 – Input screen for rectangular profile
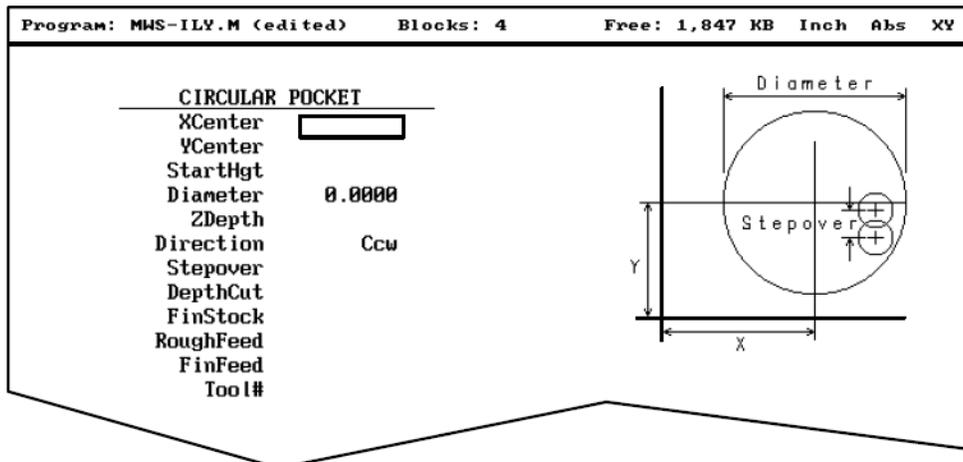
**Figure 10.13 – Input screen for circular pocket**
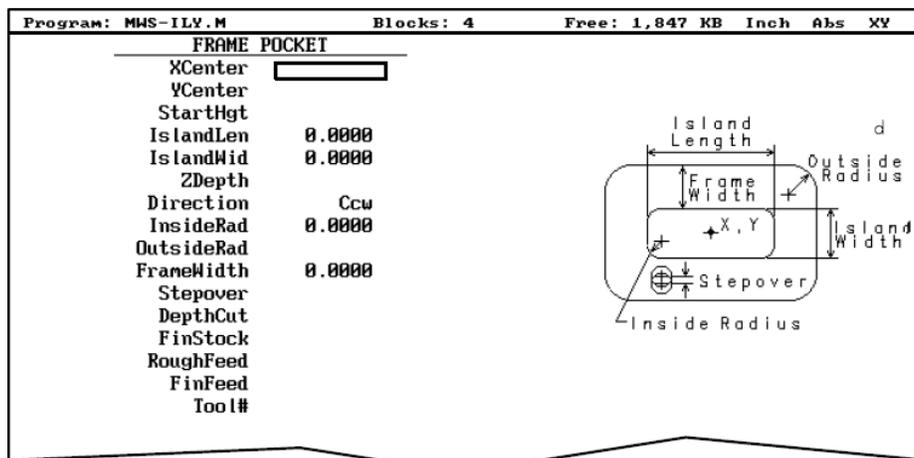
Figure 10.14 – Input screen for frame pocket

10.4.1   ANILAM CANNED CYCLE DEFINITIONS

- **Profile**: Circular or Rectangular. CNC mill cuts the outline of the shape. Cutter can be on the inside or outside of the shape. Profiles are used for milling the outside contours, slots and holes.

- **Pocket**: Circular or Rectangular. CNC mill cuts all of the material inside of the shape to a specified depth.

- **Frame**: CNC mill cutsa rectangular groove. This can have any radius on the corners (commonly used for rectangular o-ring grooves).

- **Note**: All these cycles require the shape's **centre position**, **length**, **width** or **diameter,** therefore it is most helpful to dimension drawings with these requirements in mind.

PART II - MANUFACTURING

1 POSTS

- The drawing below (Figure 1.1) shows the best practices for dimensioning a post drawing when using the machining method outlined on the following pages.
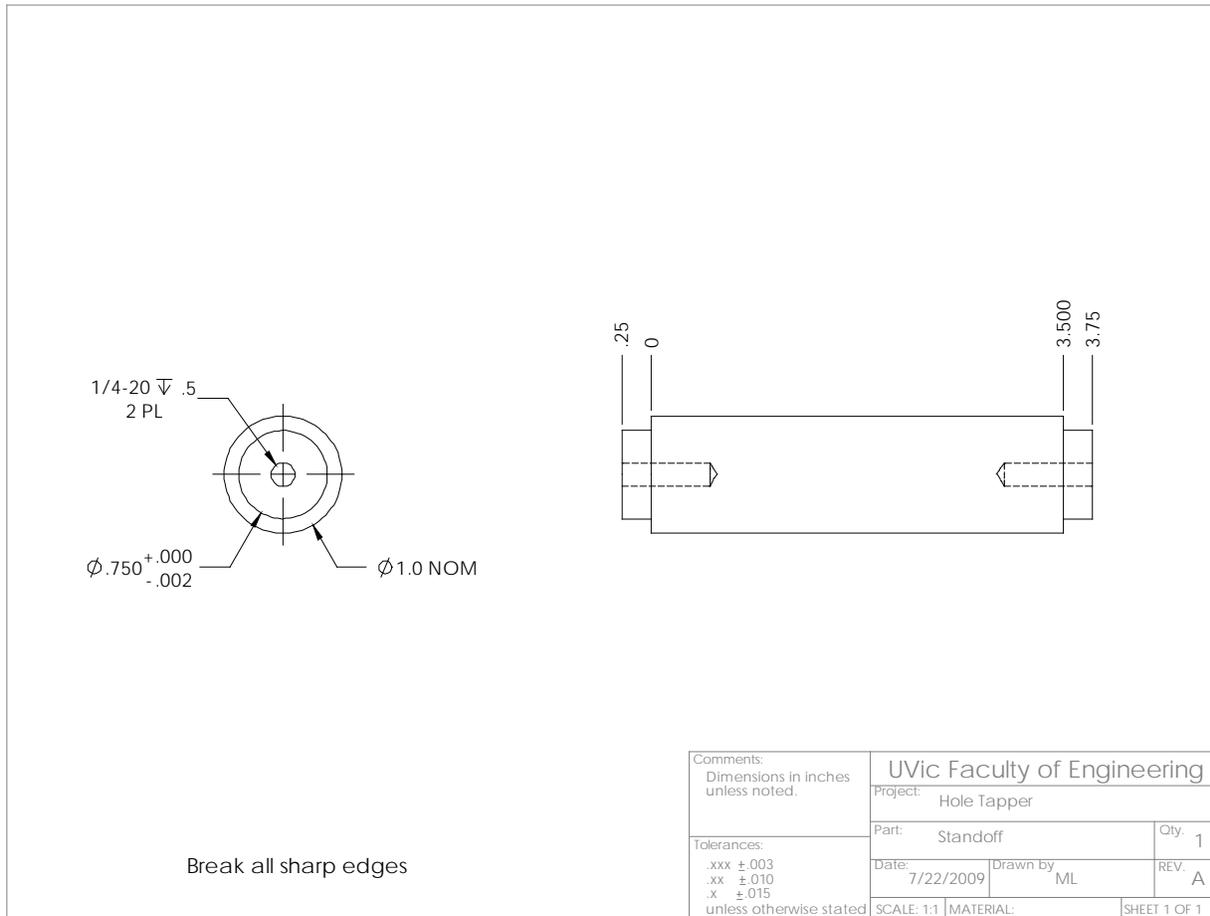

Figure 1.1 – 2D Drawing for Post

- The post drawing above shows the datum position ("zero") as the base of the left boss. This is done because the critical dimension for this part is 3.500" and not the overall length of 4.0".

- **Note:** If alignment is very critical (eg. top and bottom plates need to be aligned within ±0.003 of each other) then the outer diameter of the post must be machined first to maintain concentricity throughout the machining process. This is because stock material

is often not perfectly round. **Machining the full outer diameter length is rarely required in most applications**.

- Lathe process for machining multiple posts with bosses
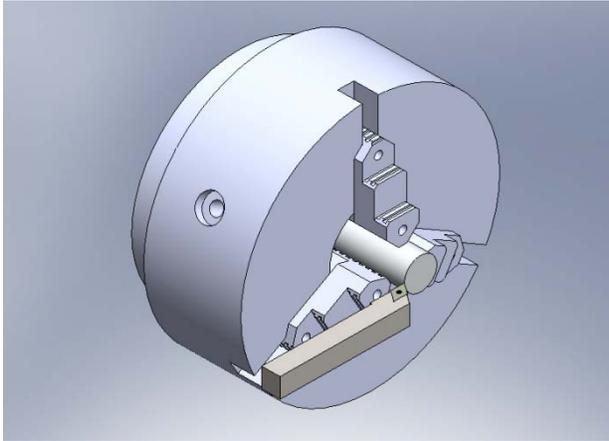
Step 1 – Face first end of post and set Z-axis to zero on lathe digital readout
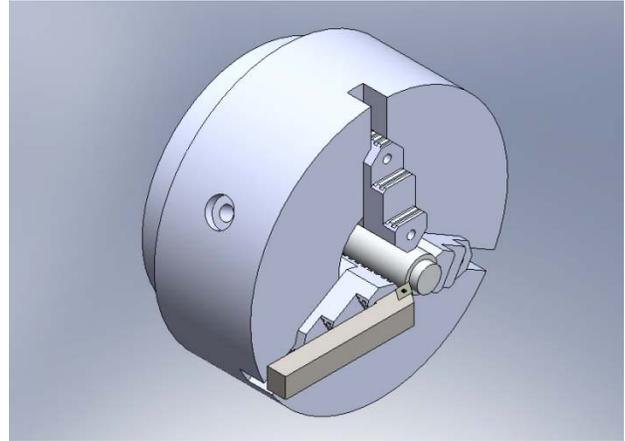

Step 2 – Turn boss to required diameter and approximate depth.
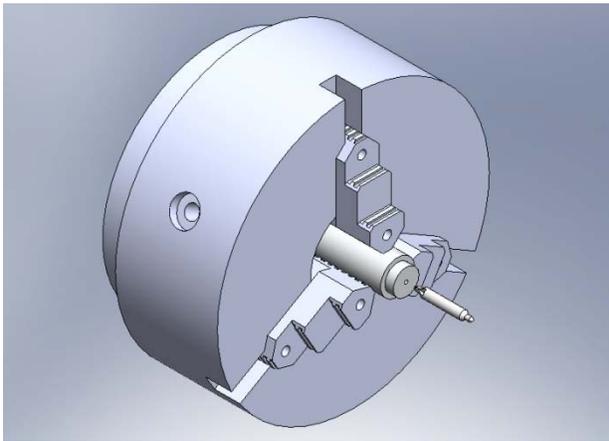

Step 3 – Centre drill
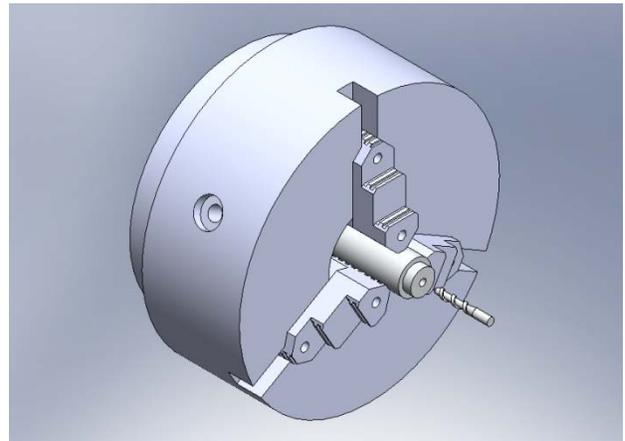

Step 4 – Drill end to tap depth
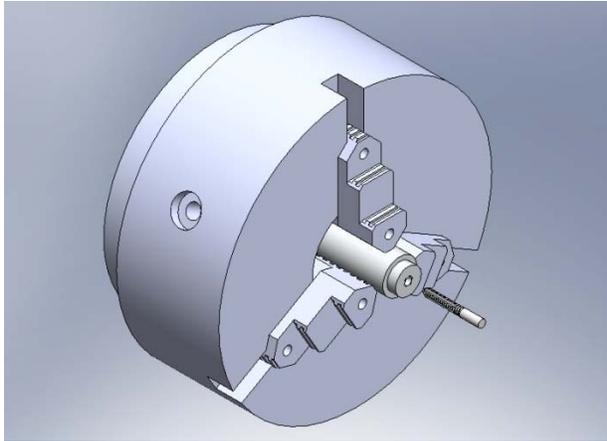
Step 5 – Tap. Repeat steps 1-5 for all posts
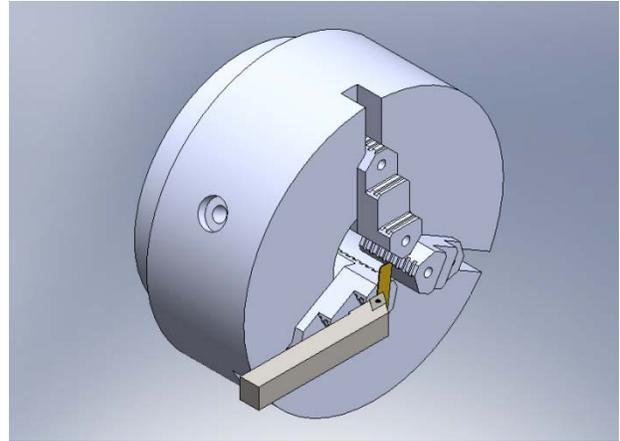
Step 6 – Zero tool against chuck using shim (make sure to account for shim thickness)
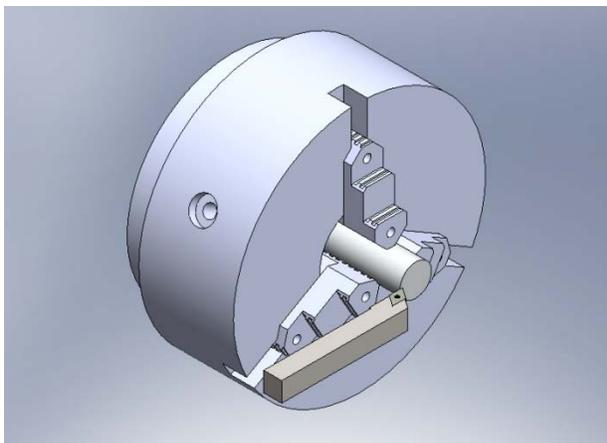

Step 7 – Face opposite end of post to approximately the total length
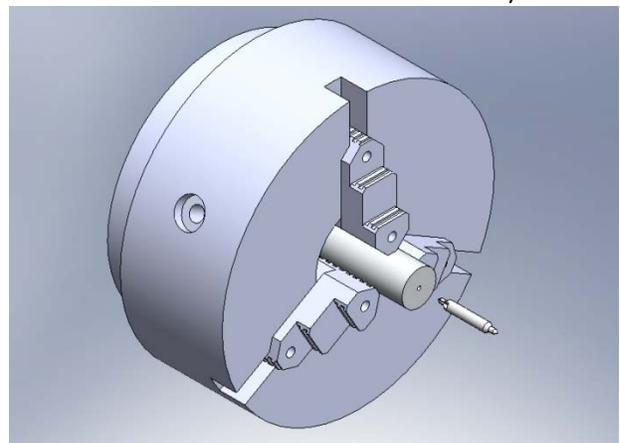
Step 8 – Centre drill. Repeat steps 7 and 8 for all identical posts
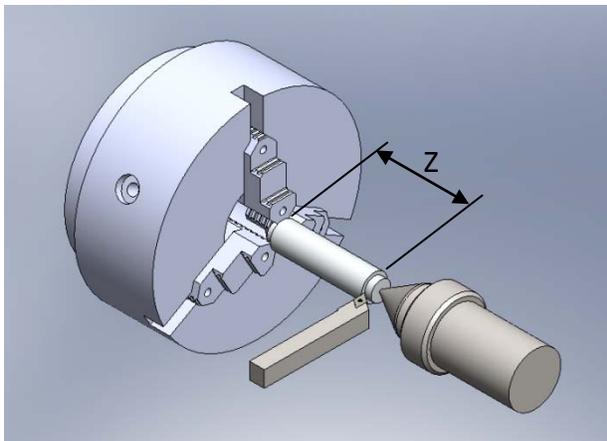

Step 9 – Turn second boss with live centre (Z = critical length of post)
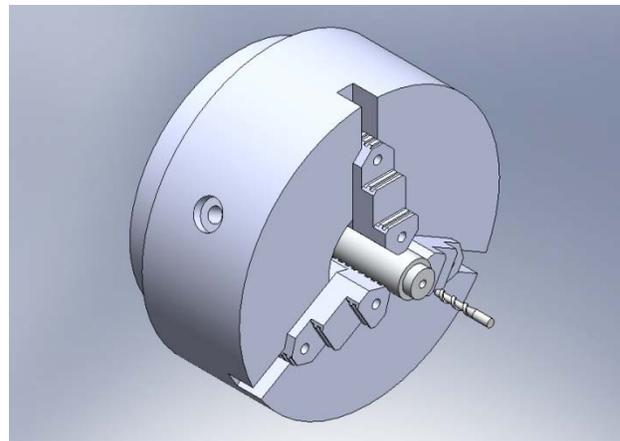
Step 10 – Drill boss of opposite end to tap depth
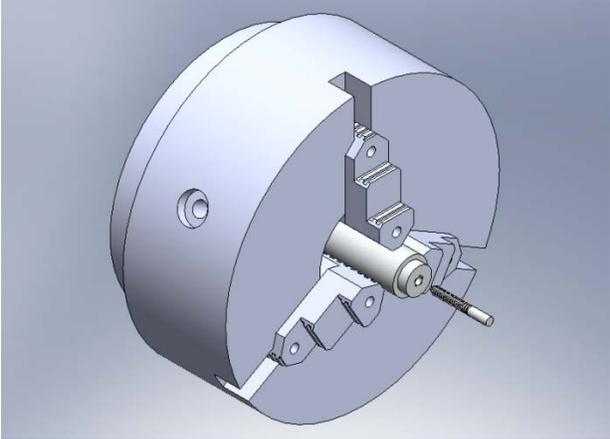
Step 11 – Tap second end
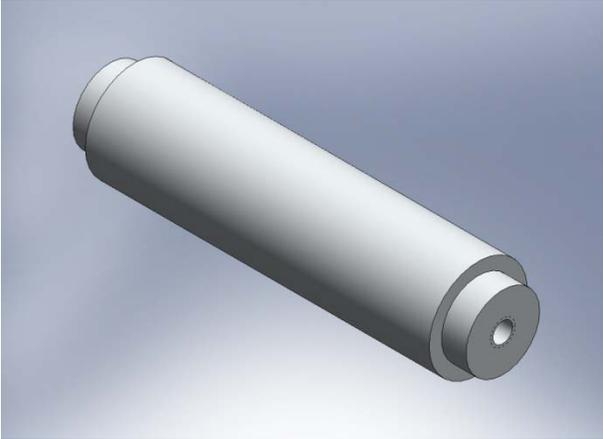
Step 12 – Chamfer all sharp edges

1.1 IMPORTANT NOTES FOR POSTS

- The overall length need not to be machined as accurately as the critical length between the bosses (shown in step 9).

- When turning the boss in step 2 it can be beneficial to leave the boss extra long. This gives the the chuck jaws more surface area to clamp when turning with the live centre. After step 9 the this extra length can be faced off easily.

2 CNC MILLING

2.1 SETUP METHODS

2.1.1 FLAT PARTS

- Flat parts are often machined on a CNC mill using a fixture plate (Shown below). This has many advantages over conventionally machined part.
    - If the part does not have any features in its ends or sides only one setup is required (**Top-down** machining). This being the rationale for not placing holes in the sides of flat parts when it can be avoided.
    - The outside of the part can be machined in one step.
    - Precise copies of the same part can be rapidly machined (i.e. left and right sides, top and bottom).
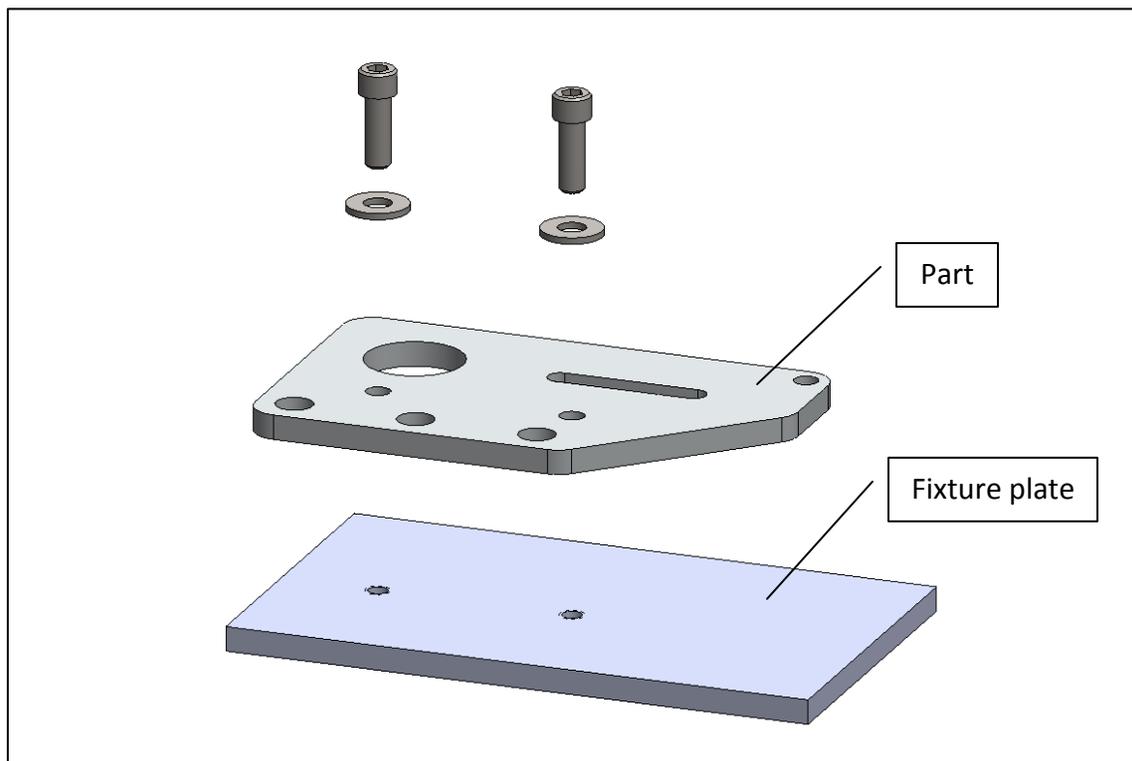

**Figure 2.1 – Flat part with fixturing plate**
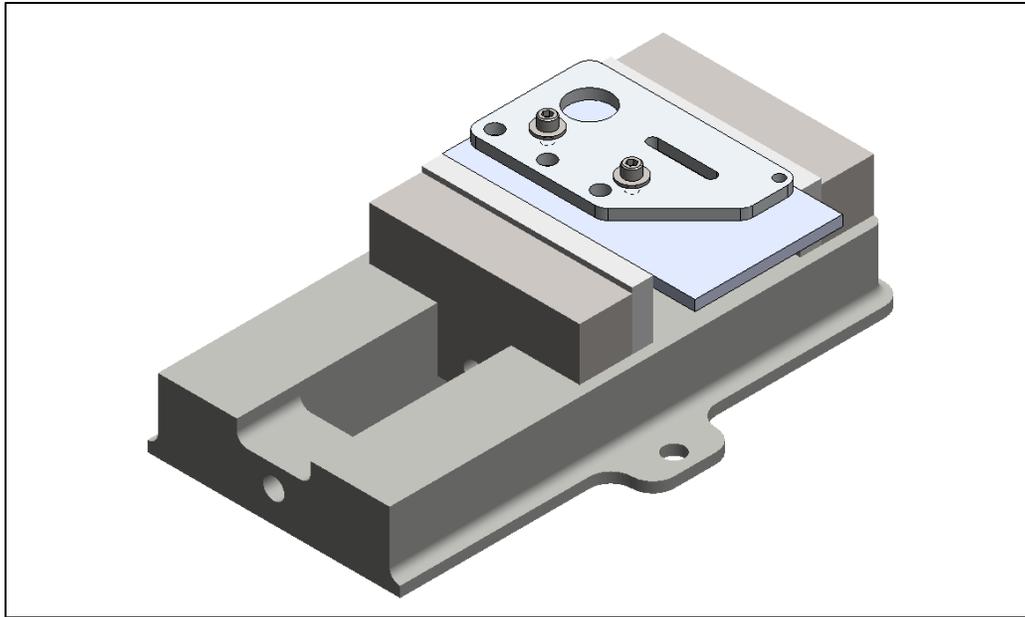
Figure 2.2 – Fixturing plate setup in vise

2.1.2 CIRCULAR PARTS

- When the milling forces will be in the Z-axis only (eg. Drilling) a circular part can be bolted down to a plate (Shown in Figure 2.3).
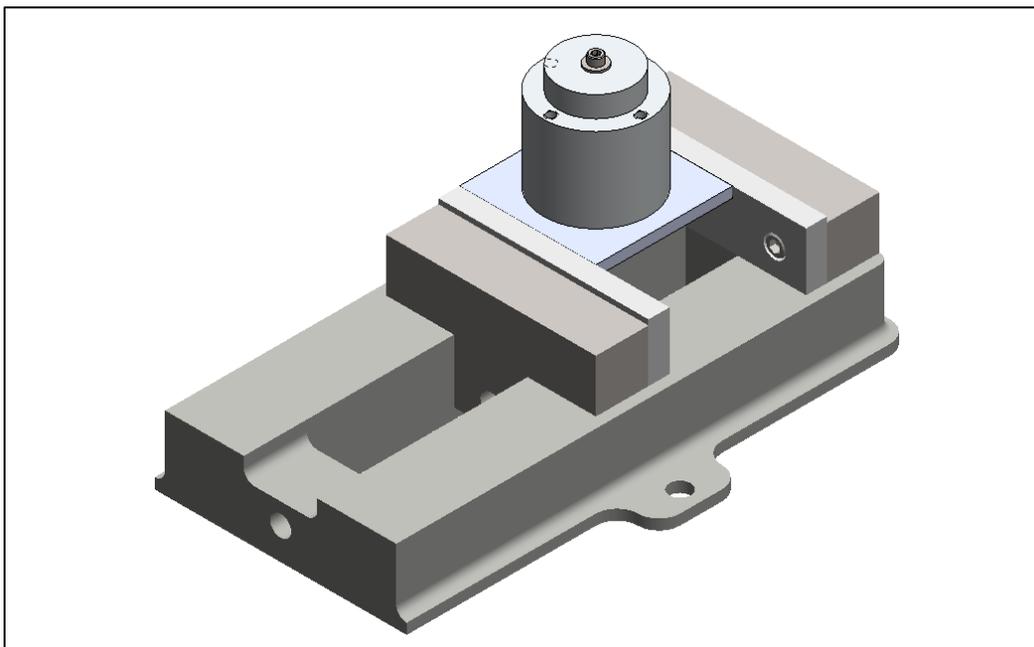
Figure 2.3 – Circular Part bolted down through center.

- In other cases where lateral and radial cutting forces will be applied to the part a jig can be used. Figure 2.4 shows an example of a very simple, effective circular holding jig which is commonly used in the shop. The scratch marks on the part and jig allow the part to be replaced into the mill in the correct orientation for post machining operations or modifications.
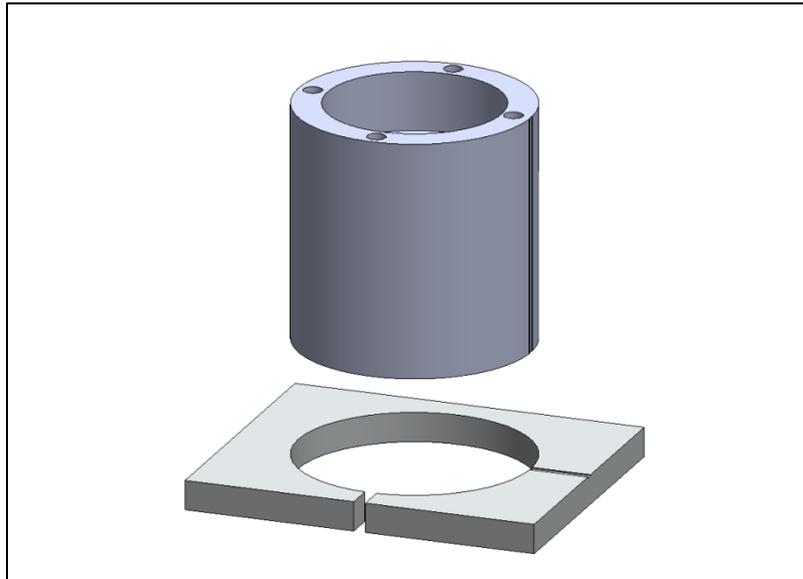
Figure 2.4 – Circular part with jig
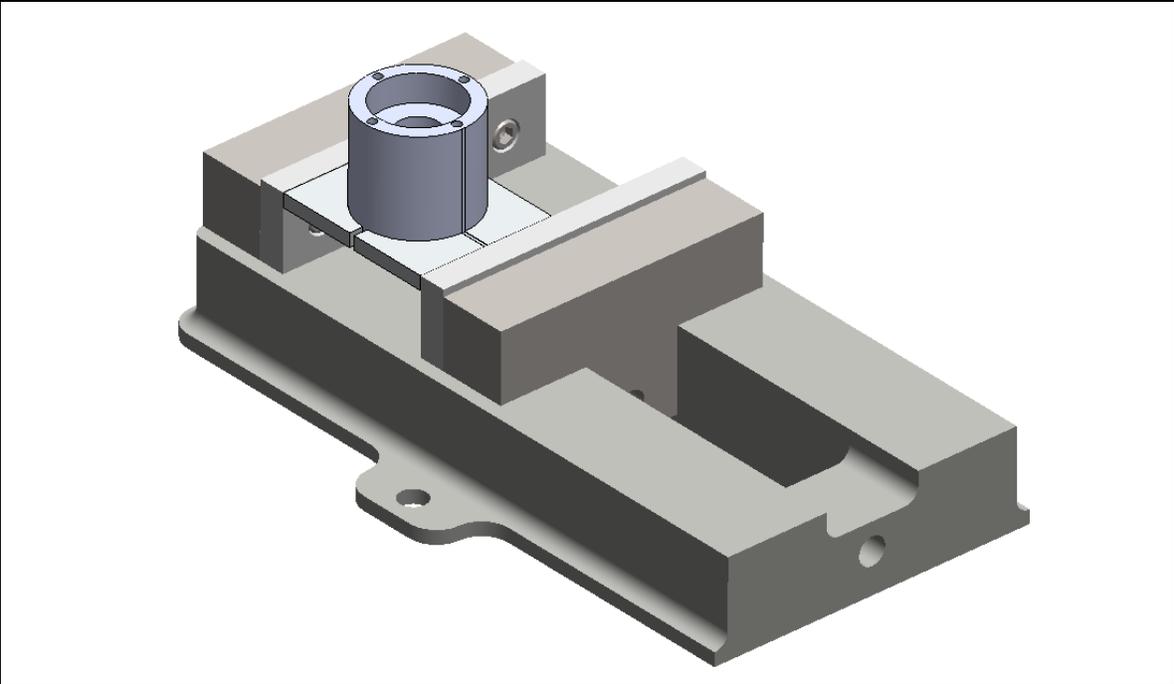
Figure 2.5 – Circular part and jig setup

3 O-RINGS

3.1 AXIAL O-RINGS

- **Reduce RPM speeds when grooving**

- Figure 3.1 shows an example axial o-ring groove.

.100±.005

$\emptyset$2.50 NOM

$\emptyset$1.580±.003

$\emptyset$1.750±.003
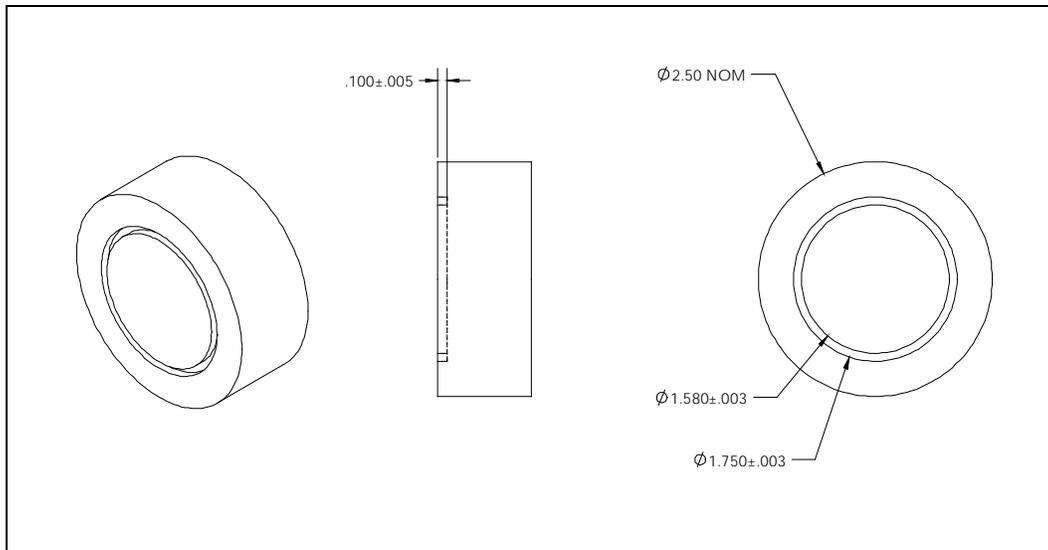
Figure 3.1 – Axial O-ring Groove

1. Determine the width of the tool being used. In this case the tool width is 0.0525".

.0525
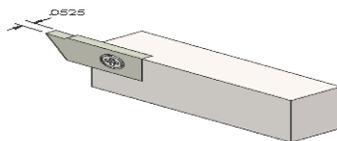
2. Using the following formula, determine the 'X' position for the O.D. of the O-ring groove

- **Work piece MUST be rotating for steps 3 and 4.**

3. Using the right side of the tool, establish the tool position on the O.D. of the work piece and enter the O.D. of the work piece into 'X' on the digital read-out (eg. 2.500").
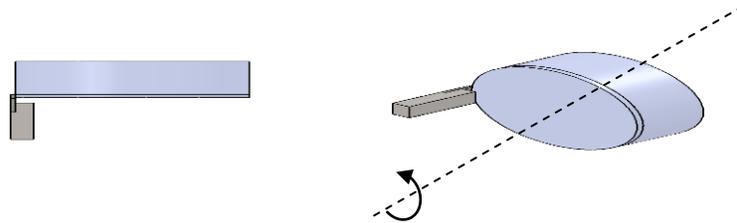
4. Using the tip of the tool, establish the tool position on the front face of the work piece by touching off lightly. Enter 0.000 into 'Z' on the digital read out.

   Touch off the tool where the O-ring groove will be located. The score mark created from the tool will disappear when the O-ring groove is cut.
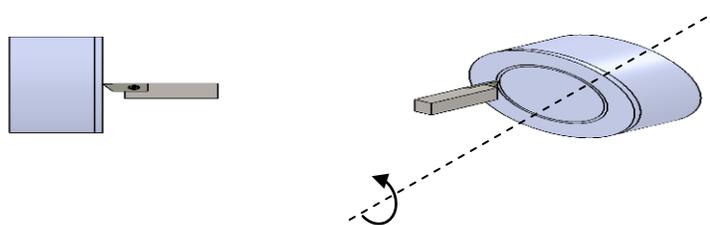
   Steps 5 to 7 will be rough cuts. Rough cuts are usually 0.010" smaller than the final tolerance. This is performed to improve surface finish and tolerances.

5. Move tool in 'X' axis to 1.590" and then move tool in 'Z' axis to -0.090".
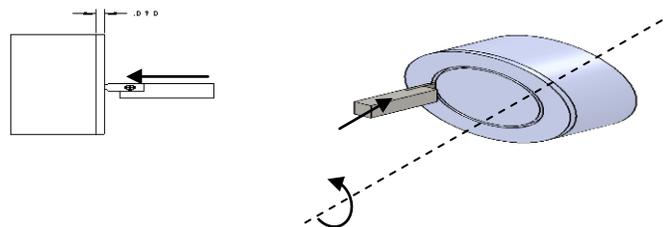
6. Move tool in 'X' axis out to 1.635". This is 0.010" smaller than the calculated value from step 2.
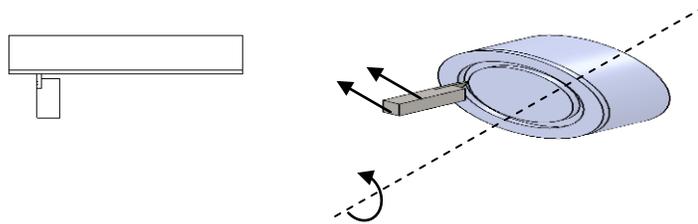
7. Perform the finishing cut. Move 'X' to 1.580", next move 'Z' in to -0.100" and then move 'X' out to 1.625".

This should be done in one fluid motion. The tool should never become stationary on the work piece when performing any cuts. This can cause chatter and excessive heat generation and when machining plastic it will create a poor surface finish and the seal may leak.

3.2 RADIAL O-RINGS

- **Reduce RPM speeds when grooving**

- Figure 3.2 shows an example radial o-ring groove.
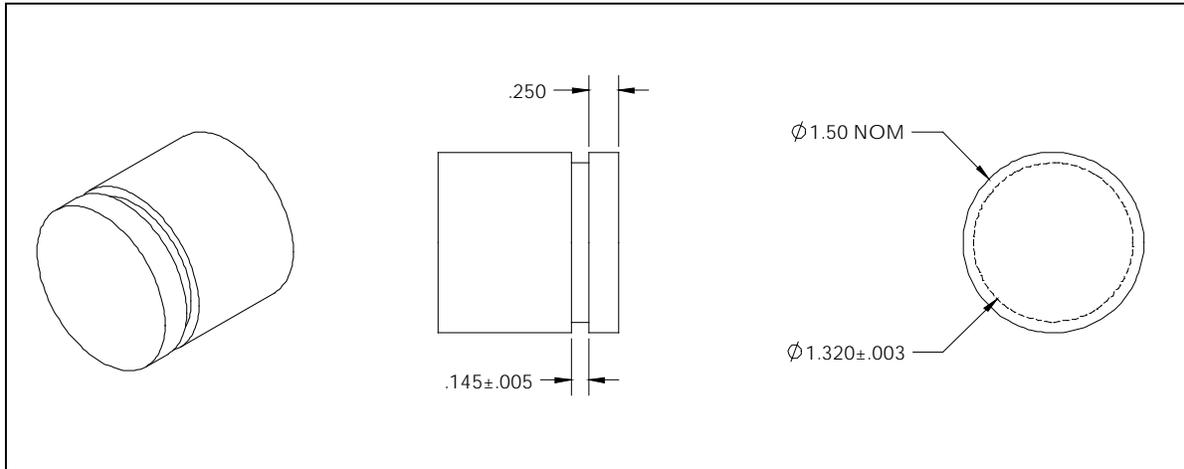
Figure 3.2 – Radial O-ring Groove

1. Determine the width of the tool being used. In this case the tool width is 0.0525".
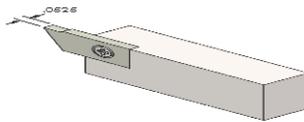

2. Use the following formula to determine the final 'Z' position for the width of the O-ring groove.

- **Work piece MUST be rotating for steps 3 and 4.**

3. Using the left side of the tool, establish the tool position on the end of the work piece and enter the width of the tool into 'Z' on the digital read-out (eg. 0.0525"). This established the right side of the tool as zero in the 'Z' axis.
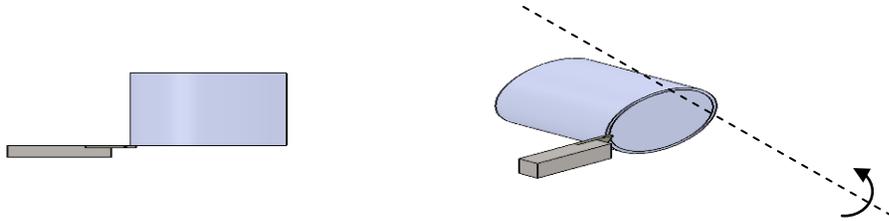
4. Using the tip of the tool, establish the tool position on the O.D. of the work piece and enter the O.D. of the work piece into 'X' on the digital read out. (eg. 1.500")

   Touch off the tool where the O-ring groove will be located. The score mark created from the tool will disappear when the O-ring groove is cut.
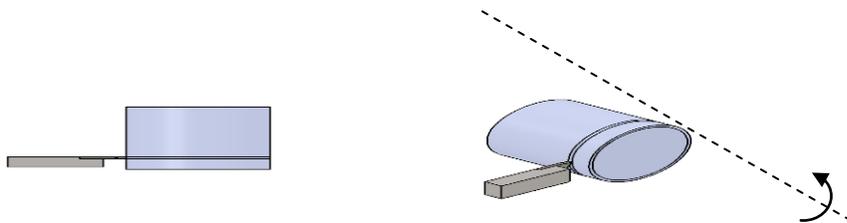
Steps 5 to 7 will be rough cuts. Rough cuts are usually 0.010" smaller than the final tolerance. This is performed to improve surface finish and tolerances.

5. Move tool in 'Z' axis to -0.260" and then move tool in 'X' axis to 1.320".
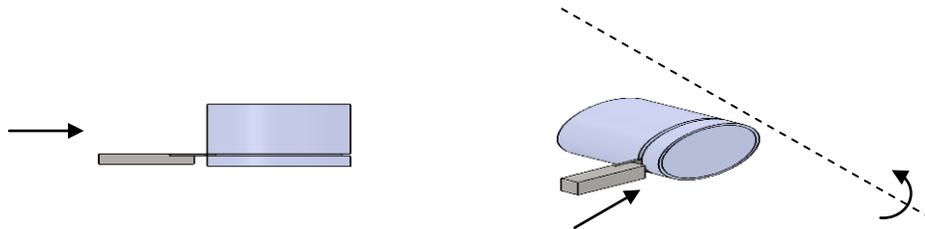
6. Move tool in 'Z' axis to -0.333". This is 0.010" smaller than the calculated value from step 2.
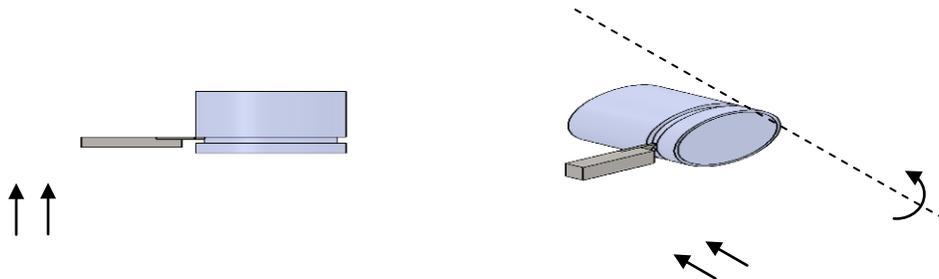
7. Perform the finishing cut. Move 'Z' to -0.250", next move 'X' in to 1.300" and then move 'Z' to -0.343".

This should be done in one fluid motion. The tool should never become stationary on the work piece when performing any cuts. This can cause chatter and excessive heat generation and when machining plastic it will create a poor surface finish and the seal may leak.

4 DRILLING AND TAPPING

- **Centre Drill**:
  - Use anytime there is a chance the drill bit may glance off the material or when holes need to be precisely positioned (ie. holes on a radius, drilling into a hard material, etc.). Centre drills are used very often when drilling in the lathe.

- **Drilling Plexiglas**:
  - Plexiglas will often to crack when the drill bit exits through the other side of the part. To prevent this, slow down the drill feed for the last 1mm or 1/16" while drill bit exits the material. This heats the Plexiglas to make the last 1/16" more pliable to prevent cracking. Use extra care when drilling holes greater than 5/16" diameter.

4.1 TAPPING

- **Taps**: Used to cut internal threads. The various types of taps are:

  Spiral point taps are the most commonly used. The first 3 to 4 threads on these taps are partial therefore they do not cut full threads for the complete length of engagement. If this is required see Bottoming taps below.

  - **Bottoming Tap**: Used to cut threads to the bottom of a blind hole. A bottoming tap does not have a tapered cutting edge therefore a plug tap must be used first. **Do not use a bottoming tap to cut threads in an unthreaded hole.**

  > **Note**: Use a bottoming tap **only when thread depth is critical** (ie. 1/2" plate requiring 3/8" thread depth). Regular taps will not cut full threads to the bottom of a hole.

- **Tap drill bit**: Specific sized drill bit for a certain tap (eg. #6 drill for a 1/4-20 tap).
- **Percentage of Thread**: The percentage of thread refers to the amount of thread in terms of the total thread depth (crest to root). Use a larger tap drill bit to lower the thread percent and vice versa. Decreasing the thread percent minimizes the torque on the tap and dramatically helps to avoid breaking the tap. It also speeds up the threading pro
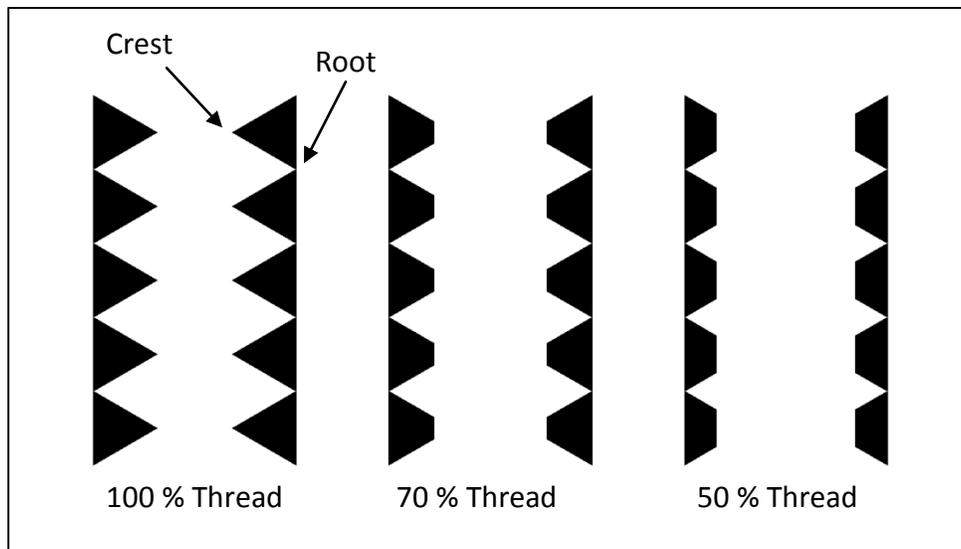
Figure 4.1 – Thread percentage example

- **TPI: T**hreads **P**er **I**nch

- Metric threads are specified by the distance between the threads

- **Die (external threads):** Used to cut external threads.

- **NPT Threads**: (National Pipe Thread) These are tapered threads for sealing fluids

- Maximum depth of tapped holes should not exceed **3 X Diameter of Tap.** Usaully twice the tap diameter is all that is required to provide maximum holding force.

APPENDICES

Tolerances Related To Machining Processes and Sizes

Tolerances

Range of Sizes From	Range of Sizes To and including									
0.010	0.599	0.00015	0.0002	0.0003	0.0005	0.0008	0.0012	0.002	0.003	0.005
0.600	0.999	0.00015	0.00025	0.0004	0.0006	0.001	0.0015	0.0025	0.004	0.006
1.000	1.499	0.0002	0.0003	0.0005	0.0008	0.0012	0.002	0.003	0.005	0.008
1.500	2.799	0.00025	0.0004	0.0006	0.001	0.0015	0.0025	0.004	0.006	0.010
2.800	4.499	0.0003	0.0005	0.0008	0.0012	0.002	0.003	0.005	0.008	0.012
4.500	7.799	0.0004	0.0006	0.001	0.0015	0.0025	0.004	0.006	0.010	0.015
7.800	13.599	0.0005	0.0008	0.0012	0.002	0.003	0.005	0.008	0.012	0.020
13.600	20.999	0.0006	0.001	0.0015	0.0025	0.004	0.006	0.010	0.015	0.025

Machining Processes (bar chart aligned to tolerance columns):

- Lapping & Honing
- Grinding, Diamond Turning & Boring
- Broaching
- Reaming
- Turning & Boring
- CNC Milling
- Manual Milling
- Drilling

Chart annotations: EXTREME CARE REQUIRED; EXTREME CARE REQUIRED; CARE REQUIRED

Tap drill and body drill sizes (from shop)

Drilling, milling speeds (from shop)

Static O-Ring Sealing

Face Seal Glands

For Internal Pressure
(outward pressure direction)
dimension the groove by its
outside diameter (H_O) and width:

(H_O) = Mean O.D. of O-ring
(see Table 4-1)
Tolerance = Minus 1% of Mean
O.D., but not more than
-.060

For External Pressure
(inward pressure direction)
dimension the groove by its
inside diameter (H_I) and width:

(H_I) = Mean I.D. of O-ring
(see Table 4-1)
Tolerance = Plus 1% of Mean
I.D., but not more than
+.060

0° to 5°*
(Typ.)

Break Corners
Approx. .005 RAD.

Surface finish X:
32 for liquids
16 for vacuum
and gases

Groove
Depth
(= Gland Depth)

Gland Detail
Finishes are RMS values

.005 Max.

.003 Max.
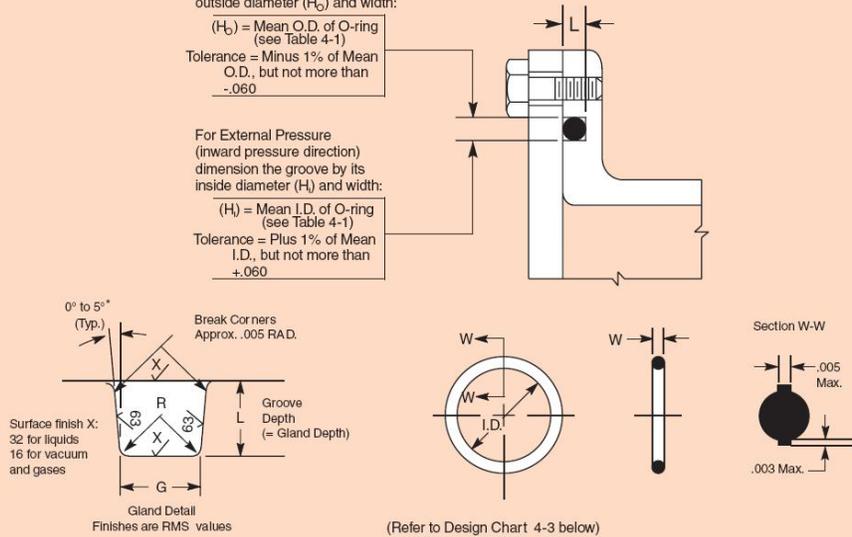
(Refer to Design Chart 4-3 below)

O-Ring Face Seal Glands These dimensions are intended primarily for face type O-ring seals and low temperature applications.

O-Ring Size Parker No. 2	W Cross Section Nominal	Actual	L Gland Depth	Squeeze Actual	%	G Groove Width Liquids	Vacuum and Gases	R Groove Radius
004 through 050	1/16	.070 ±.003 (1.78 mm)	.050 to .054	.013 to .023	19 to 32	.101 to .107	.084 to .089	.005 to .015
102 through 178	3/32	.103 ±.003 (2.62 mm)	.074 to .080	.020 to .032	20 to 30	.136 to .142	.120 to .125	.005 to .015
201 through 284	1/8	.139 ±.004 (3.53 mm)	.101 to .107	.028 to .042	20 to 30	.177 to .187	.158 to .164	.010 to .025
309 through 395	3/16	.210 ±.005 (5.33 mm)	.152 to .162	.043 to .063	21 to 30	.270 to .290	.239 to .244	.020 to .035
425 through 475	1/4	.275 ±.006 (6.99 mm)	.201 to .211	.058 to .080	21 to 29	.342 to .362	.309 to .314	.020 to .035
Special	3/8	.375 ±.007 (9.52 mm)	.276 to .286	.082 to .106	22 to 28	.475 to .485	.419 to .424	.030 to .045
Special	1/2	.500 ±.008 (12.7 mm)	.370 to .380	.112 to .138	22 to 27	.638 to .645	.560 to .565	.030 to .045

Design Chart 4-3: Design Chart for O-Ring Face Seal Glands

4-18

< Back | Section Contents | Table of Contents | Search | Next >

Parker O-Ring Handbook

Guide for Design Table 4-2

If Desired Dimension is Known for	Select Closest Dimension in Column	Read Horizontally in Column	To Determine Dimension for
Bore Dia. male gland	A	B-1 C G	Groove Dia. (male gland) Plug Dia. (male gland) Groove width
Plug Dia. male gland	C	A B-1 G	Bore Dia. (male gland) Groove (male gland) Groove width
Tube OD female gland	B	A-1 D G	Groove Dia. (female gland) Throat Dia. (female gland) Groove width
Throat Dia. female gland	D	A-1 B G	Groove Dia. (female gland) Tube OD (female gland) Groove width

Design Guide 4-2: Guide for Design Table 4-2

Industrial Static Seal Glands

Refer to Design Chart 4-2 (below) and Design Table 4-2 for dimensions

metric conversion 32√ = .8√
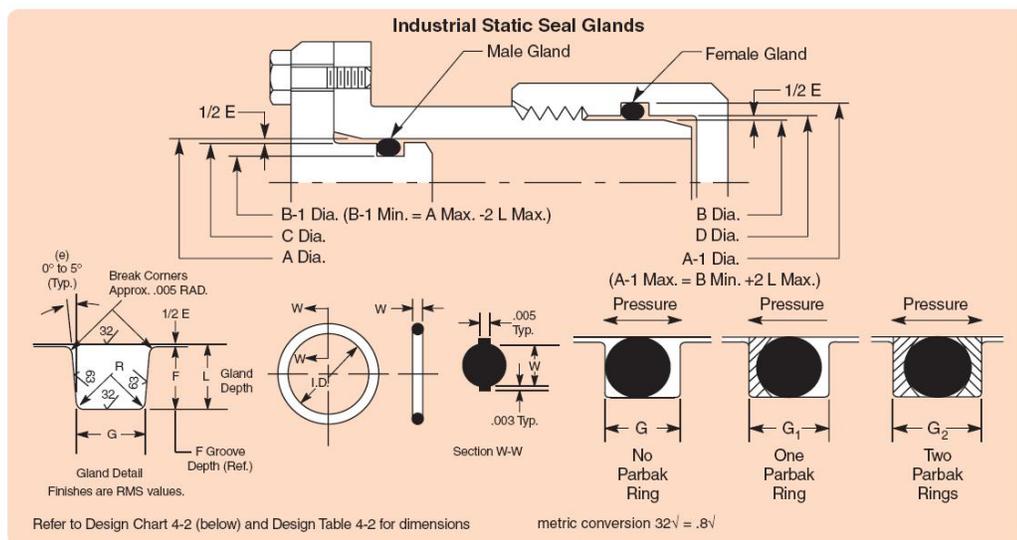
Industrial O-Ring Static Seal Glands

O-Ring 2-Size AS568B-	W Cross-Section Nominal	W Cross-Section Actual	L Gland Depth	Squeeze Actual	Squeeze %	E(a) Diametral Clearance	No Parbak Ring (G)	One Parbak Ring (G₁)	Two Parbak Ring (G₂)	R Groove Radius	Max. Eccentricity (b)
004 through 050	1/16	.070 ±.003 (1.78 mm)	.050 to .052	.015 to .023	22 to 32	.002 to .005	.093 to .098	.138 to .143	.205 to .210	.005 to .015	.002
102 through 178	3/32	.103 ±.003 (2.62 mm)	.081 to .083	.017 to .025	17 to 24	.002 to .005	.140 to .145	.171 to .176	.238 to .243	.005 to .015	.002
201 through 284	1/8	.139 ±.004 (3.53 mm)	.111 to .113	.022 to .032	16 to 23	.003 to .006	.187 to .192	.208 to .213	.275 to .280	.010 to .025	.003
309 through 395	3/16	.210 ±.005 (5.33 mm)	.170 to .173	.032 to .045	15 to 21	.003 to .006	.281 to .286	.311 to .316	.410 to .415	.020 to .035	.004
425 through 475	1/4	.275 ±.006 (6.99 mm)	.226 to .229	.040 to .055	15 to 20	.004 to .007	.375 to .380	.408 to .413	.538 to .543	.020 to .035	.005

(a) Clearance (extrusion gap) must be held to a minimum consistent with design requirements for temperature range variation.
(b) Total indicator reading between groove and adjacent bearing surface.
(c) Reduce maximum diametral clearance 50% when using silicone or fluorosilicone O-rings.
(d) For ease of assembly, when Parbaks are used, gland depth may be increased up to 5%.

Design Chart 4-2: For Industrial O-Ring Static Seal Glands

4-9

REFERENCES

Osborn, Joe. 2006. *Tips on Designing Cost Effective Machined Parts.* [Online] OMW Corporation. Available at: http://www.omwcorp.com/partdesign.html. [Accessed 19 June 2009].

McMaster-Carr. 2009. [Online] McMaster-Carr Supply Company. Available at: http://www.mcmaster.com. [Accessed 07 August 2009].

Parker O-Ring Handbook ORD 5700, Parker, 2007.

http://www.parker.com