Name:________________________ Section #______ Lab Group #______

General Lab Procedures

The laboratory work is the most important part of this course. It is expected that you will have prepared for it by **reading the lab manual BEFORE coming to the lab**. It is important to read sections of this manual so that you completely understand what is going on. This will save you a lot of time and keep you from making mistakes. This may determine whether or not you complete your project on time!

When you come to class, inform the instructor of your lab assignment for that day and she/he will give you the necessary raw materials and special tools needed. When you have finished for the day, clean up the machines you used. At the end of every class, the entire shop will be cleaned; NO ONE MAY LEAVE OR WASH UP UNTIL THE INSTRUCTOR IS SATISFIED THAT THE SHOP IS CLEAN.

Safety must be exercised at all times while working in the machine shop. If you violate safety rules, you may be requested to leave the shop, and you may not be able to complete the project on time.

One aspect of safety is keeping the walkways clear. This implies that coats and book bags should not be left strewn about in random locations. There are a number of red painted lockers installed in the hallway outside of the main door to the student shop. Please use them to stow your belongings while the class is in session. I would recommend bringing a padlock of some sort with you so that you may secure the locker while you are in class. Because I have only 40 lockers and a total of about 350 students, take your belongings and your lock with you when you leave the class. If necessary, locks may be removed from the lockers as we need them for each class and they WILL be removed at the end of class on Fri (3 PM). Any material in the lockers will end up in JEC lost and found. This is located in JEC 3018

General Safety Rules

The following is a list of *some* basic safety rules that must be followed while you are in the machine shop, it should not be considered an exhaustive list.

- ***Always wear safety glasses while in the shop.***
- People with long hair must tie it back.
- Do not wear any loose clothing or jewelry, which may become caught in moving machinery.
- Do not wear frayed and ripped clothing when you are scheduled to take part in the welding lab.
- Do not wear gloves while operating machinery (except welding equipment and grinding wheels).
- Do not wear open-toed or open backed shoes of any kind.
- Do not use any machine unless you have been instructed in the use of that equipment.
- Do not leave any machines unattended, while they are running.
- Keep your hands (and hair clothes etc.) away from moving machinery and cutters.
- Do not operate the equipment while under the influence of alcohol or drugs (legal or illegal).
- Do not run or yell unnecessarily while in the shop.
- Report all spilled fluids immediately: they are an extreme slip hazard.
- If you are uncertain about any aspect of a machining operation you wish to perform, please ask the person in charge before proceeding. No limit to the questions you may ask.
- You are required to complete (successfully) the Rensselaer Manufacturing and Prototyping Laboratories safety Orientation course and test. This is available on Percipio. To be found at RPI Info in Student resources.
- If any machine component or piece of material has sharp edges exercise great care touching it or better yet don’t touch it, particularly if it is moving.

Reading Assignments

For each day in the shop, it is necessary to read the following material in preparation for class. This is an approximation as each group may work faster or a bit slower.

	Naval Cannon	Train @ Light Saber
Instrument Reading	Micrometer Appendix, Caliper index & pages 20-24	
Layout	Sections 1.2.4 &3.1	Sections 1.2.4 & 7.1
Lab#1	Lathe/Miller Introduction	
First Day	Sections 3.1-3.3	Sections 7.2-7.4
Second Day	Sections 3.4-3.7	Sections 7.5-7.7
Third Day	Sections 3.8-3.9	Sections 7.8-7.9
Fourth Day	As needed	As needed
Lab#2	Vertical Miller Introduction	
First Day	Sections 6.2-6.5	Sections 8.2-8.4
Second Day	Section 6.6	Sections 8.5-8.7
Third Day	Section 6.7-Assembly	Sections 8.8-8.9
Lab#3	GMAW Lab, Oxyacetylene Brazing Lab, Welding Appendix	
Lab#4	Non-metallic fabrication Lab, See Appendicies	

Problems with this Manual

If something in this manual is not clear, then please let the instructors know. We will use this feedback to improve the next revision of the manual. Every time we revise the manual it can mess up the page numbering.

Problems with Tools

DO NOT assume that the last person using any given tool kit put it away with the correct size drills, etc. in place. We strongly recommend that you measure the diameter of any drill or cutting bit before using it. Failure to do so on your part is not an error on our part.

CAD-CAM

You will be given an opportunity to run both a CNC lathe and to program and run a CNC laser cutter or CNC plasma cutter. The latter might be part of your welding lab. The former will be part of the Nonmetallic Fabrication Lab.

There may also be opportunities to run a CNC miller.

Table of Contents

Appendicies

List of Figures

Shop Schedule

Lab #1	Cannon Barrel /Boiler Stack/Light Saber grip Lab
Lab #2	Canon Base / Train Cab/Light Saber cap Lab
Lab #3	Welding & Brazing Lab
Lab #4	Non-metallic Fabrication Techniques
X	Any Lab (for which equipment is available)

Group Number	Class Number 1	2	3	4	5	6	7	8	9	10	11	12
1	Introduction	Instrument Reading & Layout	1	X	1	4	2	1	1	2	3	2
2			1	4	1	1	2	1	3	2	2	X
3			1	4	1	2	2	1	1	2	3	X
4			1	4	1	X	2	1	1	2	3	2
5			1	2	3	1	1	2	1	4	2	X
6			1	2	3	1	1	2	2	1	X	4
7			1	2	2	1	3	X	1	1	2	4
8			2	2	1	1	3	1	X	4	1	2
9			2	1	1	4	1	2	3	1	2	X
10			2	1	1	4	X	1	3	2	1	2
11			2	1	2	1	3	1	1	4	2	X
12			3	1	2	2	1	2	1	1	X	4
13			3	1	2	2	1	4	2	1	1	X
14			3	1	X	2	1	4	2	1	1	2
15			X	1	3	1	1	4	2	1	2	2

Grading Policy

Your grade in this class is determined from several factors:

- Class Attendance
- Completion of the Cannon / Train /Light Saber project
- Completion of the GMAW and Brazing Lab
- Completion of the non-metallic fabrication techniques Lab
- Submission of the Report (due the last day of final exam)
- Final Exam

We require completion of all the above items. If for some reason the school is required to go remote you will be graded by the progress you have made on your project (pro rated for the amount of time you got to spend on it) as well as the Final exam and Lab report.

Final Exam

The final exam will be given on LMS during the week of April 14-19. The test may be taken beginning 11:59 PM April 14 through 11:59 PM April 19. The Lab Report is also to be submitted via LMS between 11:59 PM April 14 and 11:59 PM April 19. An email will also be sent to class members as a reminder.

Your project (cannon, train or light saber) must be checked off as being completed by the day of your final. The exam covers all material that you learned in class. This includes (but is not limited to) instrument readings, machining operations, welding, and safety rules. Reviewing this manual is a good way to study for the exam. If English is not your first language, you may wish to use a commercial dictionary for the test.

Cheating and Academic Dishonesty

Cheating is unacceptable. There is no reason why anyone should not be able to pass this class, given a reasonable amount of effort.

The following is excerpted from the Rensselaer Student Handbook. Penalties for cheating are severe, even for a one credit hour course.

> **Academic Fraud:** Alteration of documentation relating to the grading process. For example, changing exam solutions to negotiate for a higher grade or tampering with an instructor's grade book.
>
> **Collaboration:** Deliberate facilitation of academic dishonesty in any form. For example, allowing another student to observe an exam or allowing another student to "recycle" one's old term paper.
>
> **Copying:** Obtaining information pertaining to an exam question by deliberately observing the paper of another student. For example, noting which alternative a neighboring student has circled in a multiple-choice exam.
>
> **Cribbing:** Use or attempted use of prohibited materials, information, or study aids in an academic exercise. An example would be using an unauthorized formula sheet during an exam.
>
> **Fabrication:** Unauthorized falsification or invention of any information in an academic exercise. An example would be the use of "bought" or "ready-made" term papers, or falsifying lab records.
>
> **Plagiarism:** Representing the work or words of another as one's own through the omission of acknowledgement or reference. An example, using sentences verbatim from a published source in a term paper without appropriate referencing, or presenting as one's own the detailed argument of a published source.
>
> **Sabotage:** Destruction of another student's work related to an academic exercise. An example would be destroying a model, lab experiment, computer program or term paper developed by another student.
>
> **Substitution:** Utilizing a proxy, or acting as proxy, in any academic exercise. An example would be taking an exam for another student or having a homework assignment done by someone else.

The definitions and examples presented above are samples of the various types of academic dishonesty and are not to be construed as an exhaustive or exclusive list. Additionally, students who attempt to commit academic dishonesty or to assist in violation of academic dishonesty policies may be subject to two types of penalties. The instructor administers an academic penalty (i.e. failure of the course) and the student may also be subject to the procedures and penalties of the student judicial system outlined in the student handbook.

NOTE: Students who have been found in violation of academic dishonesty policies are prohibited from dropping a course to avoid the academic penalty.

Engineering Processes

Outline for Final Report – Page 1 of 2

For the Welding, Non-Metallic Fabrication Techniques , Cannon Barrel (Boiler Stack), and Cannon Base (Train Cab) Labs, discuss the following items in detail:

1. The main objective of the operation.
2. What the operation consisted of.
3. What you learned from the lab.
4. The major problems you encountered and how you solved them.

Answer the following questions in detail:

1. What is your opinion of the course?
2. Do you think the course is valuable?
3. If you could change the course in some way, what would you change?
4. If you could change the lab manual in some way, what would you change? (Be specific)

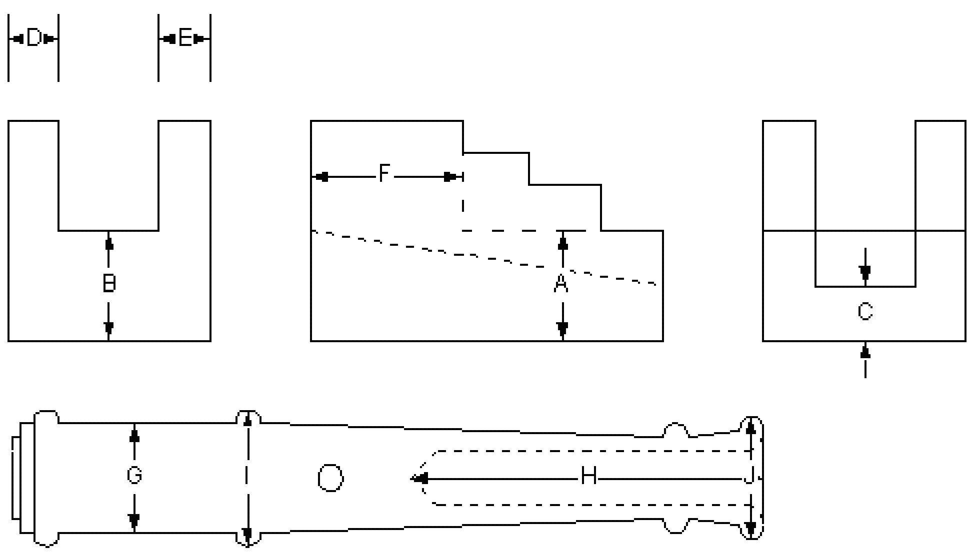
Guidelines:

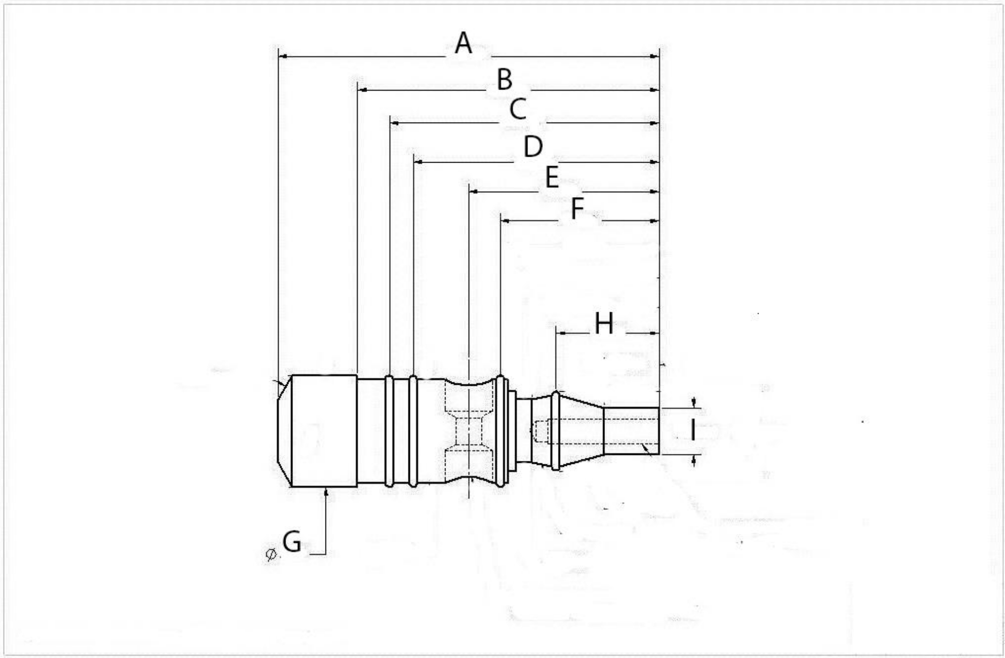
- The report must be typed and must include a title page.
- The report should be a minimum of 2-5 pages in length.
- Reports are due on the day of the final; no late reports are accepted.
- The grade for the report will be based on the organization and content.
- **Reports are to be done on an INDIVIDUAL basis. Cheating will be dealt with severely.**
- **Write as if you are corresponding to a friend who is intelligent but has not taken the class and you are describing to them what you did. It is a poor idea for your report to be shorter than the outline (do not laugh it has happened).**

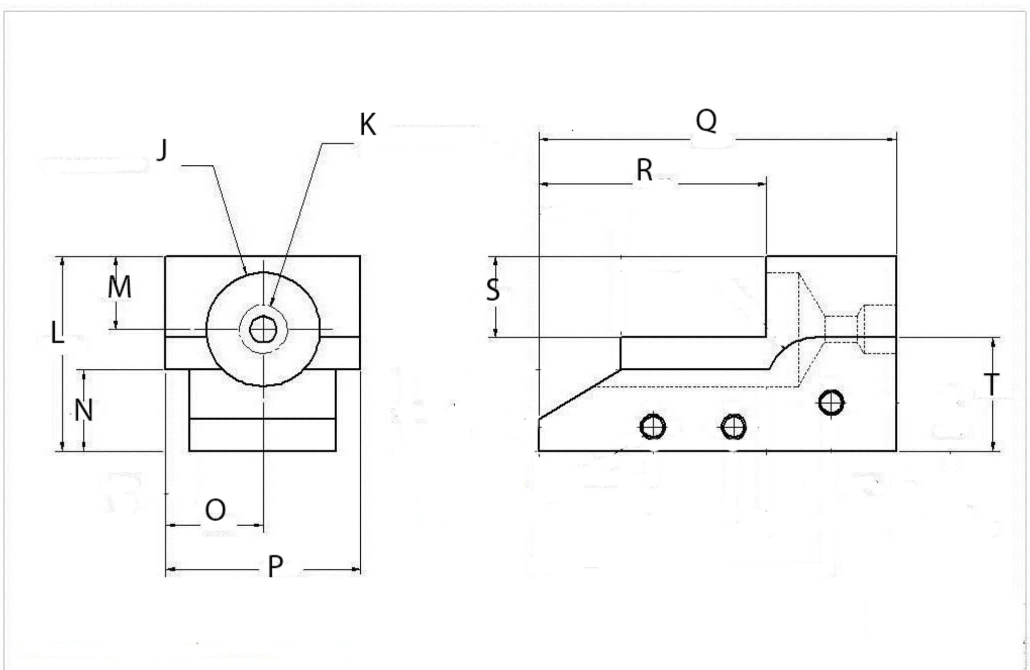
Final Exam Date: ___________________________ Location: ____________________

Engineering Processes

Outline for Final Report – Page 2 of 2




A
B
C
D
E
F
H
I
ø G


J
K
M
L
N
O
P
Q
R
S
T

Light Saber drawing for report may be found in LMS.

For the project you constructed, measure the indicated dimensions and fill in all of the information in the chart below, including units. Use a micrometer or Vernier caliper to take your measurements.

Dimension	Specified Dimension	Permitted Tolerance	Actual Dimension	Deviation	Amount out of Tolerance
A					
B					
C					
D					
E					
F					
G					
H					
I					
J					
K					
L					
M					
N					
O					
Q					
R					
S					
T					

1.0 Measuring, Layout Tools and Layout Overview:

Tools commonly used:

6 inch Scale (Ruler)

Vernier Caliper

0 to 1 inch Micrometer

Dividers

Inside Caliper

Hermaphrodite Caliper

Outside Caliper

Solid Square

Dykem Layout Fluid

Scriber

Center Punch

Ball Peen Hammer

1.1 Layout: **A structured arrangement of features, within certain limits (tolerance), on a part utilizing certain tools for purpose of location for machining operations.**

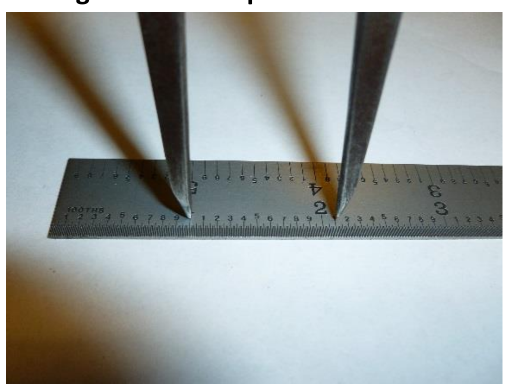
1.1.1 Graduations and Tolerance Definition: **Many tools and machines in a machine shop are graduated in 1/1000 of an inch (.0010). Some tools are graduated to 1/10,000 of an inch (.0001).**

Tolerance: refers to the amount of allowable error that is acceptable for form, fit and function. Most often seen as +/- or + or – for example .750 +/- .005 means the measurement can range from .745 to .755 and still be acceptable. Anything out of that range is regarded as out of tolerance. It is important to tolerance carefully when designing a part for example you have a 1.000 inch diameter pin that needs to fit in a hole easily. The pin may be dimensioned 1.000 +/-.001 so the hole it fits into could be 1.002 diameter +.003/-.000 The pin can measure .999 to 1.001 resulting clearance to be .001 to .004 inches. If you have a precision pin 1.000 +.0005/-.0000 and want to have an interference fit the hole diameter should be .9995 +.0002/-.0002.

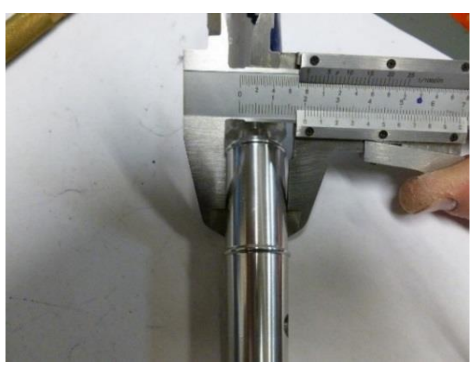
1.2 Tool Uses:

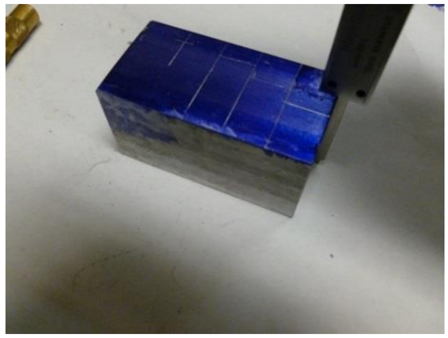
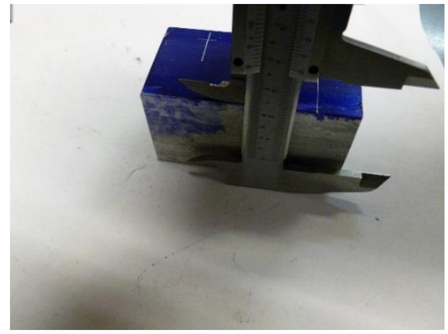
1.2.1 6 inch Scale (Ruler): Used to visually measure and to set gauges to size by use of inscribed specified graduations.

Setting Dividers to specific measurement



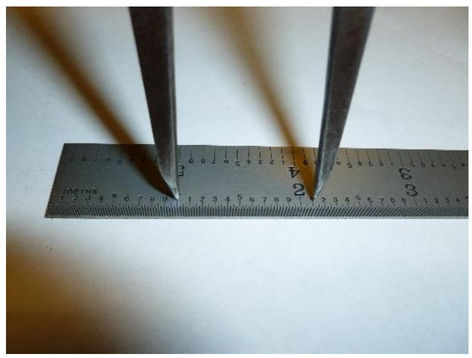
1.2.2 Vernier Caliper: Used to measure outside, inside, depth and steps. The accuracy is +/- 5/1000 inch (.005)

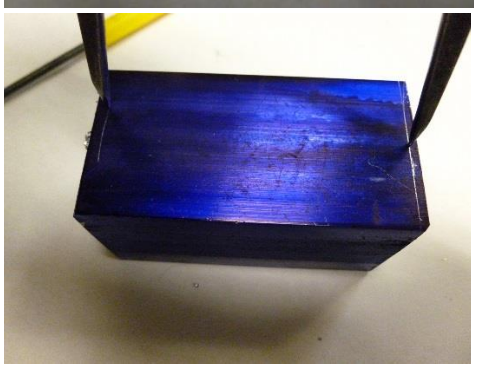


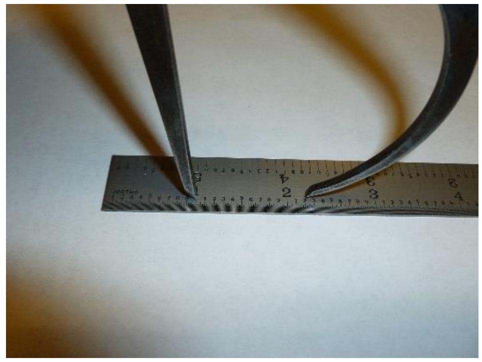
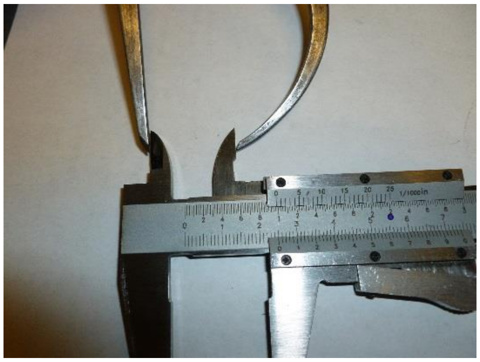
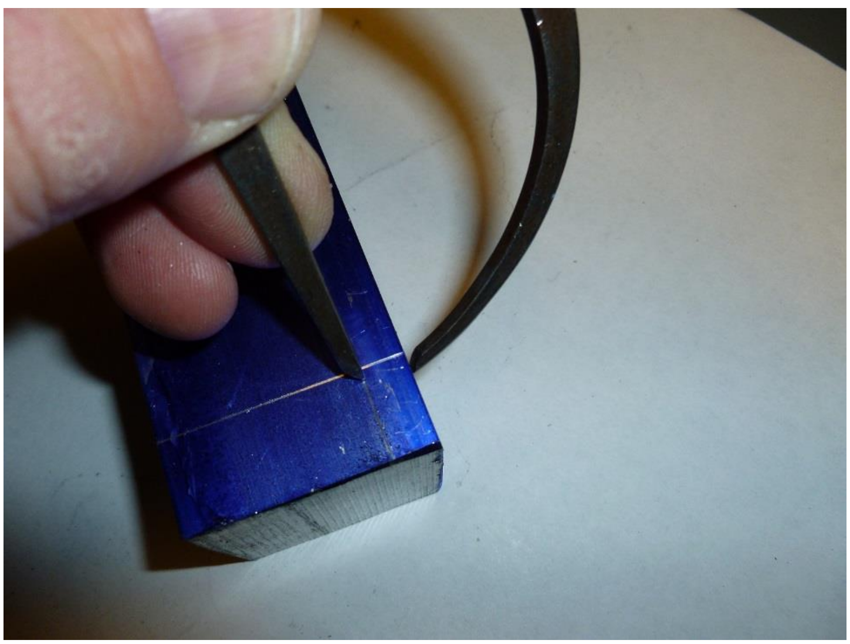
1.2.4 <u>Transfer Measurement and Layout Tools:</u> used to either transfer/gage measurements set utilizing other tools such as a ruler or scale or calipers.

<u>Dividers:</u> Used in layout to scribe arcs, locations and divisions. It has a spring loaded hinge and adjusted by rotating the knurled nut. The blue dye in picture is layout fluid (Dykem) and will be discussed later.

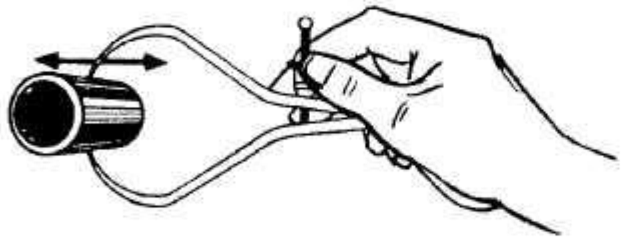
 

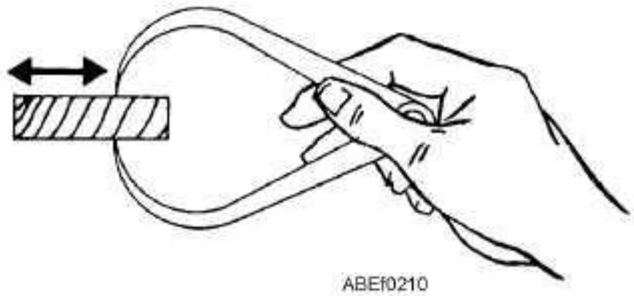


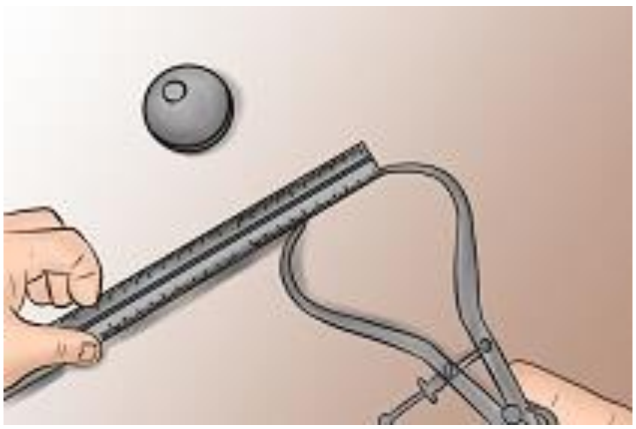
<u>Hermaphrodite Caliper:</u> A tool with two hinged legs used to lay out lines that are parallel with the edges of the work piece. It can also be used to locate the center of cylindrical shaped work pieces.

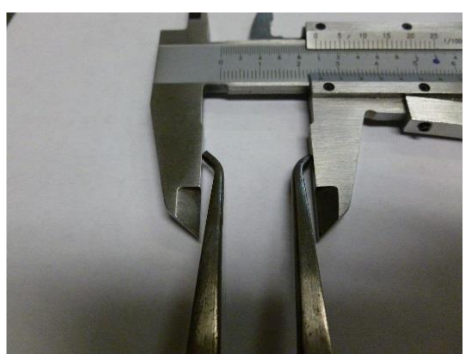
<u>Outside Caliper:</u> A measuring instrument set on adjustable legs and used to measure the outside circumference and diameter of round or cylindrical objects and structures.

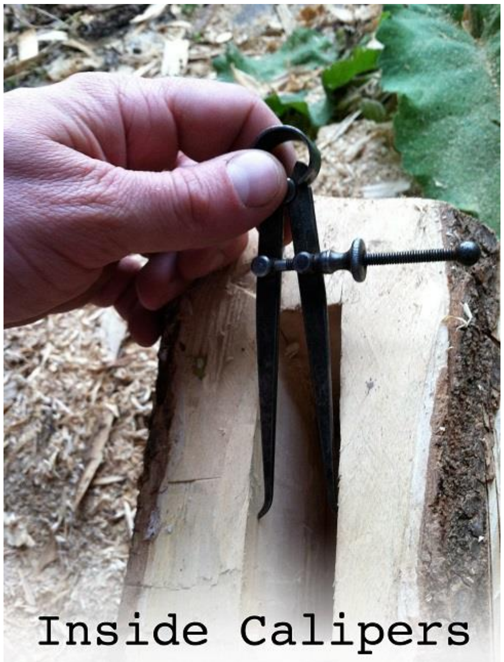




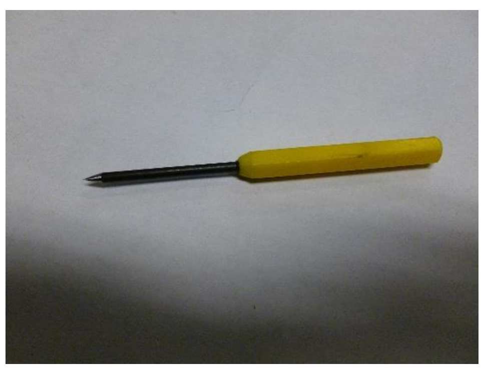


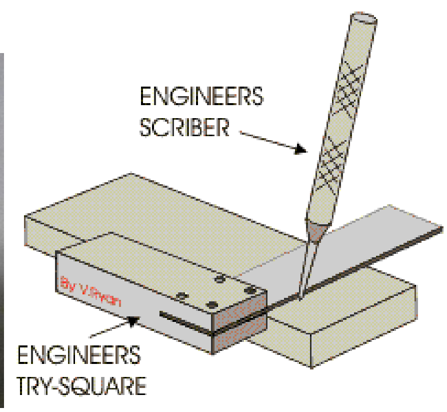
<u>Inside Caliper:</u> A caliper with pointed legs turned outward that is used for measuring inside diameters and features.





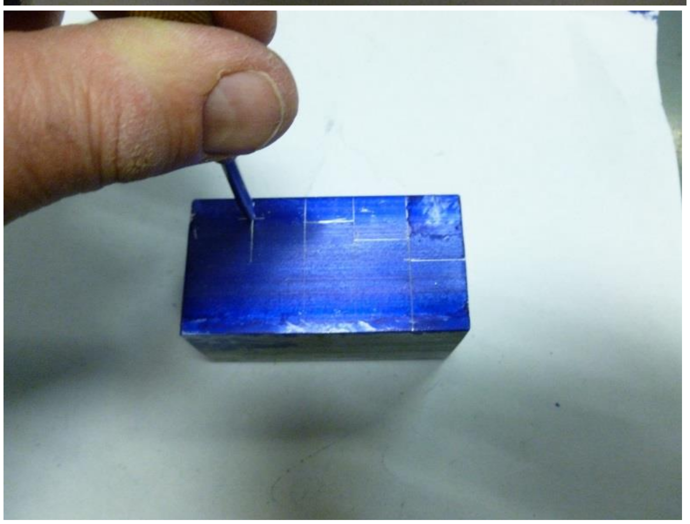
Scribe: A pointed tool used to mark guidelines on wood, metal, composites and plastic.

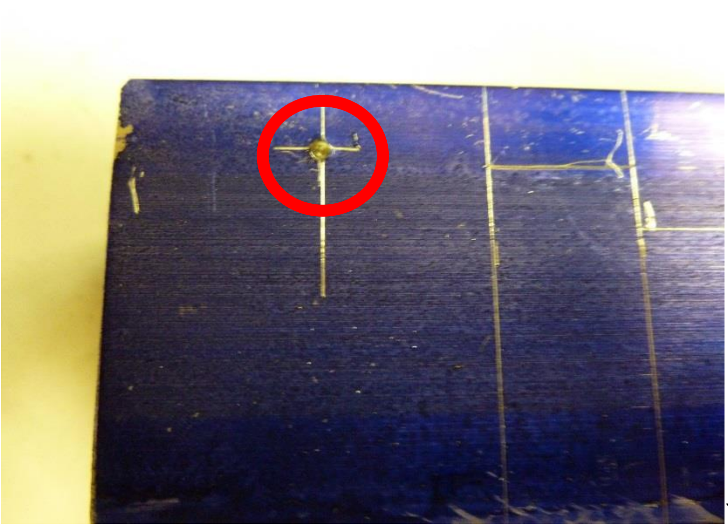




Try Square: A Try Square is a woodworking or a metal working tool used for marking and measuring a piece of material. The square refers to the tool's primary use of measuring the accuracy of a right angle (90 degrees); to try a surface is to check its straightness or correspondence to an adjoining surface. It is also used with a scribe to mark location or a line perpendicular to an edge.

<u>Center Punch:</u> a tool with a conical point for making an indentation in an object, to allow a drill to make a hole at the same spot without wandering.



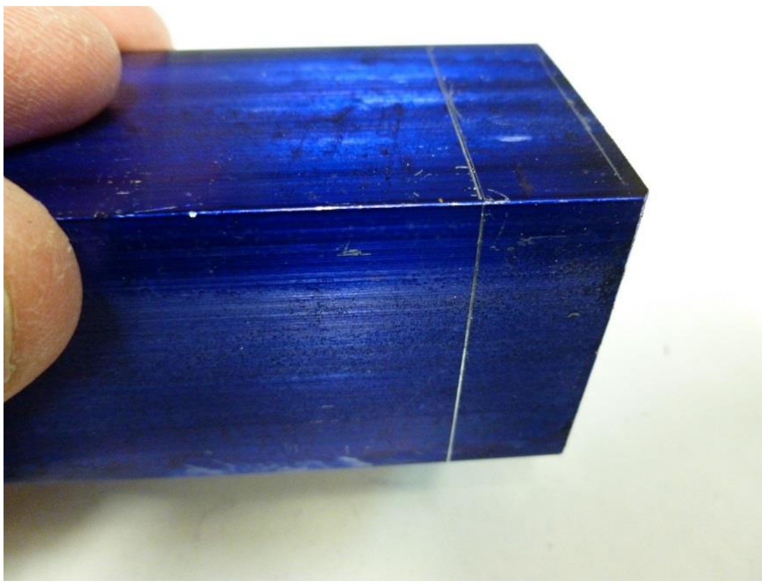


`

Ball Peen Hammer: a tool consisting of a head fastened to a handle and used for striking something. In this case used to strike Center Punch to make an indentation. The round side is used for peening (**Peening** is the process of working a metal's surface to improve its material properties, usually by mechanical means such as hammer blows)

Dykem Layout Blue (layout fluid): It is used to stain or paint a metal object with a very thin layer of dye that can be scratched off using a scriber or other sharp instrument to reveal a bright, yet very narrow line in the metal underneath.

Dykem is a brand name commonly used in industry.



Before we get into the descriptions of some of the machine operations you can get a good idea of some of them by going to You Tube, the channel link is “RPI Processes”. You will have to hunt a bit but the videos are informative particularly if you have not used some of our equipment types before.

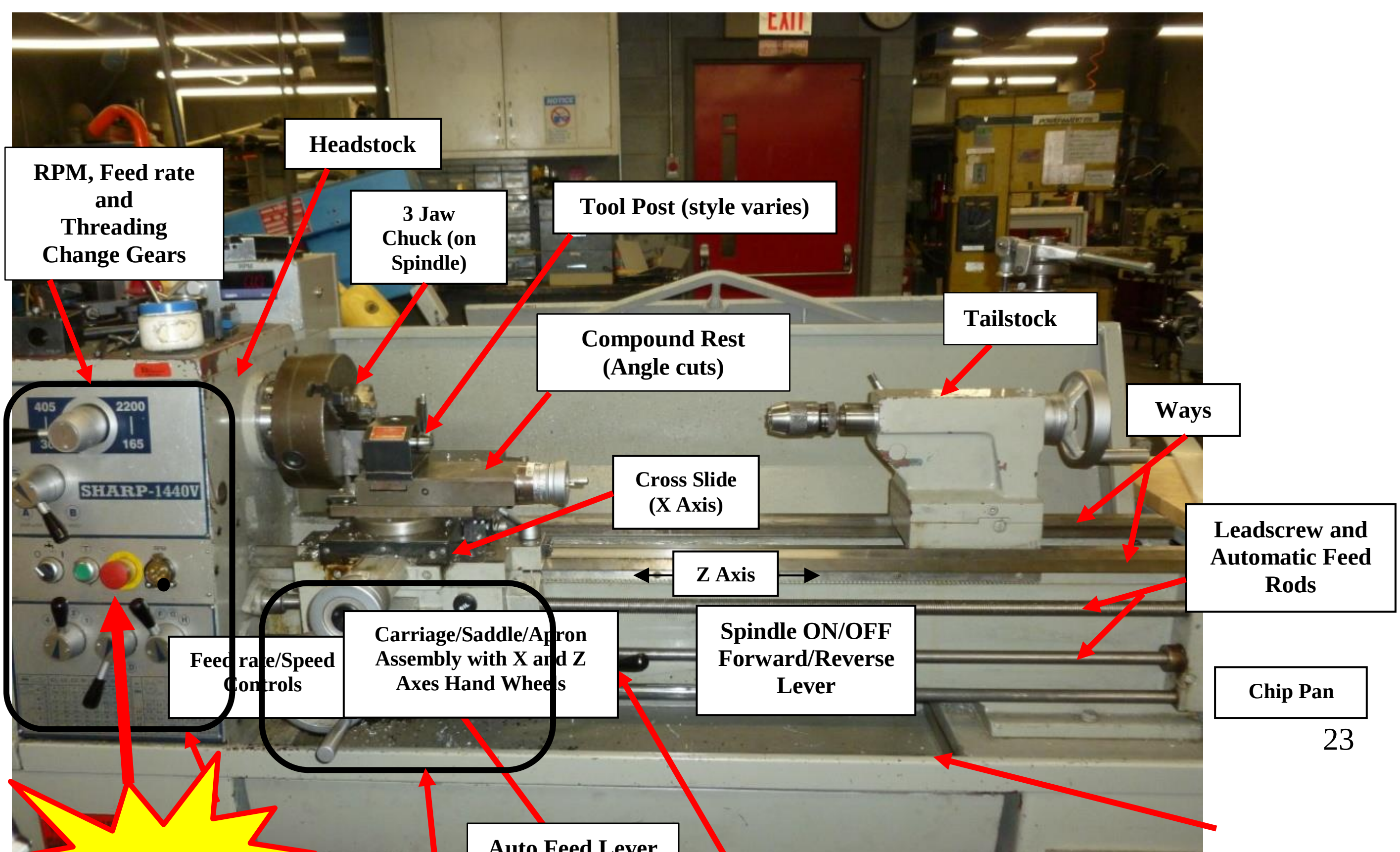
2.0 Lathe overview (parts of an Engine Lathe):

The purpose of a lathe is to rotate a part against a tool whose position it controls. It is useful for fabricating parts and/or features that have a circular cross section. The Spindle is the part of the lathe that rotates. Various work holding attachments such as three jaw chucks, collets, and centers can be held on/in the spindle and all is referenced off the Spindle centerline. The spindle is driven by an electric motor through a system of belt drives and/or gear trains. Spindle speed is controlled by varying the geometry of the drive train by changing gears and often electronically. Feeding along X and Z axes is either manual or automatic.

The Tailstock can be used to support the end of the work piece with a center, or to hold tools for drilling, reaming, threading, or cutting tapers. It can be adjusted in position along the ways to accommodate different length work pieces. The quill can be fed along the axis of rotation with the tailstock hand wheel.

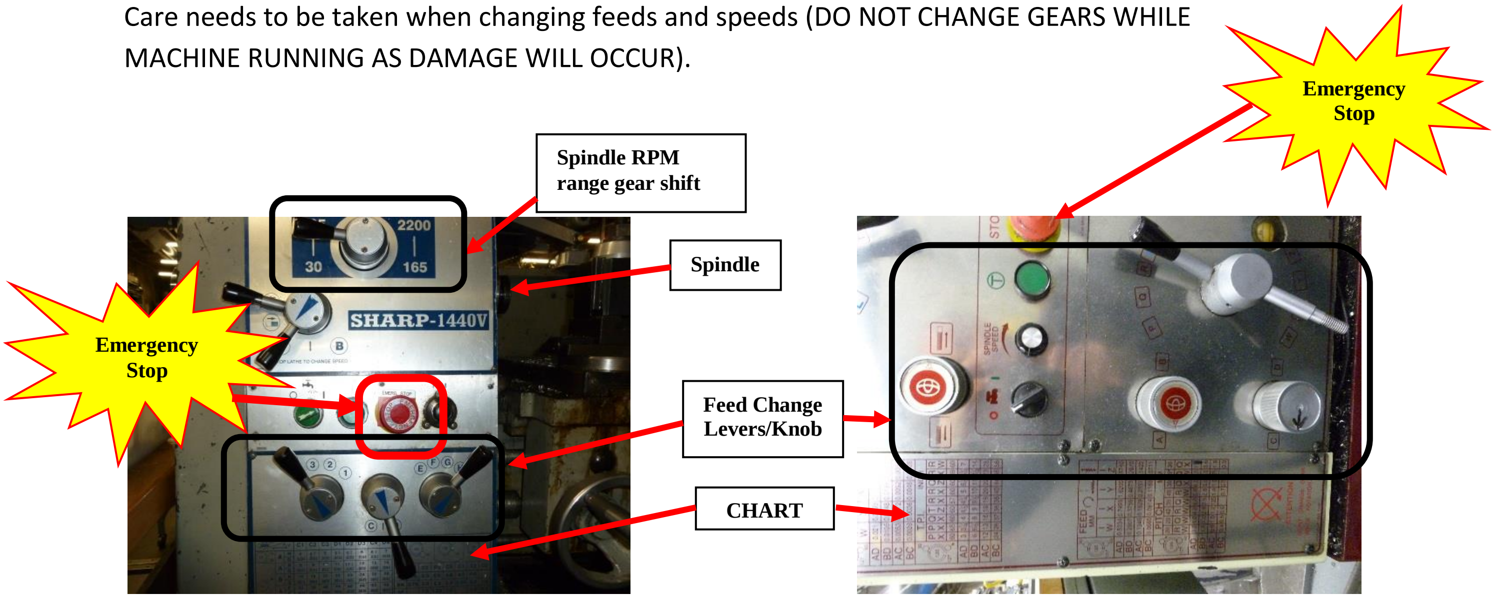
The Carriage controls and supports the cutting tool. It consists of:

- A Carriage/Saddle that mates with and slides along the Ways (Z Axis).
- An Apron that controls the feed mechanisms.
- A Cross Slide that controls transverse motion of the tool (toward or away from the operator X Axis).
- A Compound Rest that adjusts to permit angular tool movement.
- A Tool post that holds the cutting tool.



2.1 <u>Headstock:</u> (picture may vary from other lathes in the shop but process still applies).

The lathes in the shop are driven via belt and gear transmission. Control interfaces can vary. Care needs to be taken when changing feeds and speeds (DO NOT CHANGE GEARS WHILE MACHINE RUNNING AS DAMAGE WILL OCCUR).



The Headstock of the lathe contains the spindle, the controls for spindle RPM (revolutions per minute), longitudinal or carriage feed rate IPR (inches per revolution) and most important the Emergency Stop button (E-Stop). It also has a rotating spindle that either a 3 Jaw Chuck, 4 Jaw Independent Chuck or 5C Collet Closer can be mounted to and will be discussed later.

-Adjusting spindle RPM: with Emergency Stop off (pushed in), "lathe powered off" shift gears to RPM range required while "hand rotating" spindle forward and reverse until you feel gears mesh (clunks into place).

-Adjusting feed rate is also done the same way. Read the corresponding chart located on the headstock and shift the levers to the required position for feed and Carriage direction as designated on the chart before turning machine on. You will notice two numbers for example .005 (IPR or inches per revolution) and 6 (TPI or threads per inch) and is used for cutting threads to be discussed later. You will be using IPR for all of your "automatic" feeds. IPR means the linear travel of the Z Axis (Carriage) travels the selected distance per one revolution of the Spindle. The X Axis (Cross Slide) generally travels ½ the selected distance per one revolution of the Spindle.

2.1.1 <u>Spindle and Mounting Work Holding Attachments:</u>

There are 3 primary methods that are used to hold your work piece in this shop:

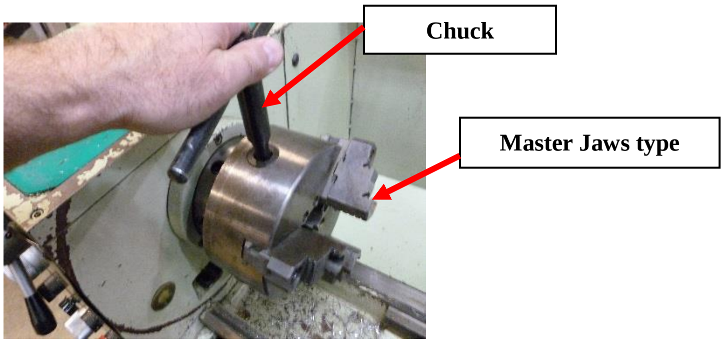
3 Jaw Universal Chuck

Independent 4 Jaw Chuck

5C Collet and Collet Closer

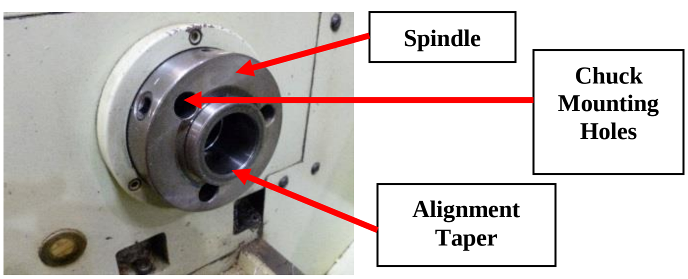
2.1.2 3 Jaw Universal Chuck and Setup:

This chuck requires a Chuck Wrench to move the 3 jaws in and out simultaneously by means of a scroll inside the main body and mating teeth on the jaws. It is self-centering so work piece must be cylindrical or hexagonal. It can hold the work piece on the outside or the inside and accommodates variable sizes. Jaws can also be reversed either by unbolting (known as a Master Jaws type) or requires different set of jaws. This shop utilizes Master Jaw type. **Chuck Key Safety: DO NOT leave inserted in the chuck/spindle. It is very DANGEROUS if machine is turned on.**



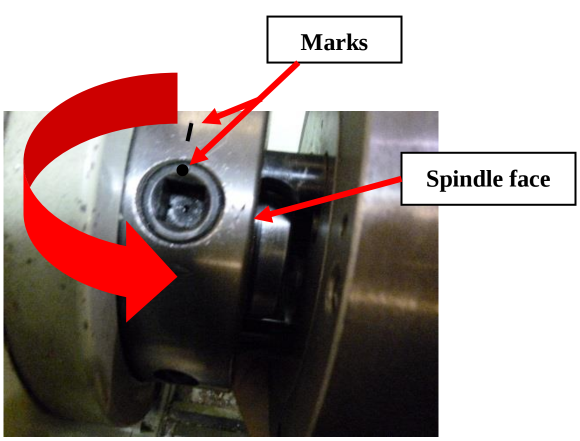
Mounting 3 Jaw Chuck to Spindle: (E-STOP IN, MACHINE POWER OFF)

a) Spindle and Chuck Mounting Holes/Taper (Area must be clean of foreign matter).



b) Chuck Cam lock Pins and Tapered Seat (Area must be clean of foreign matter).

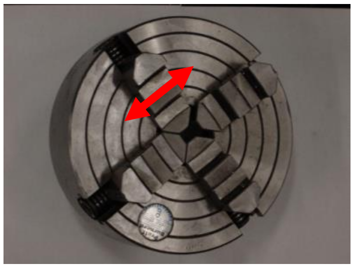
c) Using Chuck Key individually rotate all 3 Cam locks counter clockwise until marks align. Align pins on chuck with spindle holes and slide chuck up to spindle face.



d) Using Chuck Key individually rotate all Cam locks clockwise and you should feel them engage with the pin notch. Make sure they are tight.
To remove chuck repeat step "c"supporting Chuck.

2.1.3 Independent 4 Jaw Chuck:

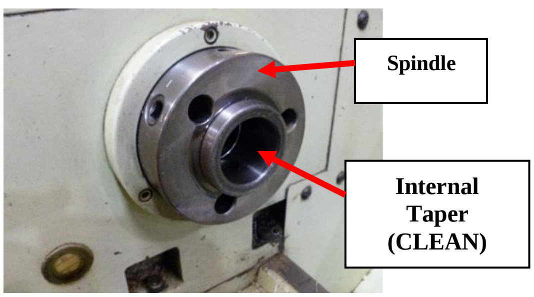
As with the 3 Jaw Universal Chuck this also requires a Chuck Key and is installed and removed the same way. The difference is that each jaw is moved independently and individually of each other. This allows for offsetting work piece and/or features on the piece for cylindrical processes. The work piece can be odd shaped, it does not have to be round or hexagonal shaped. The 4 Jaw Chuck can also hold pieces internally and externally and the jaws can be removed and reinstalled reversed in their respective guides.



2.1.4 5C Collet and Closer Mounting Procedure:

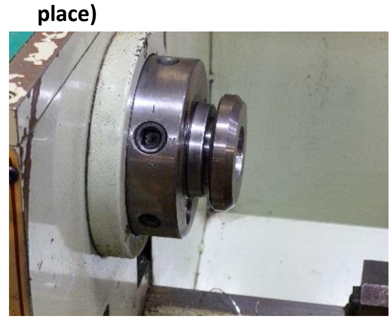
A 5C Collet is a subtype of chuck that forms a collar around the object to be held exerting a strong clamping force on the object when it is tightened, usually by means of a Tapered Adapter and Drawbar. It may be used to hold a work piece or a tool. More specific an external 5C collet is a **"size specific"** sleeve made of spring steel with one or more kerf cuts along its length to allow it to expand and contract with a (normally) cylindrical inner surface and a conical outer surface. The collet is squeezed against a matching taper in the Adapter such that its inner surface contracts to a slightly smaller diameter. Thus, squeezing the tool or work piece securing it for the process to be achieved.

a) Spindle (must be free of foreign matter)

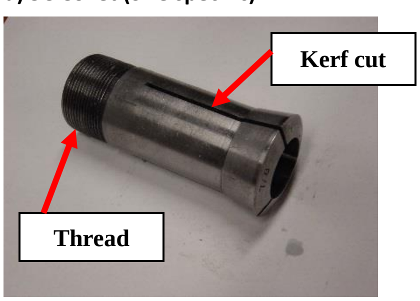


b) 5C Collet Adapter (Keyed and tapered)

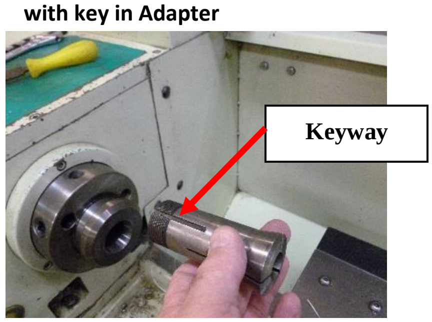
c) Adapter inserted in spindle taper (Morse taper to taper seat locks in place)



d) 5C Collet (Size specific)



e) Insert Collet into Adapter aligning keyway with key in Adapter

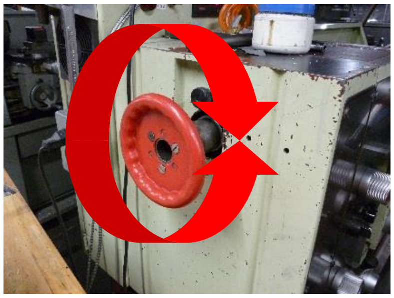


f) 5C Collet in Spindle Adapter

There are 2 different styles of Drawbars to engage collet that are used in this shop as shown on next page.

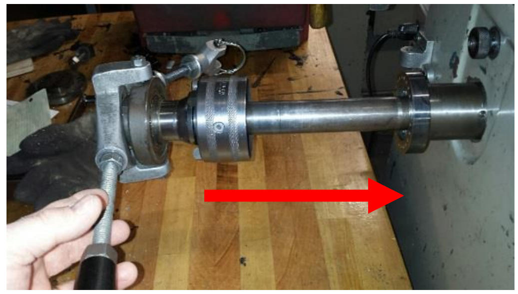
Drawbars:

g) Drawbar (Insert into spindle left side and rotate clockwise to engage collet thread drawing it into taper closing around work piece.
Rotate counter clockwise to release work piece)

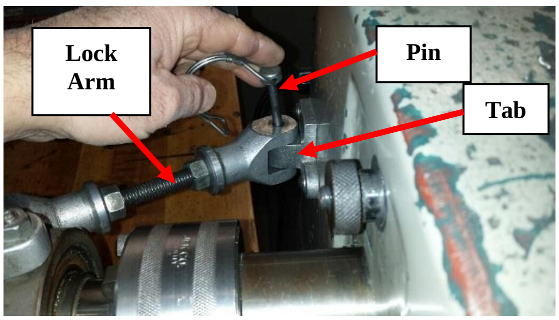


OR

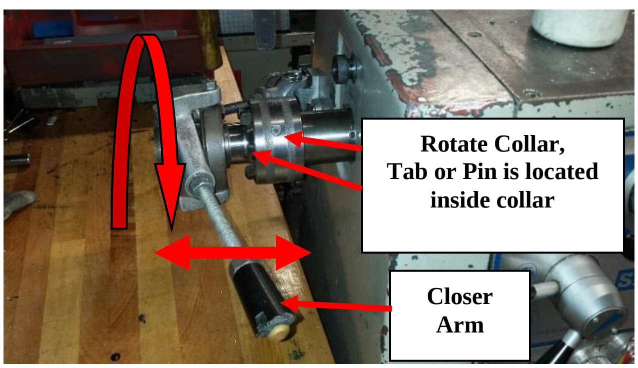
g1) Lever Type Drawbar (also inserted into spindle left side)



h) Swing Lock Arm to tab and pin in place



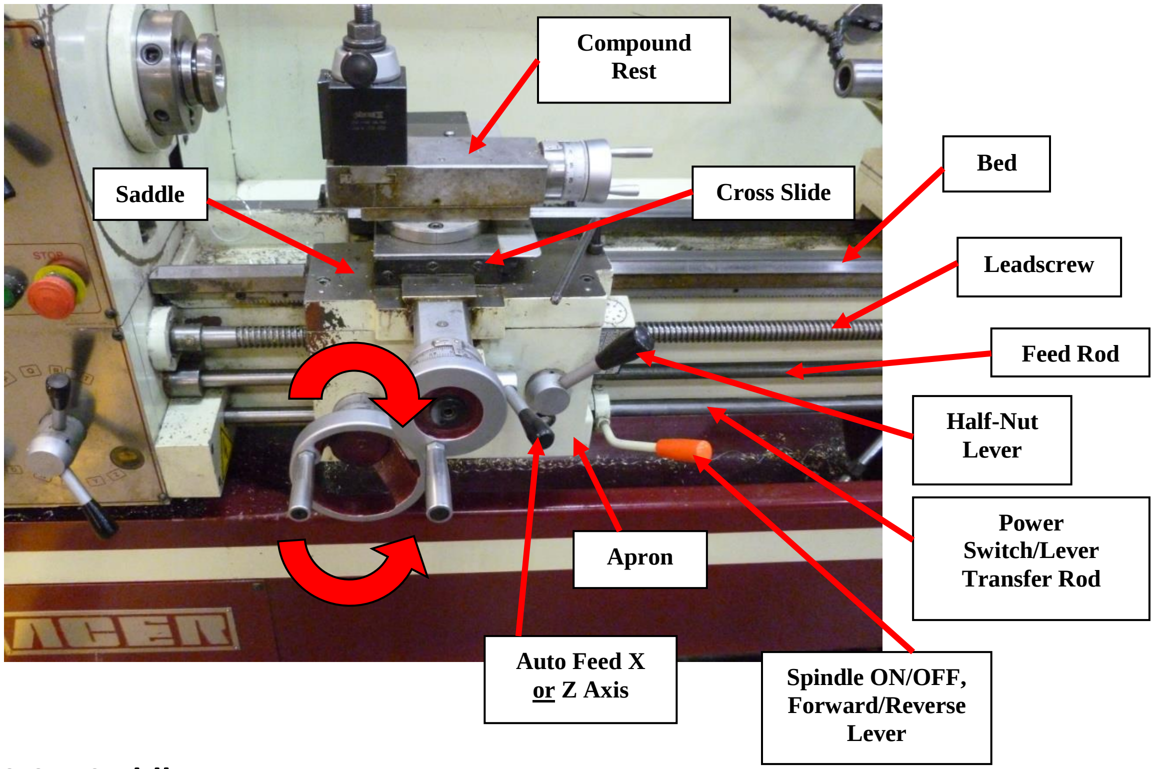
i) Unlock pin or tab, push Collar in towards lathe and rotate clockwise to engage 5C collet.
Proper setting is accomplished when Closer arm is pushed away from operator latching over with about 30 pounds force (over center/detent feeling).
Loosening or tightening Collar sets proper adjustment. Lock pin or tab when set.
To remove work piece pull Closer Arm toward operator while holding work piece.
(Removing Collet: Unlock pin or tab and Rotate Collar counter clockwise until fully unthreaded).



<u>*Removing Collet Setup:</u> with machine power off, fully unthread Collet, remove Collet, remove Collet Closer pin, pull out of spindle, hang Closer on rack, insert long enough substantial size aluminum bar to reach Collet Adapter into left end of the Spindle where the Closer was previously removed, hold Collet Adapter with right hand and slide bar with left hand hitting the back of the adapter with a rubber mallet to break the taper free. NEVER use a steel bar to remove adapter as damage will occur.

2.2 Carriage Assembly:

The whole assembly of Saddle, Apron, Cross Slide and Compound Rest is known as the Carriage.



2.2.1 Saddle:

The Saddle is the casting that fits onto the top of the Bed and slides along it. The axis of the bed is known as the “Z Axis”.

2.2.2 Apron:

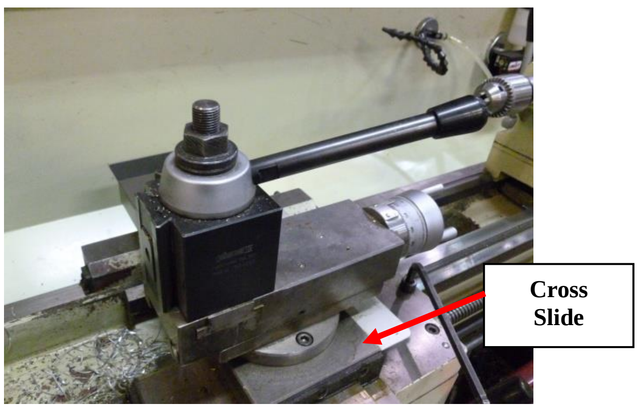
The vertical, flat and rectangular "plate" fastened to the front of the Saddle is known as the Apron and carries a selection of gears and controls that allow the carriage to be driven (by hand or power) up and down the bed which is also the Z Axis. The mechanism inside can also engage the screw cutting feed and various powered tool feeds. The Leadscrew, and sometimes a power shaft as well, are often arranged to pass through the apron and provide it with a drive for the various functions. Virtually all screw-cutting lathes have what is commonly-called a Half-Nut Lever that closes down one and sometimes two halves of a split nut to grasp the Leadscrew providing a drive for screw cutting. The Apron also provides a convenient, traversing location for the Spindle ON/OFF, Forward/Reverse Lever via a Switch/Lever Transfer Rod.

Threading/Auto feed Selection Controls:

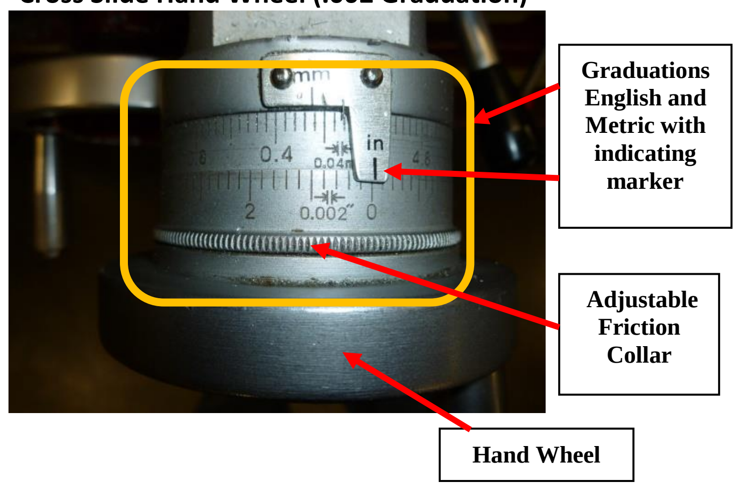
When the Auto Feed Lever is the X Axis Cross Slide will move in or out or the Z Axis Carriage will move left or right. Since the tool bit is mounted on the Tool Post on the Carriage the tool bit will move with it.
In this way, controlled cuts can be taken. The controls on the Head Stock panel (section 2.1) allow the user to select how fast the bit moves, relative to one rotation of the Spindle. Note: by selecting too fast of a cut can result in a poor finish and too slow is not always appropriate either. This will be discussed later in the manual.

2.2.3 Cross Slide (X Axis) and Hand Wheel Graduations:

Sitting on top of the Saddle is the Cross Slide. As its name implies, moves across the bed known as the X Axis driven by hand or power (Auto Feed). The hand wheel is graduated in .001 inch (1/1000). When the Hand Wheel is rotated 1 line graduation (.001 inch) it physically moves the Cross Slide .0005 (5/10000 inch, or 500 millionth inch) in the X Axis. Being that there is a centerline in the rotational axis and the Cross Slide physically moved .0005 resulting in .001 inch being removed from the work piece diameter. On the Hand Wheel below the Graduation is .002 meaning the Cross Slide physically moves .001 removing .002 from the diameter of the work piece. The Friction Collar is adjustable. While holding the hand wheel still rotate the collar with other hand. This allows operator to zero the position.



Cross Slide Hand Wheel (.002 Graduation)



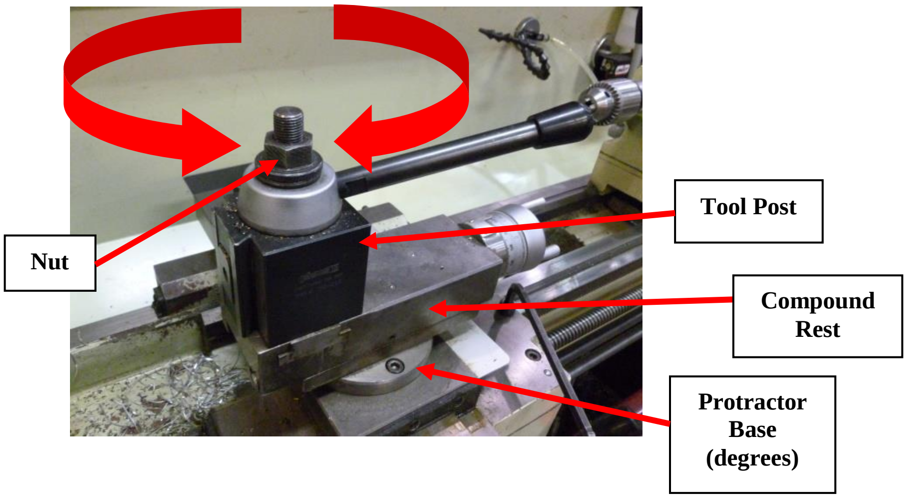
2.2.3.1 Longitudinal Feed Hand Wheel (Z Axis) and Graduations:

The graduations on this Hand Wheel are "direct" linear (.100 = .100) moving the Carriage Assembly in the Z Axis. This also has an adjustable friction collar to set zero reference.

2.2.4 Compound Rest:

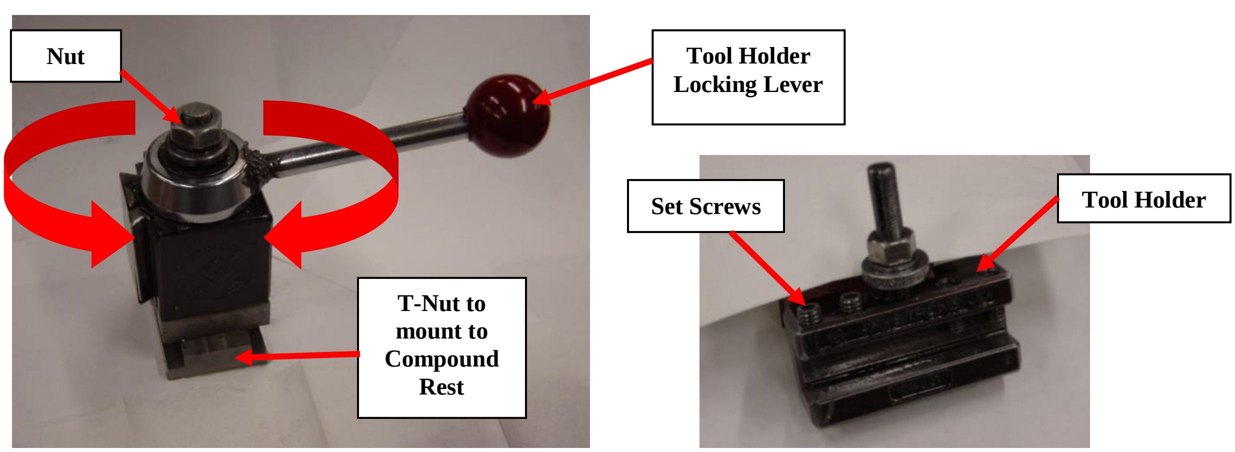
The Compound Rest is where the Tool Post is mounted. It provides a smaller amount of movement (less than the Cross Slide) along its axis via another feed screw. The Compound Rest axis can be adjusted independently of the carriage or cross-slide. It is used for turning tapers, to control depth of cut when screw cutting or precision facing, or to obtain finer feeds (under manual control) than the feed shaft permits. The Compound Rest has a protractor marked in its base enabling the operator to adjust (rotate) its axis to precise angles by loosening screws, rotating to desired angle and re-tightening them. The angles are referenced to the working plane of the work piece (Face of the work piece or centerline of the spindle).

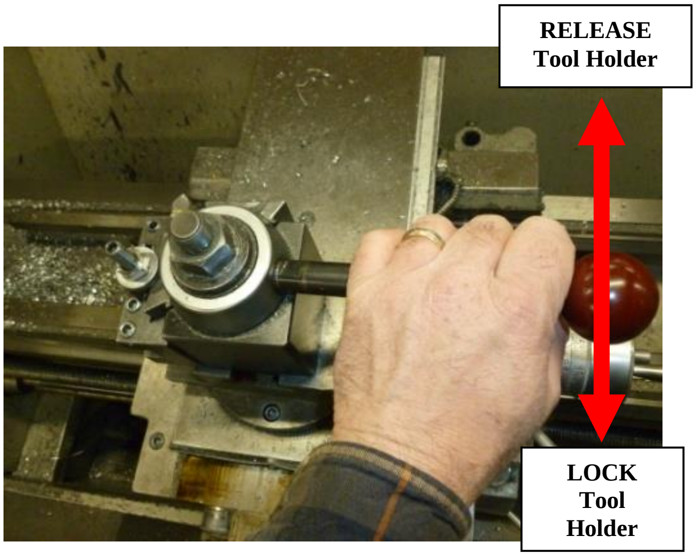
The Graduations on the hand wheel are generally .001 (1/1000) inch increments, unless otherwise specified on the adjustable friction collar (holding the hand wheel still the collar can be rotated to zero the position). The physical movement of the Compound Rest is equal to the graduation (.001 in. = .001 in.), unlike the Cross Slide physical movement. One full rotation is generally .100 (100/1000 inch). (Also see section 2.4.1)

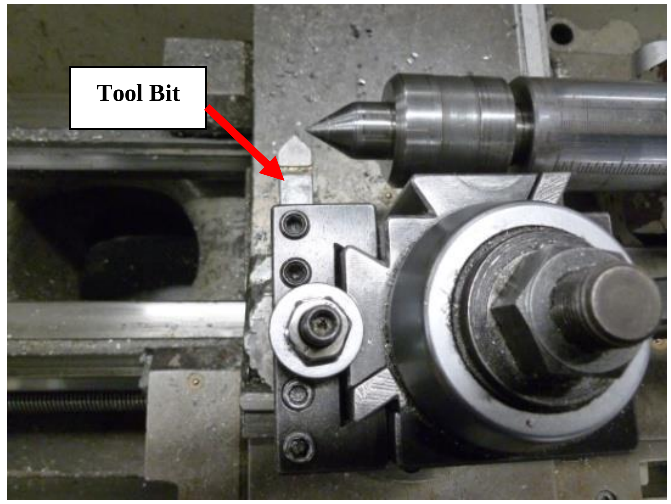


2.2.5 Tool Post:

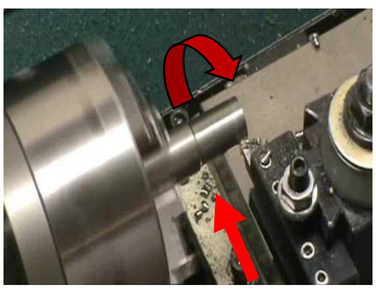
The Tool Post is the attachment point for the tool holder, in which the lathe tool bit is installed. The Nut at the top of the tool post allows the Tool Post to be rotated on the Compound Rest if necessary. The Tool Holder Locking Lever locks and releases the Tool Holder from the Tool Post. When the lever is pulled clockwise, the tool holder is locked onto the tool post. When the lever is pushed counterclockwise, the tool holder is released. The Tool Holder has Set Screws to clamp down on the shank of the tool bit holding it in place.







2.2.5.1 <u>Tools typically used with Tool Post:</u> Cutting, turning, grooving and other shape forming on a lathe is done by the stock rotating in a forward motion against the cutting tool. The cutting tool is moved in gradual increments into the stock shaving material away. As discussed prior there are graduated increments on the hand wheels used to obtain proper depth of cut and distances. (Section 2.4.2 describes how to use these tools)



Round Nose Turning Tool

Carbide Insert Parting/Grooving Tool

Knurling Tool

Ridge Forming Tool

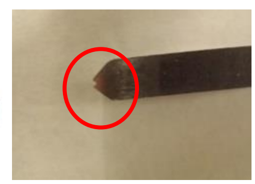
<u>Tool Uses:</u>

- **Round Nose Turning Tool:** Used in general reduction of stock diameter and Facing off work piece.
- **Parting/Grooving Tool:** Used to either cut off part to length or to create a groove into the stock by rotating the Cross Slide Hand Wheel along the X Axis, and Spindle rotating forward.

- **Knurling Tool:** Used to create either a diamond, angular or straight line pattern on the part usually for grip enhancement (hills and valleys). It is accomplished by the displacement of the material by two hardened rollers containing the pattern pressing against it. Knurl cutting for longer sections is usually accomplished using the same automatic-feed mechanisms that are used to cut screw threads and can be thought of as simply a series of threads cut at extremely coarse pitch and in both the left-hand and right-hand directions. The RPM is between 300 to 400 RPM and a flood of coolant utilized to decrease friction and clean chips away. Please see Instructor prior to performing this operation.

Knurled

Knurling Tool in use

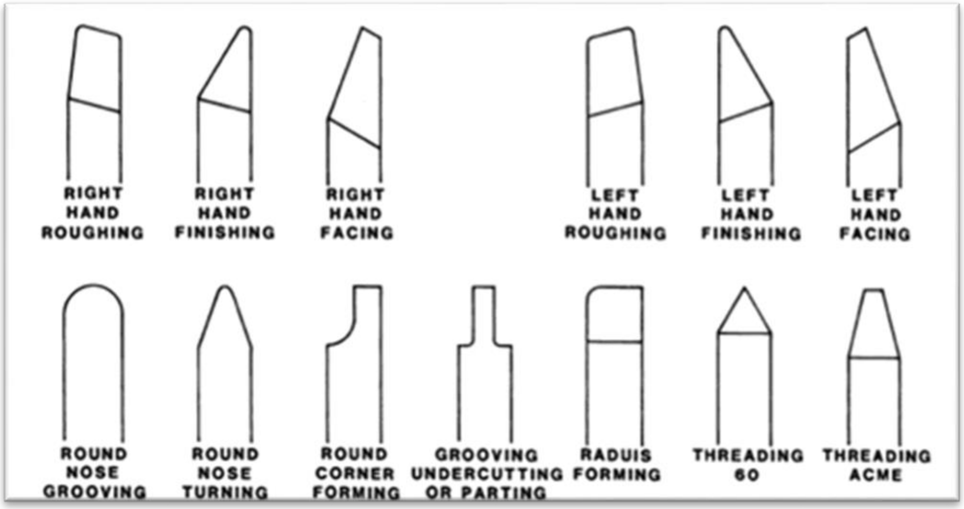


Ridge Forming Tool

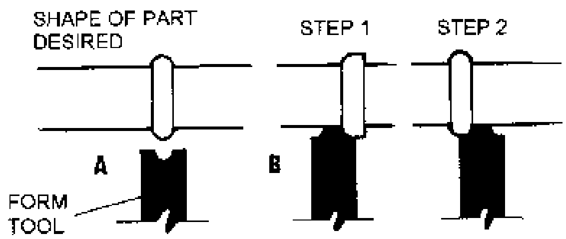
- **Ridge Forming Tool:** Used to create a raised ridged surface.

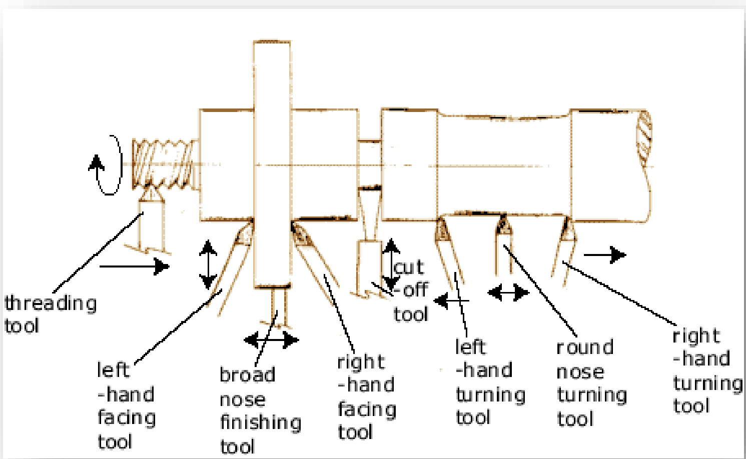
(Other shapes can be ground to produce a variety of features (see section 2.2.8.1 Cutting Tool Nomenclature for rakes, reliefs and angles).

Assorted Tools shown and application:

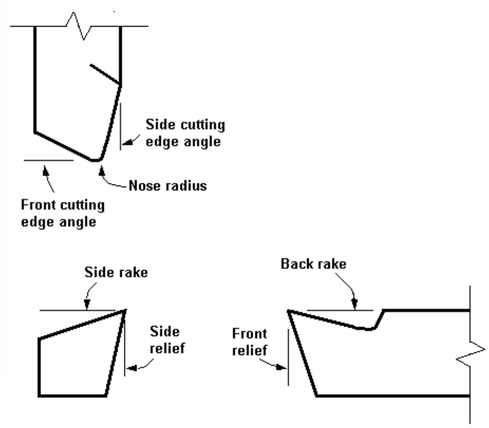


<u>Ridge Forming Tools</u>





Cutting Tool Nomenclature (See Section 1.2.8.1 for



(Setting up Cutting Tool procedure is defined in Section 1.2.8)

2.2.6 <u>Tailstock:</u>

The Tailstock Assembly is used for a variety of different tasks and slides along the Ways, the two long rails along the top of the lathe. It is mainly used for drilling holes in the work piece along the axis, and for supporting extended work pieces. The Tailstock has been set up so that any tool installed into the Quill will be centered on the axis of rotation of the Spindle in the Headstock. The Tailstock has many features such as:

<u>The Tailstock Locking Lever:</u> Pulling the lever towards the Headstock will clamp the Tailstock in desired location on the Ways and pulling away unlocks it for sliding along the Ways (Z Axis).

<u>The Quill:</u> A graduated, hollow, non-rotating spindle tube in the tailstock. It moves in a linear direction in and out via a precision Graduated Hand Wheel and threaded rod. Various tools can be inserted into the Quill as needed. It

has an internal Morse Taper and Slot matching the Taper on the tool and the Tang on the tool. The Tang inserts into the Slot in the Quill preventing rotation created by cutter force. The Quill also has engraved markings on it allowing the operator to move it a measured distance.

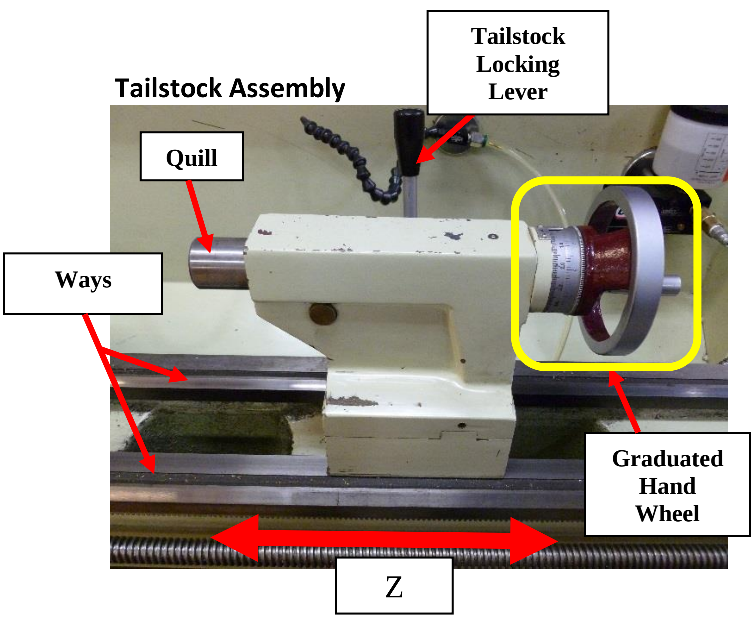
Quill with Drill Chuck Inserted

Morse Taper shank Drill Bit with Tang

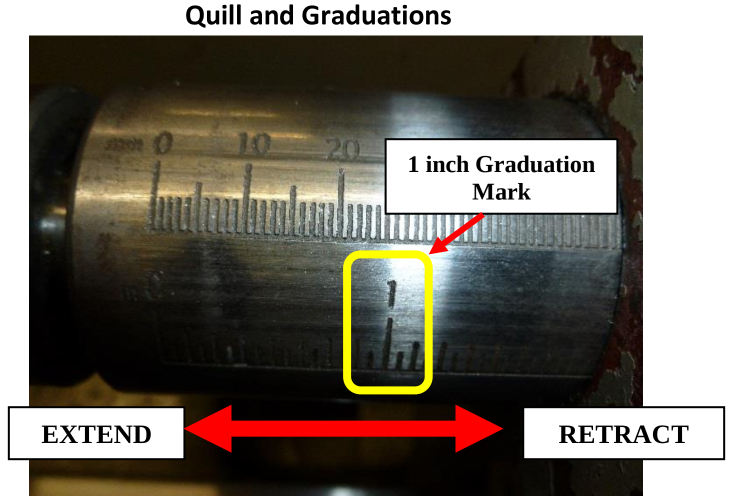
Graduated Hand Wheel: Operator can rotate it for precision movement of the Quill as it has a resettable collar generally graduated in .001 increments, (defined on collar and direct movement) either extending the Quill out from or retracting it into the tailstock.
If the Quill is retracted into the Tailstock past the 1 inch graduation, any tool installed in the Quill will be ejected. **This is how a tool is removed**. Please hold onto tool when doing so.
DO NOT "EXTEND" the Quill past the last graduation mark as it will come off the threaded rod and keyway inside the Tailstock.

Quill Locking Lever: Used to clamp Quill in place. This is a brake on the motion of the quill. Locking the Tailstock and/or the Quill in place is utilized during some processes to be discussed later.

Tailstock Assembly

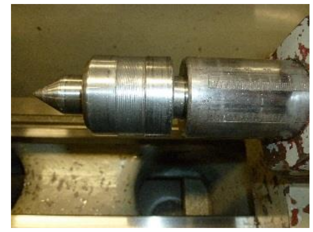


Quill and Graduations



2.2.6.1 Tools typically used with Tailstock:

Live Center **Jacobs Keyless Drill Chuck** **Jacobs Key Type Drill Chuck (installed)** **Live Center(installed)**



Center Drill

Center Drill in Keyless Drill Chuck

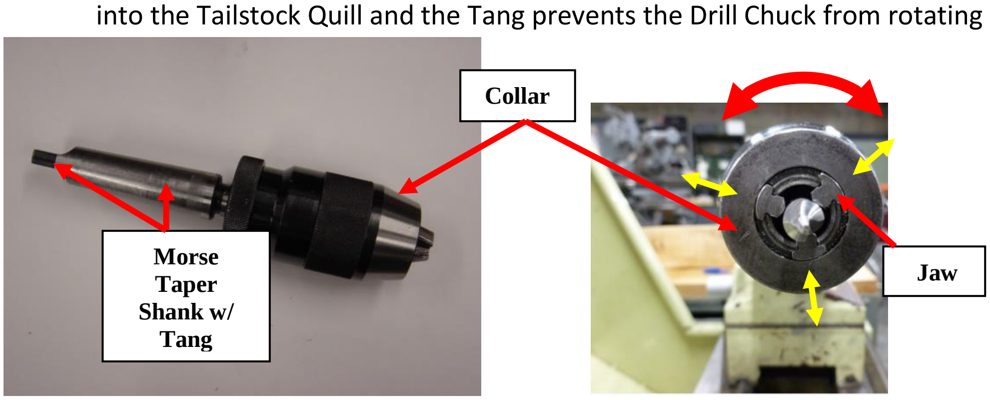
Drill Bit in Keyless Chuck

Countersink Bit

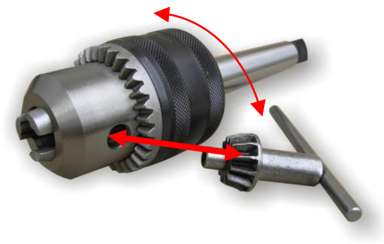
Morse Taper Shank Drill Bit

- **Live Center:** Used to support long pieces extending from the Chuck or Collet. Work piece must be Center Drilled **(section 2.2.7)** prior to utilizing Live Center. Rule: 2.5 times diameter sticking out in length requires us of Live Center.

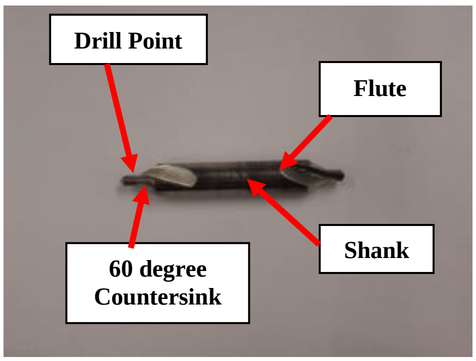
- **Keyless Drill Chuck:** Used in conjunction with the Tailstock to hold “straight shank” drill bits for making holes. It has a Morse Tapered Shank with locking Tang and 3 jaws that hold drilling tools. The jaws move simultaneously as the Collar is rotated by hand either opening or closing them. The Tapered Shank inserts into the Tailstock Quill and the Tang prevents the Drill Chuck from rotating caused by cutter force.



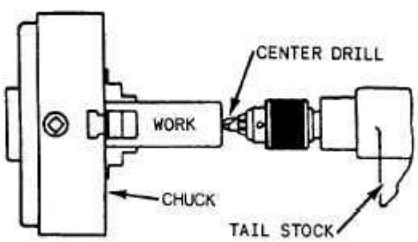
- **Key Type Drill Chuck:** Functions the same as the keyless chuck except it utilizes a key to tighten the jaws. The pilot fits into the hole and the gears mesh. Rotating the key either tightens or loosens the jaws.



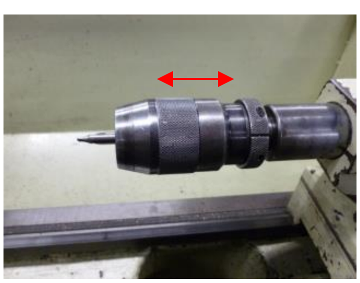
- **Center Drill:** Used to make a starter hole for a larger Drill Bit or to make a 60 degree countersink to support work piece using a Live Center. It is usually double ended and contains a cutting tip, flute to help cutting fluid reach the tip and to aid in chip removal. It also has a rigid shank in which it is held by.



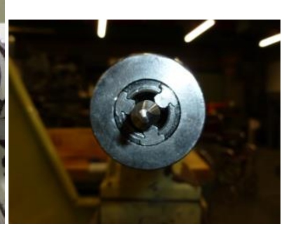
2.2.7 <u>Producing a Center Drilled Hole:</u> necessary prior to drilling a hole preventing drill from wandering off center.



(Facing work piece prior is recommended see section 2.2.9.1) with spindle off, install drill chuck in tailstock quill (tapers must be clean of debris and chips) aligning tang with slot in quill, lightly slam in by hand. It should be held in place, if not quill is retracted too far and must be extended. Insert center drill into jaws of drill chuck making sure it is centered grabbing on the shank of the center drill, rotate drill chuck collar toward operator until it is tight. Carefully slide tailstock until Center Drill is approximately ½ inch from face of work piece, lock tailstock in place, brush part face with cutting fluid, turn on spindle "forward" rotation (between 300-400 rpm), slowly rotate tailstock hand wheel until drill makes contact and continue to required depth. **Note:** chips may need to be cleared from drill to prevent breaking,
DO NOT use fingers as chips are hot and sharp. **Turn off spindle and remove with pliers or brush away.**

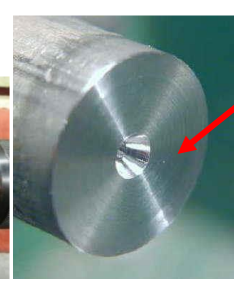


Drill inserted properly



Drill inserted IMPROPERLY

Properly Center Drilled hole



Face of work piece (Faced off)

- **Drill Bit:** (Tapered shank or Straight shank): A drill is an end-cutting tool for producing holes. It has one or more cutting edges and flutes to allow fluids to enter and chips to be ejected. The drill is composed of a shank, body and point. The body of the drill extends from the shank to the point, and contains helical flutes. Drills are size classified 4 ways; Fractional, Number (1 to 97), Metric and Letter A to Z). All of which have a size designated to them and can be found either on a Drill Chart located in the shop or more commonly found in the Machinery Hand Book. The Machinery Handbook references nearly all tooling and processes in a machine shop and is a great reference guide.

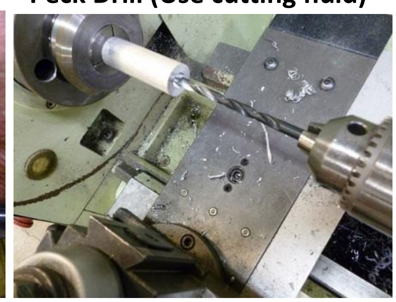
2.2.7.1 <u>Producing a Drilled Hole to depth by Peck Drilling:</u>

Center drill work piece first, turn off spindle, change to required drill bit, carefully slide tailstock until drill is approximately ½ inch from face of work piece, lock tailstock in place, brush part face with cutting fluid, turn on spindle "forward" rotation (300-400 rpm), slowly rotate tailstock hand wheel until drill makes contact and continue to drill approximately 1/8 into the part, back drill out, clear chips away with lube brush. If chips are stubborn **Turn off spindle and remove with pliers or brush away. DO NOT use fingers** as chips are hot and sharp. **Note:** chips must be cleared from drill to prevent breaking. Continue to desired depth repeating approx. 1/8 inch increments, back out clearing chip, **"Peck Drilling"** material away while continuously applying cutting fluid.

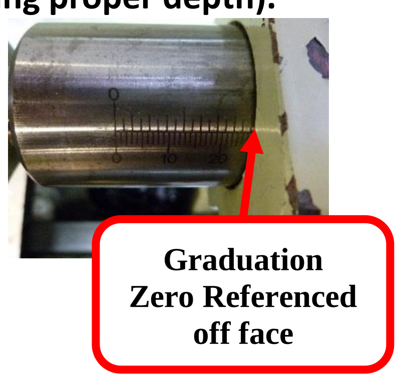
Center Drill

Change Drill Bit

Peck Drill (Use cutting fluid)

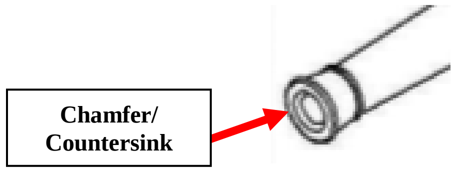


(Friction Collar on Tailstock Handle can be zeroed at point of contact for reference and the Quill also has increments to assist in obtaining proper depth).

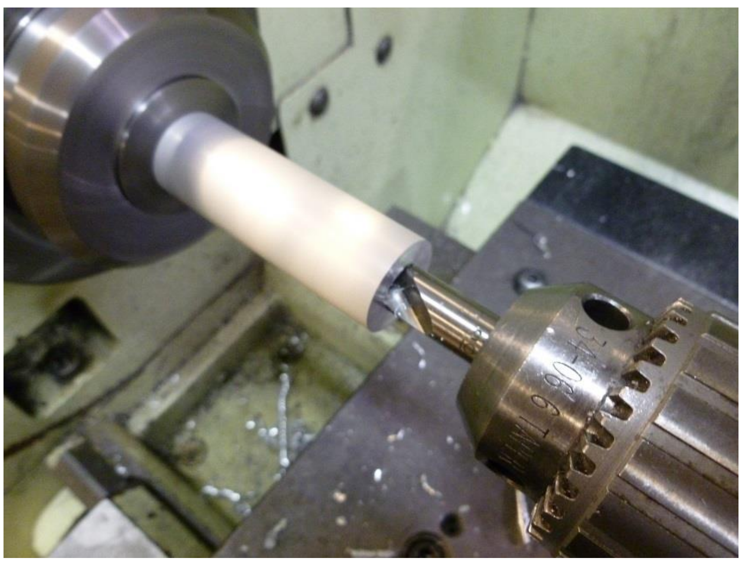


- **Countersink Bit:** Used to produce an angled or chamfered edge to a hole. A hole must be drilled prior to countersinking/chamfering. (Note: a large Center Drill may also be used to produce Countersink/Chamfer)

Countersink Bit

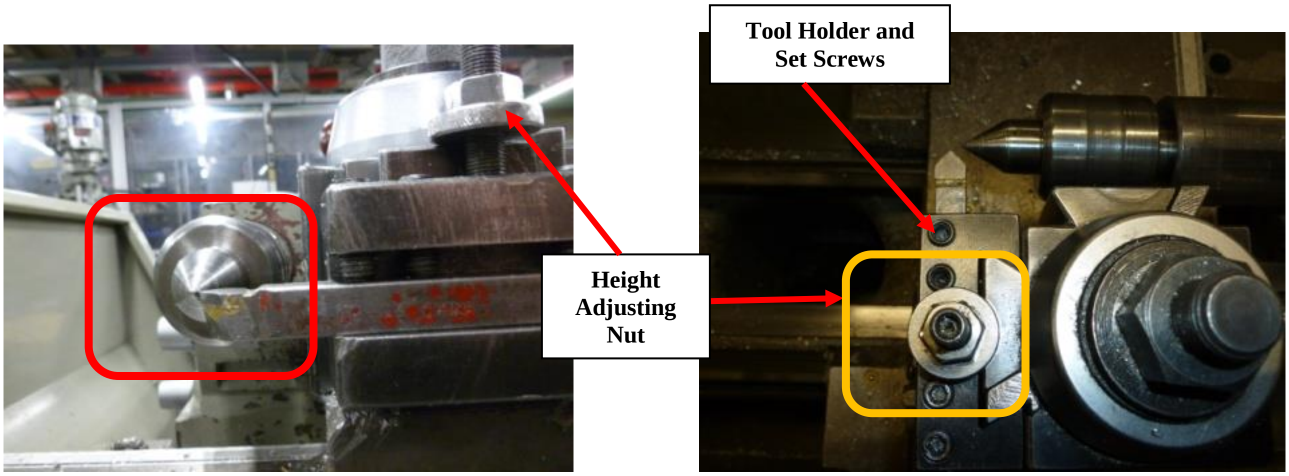


2.2.7.2 <u>Producing a Chamfer or Countersink Hole:</u> Drill desired diameter hole, turn off spindle, install Countersink in Drill Chuck, follow previous drilling procedure (315 RPM) and go to required depth and or diameter while applying cutting fluid.

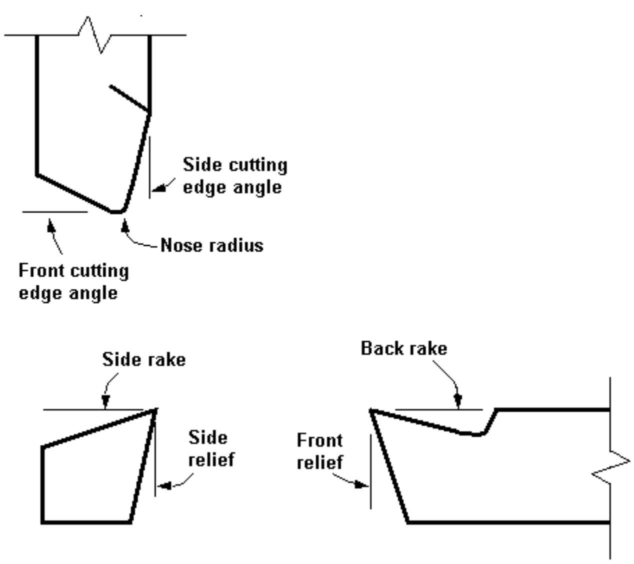


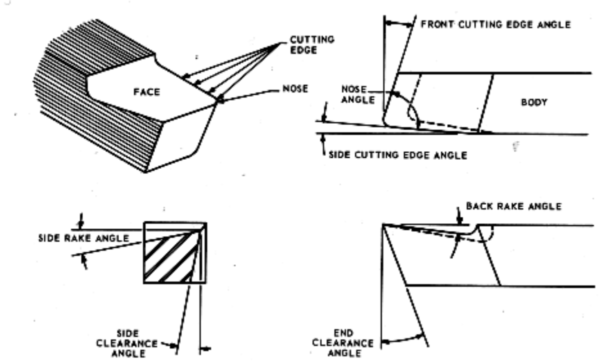
2.2.8 <u>Lathe Cutting Tool Setup:</u> The cutting tool must be setup on the centerline of the spindle. If is below the centerline the part you are turning can ride up over the cutter and break. Too high above can cause rubbing on the material as it is trying to cut with the clearance angle of the cutter. This creates massive friction, poor finish and material welding to the tool bit.

<u>Cutting Tool setup procedure:</u> (For all cutting and forming tools)With Tool Holder Locking Lever Locked, mount tool in Tool Holder tightening Set Screws using a hex key. Install Live Center in the Tailstock Quill, unlock Tool Holder Locking Lever, adjust tool to the centerline by rotating Height Adjusting Nut until proper alignment with Live Center tip is obtained. **"Lock"** Tool Holder Locking Lever and tool is now ready for cutting.



2.2.8.1 Cutting Tool Nomenclature: The purpose of side rakes, reliefs and angles is to form a single sharp cutting edge reducing rubbing/friction. Back rake is utilized to produce a number “9” shaped chip when turning as it guides the chip, breaking it to that shape (preferred chip). **Long, stringy chips are dangerous as they become entangled on rotating parts. Care must be taken when this happens. Turn off machine, and clear with pliers (NOT WITH FINGERS)**.

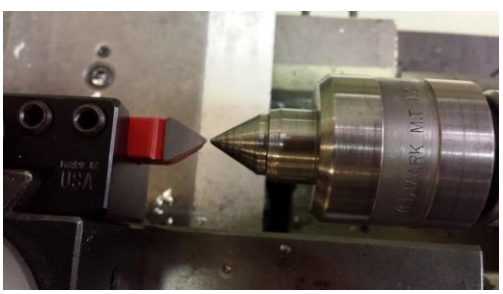




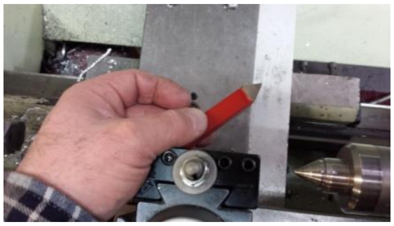
2.2.9 Facing work piece: creates a smooth, flat, face very accurately and perpendicular to the axis of a cylindrical part.

2.2.9.1 <u>Procedure for Facing Work Piece:</u>

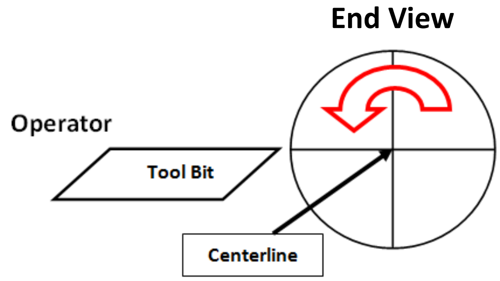
- Machine power off,
- Set tool on center: With Tool Holder positioned shown below and set screws loose, insert Cutting Tool in holder with tip facing Live Center Point (away from work piece in Collet). Adjust Height Adjustment Nut (section 2.2.8) on Holder until Tool tip is centered with Live Center point.



- Lock Tool Holder Locking Lever (section 2.2.5),Take out tool and replace it back in holder with the cutting edge facing the work piece and **TIGHTEN SET SCREWS** with Hex Key.

- Be sure body parts, tools, lathe parts, and cutter are clear from collision of rotating Collet or Chuck prior to turning on Spindle. **Tool should be positioned like this:**

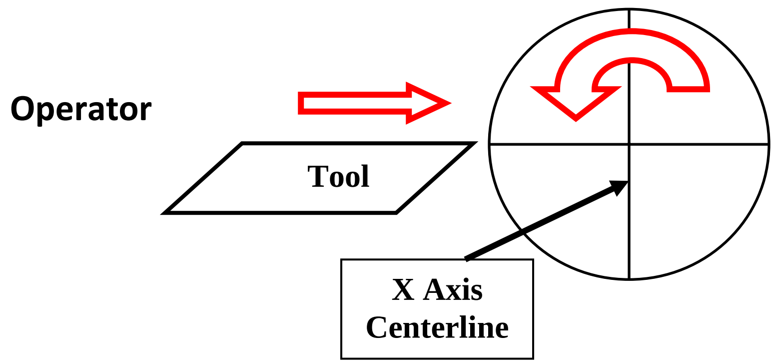


- Turn on Spindle Forward (rotates counter clockwise at 300-400 RPM for this class) using Spindle On/Off Forward/Reverse Lever (down is generally forward, **CENTER is OFF** and up is reverse).

- Tool towards operator in the X Axis. Slowly rotate Carriage Hand wheel moving Carriage towards work piece face (Z Axis) until close to face of material. Rotate Cross Slide Hand wheel until it is near part face. Now utilizing both hand wheels lightly touch the face of work piece. This is now your zero position. Back away tool towards operator in X Axis (Cross Slide Hand Wheel). Z Axis movement can either be measured by the Hand wheel graduations (set friction collar to zero or use of the Digital Readout discussed in the next section. No more than a .030 depth cut should be taken. Face to Scribed Line.

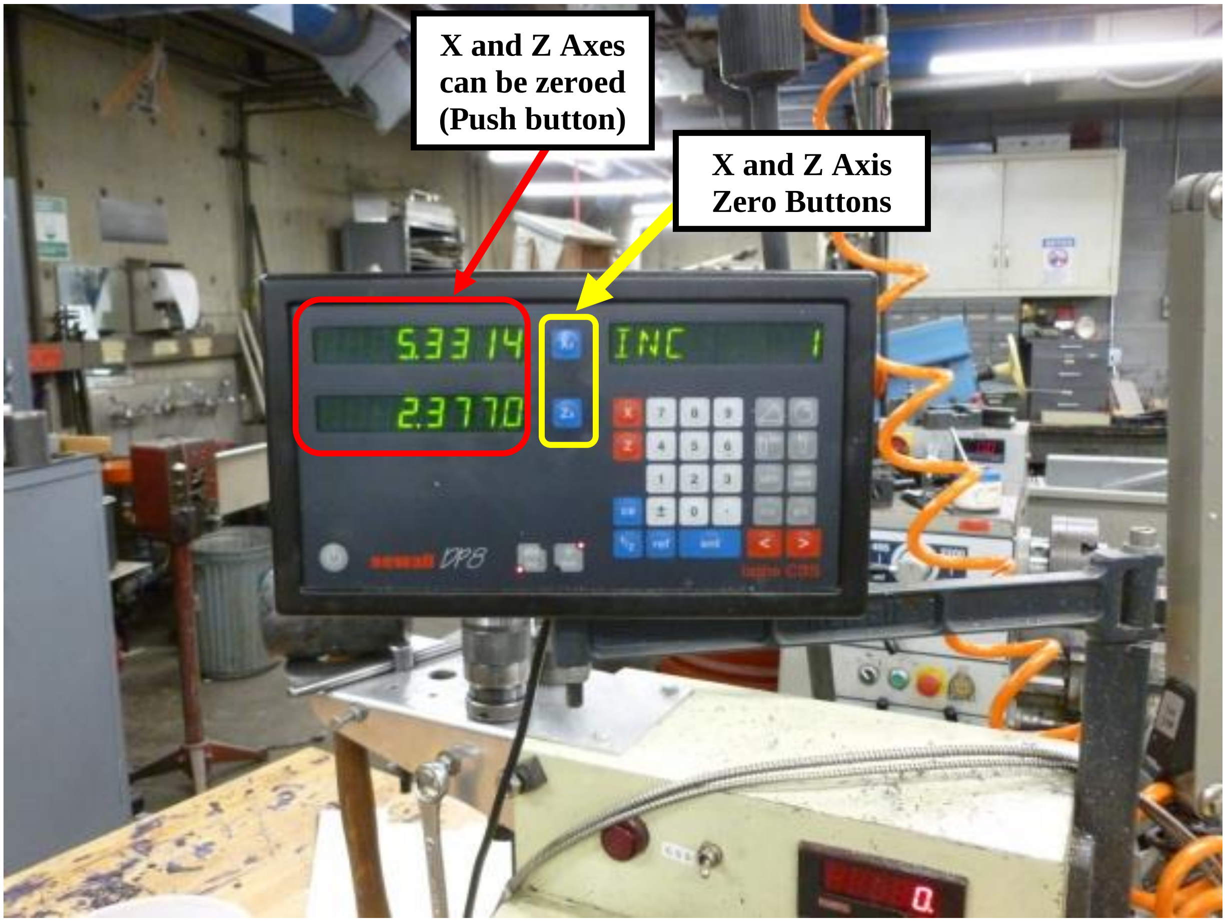
- The spindle is the centerline that all features are referenced to so do not move cutter past the X Axis centerline. If you move past the centerline it will be up cutting on the tool bit possibly breaking the tip. It also will not cut. (Cutting edge of tool bit is on top and the work piece rotates forward cutting toward the cutting edge).

Work piece rotating forward



2.3.0 Digital Readout (DRO):

The Digital Readout can be easily reset at any location during a process and is also an easy accurate method to track the position of the X Axis of the Cross Slide and Z Axis of the Carriage (NOT the Compound Rest). The X Axis is normally operating off the diameter of the work piece referring to the physical movement of the Cross Slide being ½ the graduation. For example .002 is moving the Cross Slide .001 removing .002 off the diameter of the work piece as it referenced to the centerline of the spindle. The dial readouts also measure the change in the diameter of the material mounted in the lathe.



2.4.0 <u>Turning Outer Diameters:</u> On work pieces extending in length more than 2.5 times the diameter from the face of the collet or chuck use of a Live Center is recommended. For example stock diameter is 1 inch, extending it beyond 3 inches requires the use of a Live Center (see Section 1.2.7 for Center Drilling).

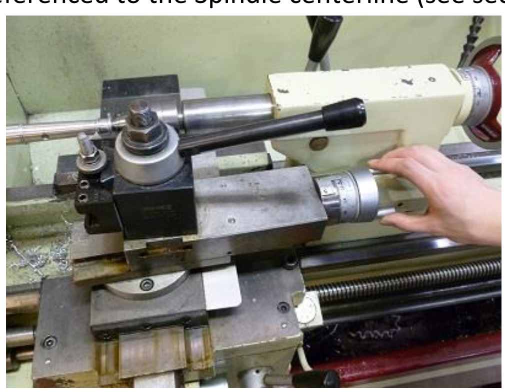
 

- After Facing work piece and center drilling if needed if using a Live Center for support,
- Spindle and Power off,
- Install work piece holding method (5C Collet shown see Section 2.1.4 or 2.1.2 for Chuck Mounting),
- Mount work piece properly in holding device (see sections 2.1.2 to 2.1.4),
- Select proper cutting tool and mount in Tool Holder, install Tool Holder on Tool Post and align tool with Live Center point (sections 2.2.5 and 2.2.8). Position Live Center to support part if needed,

Proper work piece supporting is accomplished by moving Tailstock with Live Center installed close to Faced, Center Drilled work piece face. Allow room for Cutting Tool to move afore face (may need to extend Tailstock Quill), Lock Tailstock Locking Handle, rotate Tailstock Hand Wheel (Section 2.2.6) until Live Center is "SNUG" in Center Drilled hole or Fully Drilled Hole.(DO NOT want to bend work piece).
Lock Quill Lock Handle (section 2.2.6).

- Select proper RPM (300-400 RPM for this class)
- Using the Longitudinal Hand Wheel (Z Axis), move the Carriage Assembly to the required position measuring either with the Hand Wheel graduations or use the Digital Readout (Spindle on, lightly touch tool to face of work piece using Carriage and Cross Slide Hand Wheels, set Z Zero, move tool away from part (X Axis Hand Wheel), do the same with diameter and set X Zero, move tool away from part), move to location(s), feed cutting tool in, no more than .030 per cut (X Axis, Cross Slide Hand Wheel controls depth of cut and **"graduations are diameter increments"**). Continue using Hand wheels to obtain required geometry shapes and dimensions.
- To cut angles requires the use of the Compound rest. The Hand Wheel has "direct linear graduations.

The angle increments in below photos are referenced to the Spindle centerline (see section 2.2.4 for adjusting).

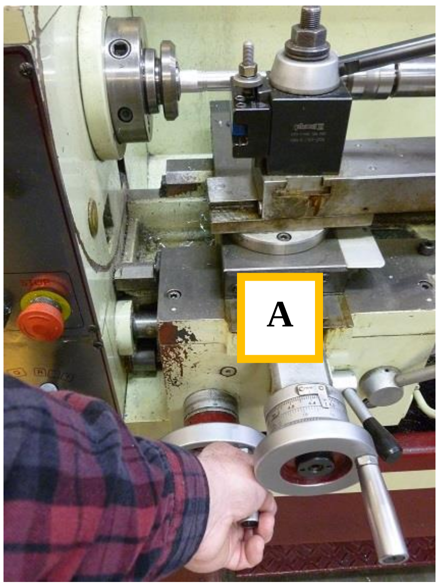
 

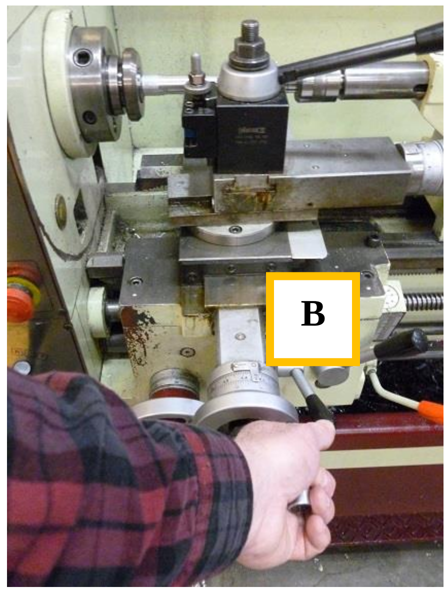
2.4.1 Cutting Angles using the Compound Rest:

- Set angle by loosening socket head cap screws. The angle increments are referenced off the working plane which is either the face of the work piece or the centerline of the spindle.

- The Carriage and Cross Slide will also be utilized to move tool bit to required location. The Cross Slide, in this instance shown, will be used for depth of cut (no more than .030) and the angle is set 2 degrees to the spindle centerline.

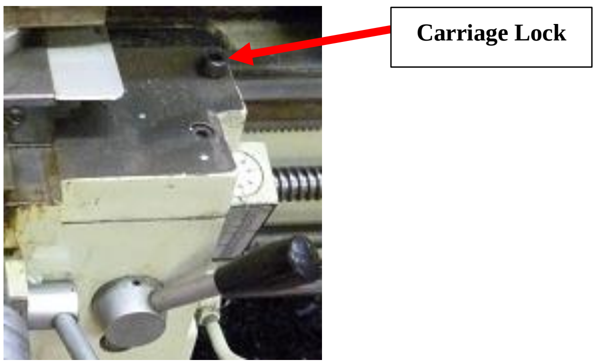
Cross Slide used to set depth of cut



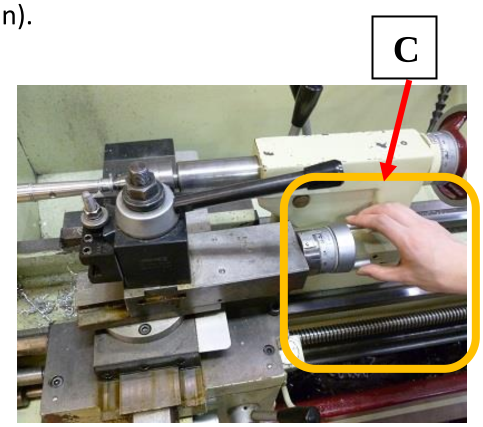


- Prior to moving tool bit to location move Compound Rest (C) towards tailstock as far as it will move in this situation by rotating the Hand Wheel counter clockwise until it stops. Rotate Hand Wheel clock wise until “0” (zero) aligns with indicator. This zeros the Compound Rest while removing Backlash or the wear or play between the thread and the nut.

- Move to location to starting position of cut, Lock Carriage in place.



- Using the Hand Wheel on the Compound Rest (C) will feed the tool bit along the required angle and the Cross Slide Hand Wheel will control the depth of cut **(graduations are diameter)**.
- Turn spindle on Forward 300-400 RPM, using Cross Slide Hand Wheel lightly touch tool to part, Zero collar or DRO, go to depth of cut, Rotate Hand Wheel slowly for a nice finish. At the end of the angled cut, back Cross Slide (X Axis) towards operator away from work piece, turn off machine, return Compound Rest to starting position rotating Hand Wheel counter clock wise. Measure work piece. Continue until dimensions are reached (remember last dial graduation).



2.4.2 <u>Parting/Grooving/Form Tool Cutting:</u> This is primarily the same process as turning for moving cutting tool to location and for depth of cut.

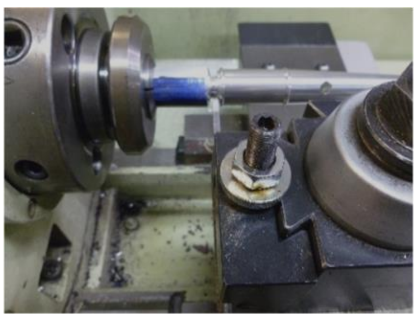
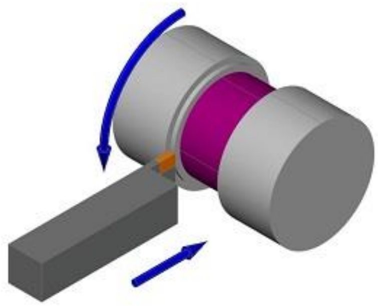
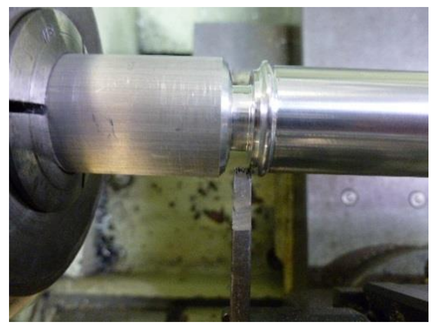
- Machine power off,
- Select tool,

- Align tool on Center,
- Use Live Center if needed,
- Move Cutting Tool to location,
- Slowly feed in X Axis Hand Wheel to required depth while generously applying proper cutting fluid.

Groove or Step: Create a notch, step or groove in the work piece (see section 2.2.5.1 for tool)

- Machine power off,
- Select tool,
- Align tool on Center,
- Use Live Center if needed,
- Move Cutting Tool to location,
- Slowly feed in X Axis Hand Wheel to required depth while generously applying proper cutting fluid.

(If wider groove is needed, back out X Axis, move Z Axis with Hand Wheel to required dimension possibly using multiple increments repeating as needed).

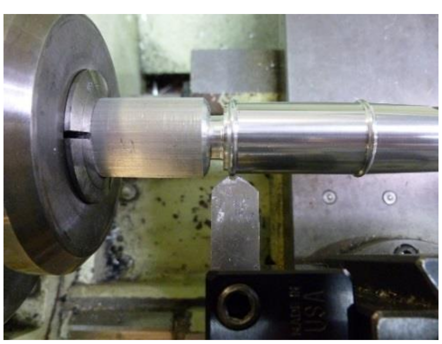
Parting: Cutting off part to a precise finished location using a Parting or Grooving Tool (see section 2.2.5.1 for tool)

- Back Live Center away from work piece if one is used and move Tailstock away also,
- Same as Grooving except work piece will be cut off and drop into bed of machine,

(DO NOT REACH FOR PIECE AS IT FALLS OFF).

- Turn off machine,
- Now you can retrieve the part.

Form Tool Cutting: (Ridge Cutting Tool in this instance) The same process used for Groove cutting.




3.0 Making the Cannon Barrel:

3.1 Cross Drilling the Barrel: Immediately after layout.

Layout

Clean the barrel stock with a rag to remove oil and dirt. Coat one side thinly with Dykem. Now, using a square, scribe a reference line approximately 1/16" – 1/8" from one end of the 7 ½" barrel stock. This line is the location of the surface of the muzzle end, and is the reference line from which you will mark layout measurements. Consult the drawing (page 52) to locate and mark the position of the transverse hole. Next center punch that mark. Now, locate the breech end of the barrel, located 5" from the reference line. Mark this location with one of the pointed feet of your dividers.

Install the "F" drill (.257" diameter) into the chuck of the drill press. The next step is to install the barrel blank into the drill jig. The drill jig (with barrel blank) is then clamped into the vice of the drill press. Position the tip of the drill tip directly over the proper drill bushing (labeled cannon) in the drill jig. Have your partner hold apply cutting fluid while you drill. Complete the hole by peck drilling.

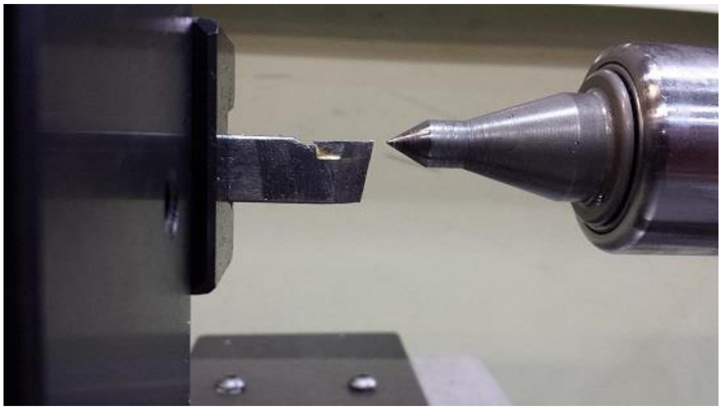
3.2 Facing Off and Axial Drilling

1. Aluminum in this shop is turned at 300-400 RPM. Set the controls on the lathe to obtain this turning speed.

2. If it is not already in place install the Collet Closer and Collet Adapter (section 2.1.4) on the lathe. Consult the instructor before attempting this. Put the collet into the collet chuck, making sure that the key and keyway are aligned.

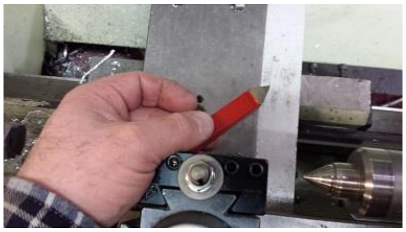
If using a 3 Jaw Chuck, MAKE SURE THE CAM LOCKS ARE FULLY TIGHTENED CLOCKWISE
(Section 2.1.2)

3. Install the round nose tool bit. Be certain that the tool bit's cutting tip is centered (section 2.2.8) In this case, position the tool holder on the side of the quick change tool post that is parallel with the long axis of the quill .



4. Once the tool bit is aligned, take the tool bit out of the tool holder by loosening the Allen screws and turn it around so that the tip of the tool is facing the collet chuck. If you do not align the tool bit correctly, you will not be able to face off your work piece completely to its center (a dimple of metal will be left at the center), and you may damage the tool bit. Also eject the live center from

the quill by retracting the quill of the tailstock. This will draw the end of the tapered shank of the live center into contact with an ejector tab which does the ejecting.



5. Install the stock into the collet (be sure no chips are in the collet) so that about 1 inch of the muzzle end (the short end) is sticking out of the collet. Install the collet into the collet chuck (be sure to align the keyway of the collet) and tighten the chuck.

6. Prior to using the auto feed, make sure you test it away from your work piece by turning on the lathe, engaging the auto feed, and checking the direction that the tool bit is moving. Face off the muzzle end of the work piece to the scribed length.

7. Extend that tailstock Quill past the 1” mark, and install the Jacob’s chuck in the quill. Be sure to seat the Jacob’s chuck with the rubber mallet. Install the center drill in the Jacob’s chuck.

8. Center drill the muzzle end up to the maximum diameter of the ½” center drill.

9. Drill to the specified depth with the 3/8” drill (see Figure 26) by using the scale on the tailstock quill as a guide. Use the depth gauge on the vernier calipers to check the precise depth of the hole.

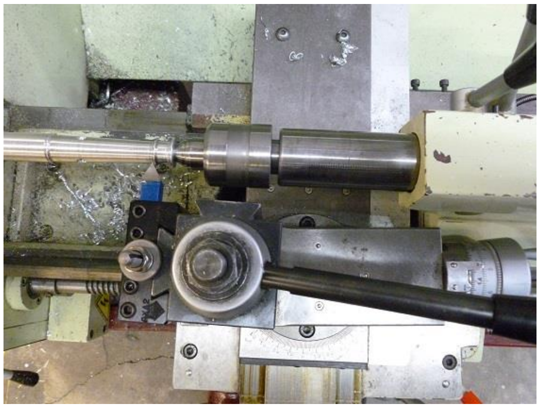
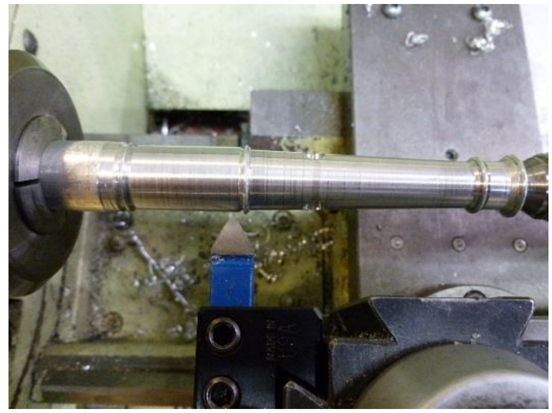
10. Now do your partner’s barrel.

11. **At this point choose the barrel to run on the CNC lathe. No additional manual steps are required until the CNC phase is complete. If you do more manual steps, so you may have to complete the barrel manually.**

3.3 Turning Outer Diameters

1. Loosen the collet chuck and extend the stock so that roughly 5 ½” of material is protruding from the collet, retighten the collet chuck. This is to ensure proper tool bit clearance with the collet chuck. Install the Live Center in the Tailstock Quill.

2. Remove the tool bit from the tool holder then place the tool holder on the tool post as shown. Orient the round nose tool bit on the tool post so that the tool bit is perpendicular to the bar stock’s axis of rotation. The center height alignment should not change.

3. Making sure that all locks on the tailstock are disengaged, extend the Tailstock Quill and the Live Center so that about 4 inches is showing on the quill. Slowly (and carefully) slide the tailstock to the point where it almost contacts the muzzle chamfer of the barrel. Slowly tighten the live center against the barrel using the Tailstock Hand Wheel. The guideline for tightening the live center is that when the headstock is in neutral, the live center and barrel should rotate together when the spindle is rotated. CAUTION: Do not over tighten the live center against the work piece, and do not extend the quill past the 4.5” mark. CONSULT THE INSTRUCTOR BEFORE TURNING ON THE LATHE.

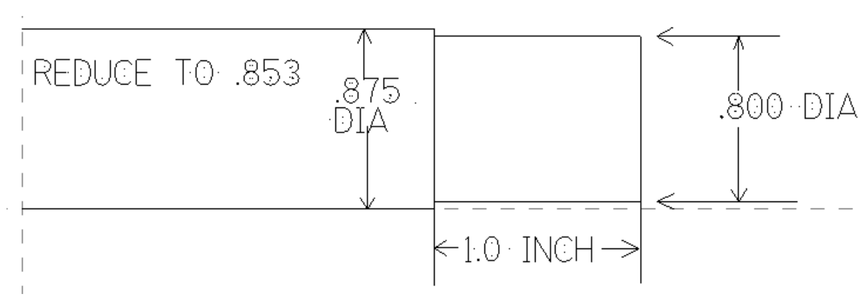
Note: Use cutting fluid for the following operations. The order of the operations given below is recommended. NEVER start the machine with the tool bit contacting the material.

4. The diameter of the entire barrel must be reduced to 0.853”. To do this, turn the barrel stock down to 0.853”, starting 5 1/8” from the muzzle end of the barrel. You will need to do this in several passes. Do not cut more than 0.020” off on any one pass. Consult the instructor for the recommended cutting speeds.

5. Turn the muzzle diameter down to 0.800” for 1 inch from the muzzle end. (Figure 30).

6. Cut to 0.680 diameter inches a section ¼” wide, beginning 3/8” from the muzzle end, and ending at 5/8” from the muzzle end. Note that the sides of the cut will be angled because of the angled sides of the tool bit. (See Figure 31 next page).

Figure 30: ***Turning the Diameter Down to 0.800” One Inch from the Muzzle End.***



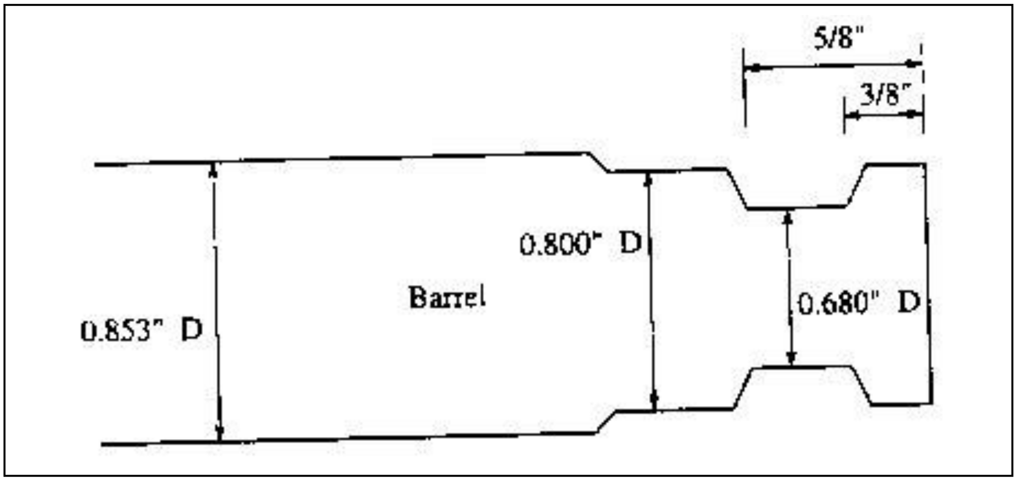
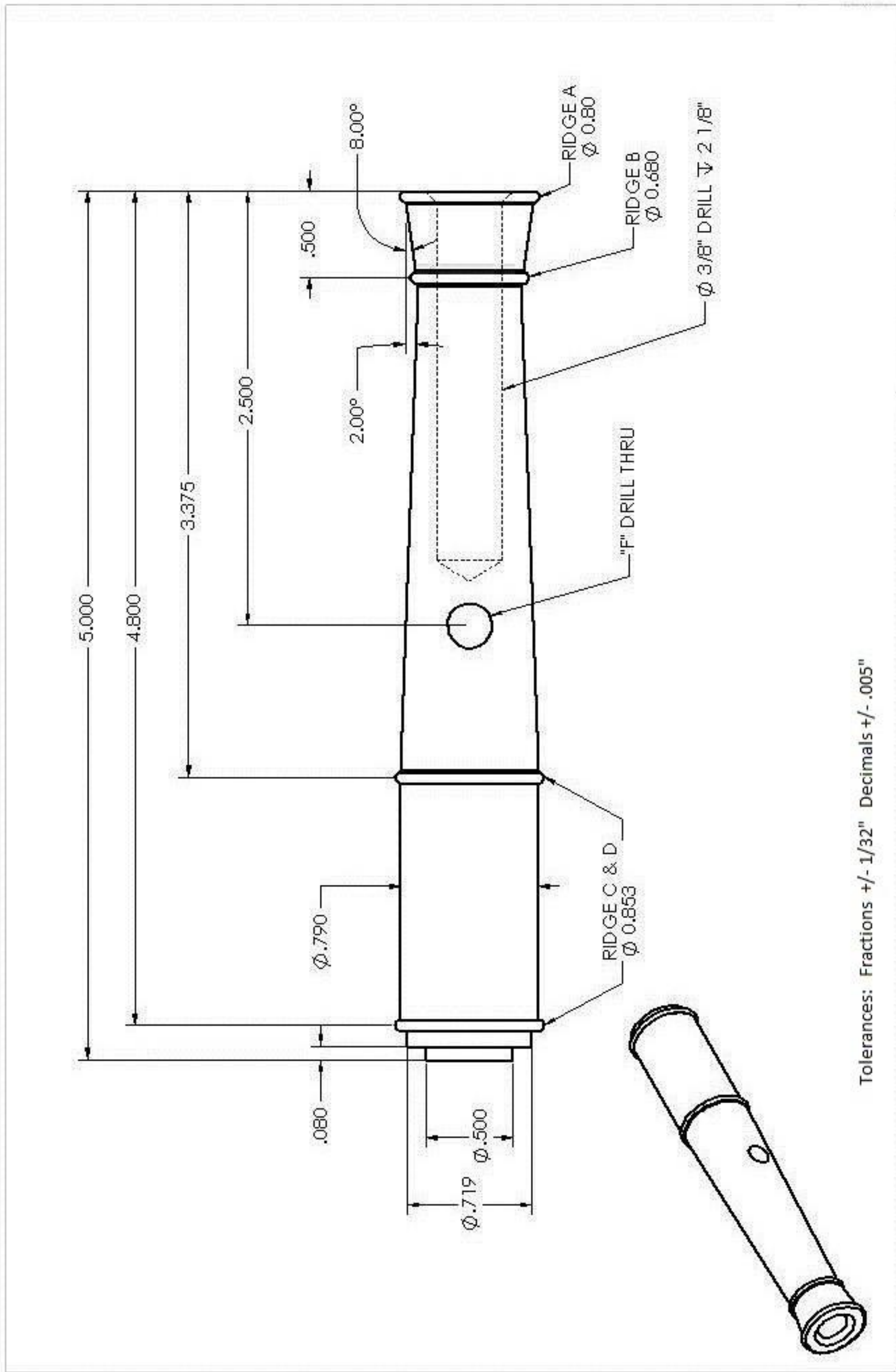


Figure 31: ***Cutting a Section to 0.680 Inches in Diameter.***

Note on tolerances- All tolerances are as follows for any dimensions given in decimal form plus/minus .005”. The tolerance for all dimensions given in fractional form plus minus 1/32”. This allows us to specify two different tolerance classes by the manner of expression of the dimension.

This is true for all drawings in this manual.

Note that one person in each lab group will be doing most of their fabrication on a CNC lathe. For this barrel or boilerstack blank, you should only faceoff, center drill, and axial drill the bar stock as well as drilling any transverse holes that may be required.



Material

7 ½" x 7/8" diameter aluminum bar stock

3.4 Cutting the Ridges

Note: The diameters of the ridges to be cut are the same as the outer diameter of the stock at these points. The diameter of the bar stock will determine the ridge diameter at this area. At the instant when the back of the ridge cutter touches the bar stock, the ridge should: 1) be round, 2) have the diameter of the bar stock at that point.

Figure 32: ***Top View of Ridge Cutter.***

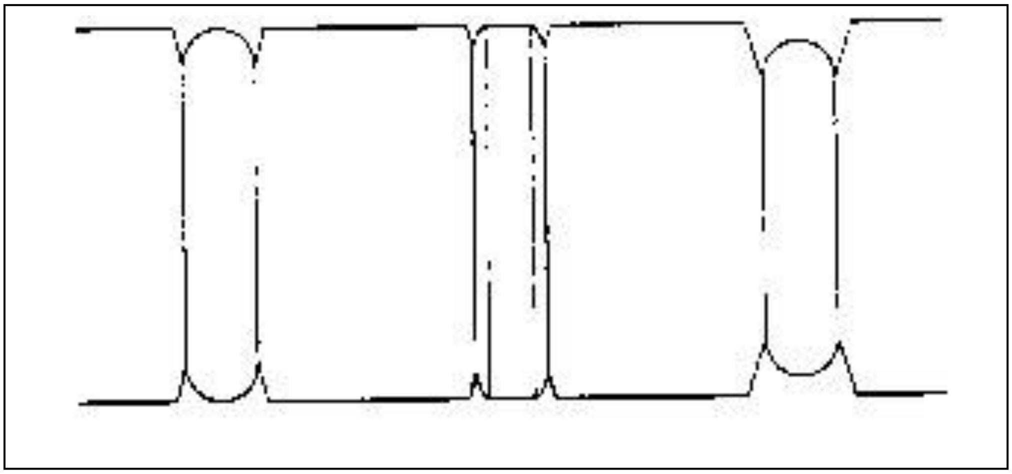


Figure 33: ***Good and Bad Ridges: X) A properly made ridge, Y) Tool bit not pushed in far enough, Z) Tool bit pushed in too far.***

1. At this point, install the form tool bit (ridge cutter – see Figure 32) into the tool holder and center the tool bit. Install the tool holder on the tool post so that the tool bit is perpendicular to the axis of rotation.

2. Cut Ridges C and D to 0.853” in diameter.

3. Cut Ridge B to 0.680” in diameter.

4. Cut Ridge A to 0.800” in diameter.

3.5 Angle Cuts and Additional Turning

1. Return the round nose tool bit to the tool holder and center it.

2. Turn down the barrel diameter between ridges C and D to 0.790”. Consult the drawing (Figure 26). If you use the auto feed on this section, pay close attention, and be sure to disengage the auto feed when the tool bit comes close to the ridge. If you do not, you will cut the ridge off.

3. Set up the lathe to make the 2deg. and 8deg. tapers on the barrel. To do this:

    (a) Loosen the two nuts (or Allen screws) in front of and behind the compound rest with the ½” wrench or suitable Allen wrench. See Figure 35.

    (b) Adjust the angle of the compound rest to 8deg. Make sure that the edge of the compound rest is parallel to the surface of the taper, which you plan to cut. Note that each tick mark on the rest is one degree.

    (c) Re-tighten the two nuts on both sides of the compound rest.

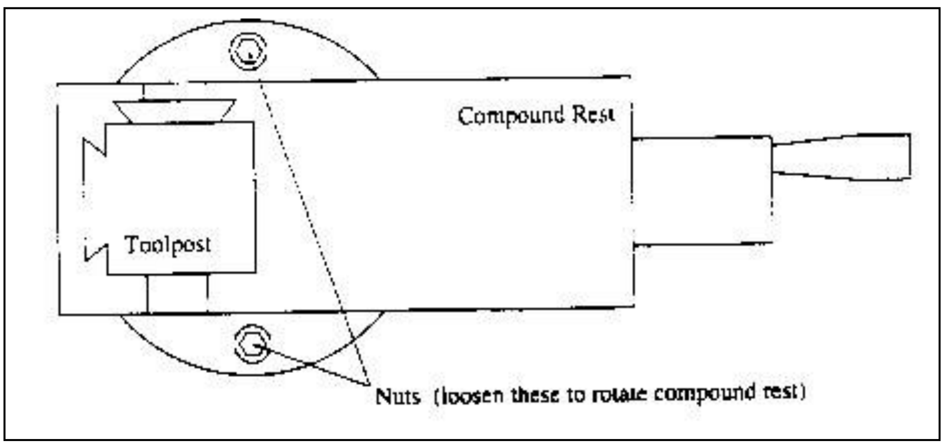


Figure 34: ***Rotating the Compound Rest.***

NOTE: Using the small hand wheel on the compound rest itself can only make the angle cuts. All other hand wheels will make perpendicular cuts. DO NOT USE AUTOFEED; auto feed controls the larger hand wheels, not the one attached to the compound feed.

4. Cut the 8^{o} taper from the base of ridge B to A, using the cross slide dial to determine the depth of cut, and the top slide dial on the compound rest to make a pass.

5. Readjust the angle on the compound rest, and cut the 2^{o} taper from the base of ridge C to B.

6. Return the compound rest to 0^{o}.

3.6 Polishing the Barrel

In order to remove any tool marks made by the round nose tool bit you must use emery paper to polish the surface of the barrel. It is easiest to polish the barrel while it is mounted on the lathe. This section is optional and the final surface finish is up to you.

For best results, tear a narrow (1/2 inch) strip of emery paper, and wrap it around the barrel. Hold one end of the emery paper in each hand, and pull it toward you while the barrel is turning. Move the paper slowly side to side to sand the entire surface.

1. Start with coarse grit (120) emery paper, and polish the surface until the scratches in the barrel are uniform in appearance.

2. Using medium grit (240) emery paper repeat the process until the surface scratches again look uniform.

3. Repeat the process with a fine grit (500) emery paper.

4. After the sanding is complete, you may want to buff your work piece. This final step will only help shine the surface. It will not remove major scratch marks from the surface. Consult the instructor before using this machine.

3.7 Finishing breach end of the Barrel and Making the Wheels

1. Once the Barrel is complete, we recommend installing the parting tool (available from the instructor or lab assistants)and using it to form the two discs on the breach end of the barrel by plunging the parting tool (slowly) straight into the aluminum. Before using the same tool to cut off the now finished barrel from the remaining stub you will be using to construct the wheels. . For the final cut it is recommended that you use manual control and a lot of cutting fluil.

2. Install the stub of the remaining 7/8" diameter aluminum stock into the collet so that about ½" of material is inside the collet. Tighten the collet.

3. Center drill the end of the stub, and drill all the way through it with the #7 twist drill (0.201"). This forms the hole required by the retaining screws, which will attach the wheels to the cannon base. Again, refer to section 4.3.

4. Sand the stock with rough emery paper lightly to remove any Dykem or dirt.

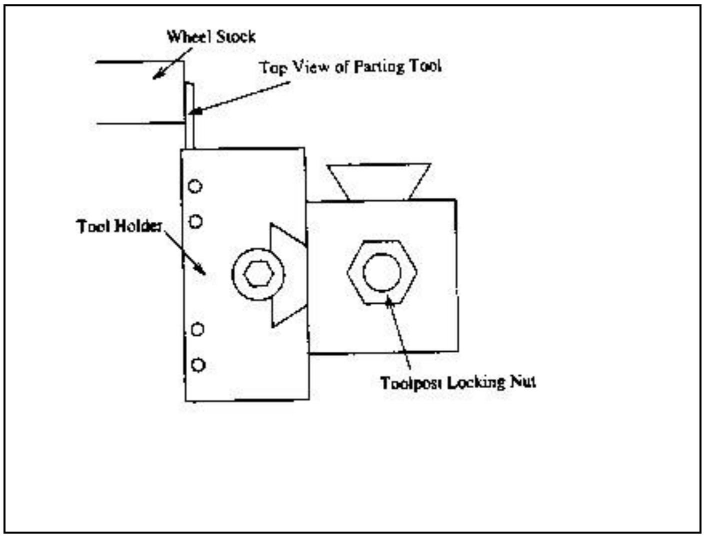


Figure 35: ***Aligning the Parting Tool So It Is Perpendicular.***

5. **This is an optional step.** If you want wheels with "treads", obtain the knurling tool from your tool kit. This tool will put diamond-shaped patterns called knurls on the surface of the aluminum. The instructor will show you how to perform this operation.

6. Obtain a wheel thickness gauge from an instructor. Be certain that the parting tool is at precisely 90deg to the longitudinal axis of material; to do this, place the side of the parting tool against the faced off end of your work piece (see Figure 36).

If there is a gap at the front edge where the parting tool touches the wheel stock, the tool post must be straightened. Loosen the nut at the top of the tool post and rotate the tool post until the parting tool touches the wheel stock all along it edge. Re-tighten the nut.

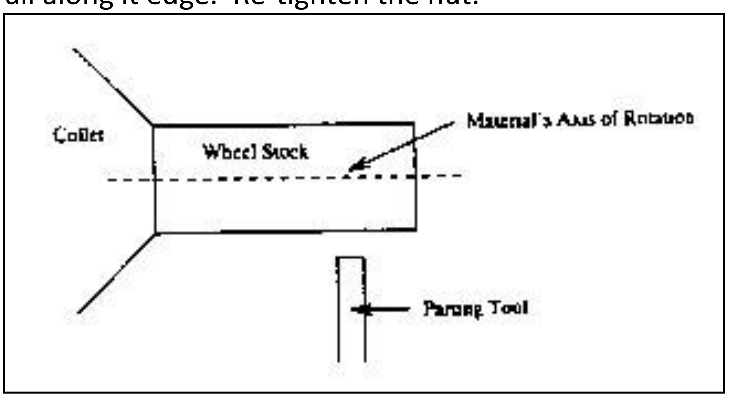


Figure 36: ***Use The Parting Tool To Cut The Wheels.***

7. Using the parting tool and wheel gauge (measure thickness of wheel with the gage it will be 7/32") and slice off four wheels from the stub (Figure 36). Be sure to center this tool as well. Do NOT use auto feed for this process. Be sure to bathe the parting tool and the aluminum piece with cutting fluid continuously as you cut into the aluminum. It is not necessary to feed the parting tool past the center of the aluminum bar stock.

3.8 Making the Trunnion: The trunnion is the pin that holds the barrel in the base.

1. Obtain a piece of ¼" diameter stock from an instructor.

2. Install the 3-jaw chuck on the lathe (see the section on changing the lathe chuck). Wipe off the jaws with a rag to remove any chips. Install the trunnion pin in the 3-jaw chuck.

3. Face off one end of the pin.

4. Scribe a line 1.5" from the faced off end.

5. Face off the other end of the pin to that line.

6. Test fit the pin in the base. The ends of the pin should both be even with the sides of the base when the pin is inserted into the hole. If the pin is too long, face off one end slightly, until the ends of the pin are flush with the sides of the base.

7. Using a knurling tool, knurl a ¼" wide strip on ONE end of the pin. This effectively expands the diameter of the pin. When this expanded end is forced into the base hole, it will create a press fit. CONSULT THE INSTRUCTOR ON HOW TO KNURL.

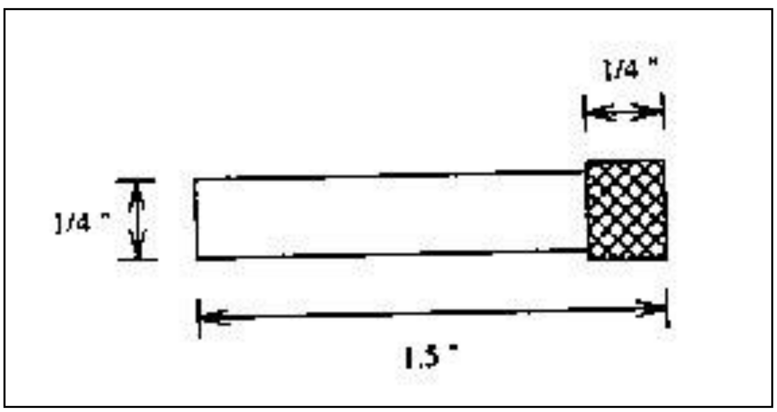


Figure 40: ***Trunnion Pin for Cannon.***

3.9 Instructions for construction of light saber grip.

The material for the light saber will come in two pieces. A cylindrical pieces which will form the grip and a rectangular one which I am arbitrarily calling the power supply.

We will deal with the grip first.

The material is saw cur and we will first install it into a collet in one of our lathes. Initially leave an inch or so sticking out of the collet and secure the collet tightly. Using the round nose tool bit face off the grip blank so that it has a smooth face.

Next install the drill chuck into the quill of the lathe tailstock. The materials being described are covered in the introductory material on the lathe much earlier in the manual.

Place a center drill into the drill chuck and tighten it in place. Now move the tailstock (with an inch or two of the quill sticking out) to a point where there is half an inch or so of distance between the tip of the drill and the face of the lightsaber grip. Lock the tailstock in place. Next drill to the point where you have made a small indent into the surface of the face of the material of the light saber grip. Retract the quill and install a #15 drill (.18" in diameter). Do not assume that the last person left the correct drill in the kit. Measure everything.

Drill to a depth of 1" into the face of the lightsaber grip.

Time to change a few settings. Begin by loosening the collet and pull the grip material out until you have between 5.5 and 6 inches of material projecting out of the collet. Tighten the collet. We will remove the drill chuck from the tailstock and replace it with a live center.

Slide the tailstock (which you have extended the quill to have about 4 inches of the quill outside of the tailstock body) so that the tip of the live center is inside of the hole you have drilled into the face of the grip. Lock the tailstock in place on the ways of the lathe and tighten the live center tip firmly into the center hole you have drilled into the face of the grip.

Put the round nosed tool bit (most of which are actually rather pointy) into the tool holder. Place the tool holder onto the tool post so that the tool is positioned at 90 degrees to the long axis of the grip. Turn on the lathe and reduce a length of 1.125 inches of the grip to a diameter of .5 inches.

Now retract the tool and traverse .625 inches left. Then using the same tool reduce a length of material (measuring.375 inches of traverse) to the same diameter of .5 inches. Retract the tool and translate .220 inches to the left. We will now begin fabricating what looks almost like cooling fins. Plunge the tool in to a change of diameter of .20 inches (slowly please). Retract the tool traverse .220 inches to the left and repeat the depth of cut. Do this 5 more times and the "cooling fins" are done. There is one more operation at this set up.

Time to change tools. Install a parting tool onto the tool post so that its long axis is at 90 degrees to the long axis of the grip material. Align the right edge of the parting tool to be coplanar to the exposed face of the grip. Translate the tool 5.125 inches to the left. Now plunge to a change of diameter of .10 inches.

Time to change things up a bit. Loosen the lock on the tailstock and slide it to the right out of the way. While you are at it remove the live center and reinstall the drill chuck.

Next loosen the collet and remove the grip blank then reinstall it so that close to 3inches of the unmachined end is sticking out of the collet and tighten the collet again. Since we still have the parting tool installed we will use it next.

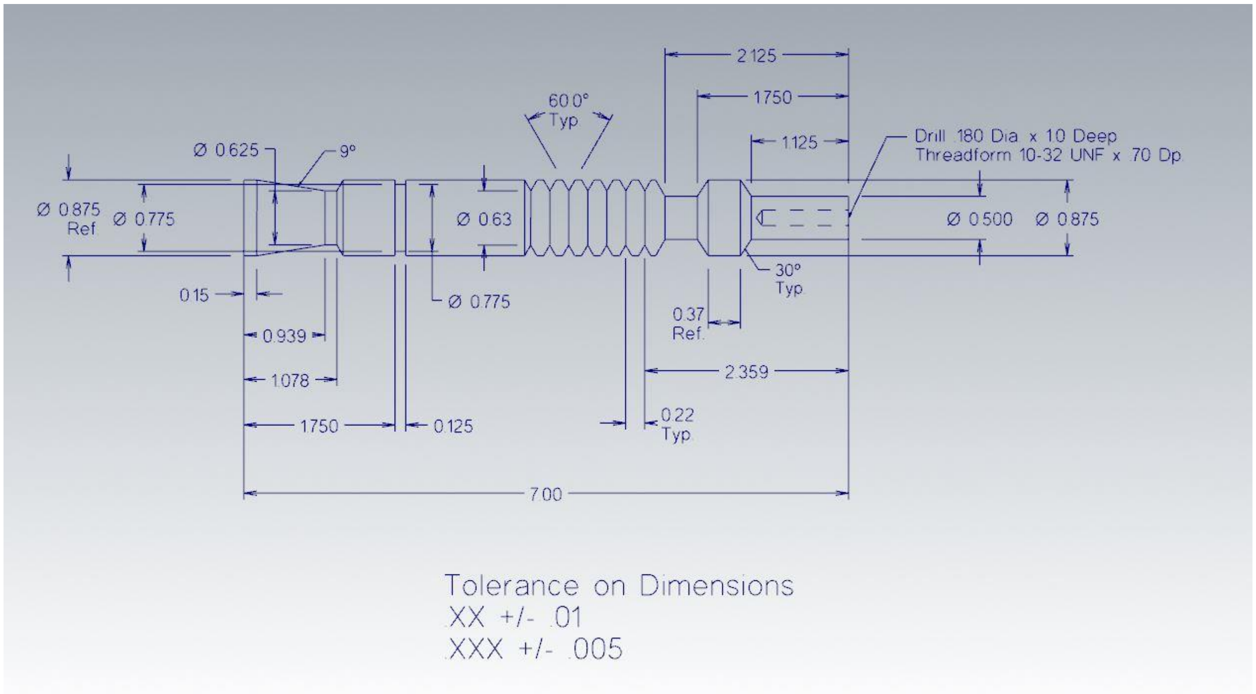
Align the parting tool with the last (only) cut you made with it so far. Move the tool .811 inches to the right and slowly plunge it so you have changed the diameter by .25 inches (leaving a diameter of .625"). Retract the tool and translate to a point where you have a distance of 1.175 inches from your initial alignment (or an additional .939 inches from the position of the cut you just finished). Part the whole remainder off. A chunk a bit less than .5 inches long should fall off.

We will now cur an angle between a starting point .152 inches from your new firing end of the saber to the parting cut closest to the firing end. Set your angle to 9 degrees and using the ROUND NOSED (ie not the parting tool) tool bit make the cut.

Now install the center drill into the drill chuck. Drill until you hit the full .5 inch diameter of the center drill.

One last operation for the lightsaber grip. Loosen the collet and reverse the grip so that 1.25 inches of the small diameter of the grip is sticking out. We will not thread the inside of the hole for a 10-32 screw which we will use to secure the "grip" to the "power supply". Ideally, this should be a press fit but we have a backup just in case someone make an error. We will show you the threading procedure. Just ask when you get to this stage.

See next page for drawing



Credits to:
Scott Yerbury
http://web.mit.edu/2.670/www/Tutorials/Machining/lathe/Description.html
littlemachineshop.com
www.weiku.com

4.0 Vertical Milling Machine – General Introduction
4.1Milling Machine Controls

4.1.1 Main Features
Figure 18 shows the main sections of a typical vertical mill used in our machine shop. Be sure to note the location of the Forward, Reverse, and Stop buttons.

Stop: This red button is located high on the mill, on the left. When you need to stop the machine, push it straight in.

Forward: This green button turns the machine on in the forward direction. When the high-speed range is selected, this is usually the button that will cause the end mill to turn in the appropriate direction.

Reverse: This black button turns the machine on in the reverse direction. When the low speed range is selected, this is usually the button that will cause the end mill to turn in the appropriate direction.

Collet Rack: This rack holds a variety of collets; these collets are designed to work with the vertical miller and are used to hold various end mills in the quill of the machine.

Quill: This tube holds a collet and spins when the machine is turned on, turning the collet (and end mill) with it.

Drawbar: This looks like a hexagonal bolt head, at the top of the miller. It is really is long bolt that extends down into the machine and has threads on the lower end. These threads engage the threads of the collet and pull the collet tight into the quill.

4.1.2 Speed Controls
Figure 19 shows the items and controls that affect the speed of rotation of the quill.
The following features are of note:

Motor: This is located at the very top of the vertical mill and is activated by the Forward or Reverse buttons. Note: if the motor is powered up but is prevented from turning, it will be damaged. If you ever turn the miller on and the motor does not spin, **IMMEDIATELY** turn the machine off and find an instructor.

Brake: This lever clamps onto the quill and prevents it from turning. This is useful when tightening the drawbar; by applying the brake, you can tighten the drawbar to the required torque.

Speed Range Selection Lever: This lever is located on the right side of the machine and is best viewed from there. It is a 3-position switch (the middle position is neutral), and it selects between high and low gear. NEVER change this lever when the motor is running.

Note: When the miller is running in high gear, the quill will rotate in the opposite direction to when it is running in low gear. For this reason, both a Forward and a Reverse switch have been provided. Use the switch, which will cause the end mill to rotate in the proper direction.

Make sure the mill is in gear before turning it on.

Speed Selector Hand Wheel: This is a dial which adjusts a continuously variable speed changer inside the machine. This allows the user to set the quill speed to any desired value within the selected gear range. This control must ONLY be moved when the machine is <u>running.</u>

Quill Speed Indicator: This dial indicates the rotation speed of the quill, in revolutions per minute. The dial will rotate as the speed selector hand wheel is turned, constantly displaying the spindle speed to the operator. There are two sets of numbers on the dial; the smaller (in value) numbers on the outside of the dial indicate the speeds for the low gear range. The higher numbers, closer to the middle of the dial, indicate the speeds for the high gear range.

Note that some of the millers have a digital speed indicator instead of the disc. This only shows the machine’s RPM when the miller is running.

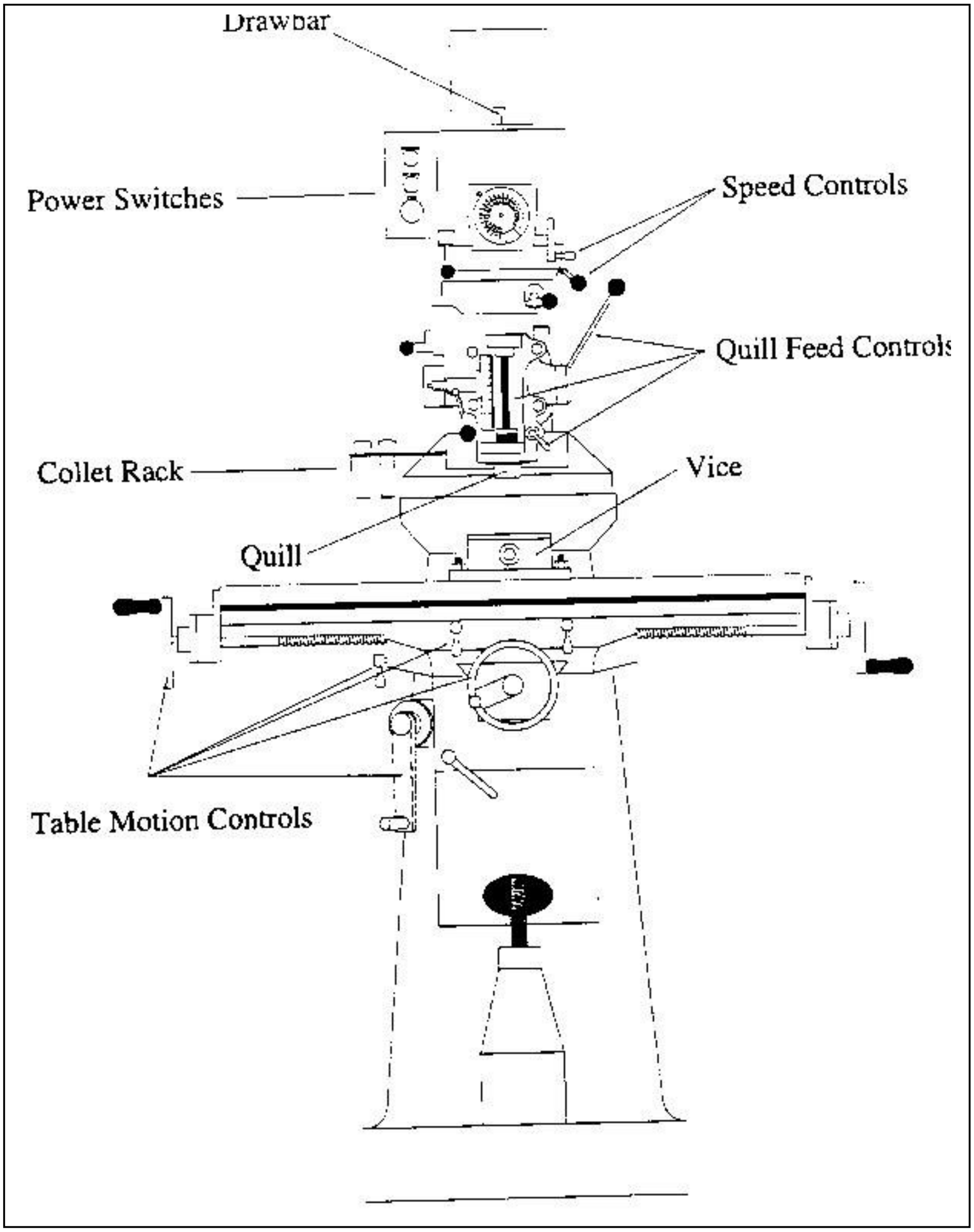


Figure 17: ***Overview of Vertical Milling Machine***

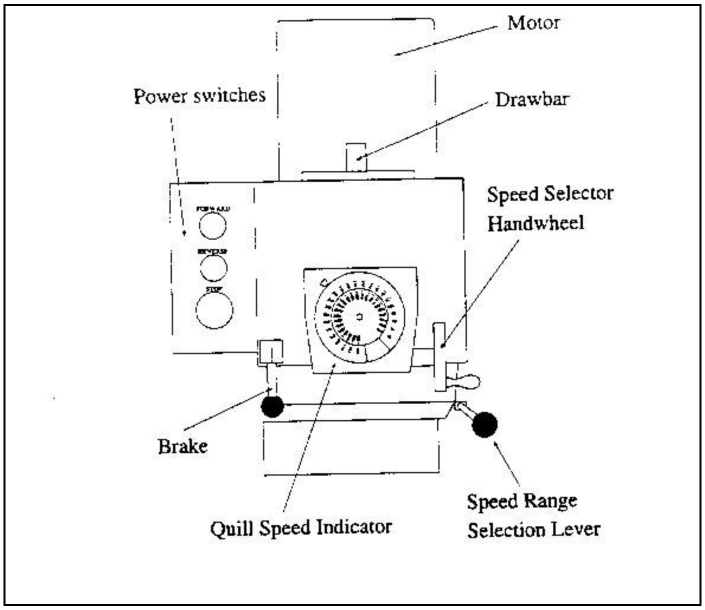


Figure 18: ***Speed Controls***

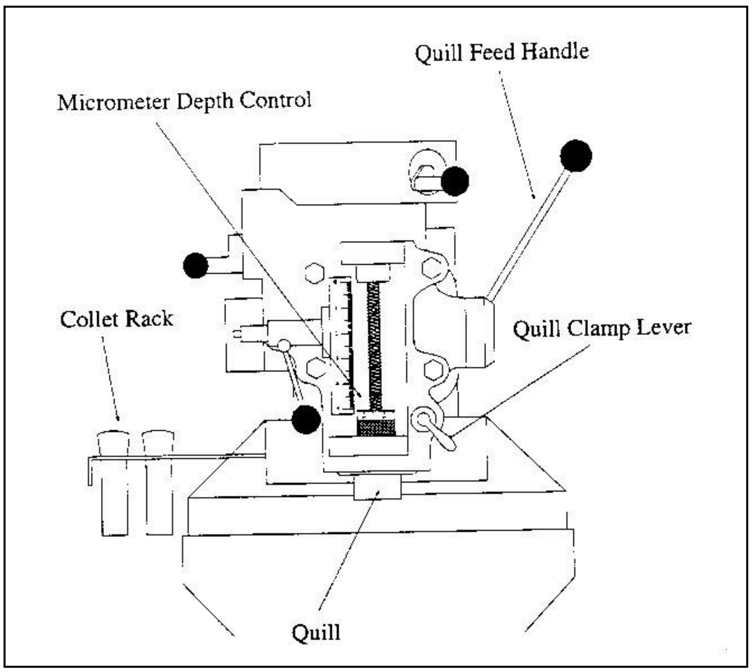


Figure 19: ***Quill Feed Controls***

4.1.3 Quill Feed Controls

The quill of the vertical miller can be raised or lowered to aid in performing some machine operations, such as drilling. Figure 19 shows the main controls, which affect this quill movement.

Quill Feed Handle: This lever moves the quill of the machine up and down, just as the lever on a drill press moves the drill chuck up and down. This handle permits the user to use the milling machine like a drill press.

When the quill feed handle is not all the way up, the drawbar may not be accessible at the top of the machine. Should you need to access the drawbar, first raise the quill all the way up.

Quill Clamp Lever: This lever is a brake for the vertical motion of the quill. This must be loosened before the quill feed handle can be used. After using the quill feed handle, the quill clamp lever should be tightened (clockwise) to prevent the quill from shifting.

Micrometer Depth Control: This consists of a threaded machine screw with a precision depth stop, which rides along the screw. When the quill feed handle is pulled, a metal bracket slides over the screw until it contacts the depth stop. The height of the depth stop can be adjusted to 0.001 inch. This permits the user to mill or drill holes to a specific depth.

4.1.4 Table Motion Controls

The miller table (see Figure 20) also has controls, which move it in a precise fashion. The following controls are important for you to be familiar with.

Vise: This is generally used to hold the work piece to be machined. It is a smooth-jawed vise, to help prevent marring the work piece. Also, the jaw face can be moved to other parts of the vise to increase its holding capacity. In some cases, for very large or bulky stock, it may be necessary to remove the vise, and to clamp the stock directly to the table.

Longitudinal Feed Hand Wheel: This hand wheel moves the table side-to-side. There are actually two of these hand wheels, one on each side of the table.

Cross Feed Hand Wheel: This hand wheel moves the table in or out from the base of the machine

Elevation Feed Handle: This handle raises or lowers the table. Turning the handle clockwise raises the table.

Table Lock Lever: These levers are brakes on the side-to-side motion of the table. These should be loosened before using the longitudinal feed hand wheels.

Saddle Lock Handle: This lever is a brake on the in-and-out motion of the table. It should be loosened before using the cross feed hand wheel.

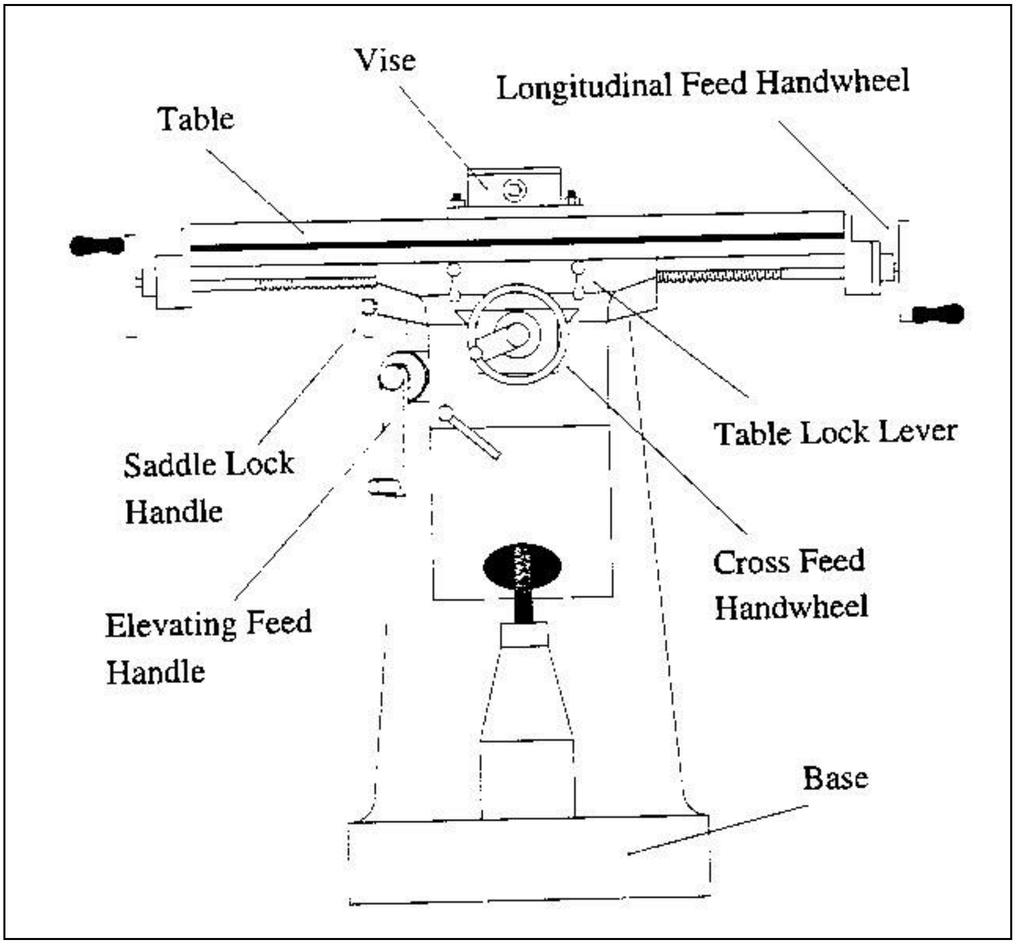


Figure 20: ***Table Motion Controls***

4.1.5 Mill Setup

Before machining any material on the vertical miller, it is necessary to set up the machine properly for the material that you are going to turn. Setting up the machine involves three main tasks:

- ✓ Installing the end mill.
- ✓ Clamping the work piece to the table.
- ✓ Setting the controls of the machine properly.

4.1.6 Installing the End Mill

1. Insert the end mill into an appropriately sized collet. The shank of the end mill should fit snugly into the collet.
2. Insert the collet into the quill of the vertical miller. Make sure that the key in the quill lines up with the keyway on the outside of the collet.
3. Tighten the drawbar on top of the machine by hand.
4. When you cannot further tighten the drawbar by hand apply the hand brake, and tighten the drawbar with a wrench. Do not apply more than 15 pounds of force to the wrench: you will over-tighten the drawbar. Also do NOT leave the wrench on the drawbar; remove it and put it away.

4.1.7 Clamping the Work Piece to the Table

Make sure the vice is free of chips and that any burrs have been filed off of the work piece to be machined. Chips and burrs will prevent the vise jaws from contacting the full surface of the work piece. This may result in the piece slipping, or the surface of the work piece being marred.

When the piece is aligned properly in the vise, tighten the vise handle firmly (clockwise). If the vise is not tight enough, the piece may slip during the milling process. Remove the vise handle and lay it on the milling table.

4.1.8 Setting the Controls

In general, the miller controls should be adjusted with the machine OFF. Any exceptions to this rule will be clearly marked on the machine. Most of the controls select gears in the miller's transmission; if you try changing gears while the machine is running, you can easily damage the miller.

If you find that a particular control resists being put into a certain position, DO NOT FORCE IT. Usually, this happens because the gears are not lined up properly. Manually rotate the quill a short distance and try again; this will move the gears enough to permit the control to be moved.

1. With the machine OFF, move the spindle speed-range selector into the appropriate speed range for the machining operation you want to perform.
2. Make sure the end mill is not contacting anything. Turn the machine on and make sure the end mill is spinning in the proper direction. Turn the speed-selector hand wheel (or potentiometer) in the appropriate direction until the end mill speed is at the desired value (approximately 1800 RPM).

4.1.9 Mill Cleanup

When you are finished for the day, you must clean up the mill. The following things should be done:

1. Remove your work piece from the miller.
2. Remove the end mill from the machine (see section 2.3.1).
3. Put the appropriate tools back into the milling toolkit check your toolkit back in.
4. Brush the chips from the boards covering the miller table into the chip recycling bin.
5. Using compressed air, blow off the accumulated chips from the top of the miller.
6. Reverse the elevating feed handle, and remove the vise handle and lay it on the milling table.
7. Sweep up the floor around the miller, and recycle the chips (blue drum for metal chips).

4.1.9.1 Removing the End Mill

The procedure for removing the end mill from the vertical miller is not quite the reverse of the installation process. This is due to the fact that when the drawbar is tightened the collet becomes wedged up into the quill.

1. Make sure that the machine is turned OFF and that the cutter is not moving.
2. Apply the hand brake, and use the wrench to loosen the drawbar. When the drawbar is loose enough to turn by hand, turn it TWO full additional turns only. This leaves most of the threads of the drawbar still engaged with the threads of the collet.
3. Hold onto the end mill, and using a lead mallet, strike the drawbar. This will dislodge the end mill. The impact force of the lead mallet will be distributed among all of the threads which are still engaged. The threads will not be damaged by the impact as the stress of the blow is distributed among nearly all of the threads. Please catch the end mill in your hand. It is quite brittle and may fracture if dropped.
4. Remove the end mill.
5. Continue to loosen the drawbar until the collet comes out of the quill.
6. Please do not drop the collet. It is very hard and somewhat brittle; it may crack if dropped.

Figure 21: ***Two-Flute End Mill***

Figure 22: ***Tap***

Figure 23: ***Tap Wrenches***

5.0 Tapping

5.1 Fundamentals

In order to fasten a machine screw into a hole, it is necessary to cut out matching threads on the inside of the hole to accommodate the threads of the screw. Without threads, the screw will not go into the hole.

Tapping is the processes of cutting the threads in a hole. The process is done with a tool called a tap (see Figure 23), which is held in a tap wrench (Figure 24). Figure 25 shows the basic machining steps needed in order to make a threaded hole for a screw.

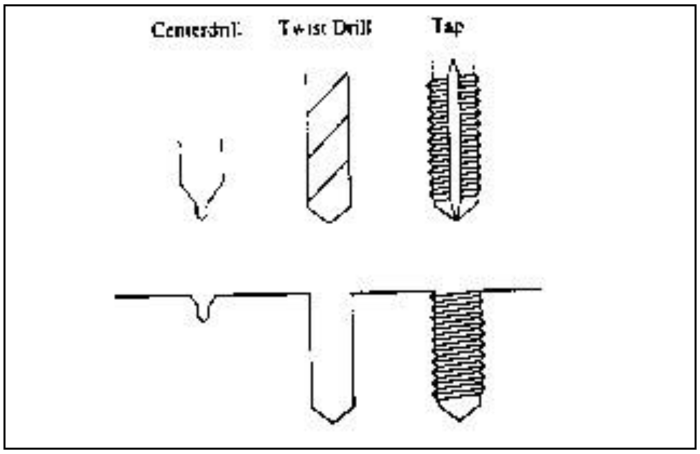


Figure 24: ***Steps in the Tapping Process: 1) Center-Drilling, 2) Drilling to Depth, 3) Tapping***

While most drills are specified only by their diameter, taps (and screws) must be specified by both their diameter and the thread pitch (the distance from a point on one thread to a corresponding point on the next thread).

For example, you will be threading a 10-32 UNF hole for the end of the boiler stack and for the six wheels. The number "10" in the "10-32" indicates the screw's diameter, and the "32" indicates that there are 32 threads per inch along the thread's length. "UNF" stands for "United National Fine".

At one time, every company made whatever arbitrary diameter and screw pitch it wanted. For an increasingly industrialized nation, this situation was unacceptable. A convention was held, and standard sizes were adopted. In most diameters, two pitches were adopted, "fine" and "coarse". The pitches were called "United" (all manufacturers agreed to abide by the convention) "National" (the agreement was nationwide) "Fine" or "Coarse". A chart of commonly used screw sizes is included for general information (**Table 1 on page 75**).

5.2 Tapping Procedure

In general, taps are made of very hard steel. They are also quite brittle, so it is easy to break them. For this reason, we will be using a thread forming tool. This tool does not cut the material but rather deforms it to produce the thread. The thread tools are less brittle than taps and thus harder to break. In addition almost no chips result making the procedure cleaner. We will frequently still refer to this step as tapping. Consult the instructor if you have a problem.

5.2.1 Cannon Base

1. Using a file, remove any burrs from the holes to be threaded.
2. Install a form tool into the tap wrench.
3. Place the base in a vise, making sure to use the aluminum or plastic guards to protect your base.
4. Brush some cutting fluid onto the form tool, and brush plenty of fluid into the hole.
5. Insert the tool through the hole in the tapping guide, and place the tip of the tool in the hole to be threaded.
6. Slide the tap guide down so that it rests on the aluminum block. Hold the tap guide flat against the block; this will ensure that the tool is perpendicular to the surface of the block.
7. Turn the tap wrench clockwise to begin forming threads. Do not press down on the tool; the tool should guide itself into the hole.

8. Once the tool is firmly embedded (5-10 threads have been formed), the tapping guide is no longer necessary. You may remove the tapping guide by unscrewing the tool, removing the guide, and reinserting the tool into the hole.
9. Continue threading the hole until there are about 4-5 threads on the form tool left showing above the surface of the block.
10. After threading the first hole, repeat the process for the other hole. Then, turn your base over and thread the holes from the other side. The form tool lacks sufficient length to go from one side to another

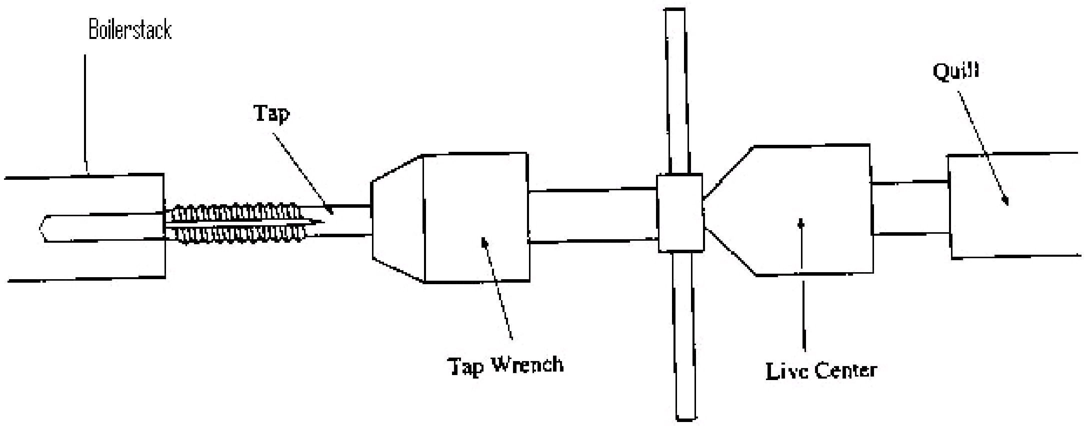


Figure 25: ***Threading the Hole in the Smoke Stack end of the Boiler stack***

5.2.2 Train Cab

1. Using a file or emery paper, remove all burs from the holes to be tapped.
2. Install a form tool into the tap wrench.
3. Place the cab on a flat surface.
4. Brush cutting fluid onto tool tap and into the hole.
5. Insert the tool through the hole in the tap guide and place the tip of the form tool into the hole to be threaded.
6. Slide the tap guide down until it rests against the aluminum block. Hold the tap guide flat against the block; this will ensure the tap is perpendicular to the surface of the block.
7. Turn the tap wrench clockwise to begin cutting threads. DO NOT press down on the tool; the tool should guide itself into the hole.
8. After 5-10 threads have been cut, the tap guide is no longer necessary. You may remove the tap guide by unscrewing the tool, removing the guide, and reinserting the tool into the hole.
9. Continue threading the hole until there are about 4-5 threads on the tool left showing above the surface. Should threading become difficult before this, back the tool out of the hole then re-lubricate the tool and the inside the hole. Reinsert the tool and continue.
10. After threading the first hole, repeat for the other holes. Then, turn your base over and thread the holes from the other side.

5.2.3 Smoke Stack & Boiler

1. Remove any burrs from the holes to be threaded. Lightly face off the end of the smoke stack if necessary.
2. Install a thread forming tool into the tap wrench. Grip the tool only by the square part on the end of the tool.
3. Install the live center into the quill of the tailstock, and extend the quill until the 2” mark is showing on the quill.
4. Brush some cutting fluid onto the tool and brush plenty of cutting fluid into the hole.

5. Place the tip of the forming tool in the hole to be threaded and slide the tailstock toward the headstock until the tip of the live center rests in the hole in the back of the tap wrench. This will ensure that the form tool is perpendicular to the end of the stock.
6. Turn the tap wrench clockwise to begin forming threads. While you are doing this, turn the tailstock hand wheel to keep the live center snug (but not tight) against the tap wrench.
7. Once the forming tool is firmly embedded (5-10 threads have been formed), the live center is no longer necessary. You may slide the tailstock away from the work piece to get it out of the way.
8. Continue threading the hole until there are about 4-5 threads on the form tool left showing outside the hole.
9. Repeat this process for the Boiler end.

Screw Size		Nonomal Body Diameter	Clearance Drill			Tap Drill		
			Dec.	Frac.	#	Dec.	Frac.	#
0-80	UNF	0.059	0.067		51	0.0469	3/64	
1-64	UNC	0.073	0.081		46	0.0595		53
1-72	UNF	0.073	0.081		46	0.0595		53
2-56	UNC	0.086	0.096		41	0.07		50
2-64	UNF	0.086	0.096		41	0.07		50
3-48	UNC	0.099	0.106		36	0.078		47
3-56	UNF	0.099	0.106		36	0.082		45
4-40	UNC	0.112	0.12		31	0.089		43
4-48	UNF	0.112	0.12		31	0.0935		42
5-40	UNC	0.125	0.136		29	0.101		38
5-44	UNF	0.125	0.136		29	0.104		37
6-32	UNC	0.138	0.147		26	0.1065		36
6-40	UNF	0.138	0.147		26	0.113		33
8-32	UNC	0.164	0.173		17	0.136		29
8-36	UNF	0.164	0.173		17	0.136		29
10-24	UNC	0.19	0.204		6	0.1495		25
10-32	UNF	0.19	0.204		6	0.159		21
12-24	UNC	0.216	0.228		1	0.177		16
12-28	UNF	0.216	0.228		1	0.182		14
1/4 - 20	UNC	0.25	0.261		G	0.201		7
1/4 - 28	UNF	0.25	0.261		G	0.213		3
5/16 - 18	UNC	0.3125	0.328			0.257		F
5/16 - 24	UNF	0.3125	0.328	21/64		0.272		I
3/8 - 16	UNC	0.375	0.386	21/64	W	0.3125	5/16	
3/8 - 24	UNF	0.375	0.386		W	0.332		Q
7/16 - 14	UNC	0.437	0.453	29/64		0.368		U
7/16 - 20	UNF	0.437	0.453	29/64		0.3906	25/64	
1/2 - 13	UNC	0.5	0.515	33/65		0.4219	27/64	
1/2 - 20	UNF	0.5	0.515	33/64		0.4531	29/64	

Table 1: ***A Chart of Commonly Used Taps and Screw Sizes.***

6.0 Milling the Cannon Base

Purpose
To manufacture the base of a model naval cannon and to become familiar with the safe operation of the horizontal and vertical milling machines

Materials

1 ½" x 1 ½" x 2 7/8" aluminum block

Tools and Equipment

Layout: Dykem, rag, scribe, combination square, center punch, hammer, 6-inch ruler, hermaphrodite caliper, divider.

Drilling: Drill press, center punch, center drill, "F" twist drill (0.257" diameter), #15 drill, drill press vice, cutting fluid.

Tapping: Thread forming tool, tap wrench, tapping guide, cutting fluid, file.

Horizontal Milling: Horizontal milling machine, combination square, clamp and mounting setup, spacer blocks, cutting fluid, file.

Band Sawing: Band saw marked "Aluminum Only", wooden push block, and cutting fluid.

Vertical Milling: Vertical mill with vise, spacer block, end mill, collet, micrometer, file, scribe edge gage.

Safety
- **Never start the vertical mill with the end mill touching the work piece.**
- **Keep your hands AWAY from the work piece until the end mill has come to a complete stop.**

General Procedure
Caution: Milling machines should be used with the greatest caution! Respect them, and follow all safety rules. Consult an instructor before turning these machines on so that YOU do not get hurt. Do not be afraid to ask questions, or have the instructor check your work or setup.

An instructor will generally assign you an order in which to perform sections 5.2 – 5.4 in order to maximize the use of the available machinery. **Otherwise, follow the order of operations as given.**

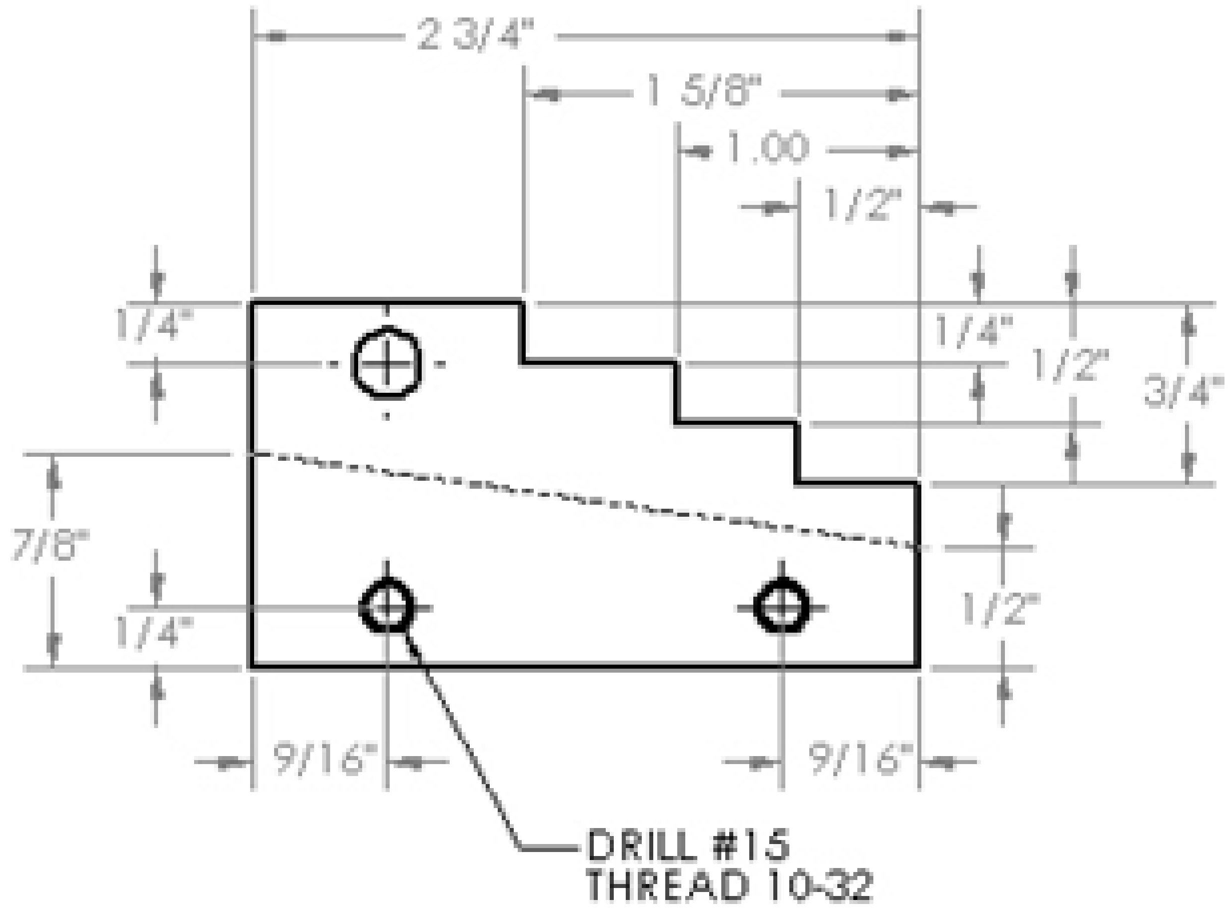
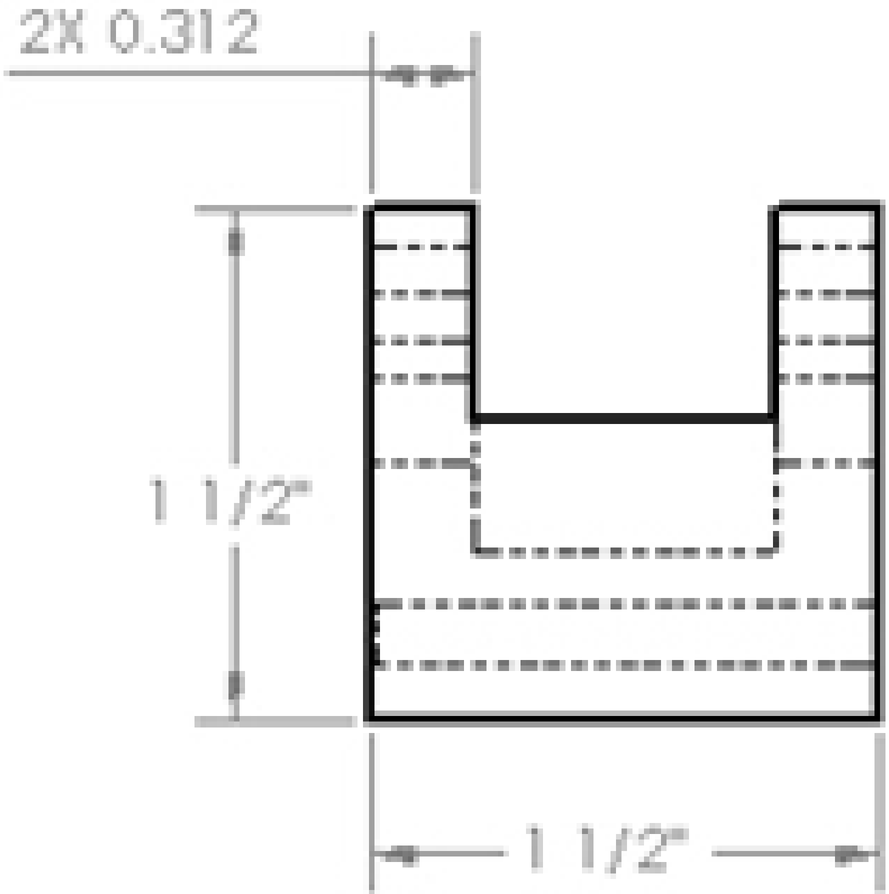


Figure 41: ***Cannon Base Side View***

Tolerance fractions +- 1/32" decimals +-.005

Figure 41b: Cannon base end view



Tolerances fractional dimensions +- 1/32"
Decimal dimensions +- .005"

6.1 Layout

1. Clean the block with a rag to remove any oil and dirt and coat two adjacent sides thinly with Dykem.
2. Using a square, mark a reference line approximately 1/16” from one end of the block. Consult the drawing (Figure 42).
3. Following the instructor’s directions, scribe on the side of the block all the lines as indicated in the side view of the drawing.
4. Using a straight edge and a scribe mark a line that runs just above the upper right corners of the “steps”.
5. Using a center punch and hammer, center punch the location of the three holes.
6. Using a hermaphrodite, scribe the lines that are 0.315” from each edge on the top of the block.

6.2 Horizontal Milling

1. Clamp your base, and your partner’s base to the table of the horizontal milling machine. Make sure that the end lines of the bases are aligned with the cutting wheels and that the blocks are square to the cutters. The instructor will explain in detail how this is done.

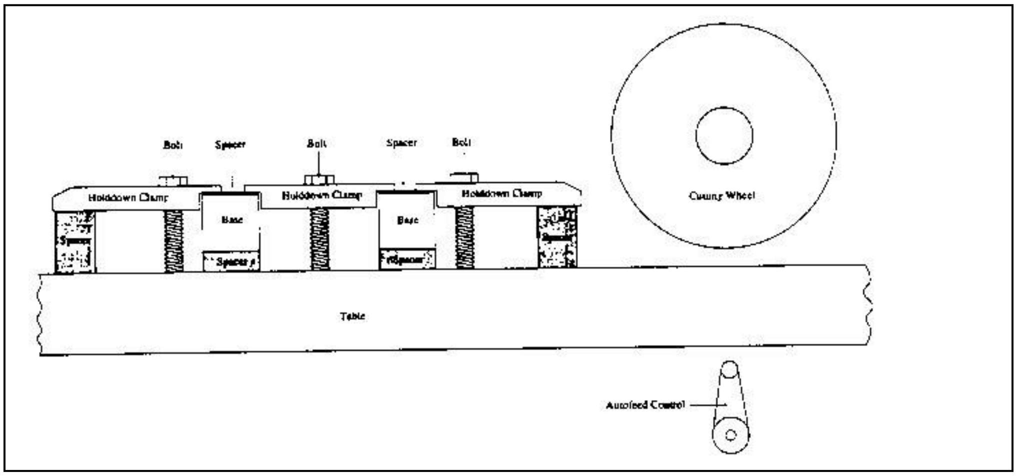


Figure 42: ***Side View of Horizontal Milling Setup.***

2. Adjust the table height so that both ends of the block will be cut at the same time, and so that the cutters will machine the entire face of the block. Make sure that the machine table will not be cut into.
3. Cut the block using conventional milling and the automatic feed. Use plenty of cutting fluid; oil the insides and the circumferences of the cutters AWAY FROM WHERE THE ALUMINUM BLOCK IS CONTACTING THEM. Keep your hands away from the cutting wheels.

6.3 Drilling Holes

When drilling holes, it is important to make sure that the twist drill is positioned so that it does not flex (i.e. bend) when it is pushed into the hole. After you center drill the position of each hole, immediately switch to the correct size twist drill and complete the through hole. This requires a lot of switching of drill bits but ensures accurate results. Re-lubricate the TIP of the drill about every ¼” of hole depth. Otherwise, the hole may be oversized, and/or not round or penetrate the block at an angle.

It is necessary to determine the actual diameter of a twist drill before you use it. Occasionally, a twist drill of the wrong size will be put into the drilling kit by mistake. To check the size of a twist drill, use a micrometer to measure the diameter. Measure across the cutting flutes, not the shank portion (it tends to get a bit abraded). Use cutting fluid when drilling.

1. When drilling, put an aluminum or plywood spacer block under the base to prevent drilling into the vice or the drill press table. (See Figure 44).
2. Center drill the barrel hole. Then stop the machine emplace an “F” twist drill and drill the hole (0.257”).
3. Drill both of the axle holes with a center drill and a #15 twist drill (0.180”) in a similar fashion. This prevents misalignment of the through hole.

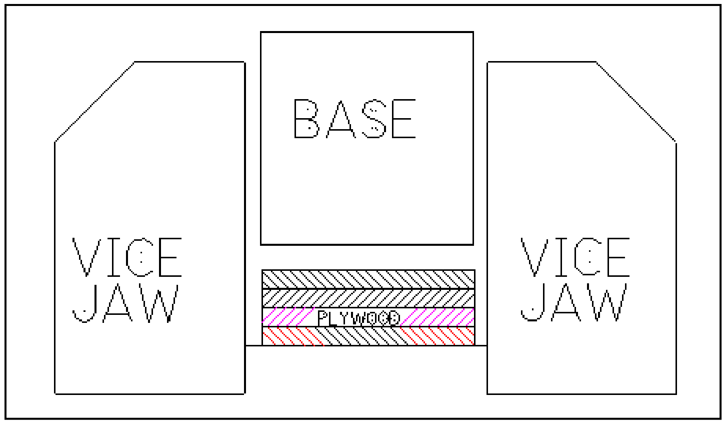


Figure 43: ***Positioning of the Base and plywood block in the Vice for Drilling.***

6.4 Tapping

Using a thread forming tool put threads in both of the holes on both sides of the cannon base. See section 5.2.1.

6.5 Band Sawing

Cut out the extra material to the upper right of the ‘steps’ with the band saw marked “Aluminum Only”. It is recommended that you cut along the diagonal line that you put on during the layout that runs above the upper right corners of the “steps”. Use a wooden push block, and cut about 1/16” – 1/8” on the scrap side of the line. This will leave excess material on your block, which you will remove later with the vertical milling machine. At this point choose which base is to be finished on a CNC machine. No additional manual steps are necessary.

6.6 Vertical Milling

1. Obtain a vertical milling kit from an instructor.
2. Install a 7/8” three-flute end mill into the milling machine. Refer to section 4.1.6
3. Make sure the vice is free of chips, and that any burrs have been filed off the blocks to be milled. Mount the blocks side by side, with a spacer block underneath. The band saw lines should be a little higher than the edges of the vise. Align the cutting plane with the edge of the blocks using the edge gage. Have an instructor check your setup before proceeding.
4. Adjust the speed of the end mill. For aluminum, the end mill should rotate at 1800 RPM.
5. Mill the “stepped” section, using cutting fluid. CAUTION: It is extremely dangerous to put your fingers or a fluid brush near a moving end mill. DO NOT DO IT! Cut all the way to the scribed lines. Use conventional milling (See Figure 47). Never cut deeper than 0.050” along the vertical axis of the miller. When using the side of the end mill to make a cut, do not cut deeper than 0.010”.
6. Mount the blocks next to each other in the vise such that the diagonal lines are even with the top of the vise. This is done so that the center slot can be milled at an angle with respect to the bottom of the base.

7. Mark the bottom of the slot on one end of the blocks. You will be milling to this line.
8. Align the 7/8" endmill with the side of the base blank using the edge gage in your tool kit. Translate the block .312" towards the center of the base blank. Lock the Y axis in place to avoid error when milling back and forth to proper depth.
9. All cuts should have a maximum depth of .05" Note: DO NOT run the machine in reverse. It will push metal around rather than cut it. The results are poor at best.

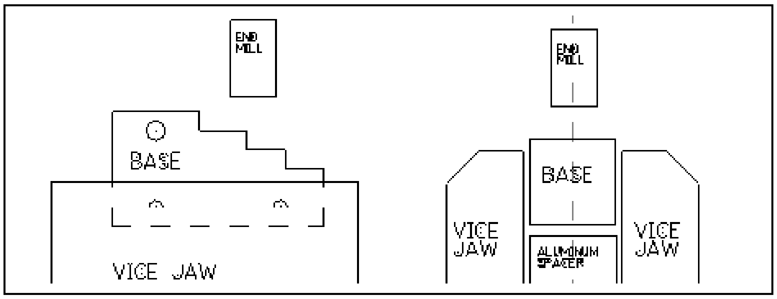


Figure 44: ***First Milling Setup for the Cannon Base.***

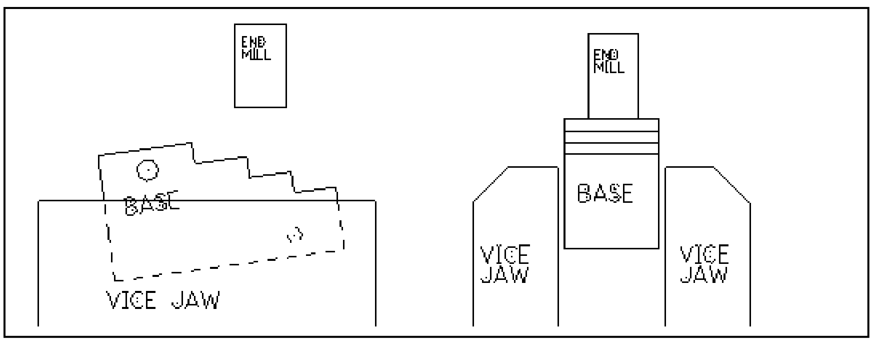


Figure 45: ***Second Milling Setup for the Cannon Base.***

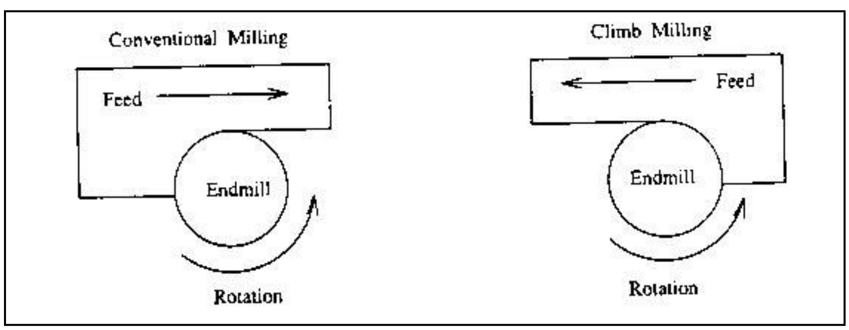


Figure 46: ***The Difference Between Conventional and Climb Milling.***

6.7 Polishing the Base

Now that the machining of the base is complete, file off any burrs that are on the edges of the base. Next, using emery paper, sand the sides of the base to remove any scratches and Dykem. Begin with coarse grit of emery paper (120) and sand until each surface looks uniform. Then proceed to finer grit of emery paper (240, 500) and repeat the process.

For the best results, use any flat surface (workbench top, table on bandsaw etc.). Set the emery paper on the surface and secure it. Hold your base and move the base against the emery paper using slow, deliberate strokes. You should sand in only one direction. The final step is to use the buffing wheels which are capable of putting a mirror finish on the finished parts (if desired). Ask instructor to show you the procedure.

6.8 Cannon Assembly

At this point, your cannon base and barrel should be polished to your satisfaction.

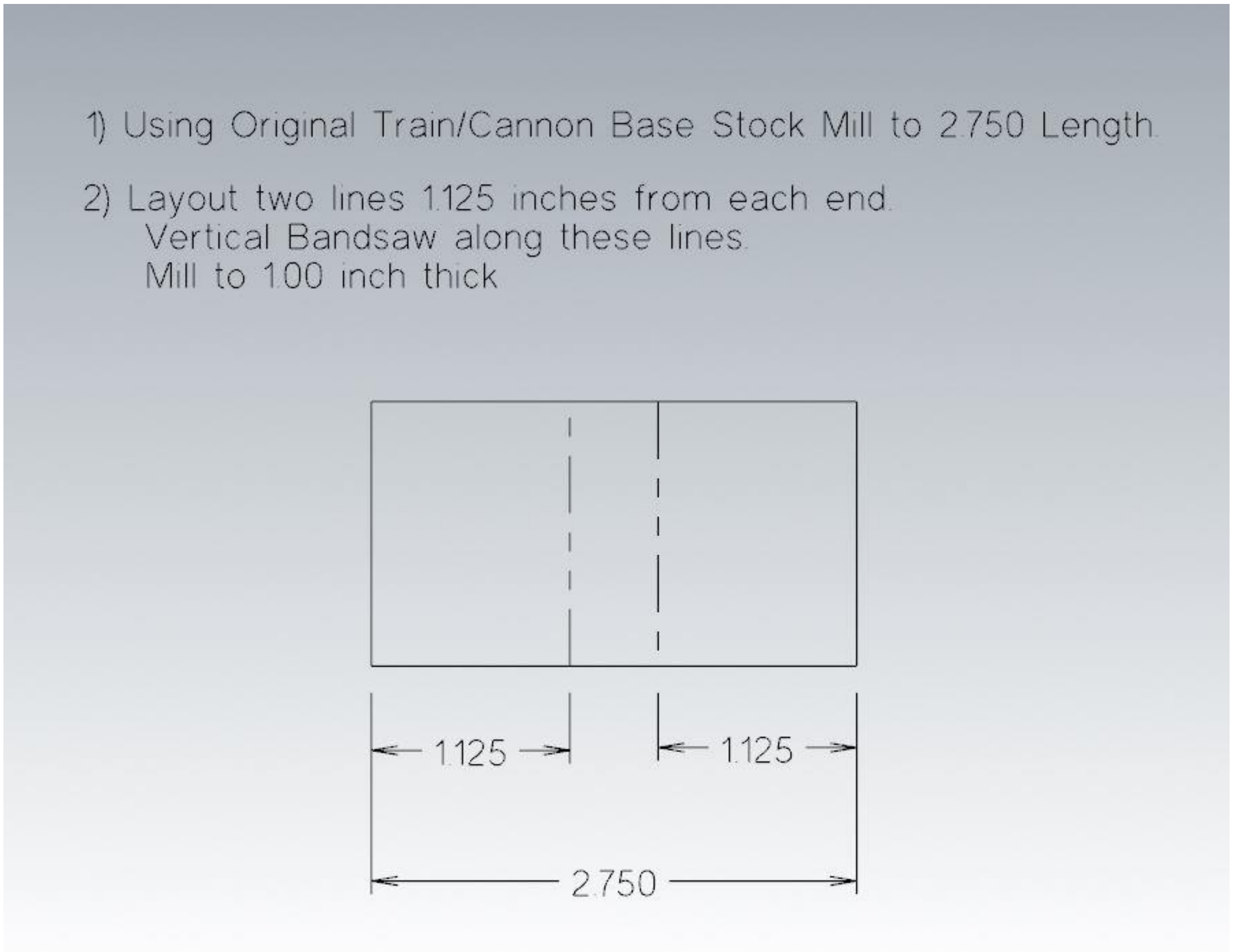
1. Using a rag, clean off any dirt from the barrel and the base.
2. Obtain the letter and number stamps from the instructor. Stamp your name (or initials, class, year, favorite saying, or anything else you like) on the bottom of your base. Use the cannon-stamping jig to support the base while you do this.
3. Put the barrel into the slot in the base, making sure the basket is facing in the correct direction, and slide the trunnion pin through the holes in the base and barrel. Make sure that the trunnion pin goes all the way through the barrel and part way into the second hole of the base.
4. Clean a milling machine vise with a rag to remove any chips. Wrap your base and barrel in a clean paper towel or rag and place it in the milling machine vice. Tighten the vise so that the trunnion pin is pushed the remainder of the way into the base.
5. Obtain 4 screws from an instructor. Using an Allen wrench, put on the wheels.
6. When your cannon is assembled, make sure that an instructor checks you off as having completed the project.

6.9 Instruction for constructing lightsaber pommel.

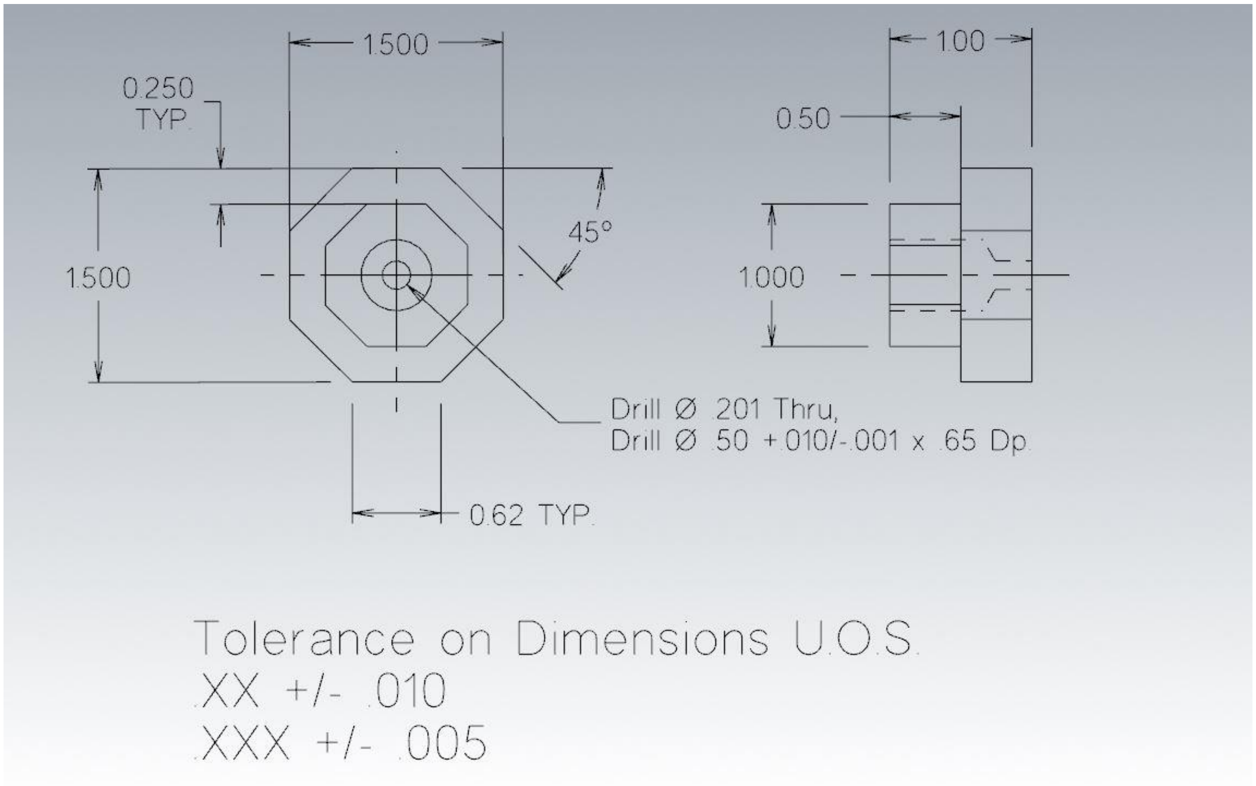
7. We will begin with a rectangular block of aluminum. First step is to run it through the horizontal miller. This will result in a block which is 1.5”x1.5”x2.75”.
8. Next step is to cut from each end pieces which are 1.5”x1.5”x approximately 1 1/16”. The block will be set on a pair of precision parallels in the vice of a vertical miller. After installing a 7/8” end mill into the spindle of the miller, the height of the block is reduced from approximately 1 1/16” to one inch.
9. Before machining further scribe diagonal lines going between each of the opposite corners. Their intersection will locate the geometric center of the upper face of the block.
10. At this point still using the vertical miller and the 7/8” dia. Cutter, we cut a step along one edge that extends .25”in from that edge and is .5” deep. Repeat for the other 3 edges.

11. Using a gage which will hold the block at a 45 degree angle with the 1 inch edges between the jaws of the vice of the vertical miller, we mill a flat down from the tip of the uppermost corner to a depth of .31 inches. Repeat for the other 3 corners. This gives us something closely resembling an octagon.
12. Returning the block to the precision parallels we orientate the corners of the interior square so that they are “pointing” to the flats of the vice jaws. Next using the 7/8” dia. end mill (indexing off of the flat of the step and the corner of the inner square) we mill in .25” towards the center of the block. This results in one face of an octagon. Repeat for the remaining 3 corners. This gives us a smaller octagon inside the larger octagon.
13. At this point, we remove the end mill and its collet and install a drill chuck. Center drill on the location of the geometric center of the block. Switch to a .5” diameter drill and drill .65 inches deep. Next, drill all the way through with a #7 drill (.201” diameter). Flip the block over and drill .25” deep with a 3/8”diameter drill. Aside from sanding it to look nice, you are done with the “power supply”.

14. See succeeding for drawings.



15.

16. 

17.

18.

19. Once you have the two main components constructed they should be a press fit. We have included a means of using a screw to assemble just in case tolerances are less than optimum.

7 Train

Manufacturing the Boiler and Smoke Stack

Purpose

To manufacture, as part of an ongoing project, the boiler and smoke stack for a model steam locomotive in order to become familiar with the operation of a engine lathe, drill press, band saw, and other related equipment, and to be able to use this equipment safely and knowledgably with the minimum amount of instruction.

Materials

7 ½" X 7/8" diameter aluminum bar stock

Tools and Equipment
Cross Drilling the Boiler: 6" Rule, hammer, center punch, center drill, #7twist drill (0.201" diameter), 3/8" twist drill, drill press vice, V-block, drill press and cutting fluid.

Facing Off and Axial Drilling: Lathe, collet and collet chuck, Jacob's Chuck with tapered shank, chuck key (if necessary), center drill, round nose tool bit, tool holder, #21 twist drill, ¼" twist drill, 90° counter sink, Allen wrenches, rubber mallet, and cutting oil.

Turning Outer Diameters and Forming Ridges: Lathe, collet and collet chuck, live center, forming tool with 1/32" x 1/16" notch, Allen wrenches, round nose tool bit, inch micrometer, Vernier calipers, emery paper, rubber mallet, and cutting fluid.

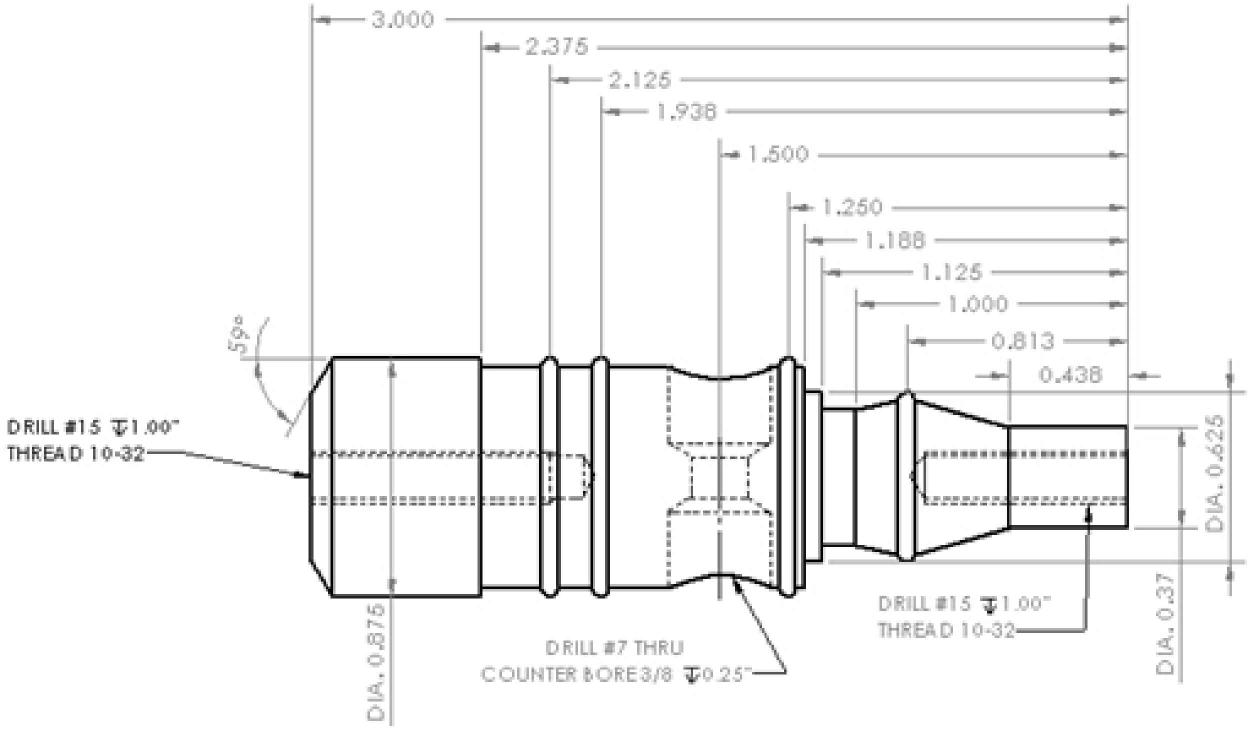
Safety

- Never start the lathe with the tool bit touching the work piece
- Make sure the chuck will not hit any part of the lathe before starting the machine
- Make sure the cam locks of the head stock are tightened CLOCKWISE.
- Hold the work piece securely while drilling holes in it.
- Always clamp round stock to prevent it from rolling while cutting it on the band saw.

General Procedure

Follow the instructions provided in this lab manual. If something does not seem right, consult the instructor. There may be things that are not completely correct, so do not be afraid to ask; you may end up saving yourself some time. PRACTICE SAFETY FIRST! All machines can be dangerous if not used properly.

Figure 47



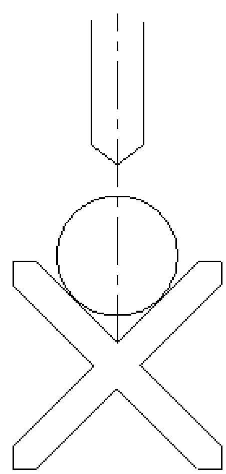
Tolerance fractions +- 1/32" decimals +- .005"

7.1 Layout

Clean the barrel stock with a rag to remove oil and dirt. Coat one side thinly with Dykem, and (using a square) mark a reference line approximately 1/16” – 1/8” from one end of the 7 ½” barrel stock. This line is the location of the surface of the bottom end of the smoke stack. It is also a reference line from which you will mark layout measurements. Consult the drawing (Figure 47) and locate and mark the position of the cross hole. Center punch that mark, next, locate the back of the boiler, located X” from the reference line. Mark this location with the pointed feet of your dividers.

7.2 Cross Drilling the Boiler

Install number 7 drill (.201” dia.) in the Jacob’s chuck of the drill press. Clamp the drill jig into the vise of the drill press. You should position the drill tip directly over the opening in the drill bushing in the jig labeled Train then lock the drill head in place. Drill the hole ,using cutting fluid. Make sure that you use cutting fluid repeatedly during the drilling process. Now go to the V- block for the counter boring procedure. Replace the #7drill with the 3/8” drill and after first carefully aligning the tip of the drill with the center of the “V” drill the first counter bore 0.25” deep. Flip the stock over and drill the second counter bore from the other side, again 0.25” deep.



7.3 Facing Off and Axial Drilling

1. Aluminum is turned at 300-400 RPM on the lathes in the machine shop. Set the controls on the lathe to obtain this turning speed.
2. If it is not already in place, you must install the collet chuck on the lathe. Consult the instructor before attempting this. MAKE SURE THE CAM LOCKS ARE TIGHTENED CLOCKWISE AND ARE TIGHT. Put the collet into the collet chuck, making sure that the key and keyway are aligned.
3. Install the round nose tool bit. Be certain that the tool bit’s cutting tip is centered as per section 1.2.2. In this case, position the tool holder on the side of the quick change tool post that is parallel with the long axis of the quill.

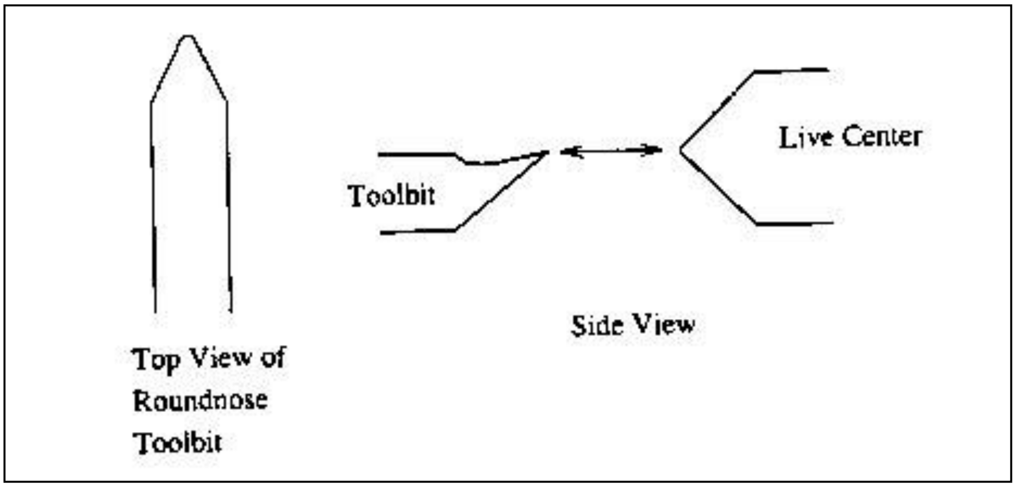


Figure 28: ***Aligning the Round Nose Tool Bit with the Live Center.***

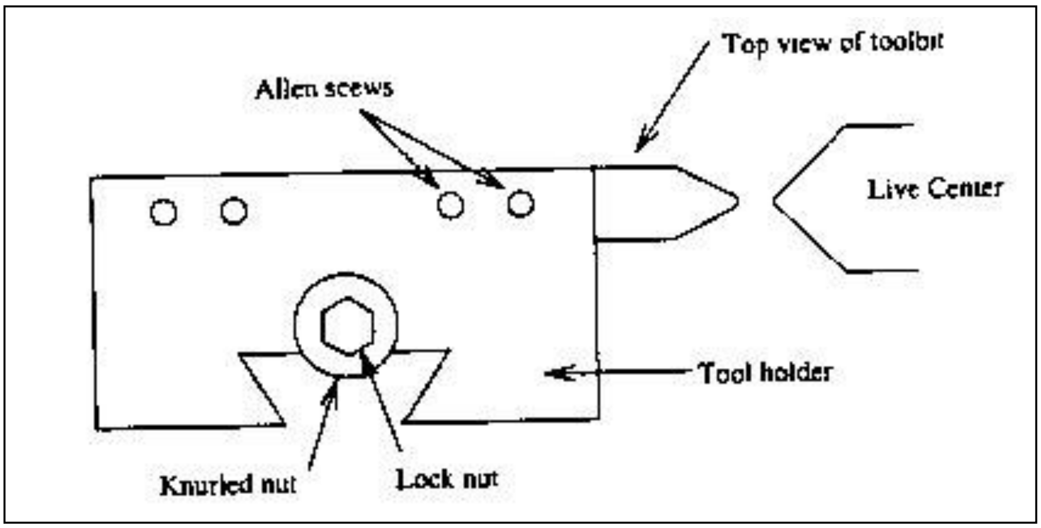


Figure 29: ***Aligning The Tool Bit For Facing Off. In order to face off the barrel, the tool bit must be repositioned to point in the opposite direction.***

4. Once the tool bit is aligned, take the tool bit out of the tool holder by loosening the Allen screws and turn it around so that the tip of the tool post is facing the collet chuck. If you do not align the tool bit correctly, you will not be able to face off your work piece completely to its center (a dimple of metal will be left at the center), and you may damage the tool bit.
5. Install the stock into the collet (be sure no chips are in the collet) so that about 1 inch of the smoke stack end (the short end) is sticking out of the collet. Install the collet into the collet chuck (be sure to align the keyway of the collet) and tighten the chuck.
6. Before using the auto feed, make sure you test it away from your work piece by turning on the lathe, engaging the auto feed, and checking the direction that the tool bit is moving. Face off the smoke stack end of the work piece to the scribed length.
7. Extend that tailstock quill past the 1" mark and install the drill chuck in the quill. Be sure to seat the drill chuck with the rubber mallet. Install the center drill in the drill chuck.
8. Center drill the smoke stack end up to 2/3 of the center drill taper.
9. Drill to the specified depth with the #15 drill (.180" dia.) (See Figure A) by using the scale on the tailstock quill as a guide. Use the depth gauge on the Vernier calipers to check the precise depth of the hole.
10. Now do your partner's boiler & smoke stack.

7.4 Turning Down the Outside Diameters; (This procedure will be only be done manually with one of the two boiler stocks, the other will be completed with the CNC Lathe)

1. Loosen the collet chuck and extend the stock so that roughly 3.5” of material is protruding from the collet; then retighten the collet chuck. This is to ensure proper tool bit clearance with the collet chuck. Install the live center in the tailstock quill.
2. Replace the tool bit in the tool holder and place the tool holder on the tool post. Check the alignment of the tool bit. Orient the round nose tool bit on the tool post so that the tool bit is perpendicular to the bar stock’s axis of rotation.
3. Making sure that all locks on the tailstock are disengaged, extend the tailstock quill and the live center so that about 4 ½” is showing on the quill. Slowly (and carefully) slide the tailstock to the point where it almost contacts the drilled hole in the smoke stack. Now, slowly tighten the live center against the smoke stack. The guideline for tightening the live center is that when the headstock is in neutral, the live center and barrel should rotate together when the headstock is turned. CAUTION: Do not over tighten the live center against the work piece, and do not extend the quill past the 5” mark. CONSULT THE INSTRUCTOR BEFORE TURNING ON THE LATHE.

Note: Use cutting fluid for the following operations. The order of the operations given below is recommended. NEVER start the machine with the tool bit contacting the material. To guard against this, put the headstock in neutral and rotate the lathe chuck by hand to make sure that nothing will hit the tool bit. Re-engage the headstock.

4. The first roughing step is to turn down the outer diameter of the smoke stack to 0.65” starting 1 1/8” from the smoke stack end of the stock.
5. Next, turn down the smaller portion of the smoke stack to 0.4” diameter starting 7/16” from the smoke stack end of the stock.

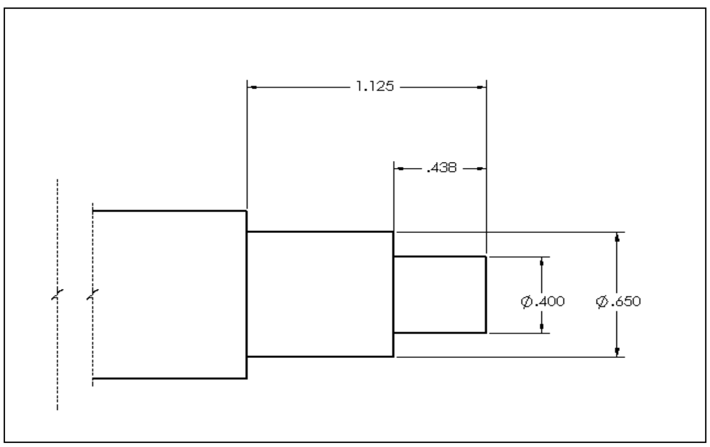


Figure 48: ***Roughing Smoke Stack.***

7.5 Cutting the Ridges

Note: The diameters of the ridges to be cut are the same as the outer diameter of the stock at these points. The diameter of the bar stock will determine the ridge diameter at this area. At the instant when the back of the ridge cutter touches the bar stock, the ridge should: 1) be round, 2) have the diameter of the bar stock at that point.

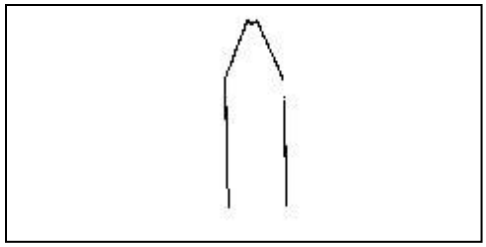


Figure 32: ***Top View of Ridge Cutter.***

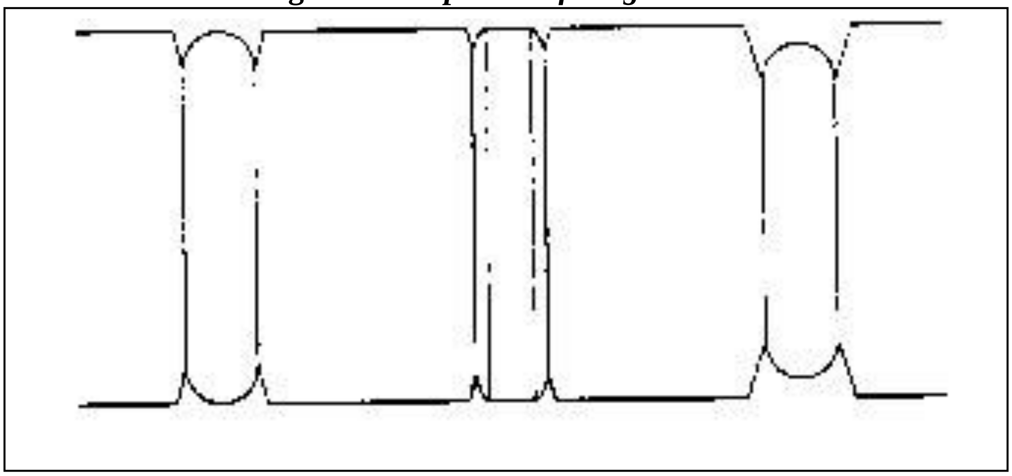


Figure 33: ***Good and Bad Ridges: X) A properly made ridge, Y) Tool bit not pushed in far enough, Z) Tool bit pushed in too far.***

1. At this point, install the form tool bit (ridge cutter – see Figure 32) into the tool holder, and center the tool bit. Install the tool holder on the tool post so that the tool bit is perpendicular to the axis of rotation.
2. Cut Smoke Stack Ridge to 0.65" in diameter.
3. Cut Boiler Ridges to .875" in diameter.

7.6 Angle Cuts and Additional Turning

1. Return the round nose tool bit to the tool holder and center it.
2. Turn down the barrel diameter between the Boiler Ridges to .812". Consult the drawing (Figure 27). If you use the auto feed on this section, pay close attention, and be sure to disengage the auto feed when the tool bit comes close to the ridge. If you do not, you will cut the ridge off.
3. Turn down the .370 diameter of Smoke Stack.
4. Set up the lathe to make the 16° taper on the Smoke Stack. To do this:

Loosen the two nuts in front of and behind the compound rest with the ½" wrench. See Figure 35.
Adjust the angle of the compound rest to 16°. Make sure that the edge of the compound rest is parallel to the surface of the taper, which you plan to cut. Note that each tick mark on the rest is one degree.
Re-tighten the two nuts on both sides of the compound rest.

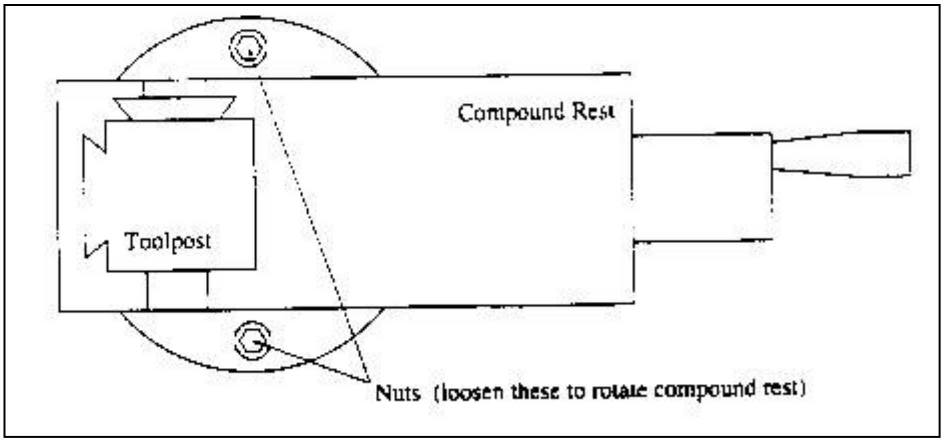


Figure 34: ***Rotating the Compound Rest.***

NOTE: Using the small hand wheel on the compound rest itself can only make the angle cuts. All other hand wheels will make perpendicular cuts. DO NOT USE AUTOFEED; auto feed controls the larger hand wheels, not the one attached to the compound feed.

5. Cut the 16° taper from the base of the smoke stack ridge the final diameter, using the cross slide dial to determine the depth of cut, and the top slide dial on the compound rest to make a pass.

This Procedure will be followed for finishing the Smoke Stack once it has been parted off.

7.7 Polishing the Boiler & Smoke Stack

In order to remove any tool marks made by the round nose tool bit, you must use emery paper to polish the surface of the barrel. It is easiest to polish the Boiler & Smoke Stack while it is mounted on the lathe. This section is optional, and the final surface finish is up to you.

For best results, tear a narrow (1/2 inch) strip of emery paper, and wrap it around the boiler stack. Hold one end of the emery paper in each hand, and pull it toward you while the part is turning. Move the paper slowly side to side to sand the entire surface.

1. Start with coarse grit (120) emery paper, and polish the surface until the scratches in the barrel are uniform in appearance.
2. Using medium grit (240) emery paper repeat the process until the surface scratches again look uniform.
3. Repeat the process with a find grit (500) emery paper.
4. After the sanding is complete, you may want to buff your work piece. This final step will only help shine the surface. It will not remove major scratch marks from the surface. Consult the instructor before using this machine.

7.8 Parting Off the Smoke Stack and Making the Wheels

1. Once the Boiler and Smoke Stack are ready for removal of the Smoke Stack, obtain a parting tool. Before Cutting, ensure the live center is not in place.
2. Install the parting tool on the tool post and ensure that it is at precisely 90° to the longitudinal axis of the material; to do this, this, place the side of the parting tool against the faced off end of your work piece (see Figure 36).

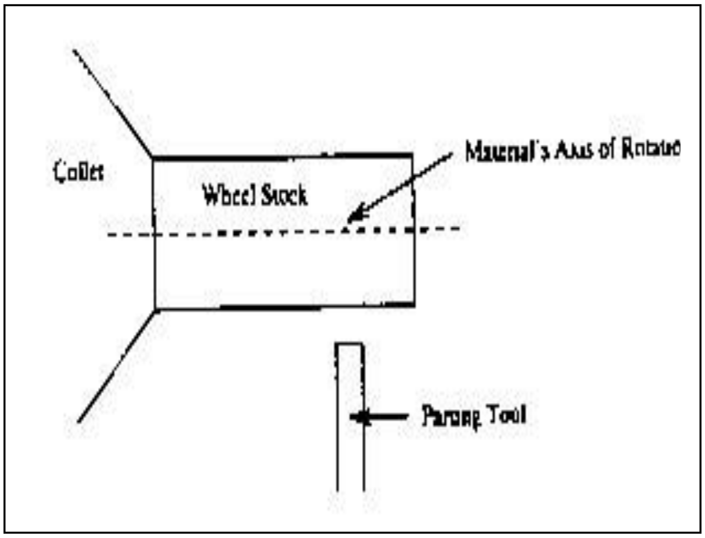


Figure 35: ***Aligning the Parting Tool So It Is Perpendicular.***

3. Using the parting tool cut off the Smoke Stack at 1” from the end of the Smoke Stack. DO NOT use auto feed for this process. Be sure to bathe the parting tool and the aluminum piece with cutting fluid continuously as you cut into the aluminum. Also, do not feed the parting tool past the center of the aluminum bar stock.
4. Remove the boiler from the collet and cut the boiler from the remainder of the stock using the band saw.
5. Reinstall the stub of the stock into the collet and face off the stock.
6. Center drill the end of the stub and drill all the way through it with the #7 twist drill (0.201”). This forms the hole required by the retaining screws, which will attach the wheels to the train cab. Again, refer to section 4.3.
7. Sand the stock with rough emery paper lightly to remove any Dykem or dirt.
8. Using the parting tool and wheel gauge, measure (7/32” from wheel gage) and slice off two wheels from the stub (Figure 35). Again, Do NOT use auto feed for this process. Be sure to bathe the parting tool and the aluminum piece with cutting fluid continuously as you cut into the aluminum.
9. Obtain a piece of 1/2” stock, replace the 7/8” collet with a 1/2” collet and repeat steps 5 through 8, producing the 4 small wheels of the train.

7.9 Finishing the Boiler & Smoke Stacks

1. Reinstall the 7/8” collet into the head stock and insert the Boiler rear end out.
2. Face off the rear end of the Boiler. Using the procedure for the angle cuts as described above, cut the 60° chamfer onto the back of the Boiler.
3. Center drill the rear end of the boiler up to 2/3 of the center drill taper.
4. Drill to the specified depth with the #15 drill (.180” dia) (see Figure 47) by using the scale on the tailstock quill as a guide. Use the depth gauge on the Vernier calipers to check the precise depth of the hole.
5. With the Boiler still in the collet, thread the #15 hole in the end of the Boiler to 10-32.
6. Remove the Boiler & 7/8” collet and replace with the 3/8” collet and install the Smoke Stack into it.
7. Using the angle cut procedure, cut the 16° taper on the top of the Smoke Stack.
8. Optionally, a ¼” drill may be used to create a bowl in the top of the Smoke Stack.

The Boiler & Smoke Stack are now finished.

8.0 Milling the Train Cab

Purpose
To manufacture the cab of a steam locomotive and to become familiar with the safe operation of the horizontal and vertical milling machines

Materials

1 ½” x 1 ½” x 2 7/8” aluminum block

Tools and Equipment

Layout: Dykem, rag, scribe, combination square, center punch, hammer, 6-inch ruler, hermaphrodite, divider.

Drilling: Drill press, center punch, center drill, “F” twist drill (0.257” diameter), #15 drill (.180” dia.), drill press vice, cutting fluid.

Tapping: Tap, tap wrench, tapping guide, cutting fluid, file.

Horizontal Milling: Horizontal miller, combination square, clamp and mounting setup, spacer blocks, cutting oil, file

Band Sawing: Band saw marked “Aluminum Only”, wooden push block, and cutting fluid.

Vertical Milling: Vertical miller with vise, spacer block, 7/8”end mill, collet, micrometer, file, scribe.

Safety
- **Never start the vertical miller with the end mill touching the work piece.**
- **Keep your hands AWAY from the work piece until the end mill has come to a complete stop.**

General Procedure
Caution: Milling machines should be used with the greatest caution! Respect them, and follow all safety rules. Consult an instructor before turning these machines on so that YOU do not get hurt. Do not be afraid to ask questions, or have the instructor check your work or setup.

An instructor will generally assign you an order in which to perform sections 8.2 – 8.4 in order to maximize the use of the available machinery. **Otherwise, follow the order of operations as given.**

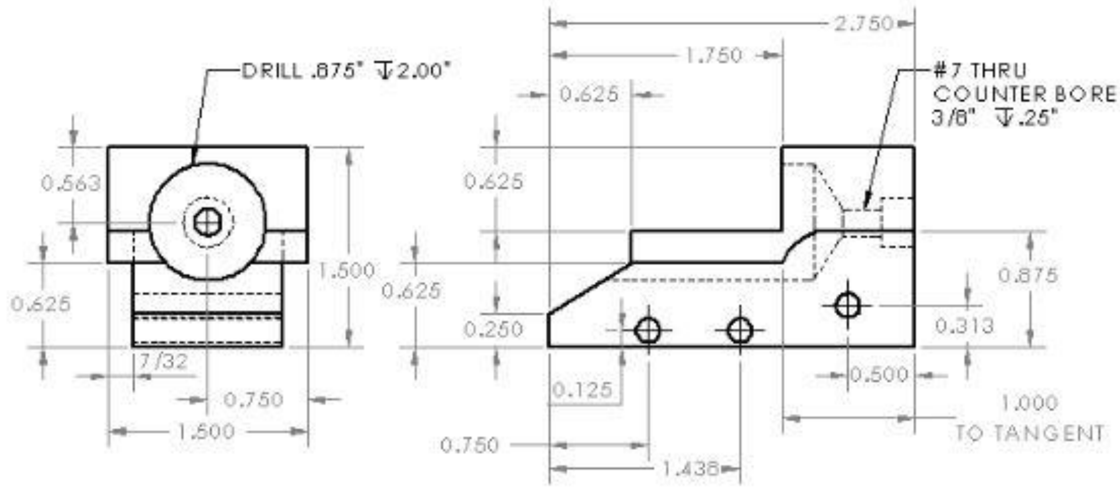


Figure 49: ***Train Cab Machining Diagram.***

Tolerances fractions +- 1/32" decimals +-.005"

8.1 Layout

1. Clean the block with a rag to remove any oil and dirt and coat two adjacent sides thinly with Dykem.
2. Using a square, mark a reference line approximately 1/16" from one end of the block. Consult the drawing (Figure 42).
3. Following the instructor's directions, scribe on the side of the block all the lines as indicated in the side view of the drawing.
4. Using a center punch and hammer, center punch the location of the three holes.

8.2 Horizontal Milling

1. Clamp your cab, and your partner's cab, to the table of the horizontal milling machine. Make sure that the end lines of the cabs are aligned with the cutting wheels, and that the blocks are square to the cutters. The instructor will explain in detail how this is done.

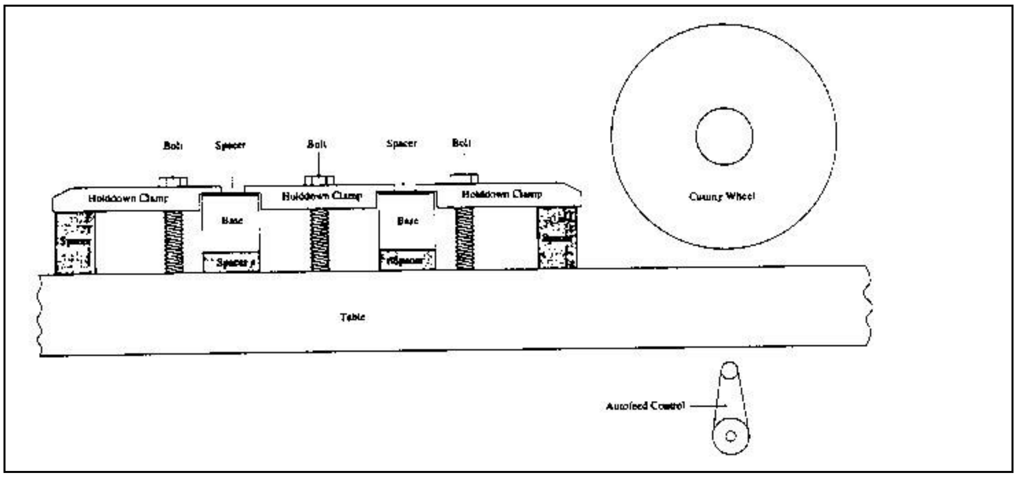


Figure 42: ***Side View of Horizontal Milling Setup.***

2. Adjust the table height so that both ends of the block will be cut at the same time, and so that the cutters will machine the entire face of the block. Make sure that the machine table will not be cut into.
3. Cut the block using conventional milling and the automatic feed. Use plenty of cutting fluid; also coat the insides and the circumferences of the cutters AWAY FROM WHERE THE ALUMINUM BLOCK IS CONTACTING THEM. Keep your hands away from the cutting wheels.

8.3 Drilling Holes

When drilling holes, it is important to make sure that the twist drill is positioned so that it does not flex (i.e. bend) when it is pushed into the hole. This can best be done by center drilling a hole location then immediately switching to the appropriately sized twist drill for the through hole. Drill the hole re-lubricating the tip of the drill every1/4"of depth. Otherwise the hole may be oversized, and/or not round or penetrate the block at an angle.

It is necessary to determine the actual diameter of a twist drill before you use it. Occasionally, a twist drill of the wrong size will be put into the drilling kit by mistake. To check the size of a twist drill, use a micrometer to measure the diameter. Measure across the cutting flutes, not the shank portion.

When drilling, be certain to use adequate (LARGE) amounts of cutting fluid to prevent galling and assist in chip removal. This is especially true when drilling with large (>1/2") drill bits.

1. Prior to drilling, measure, scribe, and center punch the location of the boiler (7/8 inch) hole .
2. Using the drill press, center drill the three cross-hole locations. When drilling, put a plywood spacer under the base to prevent drilling into the vice of the vertical miller. (See Figure 50).
3. Drill the axle holes with a #15 twist drill (0.180").
4. At this point choose which train cab will be finished on the CNC vertical miller.
5. Unclamp the block and place it (front end up) in the vice of a vertical miller. Using a pair of precision parallels in place of the plywood block used previously.
6. Center drill the location of the boiler hole.
7. Pre-Drill the boiler hole using a 1/2" twist drill in a ½" collet.
8. Remove ½"drill & install 7/8" Silver & Demming drill bit directly into a ½" collet
9. Drill the 7/8" hole down 2.25 inches.
10. Remove the 7/8" drill bit and install the drill chuck directly into the spindle.
11. Drill with a #7 (.201 diameter) twist drill through the remaining material.
12. Turn over the block and counter bore using 3/8" twist drill to a ¼" depth.

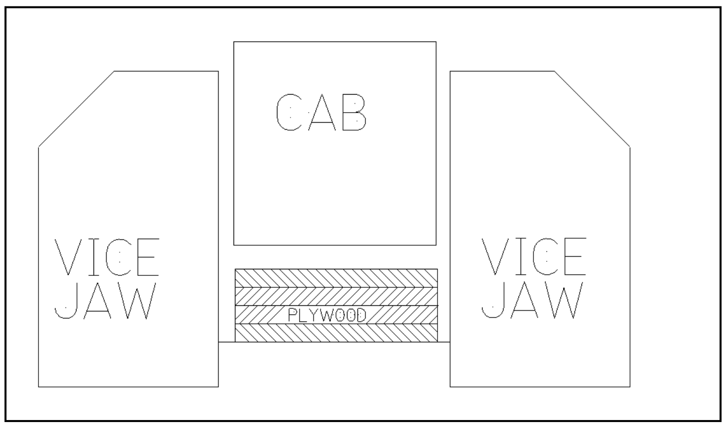


Figure 50: ***Positioning of the Cab and Plywood Spacer in the Vice for Drilling.***

8.4 Tapping

Please see Section 5 and especially section 5.2.2 and 5.2.3.

8.5 Band sawing

Cut out the 5/8 inch x 1 3/4 inch slot with the band saw marked "Aluminum Only". Use a wooden push block, and cut about 1/16" – 1/8" on the scrap side of the lines. This will leave excess material on your block, which you will remove later with the vertical milling machine.

8.6 Vertical Milling

1. Obtain a vertical milling kit from an instructor.
2. Install a 7/8" two-flute end mill into the milling machine. Refer to section 2.2.1.

3. Make sure the vice is free of chips, and that any burrs have been filed off the blocks to be milled. Mount the blocks side by side, with a spacer block underneath. The band saw lines should be a little higher than the edges of the vise. Have an instructor check your setup before proceeding.
4. Adjust the speed of the end mill. For aluminum, the end mill should rotate at about 1800 RPM.
5. Mill the 5/8" section, using cutting fluid. CAUTION: It is extremely dangerous to put your fingers or a fluid brush near a moving end mill. DO NOT DO IT! Cut all the way to the scribed lines. Use conventional milling (See Figure 46). Never cut deeper than 0.050" along the vertical

axis of the miller. When using the side of the end mill to make a cut. Do not cut deeper than 0.010".

6. Mount the blocks next to each other in the vise such that the diagonal lines are parallel with the top of the vise. This is done so that the "cow catcher" can be milled at an angle with respect to the bottom of the cab.
7. Next, place one cab into the vice and mill the side profiles of the cab using a cut depth totaling 7/32" (maximum of .05" at a time). First mill the straight section to depth on one side and then cut the recess for the large wheel. Flip the cab over and repeat for the opposite side.

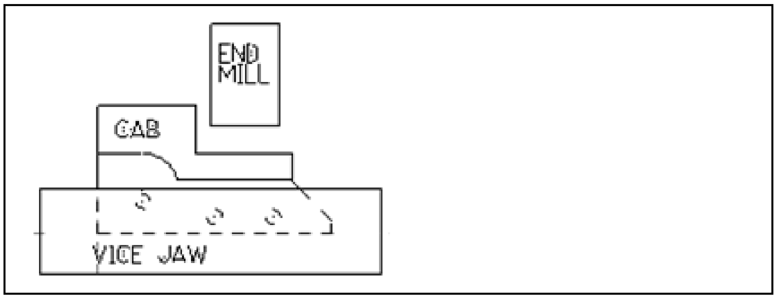


Figure 51: ***First Milling Setup for the Train Cab.***

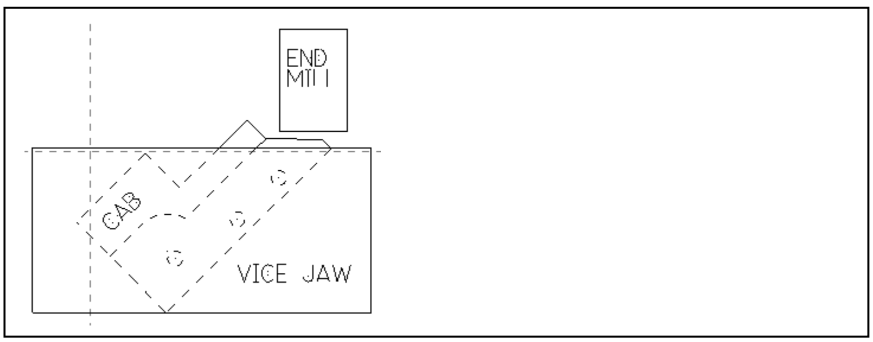


Figure 52: ***Second Milling Setup for the Train Cab.***

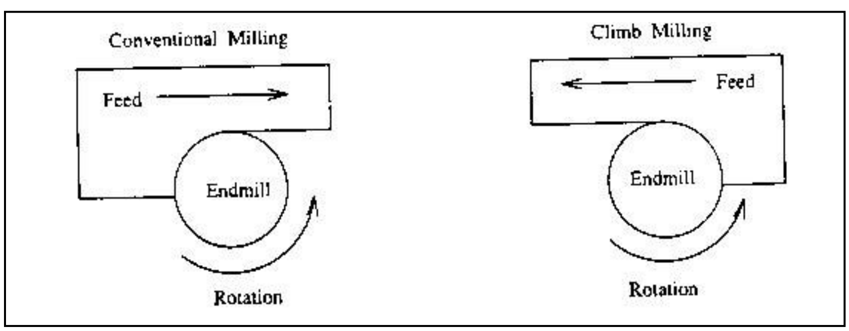


Figure 46: ***The Difference Between Conventional and Climb Milling.***

8.7 Polishing the Cab

Now that the machining of the cab is complete, file off any burrs that are on the edges of the cab. Using emery paper, sand the sides of the cab to remove any scratches and Dykem. Begin with coarse grit of emery paper (120), and sand until each surface looks uniform. Then proceed to finer grit of emery paper (240, 500), and repeat the process.

For the best results, use any flat surface (wooden work bench, band saw table etc.). Set the emery paper on the surface, and secure it. Hold your cab, and move the cab against the emery paper, using slow, deliberate strokes. Sand in only one direction.

Finally use the buffing wheels if desired. We require only that all bluing and large burrs be removed.

8.8Train Assembly

At this point, your train cab, boiler and smoke stack should be polished to your satisfaction.

1. Using a rag, clean off any dirt from the boiler, smoke stack, and cab.
2. Obtain the letter and number stamps from the instructor. Stamp your name (or initials, class, year, favorite saying, or anything else you like) on the bottom of your cab. Use the cannon-stamping jig to support the cab while you do this.
3. Obtain 8 screws from and instructor. Using an Allen wrench, put on the wheels.
4. Place the smoke stack into the accepting counter bore in the boiler and attach using a screw from the opposite side.
5. Place the boiler & smoke stack into the groove of the cab and ensure the smoke stack is perpendicular to the top of the cab. Place the remaining screw through the counter bore in the back of the cab and into the boiler. Should it not reach, remove the boiler and wheels, drill the counter bore deeper, and replace boiler.
6. When your train is assembled, make sure that an instructor checks you off as having completed the project.

9.0 Welding Lab

All of the following should be accomplished only while under the supervision of a lab instructor.

Purpose

To practice arc initiation, fabrication, and testing of a butt-weld

Material

3/16” x 1 ½” x 2” hot rolled steel plates.

Tools and Equipment

Gas Metal Arc Welder, breaker bar, vice, pliers, welder’s helmet, gloves, apron

Safety

- **Make sure there is proper ventilation in the area that you are welding in.**
- **Always wear a functional welding helmet while welding.**
- **Always wear a leather apron and leather gloves while welding. Long sleeved shirts are also recommended.**
- **People wearing nylon-topped shoes should wear a pair of leather spats when welding.**
- **Do not attempt to touch recently welded objects (even with gloves) until they have cooled.**

Procedure

1. Ensure that the ventilation system is on.
2. Attach the ground cable to the clamp on the workbench, if it has not already been done.
3. Place a piece of scrap steel on the welding bench.
4. Adjust the controls on the welding machine to the proper settings.
5. Turn on the shielding gas supply to the welding machine.
6. After the safety of all individuals in the welding area has been checked, initiate an arc, and practice laying a short bead on the scrap metal. Remember to maintain a proper arc gap, and move the welding gun slowly, in a circular motion.
7. Practice until the weld bead looks uniform along its entire length.
8. Make a butt weld by laying two test plates side by side and laying a short bead 1 to 2 inches long on the joint between them.
9. The instructor will show you how to test the welds.
10. At this point, you have a welding project to do in order to practice your welding technique. The instructor will show you what materials are available for your project and what she/he expects. You may work together or individually.

9.1 Plasma Cutting-As part of your welding lab you will also use both a manually controlled and a CNC plasma cutting table. Explanations will be given by your instructor or laboratory assistant.

Oxyacetylene Brazing

Purpose

To practice adjusting regulators, lighting the torch, adjusting the flame, brazing and bend testing

Tools and Equipment

Oxygen and acetylene tanks with regulators, welding torch and tip, spark lighter, brazing goggles, welder's apron, and gloves.

Materials

1/16" x ½" x 6" steel pieces, 1/16" brass brazing rods, brazing flux.

Safety

- **Make sure there is proper ventilation in the area that you are brazing in.**
- **NEVER set the acetylene pressure above 15 psi.**
- **Always wear brazing goggles before lighting torch.**
- **Point the torch in a safe direction before lighting it.**

Procedure

1. Make sure that everyone is wearing a leather apron, leather gloves, and brazing goggles.
2. Open main valves of acetylene and oxygen tanks. Adjust regulators to provide proper working pressure for the gases (acetylene = 5 PSI, oxygen = 20 PSI).
3. Place two pieces of steel on a firebrick so that they overlap each other.
4. Open the acetylene needle valve on the torch (red hose = acetylene) approximately ¾ of a turn, and light the torch with a spark lighter.
5. Adjust the acetylene needle valve until there is a gap of about ¼" between the tip of the torch and the base of the flame.
6. SLOWLY open the oxygen needle valve on the torch. Keep adding oxygen until the flame just turns from orange to blue. This will result in a carburizing flame. Note the three different zones of the flame; the inner cone, the acetylene feather, and the outer envelope.
7. Heat the tip of the brazing rod until it just begins to melt (a few seconds) and dip it into the brazing flux. This will cause some flux to stick to the end of the rod.
8. Fan the flame over both pieces of steel. Make sure both pieces of steel are heated.

9. Touch the brazing rod to the edge where the pieces overlap. Concentrate the flame on this point, until the brazing rod begins to melt. As it melts, feed more brazing rod into the joint.
10. More flux can be added at any time by dipping the already hot brazing rod into the container of flux. Be sure to add brazing rod to the other edge where the pieces touch.
11. In order to draw the brass into a joint, heat the part of steel, that lies in the direction in which you want the brass to flow.
12. When extinguishing the flame, turn off the oxygen needle valve first; then the acetylene needle valve.

10.0 Plastic Welding Lab

Purpose
To practice adjusting the controls of the plastic welder, making the welds and strength testing

Tools and Equipment
Plastic Welder, clean rag or paper towel

Materials
Pieces of PVC, Lexan, or acrylic, 1/8” welding rod (PVC or acrylic)

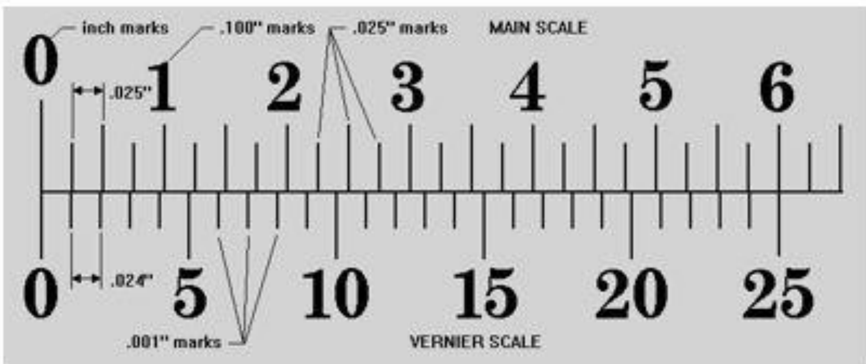
Safety
- **Make sure there is proper ventilation in the area that you are plastic welding in.**
- **Keep your fingers as far away from the hot air stream as possible.**
- **Do not touch the metal portions of the plastic welding gun.**

Procedure
1. Open the main air valve on the plastic welding apparatus FIRST. Adjust the air pressure to 3 PSI.
2. Once the air is flowing, turn on the heating element. The welder will take about 10 minutes to heat up.
3. Clean the pieces of plastic with the rag. Make sure the pieces are free of dirt, chips, and oil.
4. Hold the pieces of plastic together in the desired configuration, and use the tacking tip of the welding gun to tack the pieces together.
5. After the pieces are tacked together, use the appropriate type of welding rod to weld the pieces together.
6. The instructor will show you to test the strength of the weld.
7. At this point, you have a welding project to do, in order to practice your welding technique. The instructor will show you what materials are available for your project, and what she/he expects. You may work together or individually.
8. When turning the plastic welder off, turn off the heating element FIRST. Wait until the air coming out of the gun is at room temperature (about 10 minutes) before turning off the air valve.

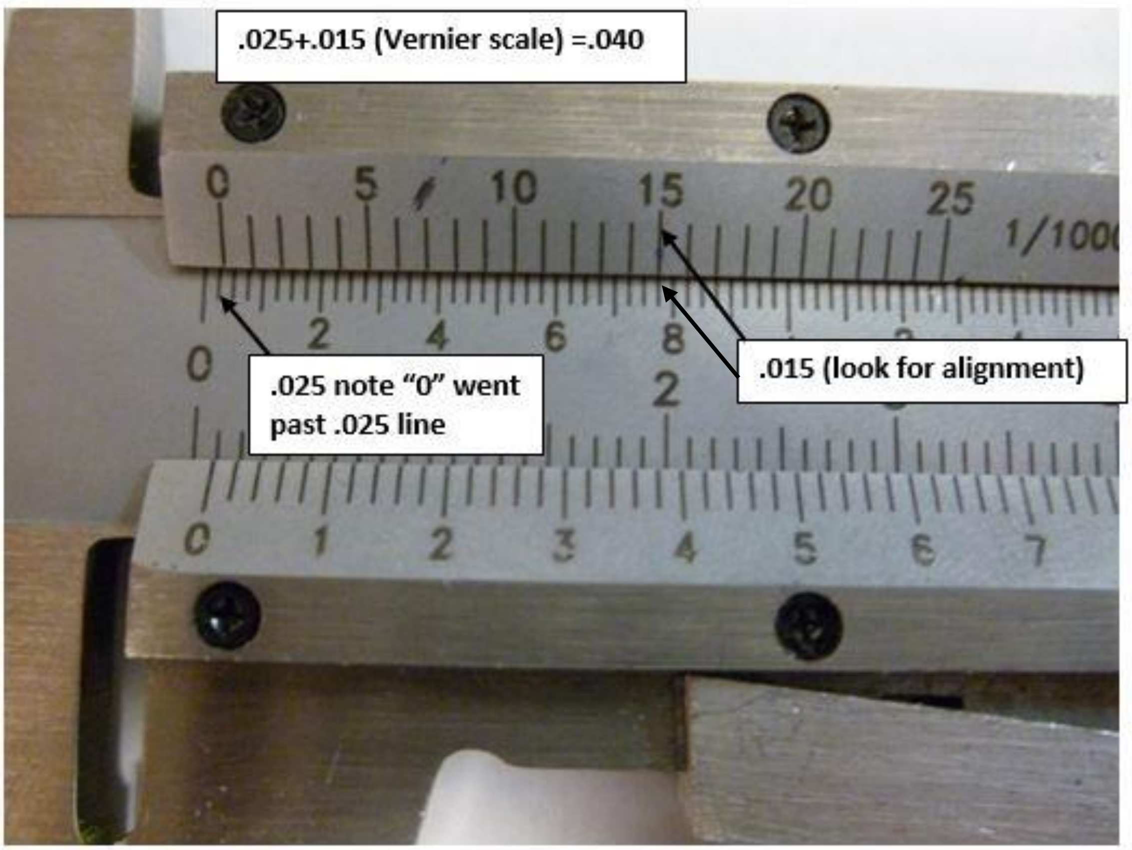
Laser Cutter- This is a CNC device akin to a printer. Instead of an ink jet it uses a laser beam. The Corel Draw program which runs it accepts a wide variety of CAD formats as inputs. This will be included in your welding lab. Part of this includes an introduction to CAD(specifically Mastercam).

Appendix A: Reading a Micrometer and Vernier Caliper

1.2.2.1 <u>How to read Vernier Caliper Scale:</u>

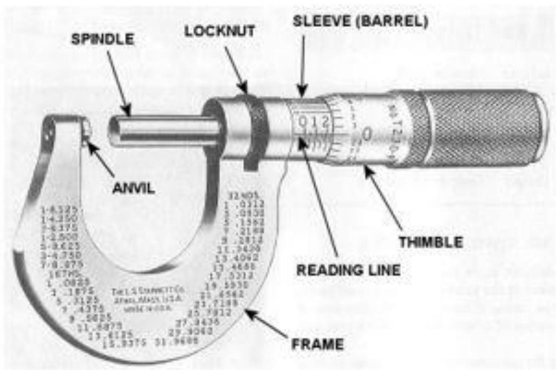


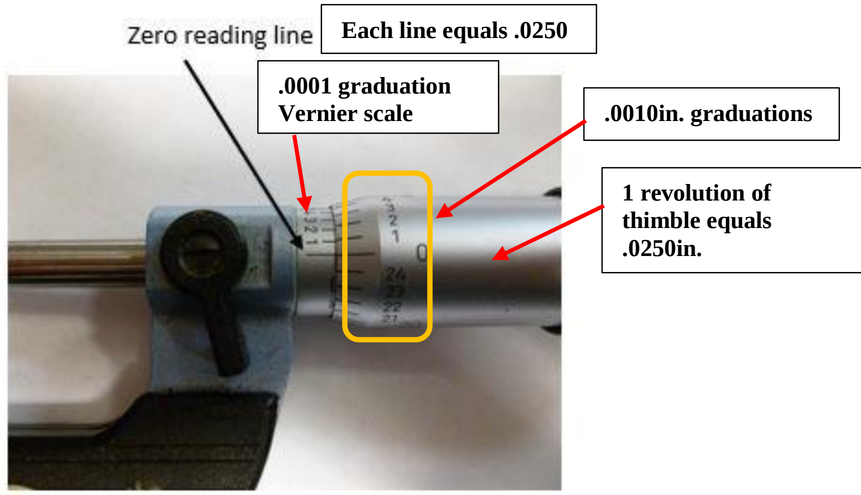
Sample



1.2.3 <u>Micrometer Features and Reading Micrometer Scale:</u> a device incorporating a calibrated screw, sleeve and rotating thimble with a Vernier scale. They are used for **precise** measurement of components (diameter, length and thickness) within its range. The accuracy is 1/10,000 inch (.0001), they must be handled with care and not overtightened. Most incorporate a friction/ratchet thimble to set proper force between object, spindle and anvil.

<u>0 to 1 inch Micrometer</u>





How to read an Outside Micrometer

The pitch of the screw thread on a standard spindle is 40 threads per inch. One revolution of the thimble advances the spindle face toward or away from the anvil face precisely 1/40" or 0.025" equaling the distance between two graduations on the sleeve. The reading line on the sleeve is divided into 40 equal parts by vertical lines that correspond to the number of threads on the spindle. Therefore, each vertical line designates 1/40 inches or 0.025". Every fourth line, which is longer than the others, designates 0.100 inches and is numbered. The beveled edge of the thimble is divided into 25 equal parts with each line representing 0.001" and every line may or may not be numbered however every fifth line is numbered consecutively. To read the micrometer in thousandths, multiply the number of vertical divisions visible on the sleeve by 0.025 inches, and to this add the number of thousandths indicated by the line on the thimble which best coincides with the central long line on the sleeve.

How to read a Micrometer graduated in 0.001"

Micrometer Thimble showing 0.276 inch

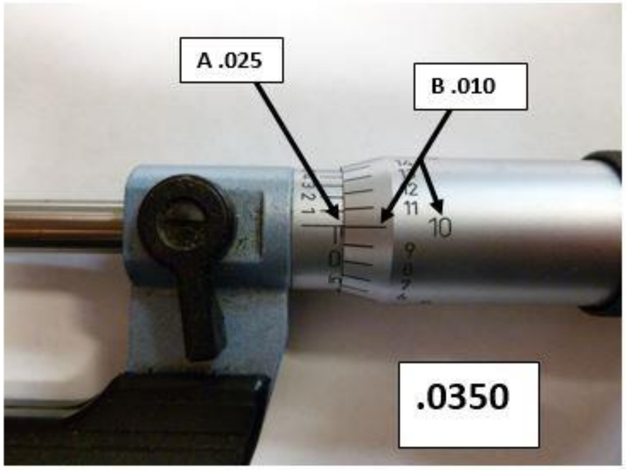
In the above picture, the thimble is positioned to where it is setting between the 2nd and 3rd numbered graduation thus 2 x 0.100in = 0.200in. Past that there are three additional sub-divisions, which is 3 x .025in. = 0.075in. Lastly the graduation 1 on the thimble is the closest to the central long line on the sleeve therefore 1 x 0.001in. = 0.001in. The reading thus would be 0.200in. + 0.075in + 0.001in, totaling to 0.276in.

How to Read an Outside Micrometer Graduated in 0.0001" (using the extra Vernier scale)

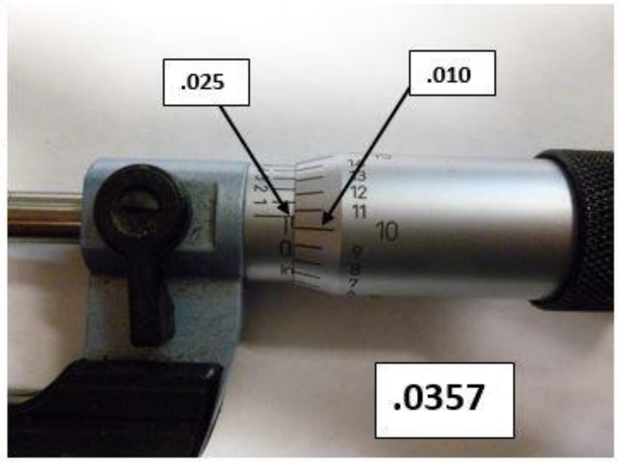
Many micrometers include a Vernier scale on the sleeve in addition to the regular graduations. This allows measurements within 0.0001 inches on inch-system micrometers.

The additional digit of these micrometers is obtained by finding the line on the sleeve that best coincides with the line on the thimble. The number of this coinciding Vernier line represents the additional digit.

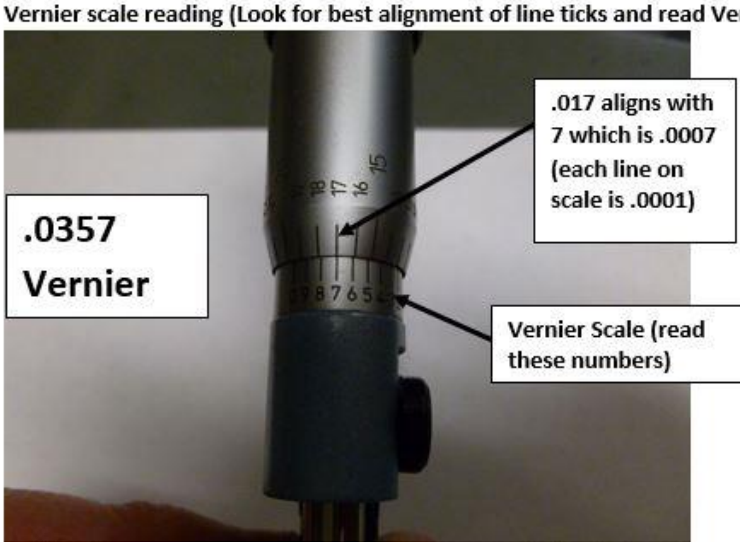
.0250+.0100=.0350 Note: ("0" went past reading line .010)



.0250+.0100 (went past)+Vernier scale (next picture)=.0357

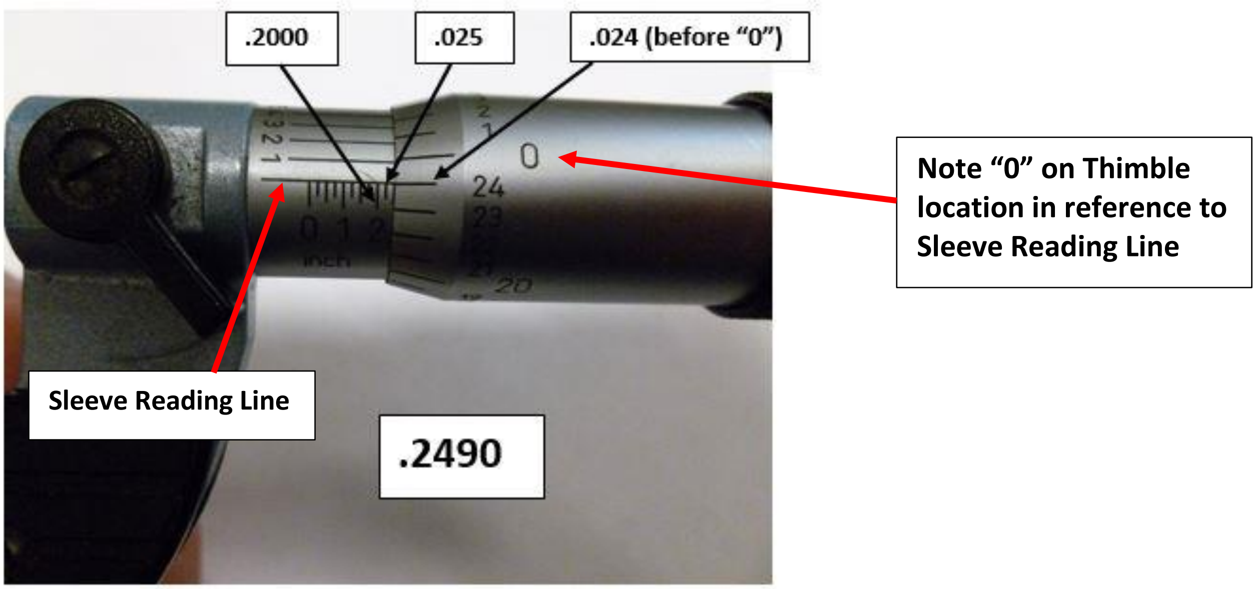


.0375 continued (0 went past reading line .010+) so now you must add Vernier scale reading (Look for best alignment of line ticks and read Vernier scale)

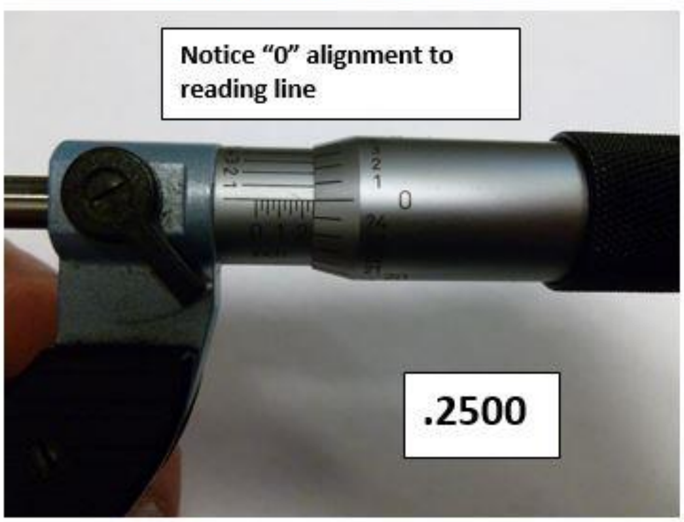


<u>More Examples:</u>

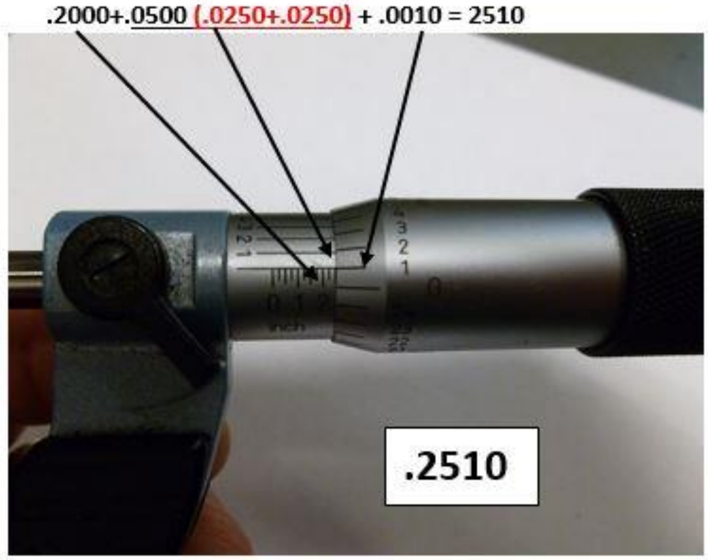
.2000+.0250+.0240=.2490



.2000+.0500 (.0250+.0250 two lines past) =.2500



.2000+<u>.0500</u> <u>(.0250+.0250)</u> + .0010 = 2510



Standard Inch Micrometer (graduated in thousandths of an inch)

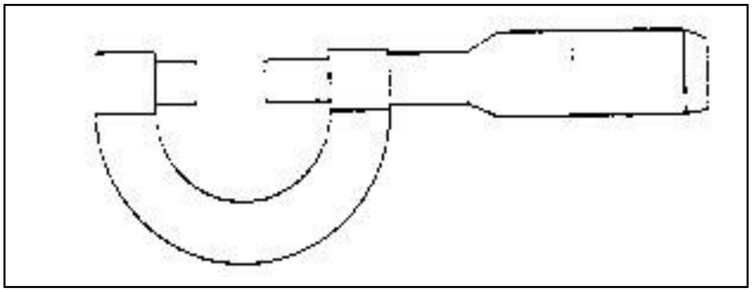


Figure 53: ***Inch Micrometer***

In order to understand the principle of the inch micrometer, the student should be familiar with two important terms concerning screw threads:

Pitch: The distance from a point on one thread to a corresponding point on the next thread. For inch threads, this is expressed as 1/N (where N is the number of threads per inch). For metric thread, pitch is expressed in millimeters.

Lead: The distance a screw thread advances axially in one complete revolution or turn.

Since there are 40 threads per inch on the micrometer, the pitch is 1/40 (0.025) inch. Thus, one complete revolution for the thimble will either increase or decrease the distance between the measuring faces by 1/40 (0.025) inch. The 1-inch distance marked on the micrometer sleeve is divided into 40 equal divisions, each of which equals 1/40 (0.025) inch.

If the micrometer is closed until the measuring faces just touch, the zero line on the thimble should line up with the index line on the sleeve. If the thimble is rotated counter-clockwise one complete revolution, it will be noted that one line has appeared on the sleeve. Each line on the sleeve indicates 0.025 inches. Thus, if three lines were showing on the sleeve, the micrometer would have opened 3 x 0.025, or 0.075 inches.

Every fourth line on the sleeve is longer than the others and is numbered to permit easy reading. Each numbered line indicated a distance of about 0.100 inch. For example the #4 showing on the sleeve indicates a distance between the measure faces of 4 x 0.100 or 0.400 inches.

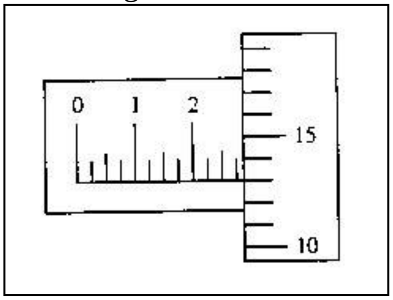
The thimble has 25 equal divisions about its circumference. Since one turn moves the thimble 0.025 inches, one division would represent 1/25 of 0.025 or 0.001 inch. Therefore, each line on the thimble represents 0.001 inch.

Reading a Standard Inch Micrometer

1. Note the last number showing on the sleeve. Multiply this number by 0.100.
2. Note the number of small lines visible to the right of the last known shown. Multiply this number by 0.025.
3. Add the number of divisions on the thimble from zero to the line that coincides with the index line on the sleeve.

Example 1

See the figure below. The above steps are followed in order to obtain the reading.



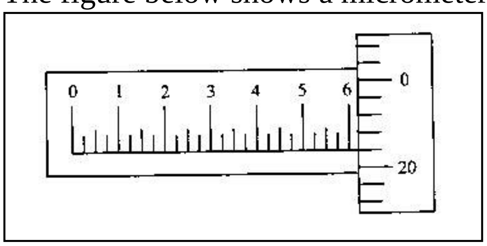
Answer:________________

An inch micrometer reading of 0.288 inch

#2 shown on the sleeve:	2 x 0.100 = 0.200
3 lines visible past the number:	3 x 0.025 = 0.075
#13 line on thimble coincides with the index line:	13 x 0.001= 0.013
Adding up these numbers, the total reading is:	0.288 inch

Example 2

The figure below shows a micrometer with a different reading.



Answer:________________

An inch micrometer reading of 0.621 inch

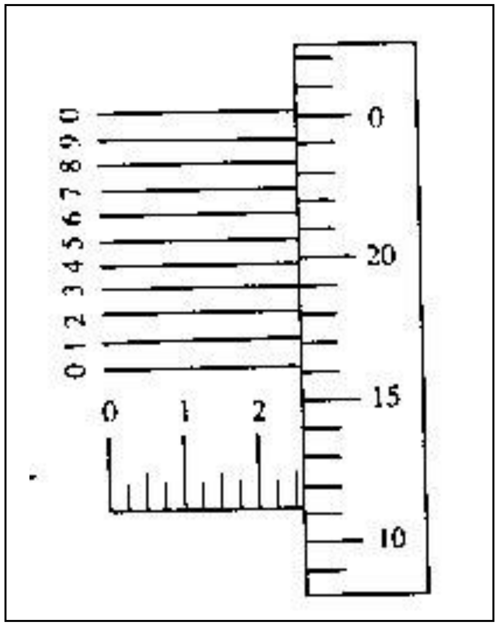
#6 shown on the sleeve:	6 x 0.100 = 0.600
0 lines visible past the number:	0 x 0.025 = 0.000
#21 line on thimble coincides with index line:	21 x 0.001 = 0.021
Adding up these numbers, the total reading is:	0.621 inch

Inch Micrometer (graduated in ten-thousandths of an inch)

Some inch micrometers have, in addition to the graduations found on a standard micrometer, a Vernier scale on the sleeve. This Vernier scale consists of 10 divisions, which run parallel to and above the index line. It will be noted that these 10 divisions on the sleeve occupy the same distance as 9 divisions (0.009) on the thimble. One division on the Vernier scale represents 1/10 x 0.009 or 0.0009 inch. Since one graduation on the thimble represents 0.001 or 0.0010 inch, the difference between one thimble division and one Vernier scale division represents 0.0010 – 0.0009 or 0.0001 inch. Therefore, each division on the Vernier scale has a value of 0.0001 inch.

Reading a Micrometer to Ten-Thousandths of an Inch

1. Read the micrometer in the same manner as you would a standard inch micrometer.



2. Note the line on the Vernier scale that coincides with a line on the thimble. This line will indicate the number of ten-thousandths that must be added to the above reading.

Examples

Answer:________________

A micrometer reading of 0.2613 inch

See the figure above. The micrometer is read as follows:

#2 shown on the sleeve:	2 x 0.100 =	0.200
2 lines visible past the number:	2 x 0.025 =	0.050
#11 line on the thimble coincides with the index line:	11 x 0.001=	0.011
#3 line on the Vernier scale coincides with a line on the thimble:	3 x 0.0001=	0.0003
Adding up these numbers, the total reading is:		0.2613 inch

Reading a Vernier Caliper

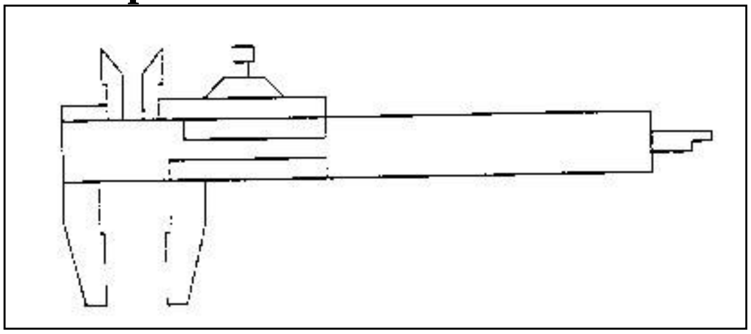


Figure 54: ***Vernier Calipers.***

A Vernier caliper is a tool, which can measure objects to the nearest thousandth of an inch, just as an inch micrometer can.

One advantage of the Vernier caliper is that it can be used to measure larger objects than an inch micrometer can. The inch micrometer is limited to measuring objects that are under an inch in size. The Vernier caliper, on the other hand, can measure objects which are several inches long.

In addition to measuring outside diameters, the Vernier caliper can also be used to measure inside diameters. Thus, the diameter of a hole can be determined using this tool. An additional feature of the Vernier caliper is that it also has a depth gauge; thus, the depth of a hole can be measured as well.

Measuring outside diameters, inside diameters, and depth are done with different parts of the Vernier caliper. Once the measurement is taken however, reading the instrument is done in exactly the same way.

If you look at the Vernier caliper, you will note that one jaw is fixed, while the other jaw slides. A Vernier scale, numbered 0-25, is attached to the sliding jaw, and moves with it. In order to take a reading, it is necessary to determine the position of the zero mark of this scale, relative to the scale on the flat, fixed portion of the caliper.

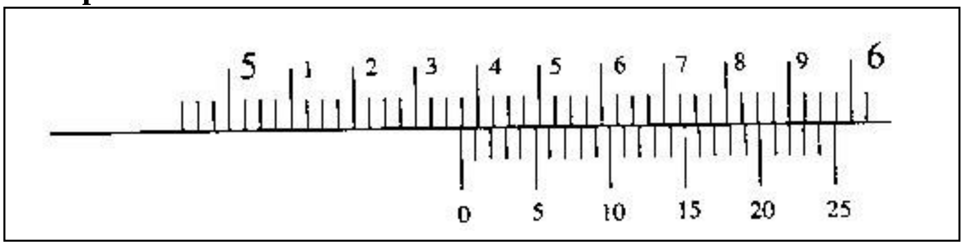
The scale on the flat, fixed portion of the caliper is divided into inches, and tenths of an inch. Each inch mark is noted with large numerals, while each tenth of an inch is marked with smaller numerals. Each tenth of an inch is also segmented into four equal parts. Thus, each small line on this scale is ¼ x 1/10 = 1/40, or 0.025 inch.

Using just the fixed scale on the caliper, it is possible to obtain the position to 0.025 inch. In order to obtain the position to 0.001 inch, we must use the Vernier scale. The Vernier scale is divided into 25 segments, and is used just like the Vernier scale on the Vernier micrometer. By finding the line on the Vernier scale, which matches up with a line on the fixed scale, we determine the number of thousandths of an inch, which must be added to our result to get the final reading.

Reading a Vernier Caliper

1. Note the number of large digits to the left of the 0 mark on the Vernier scale. Multiply this number by 1.
2. Note the number of small digits to the left of the 0 mark on the Vernier scale. Multiply this number by 0.1.
3. Note the number of small divisions past the small last small digit that the 0 mark is to the right of. Multiply this number by 0.025.
4. Note the line on the Vernier scale that coincides with a line on the fixed scale. (Only one line will match although some others may come close.) This line will indicate the number of thousandths.
5. Add all of these numbers together to obtain the reading.

Example 1

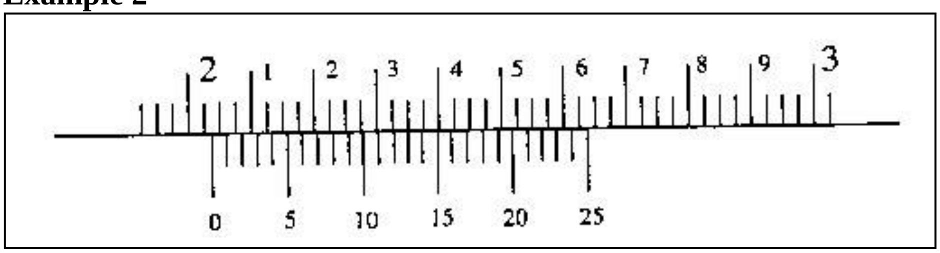


A Vernier caliper reading of 5.372 inches.

See the figure above. The steps are followed in order to get the reading:

0 mark is to the right of the large digit 5:	5 x 1 = 5.000
0 mark is to the right of the small digit 3:	3 x 0.1= 0.300
0 mark is to the right of 2 small divisions:	2 x 0.025 = 0.050
#22 line on the Vernier scale coincides with a line on the fixed scale:	22 x 0.001= <u>0.022</u>
Adding up these numbers, the total reading is:	5.372 inch

Example 2



A Vernier caliper reading of 2.038 inches

See the above figure. Again, the steps are followed in order to get the reading:

0 mark is to the right of the large digit 2:	2 x 1 = 2.000
0 mark is to the right of no small digits:	0 x 0.1 = 0.000
0 mark is to the right of 1 small divisions:	1 x 0.025 = 0.025
#13 line on the Vernier scale coincides with a line on the fixed scale:	13 x 0.001 = <u>0.013</u>
Adding up these numbers, the total reading is:	2.038 inch

Appendix B: Grinding

B.1 Tool Bit Grinding

The lathe tool bit, or cutter bit, is the part of the lathe that cuts the metal to be removed in order to bring the work piece to the desired size and shape. The tool bit is usually made of high-speed steel or tungsten carbide and is held in a lathe tool holder.

High-speed steel cutter bits are hardened and are ready for use when properly ground. Correct grinding of the lathe tool cutter bit is essential for good lathe work because a properly ground cutter bit will produce better results, will last longer, and will cut more readily than a tool bit which has been improperly ground. Tungsten carbide bits can be re-sharpened but require a silicon carbide grinding wheel to do so.

Correct grinding of the lathe tool cutter bit involves grinding the correct angles on the tool bit for the turning job that is to be done, and for the material that is to be turned.

Angles of the Lathe Tool Bit

Several definitions are provided below, along with diagrams to illustrate the terms.

Tool Angle: (also called the included angle, or angle of keenness) this is the included angle of the cutting edge formed by the top surface and the side surface of the cutting bit.

Different materials will require different tool angles. For machining soft steel, an angle of 61 degrees is the most efficient. For ordinary cast iron, the included angle should be approximately 71 degrees. However, for machining chilled iron or very hard grades of cast iron, the tool angle may be as great as 85 degrees.

Side Clearance: This is the angle between the side surface of the cutting bit and the vertical. Note that there is side clearance on both sides of a cutting bit. This permits the cutting edge to advance freely without the heel of the tool rubbing against the work piece.

The side clearance should be from 3 degrees to 10 degrees, depending on the amount used, and on the nature of the work.

Front Clearance: This is the angle between the front edge of the cutting bit and the line tangent to the work surface (usually the vertical). This permits the cutting edge to cut freely as the tool bit is fed into the work piece.

The front clearance should be from 3 degrees to 15 degrees, depending on the nature of the work, and on the height of the cutter bit.

Back Rake: The angle between the top surface of the cutting bit (at the tool tip) and the horizontal, as viewed from the side of the cutting bit.

Side Rake: The angle between the top surface of the cutting bit (at the tool tip) and the horizontal, as viewed from the front or back of the cutting bit.

See Figures 61 and 62. These figures illustrate the various clearance and rake angles of the tool bits.

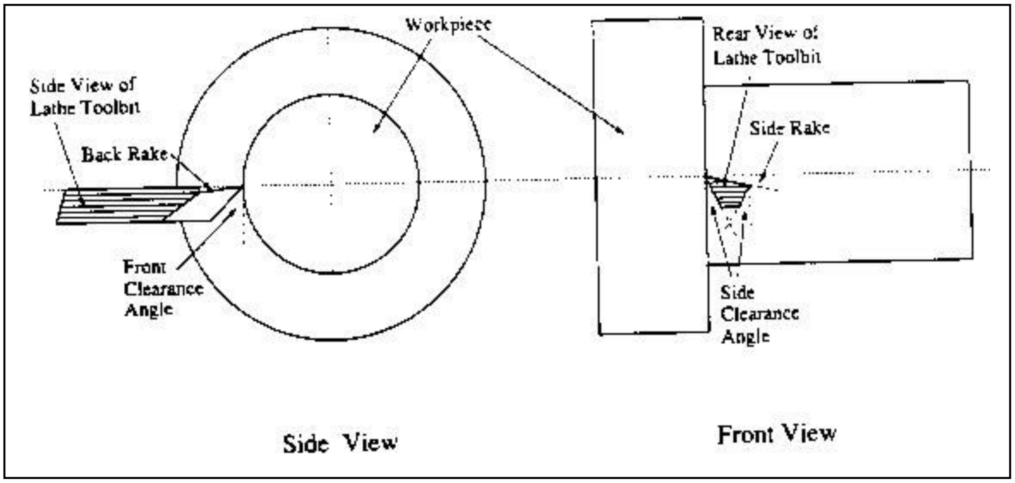


Figure 55: ***Rake and Clearance Angles of the Lathe Tool Bit.***

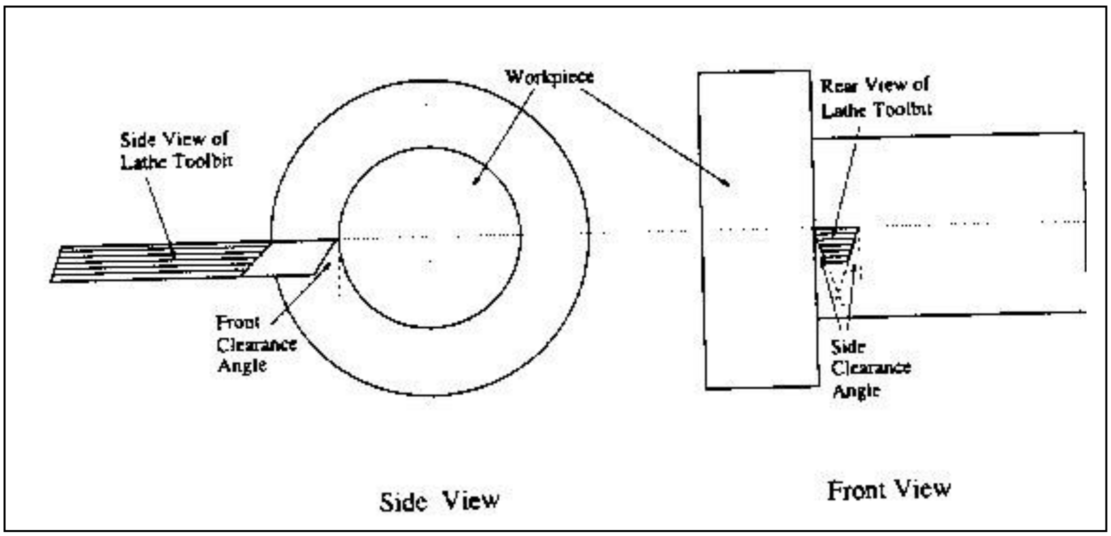


Figure 56: ***A Lathe Tool Bit with Zero Rake and Zero Back Rake.***

Tool Bit Gauge

A cutter bit grinding gauge is shown in Figure 63. It is a helpful tool for the beginner in grinding the correct angle on the various faces of the cutter bit. This gauge can easily be made of sheet metal, using the figure as a pattern, which is full size. If you have time, it would be a useful tool to construct.

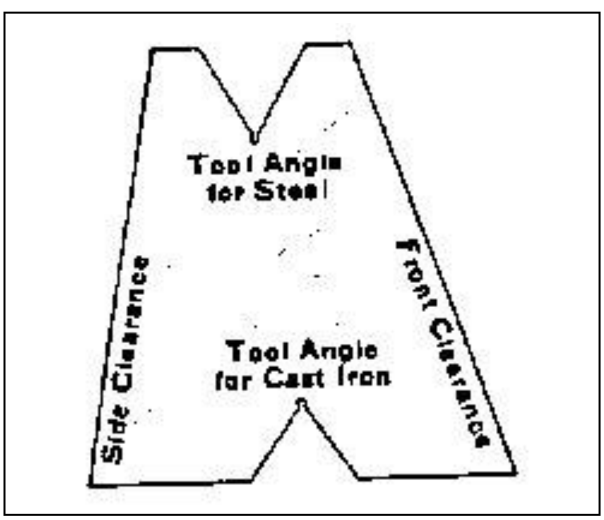


Figure 57: ***Pattern for Making a Tool Bit Grinding Gauge***

Grinding a Round-Nose Tool Bit

The following illustrations show each step in the grinding of a round-nose turning tool for general machine work. The various steps in grinding the cutter bit are as follows:

Grind the left side of the cutter bit, holding the cutter bit at the correct angle against the wheel to form the side clearance. See the figure below. Use a coarse grinding wheel to remove most of the metal, and then finish on the side of the fine-grinding wheel to produce a straight surface. (If ground on the periphery of a small diameter wheel, the cutting edge will be undercut, and it will not have the correct angle.) Dip the cutter bit into water frequently while grinding to prevent the bit from overheating.

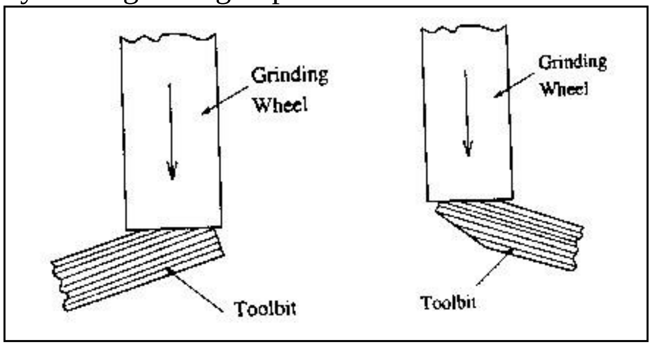


Figure 58 Grinding Side Clearance Angles

Grind the right side of the cutter bit, holding it at the required angle in order to form the right side clearance. See the above figure.

Grind the radius by rounding on the end of the cutter bit (see below). A small radius (approximately 1/32”) is preferable, as a larger radius may cause chatter. Hold the cutter bit lightly against the wheel and turn from side to side to produce the desired radius. Be careful to hold the cutter bit at the correct angle to obtain the proper front clearance.

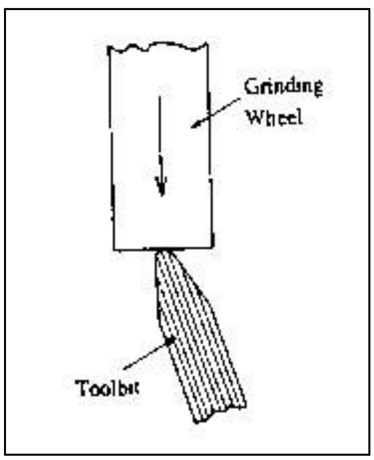


Figure 59 grinding Tool Bit Tip Radius

Hold the cutter bit at an angle as shown while grinding the radius on the end of the cutter bit, in order to form the required front clearance. See below.

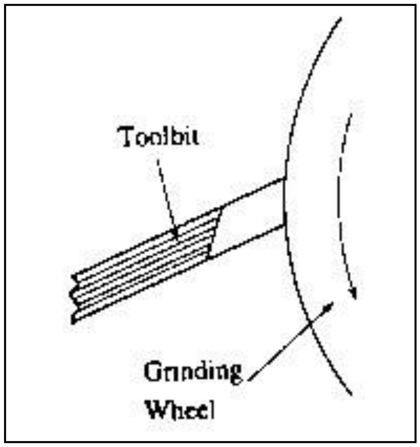


Figure 60 Grinding Front Clearance Angle

Grind the top of the cutter bit, holding the cutter bit at the required angles in order to form the necessary side rake and back rake.

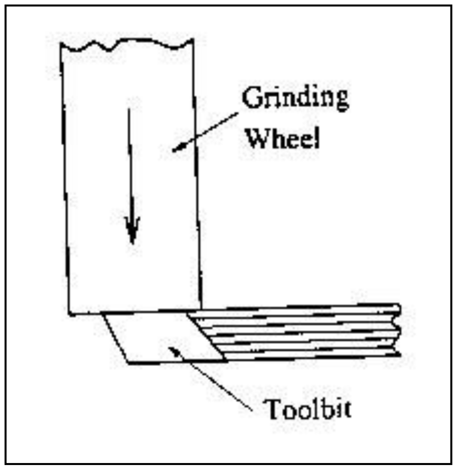


Figure 61 Grinding Side and Back Rake Angles

Types of Lathe Tool Bits

The illustrations in Figure 65 show the most popular shapes of ground lathe tool cutter bits and their applications.

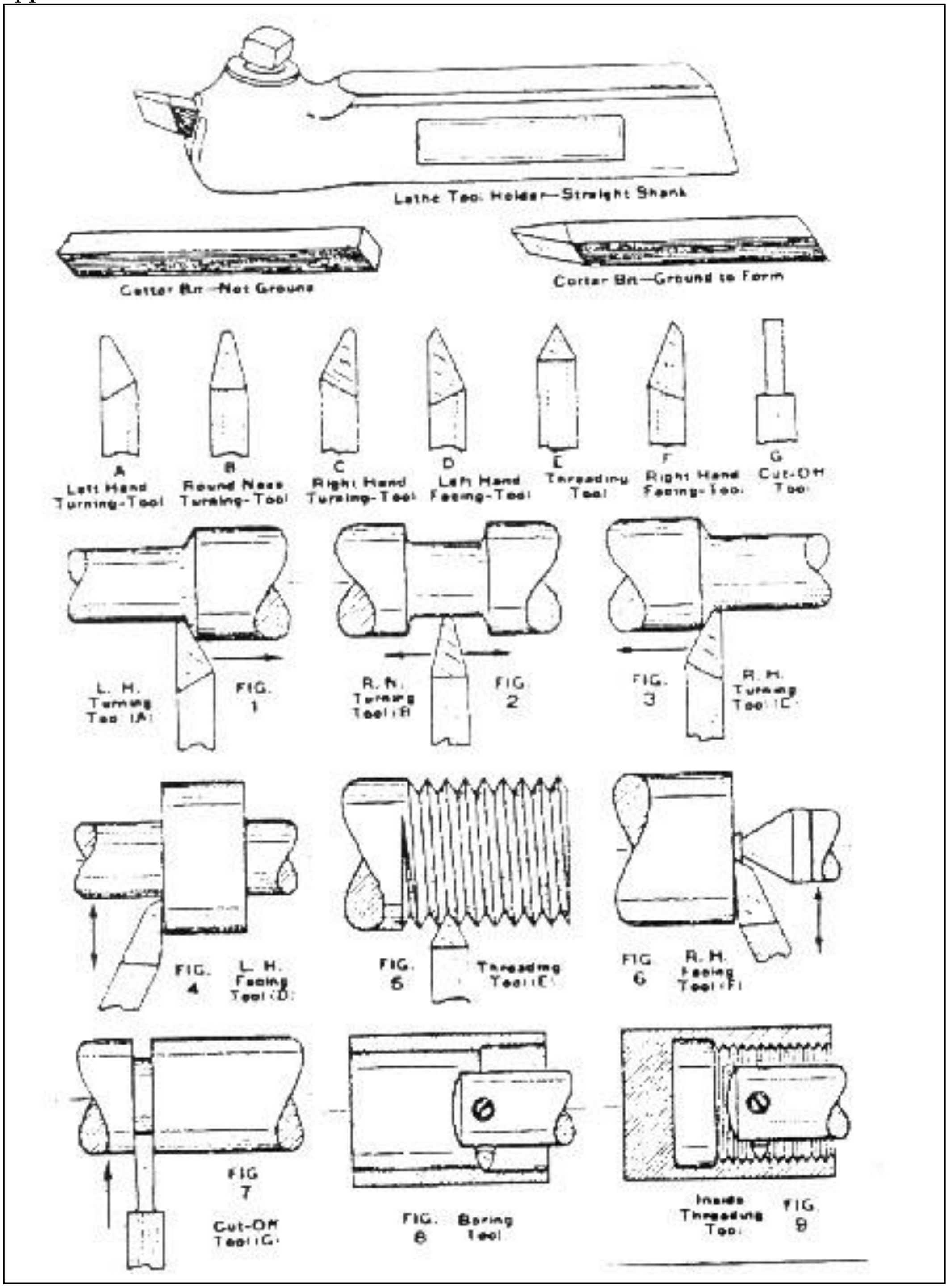


Figure 62: ***Various Types of Lathe Tool Bits.***

B.2 Surface Grinding

Dressing the Wheel

As a grinder is operated the wheel will gradually become impregnated with small steel filings. It may begin to develop irregularities in its surface geometry from hitting one or more high spots in steel, causing one part of the wheel to wear a bit more than the rest of the wheel. When this happens it becomes necessary to remove the outer layer of the wheel and square up the edge. This is referred to as 'dressing' the grinding wheel. This is done in the following manner.

- Mount a small industrial diamond in a steel holder, and then mount the holder on a magnetic chuck.
- Next, start the grinding wheel and slowly move it down until it just touches the tip of the diamond.

CAUTION: Never stand in line with the wheel because the material of the wheel is quite brittle. If there is any accident causing wheel breakage the wheel will shatter and the pieces will fly along paths in line with the wheel.

- Slowly move the grinding wheel back and forth to clean the entire surface of the grinding wheel face. Feed the grinding wheel down in increments of 0.001" per pass.
- Repeat this at least four times or until the wheel's face is clean. The face will be of uniform color with no dark areas present at the edges of the wheel's face.
- Once the wheel is dressed, shut it off and wait until is stops rotating. Then remove the dressing assembly.

Grinding the Work Piece
Be sure that no small burrs remain on the edges of the work piece you intend to grind. If there are any burrs, remove them with a file. This will enable maximum surface area to be in contact with the magnetic chuck so as to provide the best grip on the steel.

Once the work piece is secured to the magnetic chuck, start the coolant pump, but do not turn on the valve for the coolant flow. Now start the grinding wheel. Slowly bring the wheel down until it just barely touches the steel. Now turn on the coolant flow and make a series of cutting passes over the surface of the steel until you have covered about ½ to 1/3 of the surface. Now lower the wheel another 0.001 inches and go over the same surface again. Repeat this process until the surface is bright and uniform. For the final pass, use a cut of just 0.005 inches.

In order to produce a really good finish, you would at this point dress the wheel again and take off only 0.0002 inches then 0.0001 inches per pass. Note that too fast a traverse across the piece of 0.003 inches or more metal removal may cause the wheel to stall or burn the work. The wheel may also tend to stall if it needs to be dressed again. Too deep a cut may result in the work piece being thrown from the magnetic chuck.

Appendix C: Welding

The primary objective of any welding operation is to produce a weld that has the same properties as the base metal. While the perfect weld can never theoretically be achieved, in practice we can come very close and impurities in the weld can be kept to a minimum if proper care is taken.

In terms of impurities in welding, the biggest obstacle, which must be overcome, next to surface dirt and grime, is the atmosphere. When the metal is heated to its molten state, the molten puddle absorbs oxygen and nitrogen from the atmosphere, and upon cooling the metal becomes weak and porous. Since the purpose of welding is to join two pieces of metal together or to strengthen an existing piece this is obviously an unwanted condition, and contamination by the atmosphere should be controlled.

There are several different ways of accomplishing this task and each has its own application. Usually some sort of flux is used when welding, which when burned in addition to cleaning the base metal surface, provides a gaseous shield from the atmosphere.

There are several types of welding techniques as well. While not all of them will be discussed here, it is the purpose of this article to give the student a basic knowledge of the most prevalent welding techniques in use today. Three techniques will be considered: Electric Arc welding, Oxyacetylene Brazing, and Gas Metal Arc Welding (GMAW).

C.1 Electric Arc Welding

The heat generated for Electric Arc Welding (sometimes called stick welding) comes from an arc which develops when electricity jumps across an air gap between the end of an electrode and the base metal. The air gap creates a high resistance to current flow, which generates an intense amount of heat capable of producing temperatures anywhere in the range of 6000 degrees F to 10,000 degrees F (approximately 3300^o C to 5500^o C). Either provides welding current in AC or DC source.

If a direct current source is used, welding may be performed with either straight or reverse polarity. Straight polarity simply means that the electrode is connected to the negative (-) terminal of the current source and the base metal is connected to the positive (+). For terminal reverse polarity, the opposite is true. The reason for the difference is that electrons flow from the negative terminal to the positive; depending on the type of electrode used and the weld penetration, it may be desirable to have the base metal hotter than the electrode or vice versa. Tripping a switch on the welding machine itself can usually change polarity.

The Electrode

The electrode is a coated metal wire having the same composition as the base metal. When an arc is formed between the electrode and the base metal, both the electrode and the base metal are melted. The melted electrode flows into the molten base metal and becomes a part of it. Most electrodes are designed for either AC or DC welding, but a few work equally well for both.

There are two kinds of electrodes: bare and shielded. Originally, a bare electrode was just an uncoated metal rod, but today they have a light coating. Their use is limited however, because they have a tendency to produce brittle welds with low strength, and they are difficult to weld with. A shielded electrode has a heavy coating made of various substances, each of which performs a particular function in the welding process, including:

- To act as a cleansing and de-oxidizing agent in the molten metal.
- To release an inert gas to protect the weld from oxygen and nitrogen in the atmosphere.
- To form a slag over the deposited metal, which further protects the weld from the atmosphere until it has cooled sufficiently to prevent contamination.
- To provide easier arc starting and stabilize the arc.
- To permit better penetration of the weld into the base metal.

A coated electrode in the process of welding is shown in Figure 65. As a rule of thumb, the electrode never has a larger diameter than the thickness of the metal to be welded.

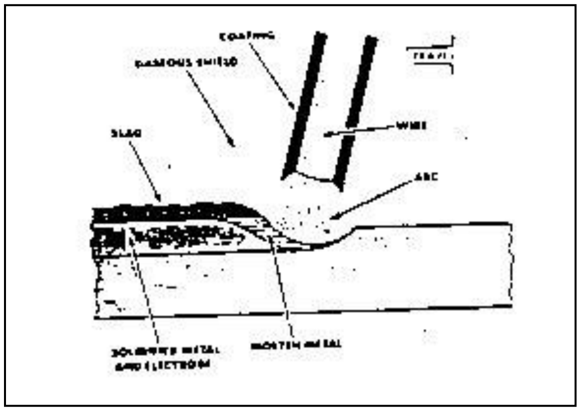


Figure 63: ***The Arc Welding Process.***

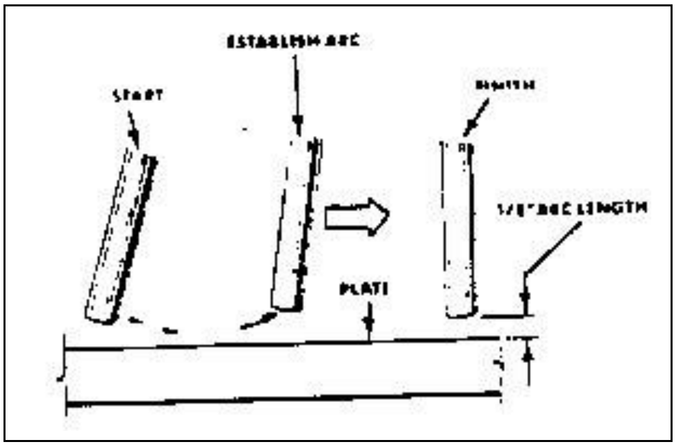


Figure 64: ***Proper Arc Gap.***

For the beginning welder, the easiest way to strike an arc is by using a scratching method similar to striking a match. Upon contact with the metal, the electrode should be raised to a height approximately equal to the diameter of the electrode (Figure 66). Otherwise, the electrode will stick to the metal and within a short time weld itself to it.

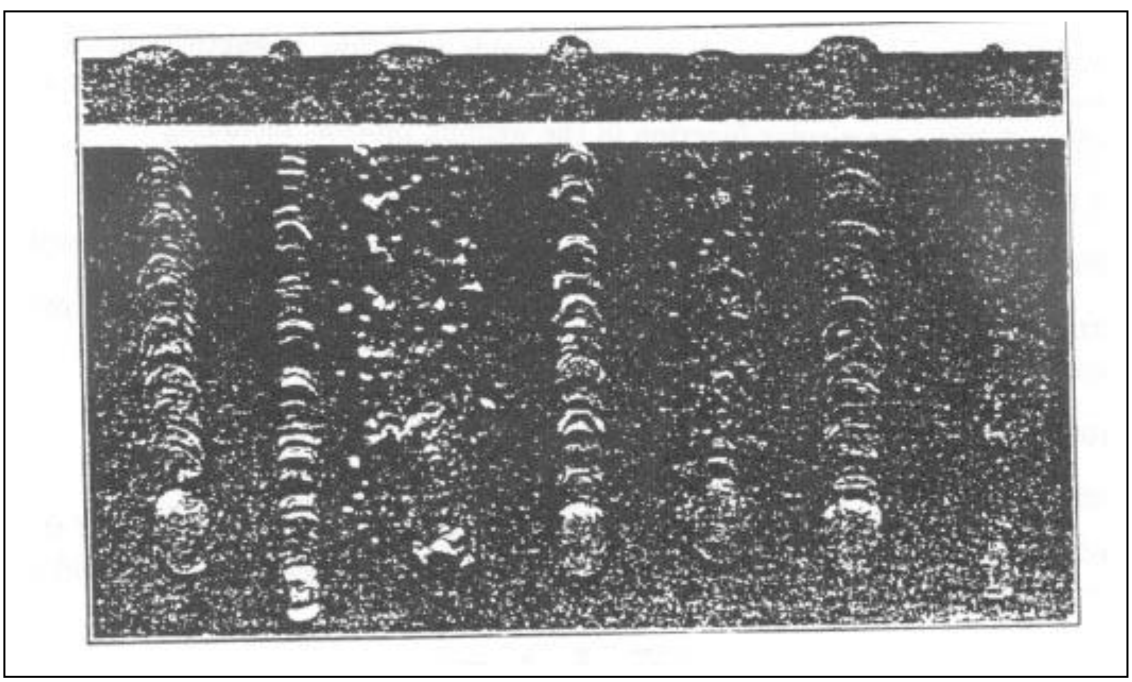


Figure 65: ***Examples of Proper and Improper Weld Beads (From Left to Right: Current, Voltage and Speed Correct; Current too low; Current too high; Voltage too low; Voltage too high; Speed too slow; Speed too fast)***

Running a Bead
To run a continuous bead on a flat surface, the electrode should be held at an angle of approximately 15^{o} from the vertical. After striking an arc, the electrode should be moved from left to right (for a right handed person) slowly enough to allow the deposited metal to penetrate into the base metal. A slight weaving or circular motion may help to distribute the deposited metal more evenly. At the same time, the electrode should be continuously fed towards the molten pool in order to maintain the proper arc length. The proper arc length will have a constant crackling or frying noise. An arc that is too short will have a louder, popping noise. Too long an arc will have somewhat of a humming sound. Examples of properly and improperly formed beads are shown in Figure 67.

When the weld is complete, the slag should be removed by striking the weld with a chipping hammer in a direction away from the body, eyes and face. Brushing with a stiff wire brush should follow chipping. Always wear protective eye shields (goggles) when removing slag from the weld bead.

C.2 Brazing
Brazing as defined by the American Welding Society (AWS) is a process in which the base metal is heated to temperatures above 800 degrees F, and which uses a non-ferrous filler metal having a melting point below that of the base metal. Only the filler metal is melted. Coalescence of the metals is produced by capillary action between the closely fitted weld joint.

There are two major advantages of brazing. The first is that many dissimilar metals can be joined together. The second is that the mechanical properties of the base metal are changed very little since only the filler metal is melted. Brazed joints, however, although most have a relatively high tensile strength, do not possess the full strength properties of other conventional welding techniques. For this reason, joint design in brazing is very important.

The two basic joints used for brazing are the butt and lap. The lap is more common because if offers the greatest strength due to the increased surface area. For maximum efficiency, it is recommended that the overlap be at least 3 times the thickness of the thinnest member. The only disadvantage of the lap joint is that the metal thickness at the joint is increased.

The heat required for brazing can be applied in many ways, but for most manual application, a gas torch is considered most practical. For oxyacetylene brazing, the flame should be set slightly carbonizing, and only the outer envelope of the flame and not the inner cone should be used. The filler metal should be chosen so that its melting temperature is lower than that of the base metal. 50 degrees F or lower is usually sufficient. At the same time, the lowest possible brazing temperatures are preferred because of high temperature effects on the base metal such as grain growth, warpage and hardness reduction.

Surface preparation is very important and is a determining factor in the resulting strength of the brazed joint. The base metal must be clean and free from surface oxides because capillary action is only possible when surfaces are completely free of foreign substances. Dirt, oil and grease can be removed by immersing the metal in some commercial cleaning solvent and sanding, grinding or wire brushing can remove surface oxides. After the surfaces have been cleaned, flux should be applied to both the base metal and the filler metal. The work (joint) is then preheated by placing the torch over the entire surface to bring it up to a uniform temperature. When the flux becomes completely fluid and the surface becomes hot enough, the filler metal should be touched to the joint and applied until it flows completely through the joint. Do not apply the inner cone of the flame directly to the work and make sure the flame is slightly reducing. When brazing is completed, removing all flux residues should clean the joint, so that corrosion will not set in.

Lighting the Torch

The oxygen and acetylene cylinder valves should be opened slowly so that the cylinder gas pressures are shown by the high pressure gauges. With the torch acetylene needle valve closed and the torch oxygen valve opened, the oxygen regulator should be adjusted until the low pressure gauge indicates the desired pressure (20 psi). With both torch needle valves closed, the acetylene should now be adjusted until the low pressure gauge indicates the desired pressure (5 psi). Working pressures are dependent on the torch size and may vary.

CAUTION: Acetylene gas is a highly unstable compound, which tends to dissociate when subjected to pressures greater than 15 psi. This can cause a serious explosion.

Next the acetylene needle valve should be opened approximately three quarters of a turn and with a spark-lighter held about one inch away from the tip of the acetylene should then be ignited. If not enough acetylene is flowing; the flames will produce a lot of smoke. The acetylene needle valve should be adjusted until the flame produces a gap of about ¼ inch between the tip of the torch and the flame. This will give the correct working pressure regardless of the tip size being used.

CAUTION: Never light the torch with a match and always point the torch tip downward to protect the eyes and face.

Adjusting the Flame

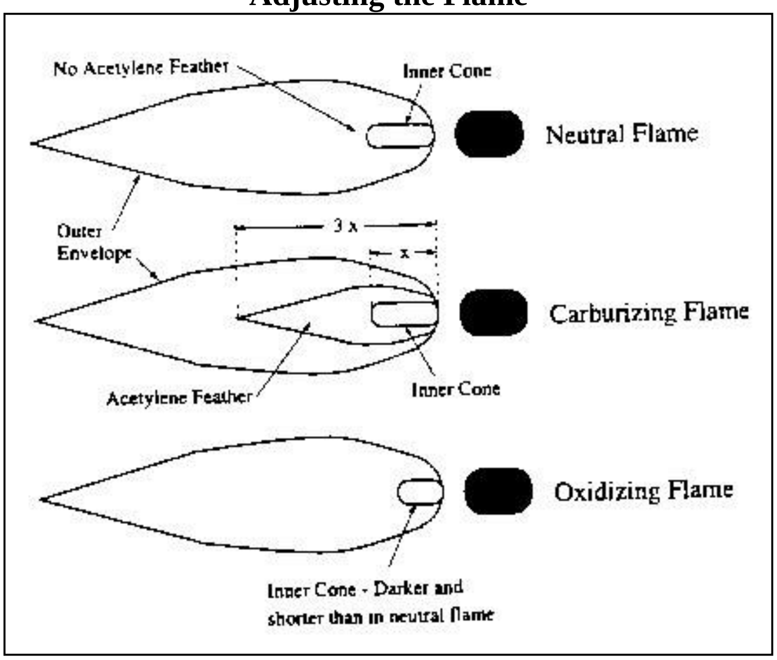


Figure 66: ***Examples of the Different Types of Flames.***

With the acetylene still burning, the oxygen needle valve should be adjusted until the yellow feather in the acetylene flame just disappears into the end of the inner cone. This produces a neutral flame due to the equal amounts of acetylene and oxygen burning. The neutral flame is used for most welding operations.

Any variation from the one-to-one mixture of gases will cause the flame characteristics to change. When there is excess acetylene has present, the flame is called carburizing or reducing. This can be identified by the existence of three flame zones instead of two. The end of the white inner cone will no longer be well defined and an intermediate, almost colorless zone will surround it with a feathery edge (the acetylene feather) in addition to the bluish outer envelope. Welding supply manufacturers will often specify the excess amount of acetylene in terms of the length of the inner cone (2x, etc.). When there is excess oxygen present, the flame is referred to as oxidizing. This flame resembles the neutral flame but has a shorter and more pointed inner cone with an almost purple color (Figure 69).

Shutting Off the Torch

When the weld has been completed, the following sequence of steps should be followed in order to extinguish the flame.

1. Close the oxygen valve first. This will immediately result in the pure yellow acetylene flame reappearing.
2. Close the acetylene needle valve. This will extinguish the flame.
3. If the entire brazing unit is to be shut down, close both the acetylene and oxygen cylinder valves then reopen the needle valves to exhaust the pressure on the working gauges. The second part of the preceding sentence is not critical and may be considered optional. Re-close the needle valves.

C.3 Gas Metal Arc Welding

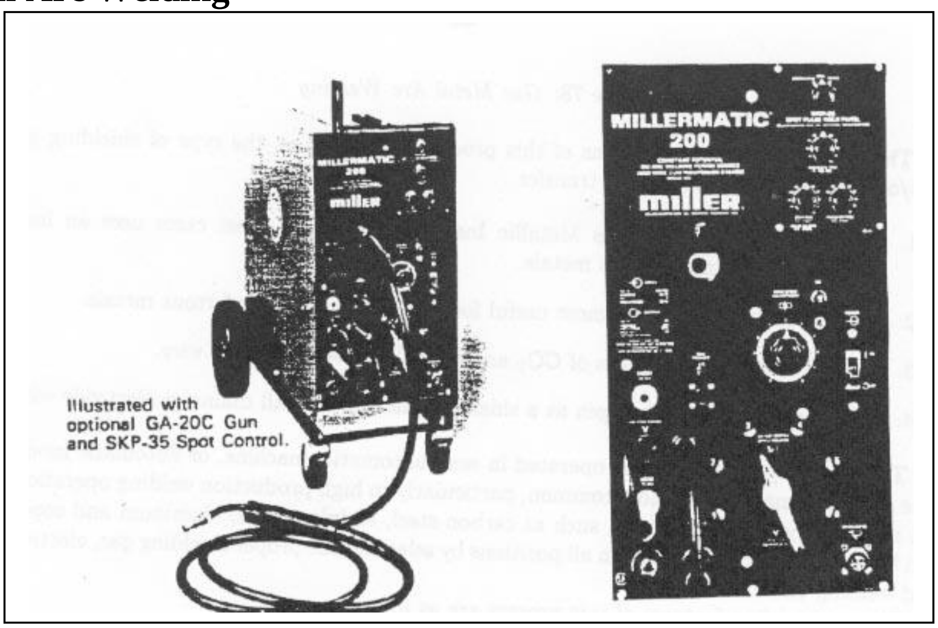


Figure 67: ***Front View of GMAW Welding Machine.***

The gas metal arc welding has grown more rapidly than any other type of welding in the last 20 years. GMAW has now become the major production welding process in most of the industrial applications around the world today.

In GMA welding (Figure 74), an electric arc is struck between the work piece and consumable wire electrode that is fed continuously through the torch at controlled speeds. Shielding gas is fed simultaneously through the torch into the weld zone, surrounding the wire and protecting the weld from the contaminating effects of the atmosphere.

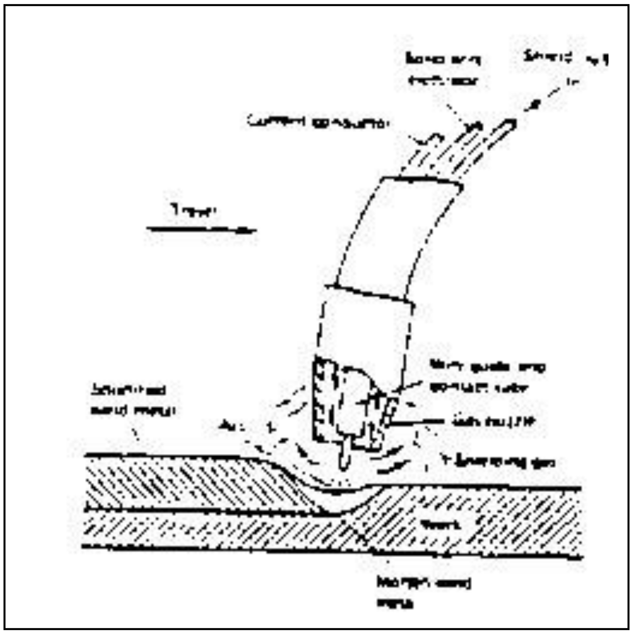


Figure 68: ***Gas Metal Arc Welding.***

There are four major variations of this process depending on the type of shielding gas and/or the type of electrode wire transfer.

1. GMAW, formerly known as Metallic Inert Gas welding or MIG welding. Will in most cases use an inert shielding gas on non-ferrous metals.
2. Short circulating transfer is most useful for all position work on ferrous metals.
3. GMAW uses a shielding gas of CO2 and a small ferrous electrode wire on ferrous base metals..
4. Spray arc-using argon/oxygen as a shielding gas and a small diameter electrode wire.

The GMAW process may be operated in semi-automatic or automatic modes. The semi-automatic is the most common, particularly in high production welding operations. All industrially common metals, such as carbon steel, stainless steel, aluminum and copper can be welded with this process in all positions by selecting the proper shielding gas, electrode and welding condition.

The outstanding features of this process are as follows:

1. It is able to make good quality welds on almost every metal or alloy used in industry today.
2. Minimum post weld cleaning is necessary.
3. The arc and molten puddle are clearly visible to the welder.
4. Welding is possible in all positions depending on various process conditions.
5. High welding deposition rates and speeds make the process economical.
6. There is no heavy slag produced, therefore less possibility of slag inclusions. In some conditions when welding carbon steel, a surface discoloration takes place that appears as a light slag.

A typical GMAW setup (Figure 73) consists of a constant DC power source, an electrode wire drive unit and control box, shielding gas cylinders with regulatory flow meter, power and gas cables and hoses, and the welding gun. The filler metal or wire can be supplied to the weld either from an external spool (25 pound, 40 pound) or from a 5-pound spool located inside the gun. The consumable wire electrode is fed from a spool through a torch or welding gun. The wire passes through a contact tube in the gun where it picks up the welding current. This current level is transmitted to the wire feed motor, which determines the wire feed speed. Once the arc has been established, the filler metal is transferred across the arc in the form of fine spray or plasma. The arc can be regulated to give a wide variation of forms, depending upon the kind of metal being welded, the thickness of the work piece and the type of weld desired.

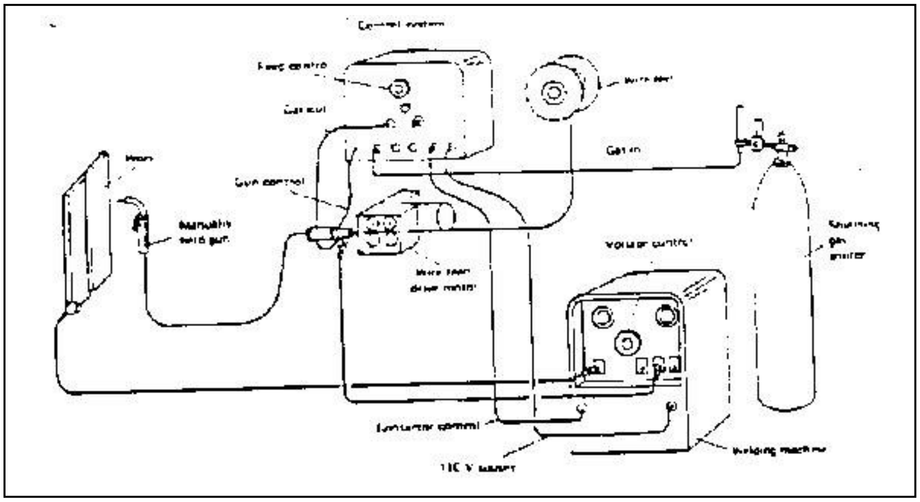


Figure 69 : ***Common Setup for the GMAW Process***

Electrode Wires

One of the most important factors to consider in GMA welding is the correct filler wire. The filler wire in combination with the shielding gas will produce a weld bead that must have the proper characteristics of the structure being welded. There are five major factors that influence the choice of filler wire to be used:

1. Chemical composition of the metal to be welded.
2. Mechanical properties of the metal to be welded.
3. Type of shielding gas to be used.
4. Application requirements of the finished product.
5. Type of weld joint design.

After many years of development, wire electrodes are now manufactured that continually produce excellent results on a number of metals and joint designs. Although there is no industry-wide set of specifications, most electrode wires conform to an American Welding Society standard.

Ferrous Metals

When GMA welding carbon steels, the primary function of the alloying additions is to control the de-oxidation of the weld puddle and help determine the weld mechanical properties. The removal of oxygen from the puddle eliminates the chance of weld metal porosity and causes the formation of a fine slag or glass on the surface of the bead.

Alloying Elements Added to GMAW Wires

Silicon (Si): Most commonly used as a de-oxidizer, most wires contain 0.40 to 1.00 percent Si depending upon their intended use. Increasing amounts of Si will increase the strength of the weld with a small decrease in ductility and toughness.

Manganese (Mn): Most often used as a de-oxidizer and strengthener in amounts of 1.00 to 2.00 percent in mild steel wires. Mn will increase the weld metal strength to an even greater degree than Si.

Aluminum (Al), Titanium (Ti) and Zirconium (Zr): These elements are all strong de-oxidizers. Very small additions, not more than 0.20 percent combined will also cause an increase in weld metal strength.

Carbon (C): Carbon, more than any other element influences the structure and mechanical properties of the weld. In most GMAW wires, the carbon content usually ranges between 0.05 and 0.12 percent. Carbon content of 0.12 percent or less provides the necessary weld strength without affecting ductility, toughness or porosity.

Others: Nickel, chromium and molybdenum are often added to improve the mechanical and corrosion resistant properties of the finished weld.

Non-Ferrous Metals
The primary elements used in aluminum alloy wire are magnesium, manganese, zinc, silicon, and copper. The major reason for adding these elements is to increase the strength of the pure aluminum. Most aluminum-alloy wires contain a number of elements that will improve the weld properties, increase corrosion resistance and increase weld ability. The most popular general-purpose electrode wires are magnesium 5356 and silicon 4043 aluminum alloys.

Shielding Gases
The purpose of the shielding gas is to displace the air in the weld zone, thereby preventing contamination of the molten weld metal. In general, nitrogen, oxygen and water vapor are responsible for most of the contamination in the weld zone.

Nitrogen trapped in the weld causes cracking and reduction in ductility and impact strength. Excess oxygen in a weld combines with iron to form iron oxide, resulting in reducing physical and mechanical properties. Trapped oxygen will cause porosity and inclusions in most weld beads.

To avoid problems of contamination of the weld puddle, three main gases or combinations are used to shield the GMAW process: argon, helium and carbon dioxide. In some cases, small amounts of oxygen are combined with one of the major gases. Compensation for the oxidizing tendencies of CO_2 and oxygen are made by special wire electrode formulas. Argon, helium and carbon dioxide can be used alone or in combination to provide defect-free welds in a variety of materials and applications.

The shielding gas will affect the following aspects of the welding operation and the final weld produced:

1. Arc characteristics.
2. Mode of metal transfer.
3. Penetration and weld bead profile.
4. Speed of travel.
5. Tendency of undercutting.
6. Cleaning action.
7. Volume of spatter.

Argon and Helium Shielding Gases
Argon and helium are inert gases and are primarily used in the welding of non-ferrous metals, stainless steel and low alloy steels. The major difference between argon and helium is density, with argon being much heavier, thus more effective in shielding the weld zone in a flat position weld. Helium, because it is lighter in density, would require two or three times the flow rate to provide the same protection to a weld: therefore, its most economical use is in overhead welds.

Helium possesses a higher thermal conductivity than argon and therefore the bead contour tends to be deeper and broader, which tends to produce a narrow bead with less penetration. At a given wire feed speed, the voltage of the argon will be noticeably less than that of the helium arc. The arc will remain more stable with the argon shield and thus fewer spatters and a better weld bead appearance are produced.

Oxygen and CO_2 Additions to Argon and Helium
Argon and helium produce excellent results with non-ferrous metals but less than satisfactory welds with ferrous metals. Generally 3 % oxygen or 9% CO_2 will give good results and compensates for such

variables as parent metal surface conditions, joint design and position welding technique and base metal composition.

Carbon Dioxide
Carbon Dioxide (CO2) is used in its pure form for GMAW of carbon and low alloy steels. Higher welding speeds, greater joint penetration and lower costs are the major advantages that have made CO2 so popular for most ferrous GMA welding.
With the CO2 shield, metal transfer is either of the short circulating or globular mode, which is quite harsh and produces a much higher spatter count than an argon shield. When compared to argon, CO2 produces excellent penetration with rougher bead surface and less "washing" at the extremity of the weld bead. Sound weld deposits may be achieved, but mechanical properties may be adversely affected due to the oxidizing nature of the arc.

Power Source
Special welding machines, direct current, constant potential, are most commonly used for the GMAW process. This type of machine provides a relatively constant voltage to the arc during welding. A constant potential machine quickly increases or decreases the current (wire burn-off rate) depending on the arc length change. The wire burn-off rate will adjust automatically to the original arc length.

Before welding begins, the operator may set the arc length by making the proper adjustment of the output voltage. At the same time, the wire-feed speed the operator selects prior to welding determines the arc current. Both the arc voltage and current can change over a wide range before the arc length will be altered to cause stubbing or burn-back into the guide tube.

The self-correcting arc length feature of the constant voltage power supply is the key to producing stable welding conditions. Additional characteristics are also built in to control the arc heat, spatter, and other variables.

Arc voltage is the voltage between the end of the contact tip and the work piece; it's not directly read on any voltmeter. As stated, the welding voltage is the arc length, (electrode stick out) which has a very important effect on the type of metal transfer desired. Short arc welding requires relatively low voltages whole spray arc welding requires much higher voltages. To provide the best operation, arc voltage must increase as the electrode wire speed increases. The constant potential power source provides a volt/amperage curve that is essentially flat. The voltage remains relatively constant while the current changes. The current can rise to very high values; extreme spatter will occur at the arc. The spatter is caused by the pinching off of liquid globules at the tip of the square of the current flowing through any conductor. The higher the current rises, the more violent the pinch effect at the arc and the greater the spatter. Two controls that are used to overcome the spatter, create a flatter puddle, and provide a smoother flowing arc, are slope and inductance.

Slope refers to the reduction in output voltage with increasing current values. In fact, a constant voltage power supply produces a decreasing voltage and an increasing amperage curve. In general, the greater the slope, the lower the short-circuit current during short arc welding and the lower the spatter. The short circuit must be high enough to melt off the molten drops from the electrode wire.

NOTE: Constant voltage power sources have extremely high short circuit current, which could reach several thousand amperes. For this reason, manufacturers of CV power supplies have recommended that stick electrodes never be used with constant potential welding machines.

Slope determines the maximum current attainable during a short circuit transfer at the arc during "short arc" welding. If the current is controlled, the pinch force is controlled as well the amount of spatter at the weld. Slope also will provide a smoother arc when welding aluminum using the GMAW process.

Inductance controls the rate of current rise. Increasing the inductance causes current to rise more slowly; thus, it takes longer for the current to rise to a given value. The prolonged time assists in making the puddle more fluid, which produces a flatter and smoother weld. Inductance also decreases the frequency of short-circuiting. This reduces spatter because the short circuit has time to clear.

Wire Feed and Control Units

The purpose of the wire-feeding unit is to automatically drive a small-diameter wire spool or coil through the cable assembly to the gun and to the welding arc. The constant potential power supply requires a constant speed of wire feed, which may be adjusted for different welding conditions. Many wire feed arrangements from very small units have the wire spool contained in the welding gun, whereas large heavy-duty units use heavy motors and rheostats to drive a large coil of wire. All wire feed units allow for motor speed adjustment, which in turn controls the wire feed speed and the welding current. Built in to all wire feeders are controls for starting and stopping the wire as well as a welding power contractor and gas control valve energizer. Wire feeder and control units are usually sold as part of a total unit with the cables, hoses, and gun or torch.

Welding Torches and Guns

Since the GMAW process may be operated in semi-automatic, machine, and automatic modes, the welding torch or gun will vary considerably from one process to another. The primary purpose of the torch or gun is to deliver the electrode wire and shielding gas from the wire feeder and the welding current from the power supply to the welding zone.

The gun must be well-built, light and designed so nozzles and guide tubes can be easily replaced or adjusted. Many different types of GMAW guns and torches are available on the market today. Lightweight guns are used mainly for the fabrication of light gauge material in out-of-position. Heavy units are used for many automatic operations where the operator controls a large piece of equipment that holds the gun or torch in a particular position.

Cable Assemblies

Most cable assemblies are sold as a unit with the GMAW torch or gun. These assemblies come in various lengths and weights with all the cables, liners, and hoses forming a single unit or separated into individual parts. The major parts of the unit include a power cable, and electrode wire liner, and a gas hose. The power cable must match the torch or gun and therefore they are sold as a unit.

Metal Transfers

Metal transfers, when welding with a consumable electrode, are generally divided into three groups: spray transfer, globular transfer, and short-circuiting transfer. The type of metal transfer will depend on such things as electrode wire size, shielding gas, welding current, and arc voltage. When using the GMAW process, spray transfer is experienced only when argon or argon plus oxygen gas mixtures are used. When CO2, or argon plus CO2 gas mixtures are used, a globular or short circuiting transfer may be obtained.

The metal transfer and the finished bead depend on the following variables: voltage and amperage setting, diameter of the electrode wire, electrode composition, electrode extension, shielding gas, and power supply characteristics.

The operator depending upon the welding conditions sets the rate of feed of the welding wire. Different arc characteristics can be obtained by varying the proportions of argon and other gas combinations being used. The power sources vary with the material being welded and the type of equipment being used.

Arc Power and Polarity

For the vast majority of GMAW applications, direct current reverse polarity (electrode positive) is used. The setup gives a stable arc, smooth metal transfer, low spatter loss, and good bead characteristics for the entire range of welding currents used. Direct current straight polarity is seldom used in GMAW applications as the arc becomes very unstable. Alternating current is never used with the GMAW process.

C.4 Safety

The gas metal arc welding process produces twice as much radiant energy as do coated electrodes of equal current. It is therefore imperative for the welder to protect him/herself in the following manner:

1. Wear a darker shade of lens that still permits adequate vision, usually lens shade number 11 or 12.
2. Cover exposed skin completely while welding to avoid being burned by ultraviolet rays.
3. Wear dark clothing to reduce the amount of radiation reflected on the face underneath the helmet.
4. Avoid using any common degreasers or any substance containing trichloroethylene vapor on the metal to be welded. Experiments have shown that radiation from this welding process decomposes such substances very rapidly and that when decomposed they give off highly concentrated fumes which can be toxic.

Appendix D: Plastic Welding

Although general procedures for welding plastics are similar to welding metals, there is a definite technique, which must be learned before successful plastic welds can be made. Since the quality of the finished weld is directly proportional to the skill of the welder, the beginner should first become familiar with thermoplastics and their properties. Only after this familiarity is gained should the beginner attempt to acquaint himself with the operation and maintenance of today's equipment and the modern techniques of welding,

Preparation of Materials

In common with all successful endeavors, sound plastic welds start with proper planning and preparation. Size and shape of the welding project; type, shape and thickness of the material to be welded; stresses which the completed project will be subjected to; position of materials for welding; unusual circumstances surrounding the welding; and many other factors influence the selection of the type of weld to be used. This is often determined by the project engineer, job supervisor, or by the welding operator according to his ability and experience.

D.1 Types of Welds

Although welding procedure differs, both metal welders and plastic welders use the same basic types of welds. Some of the most commonly used plastic welds are shown below in Figure 73.

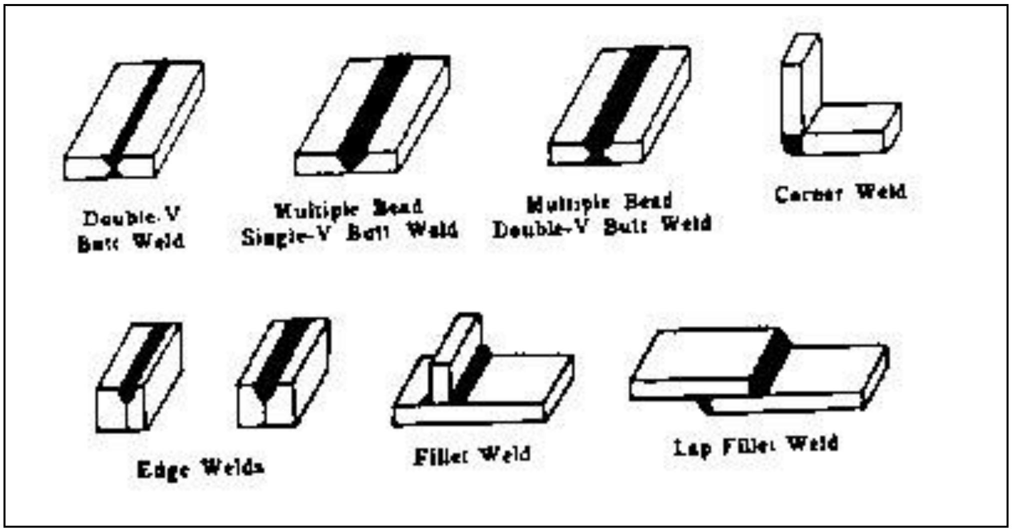


Figure 70: ***Types of Plastic Welds***

Butt Welds, Edge Welds, and Corner Welds
To prepare material for the welding of butt joints (butt welds), edge welds and certain types of corner welds, bevel the edge of both pieces, using a saw, jointer, sander or block plane. Do not bevel to a feather edge; leave about 1/32" flat. The two pieces when placed together should now have a "V" groove with an angle of 60 degrees. Most shops will operate at 50-55 degrees to cut the number of passes.

In order to create a good bond in the finished weld, the two surfaces to be joined must be free of dirt, dust, oil, moisture and loose particle of material. Wipe edges using a clean cloth. Do not use solvents to clean beveled edges since they tend to soften the edges causing a poor finished weld.

Place the pieces to be welding together. If pieces are to be tacked together with a tacking rod, leave a root gap of 1/64" – 1/32" between the pieces. Do not leave a root gap when using the tacking tip for the tacking operation.

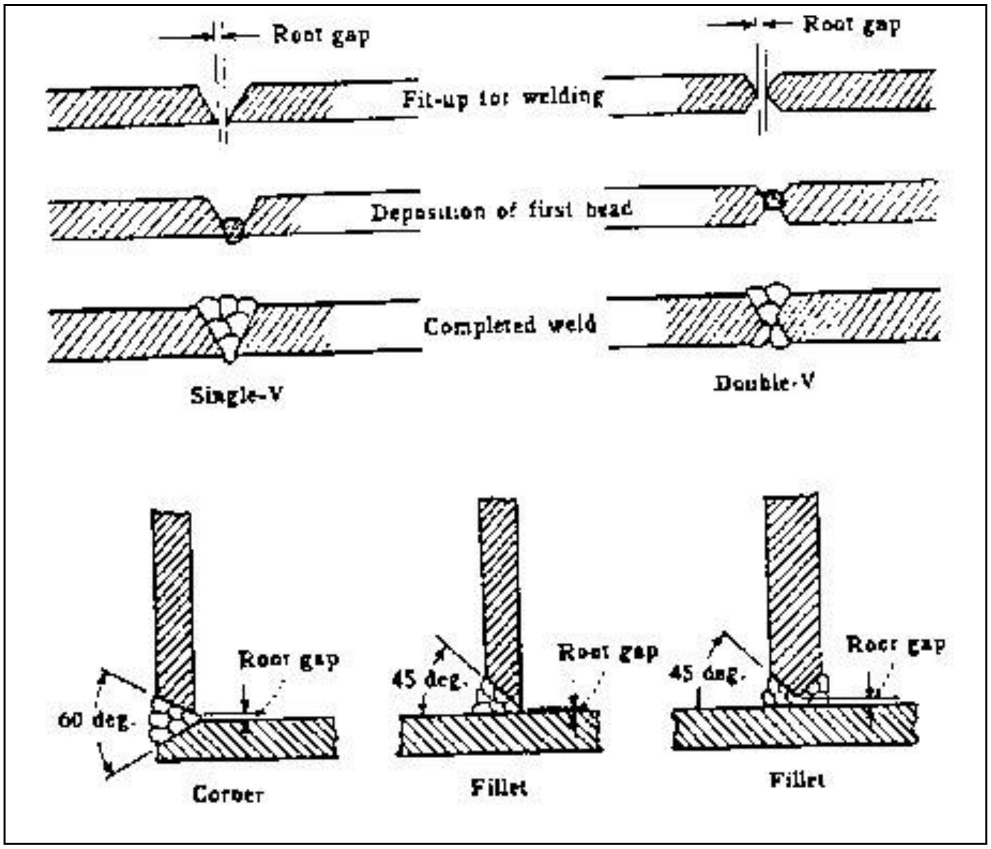


Figure 71: ***Beveling and Preparation***

Lap Welds: Lap welding requires little preliminary preparation since the pieces to be joined are placed atop the other. As with the welds described above, surfaces must be clean and free of all dirt, dust, oil, moisture and loose particles of material. To hold pieces firmly together for welding, "C" clamps may be used or a tack weld applied.

Fillet Welds: Fillet welds and lap fillet welds require little preliminary preparation. Pieces to be joined must be clean and free of all dirt. Pieces to be joined must be held securely in the desired position, using clamps, blocks, tack welds or hand. When making fillet welds with one or both edges beveled, be sure to leave a root gap of 1/64" – 1/32" if a tacking tip is not used.

Rosette Welds: Rosette welds are similar to lap welds. Little preliminary preparation is required other than cleaning and drilling of holes of the desired size and position. To drill holes, use a hand or electric drill.

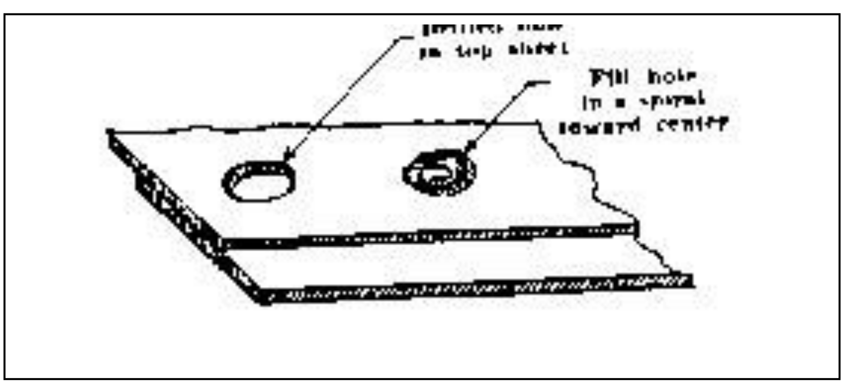


Figure 7: ***A Rosette Weld.***

Back Welding Cemented Joints
Before back welding cemented joints, allow cement to cure for at least six hours. Be sure all cement residue is removed with a knife, sand paper, emery cloth, wire brush or router.

D.2 Tacking
Just as in metal welding, tacking is a method of superficially joining together pieces to be welded in order to hold them in position for final welding. With plastic welding, this may be accomplished either by the use of a small diameter rod or with a tacking tip on the operator's torch.

Tack welding with a rod is similar to hand welding except that the use of smaller welding or tacking rod allows greater welding speed. (See the discussion on hand welding.)

The tacking tip is a pointed shoe, which is attached to the welding torch and heated with the hot gas from the torch. By applying pressure on this pointed tip, material softened by the heat is fused together. Advantages of the tacking tip are primarily its great speed and neatness. Use of the tracking tip also eliminates a potential source of weakness in completed welds caused by rod tacks left in place. Most important, jigs and clamps are not necessary, and one hand is free to hold work together.

The pointed tip is held by the operator at an angle of approximately 80 degrees and placed directly on the joint to be tacked. Then, much as if it were pencil, the operator slowly draws a line along the joint.

When pieces to be welded are larger or unwieldy, short tacks are often made at strategic points, such as corners at regular intervals. These short tacks help to hold in place the pieces to be tacked. With the partially tacked pieces in the proper position, the tacking tip may be drawn around the entire joint creating a continuous seal. The resultant tack will hold together large pieces of material sufficiently well so they can be moved and handled without coming apart. If the welder wishes to reposition the pieces, the tack weld may be broken and re-tacked.

The welder is now ready to make a completed weld using one of the methods shown in the following sections. Since the pieces to be welded are held in place by the tack weld, no jigs, or clamps are necessary and the operator has both hands free. NOTE: Tacking produces only a superficial welds which has little strength. It should not be considered a complete weld.

D.3 Hand Welding
In the welding of plastics, materials are fused together by a proper combination of heat and pressure. With the conventional hand welding method, this combination is achieved by applying pressure on the welding rod with one hand while at the same time applying heat to the rod and base material with hot gas from the welding torch. Successful welds require that both pressure and heat be kept constant and in proper balance. Too much pressure on the rod tends to stretch the bead and produce unsatisfactory results. Too much heat will char, melt or distort the materials.

Preparation for Welding
With the torch ready for welding (tip inserted, welding gas and current turned on) check the temperature by holding the bulb of the thermometer ¼" from the end of the tip. When welding PVC the correct temperature may be easily determined by holding the tip ¼" from the material and counting off seconds in the following manner: (slowly) one and two and three and four. At the count of four, the material should show a faint yellowish tinge. Adjust the temperature accordingly.

Select the proper filler rod. With a sharp knife or cutting pliers, cut the filler rod to the desired length (slightly longer than the length of the intended weld) at an angle of 60 degrees. This provides a thin wedge, which is easily heated and facilitates starting the weld.

Starting the Weld
Holding the torch with the tip ¼" – ¾" from the material to be welded, preheat the starting area on the base material and rod until it appears shiny and becomes tacky. Hold the rod at an angle of 90 degrees to the base material and move it up and down slowly so it barely touches the base material. When heated sufficiently, the rod will stick to the base material. To maintain the correct balance of heat, the torch should now be moved in a vertical fanning or weaving motion so as to heat both the rod and the base material equally. At the same time, the rod should bend and begin to move forward. Overheated welding rod becomes rubbery and makes application of even pressure virtually impossible.

When welding plastics, a good start is essential. It is at the starting point that welds most frequently fail. For this reason, starting points on multiple bead welds should be staggered whenever possible.

Continuing the Weld
Once the weld has been started, the torch should continue to fan the rod to the base material with approximately two full oscillations per second. To compensate for this difference in bulk, the arc of the fanning motion should be concentrated on the base material approximately 60% of the time when using 1/8" rod; and approximately 40% of the time when using 5/32" rod. The fanning motion should heat ½" of the welding rod and 3/8" forward of the rod on the base material. Average welding speed should be 4" – 6" per minute.

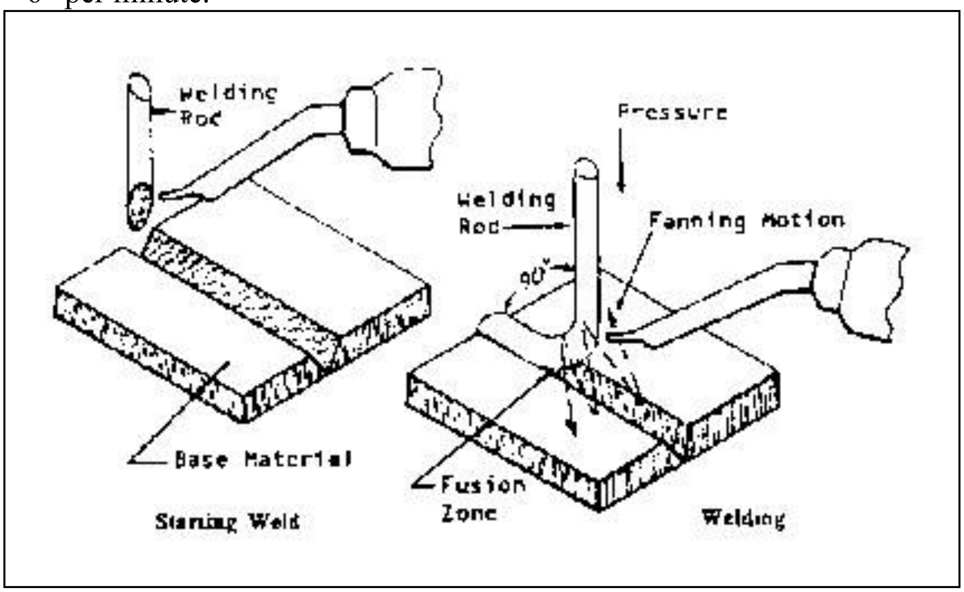


Figure 73: ***Plastic Welding***

Correct Angle of Welding Rod
When welding PVC, the rod should be held at an angle of 90 degrees to the base material. Although greater welding speed can be obtained by leaning the rod past the perpendicular (away from the direction of welding), the resultant stretching of the rod produces checks and cracks in the finished weld upon cooling. In order to exert sufficient pressure on polyethylene rod, it must be fed into the weld bed at an angle of 45 degrees to the direction of the weld with the upper part of the rod looping away from the direction of the weld. For fillet welds, the rod should be held in such a way that it bisects the angle between the two surfaces. In most cases this will be 45 degrees. It will be essential to preheat all surfaces being joined. When butt-welding PVC pipe, the welding rod should always point towards the center of the pipe to prevent stretched welds.

In the process of welding, the rod will of course eventually be used up, making it necessary for the welder to renew his grip on the rod. Unless this is performed carefully the sudden release of pressure may cause the rod to lift away from the weld bed, causing air to become trapped under the weld resulting in a weak weld…often in a complete weld failure. To eliminate this possibility, place the 4th and 5th fingers on either side to maintain pressure while repositioning the thumb and forefinger. If this movement is too difficult, place the 3rd and 4th finger on top of the already deposited bead (it should be cool since only the bottom part is exposed to heat) and hold the rod down while repositioning the thumb and forefinger. Then resume normal pressure. When using the latter method, certain caution should be taken to turn the torch away from the working area to eliminate any danger of burning fingers.

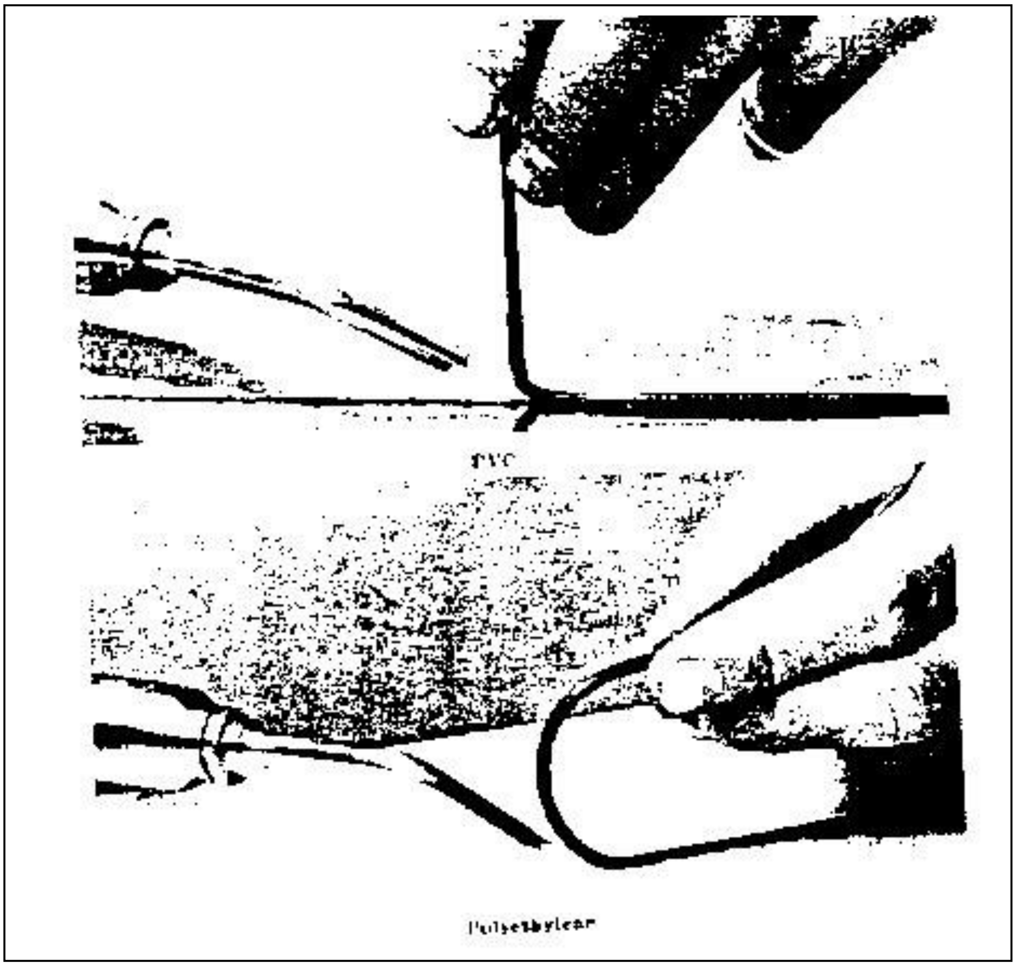


Figure 74: ***Correct Angle of Welding Rod.***

Finishing the Weld

When a weld is to be terminated, stop all forward motion and direct a quick application of heat directly at the intersection of the rod and base material. Remove the heat; maintain downward pressure on the rod for several seconds to prevent possibility of the bead being pulled from its bed. Then release the downward pressure; twist the rod with the fingers until it breaks. If a continuation weld is to be made, cutting at an angle of 30 degrees should terminate the deposited bead with a sharp knife or cutting pliers after allowing it to cool for several seconds under pressure. When joining one rod to another in a continuation weld cut the new row at 60 degrees. Heat the 60-degree angle surface of the new rod and weld on an angle of old rod so that the pieces joined together appear to be almost one piece. Never splice welds overlapping side by side. When terminating a weld, as in the case of pipe welding, the weld should always be lapped on top (not beside) of itself for a distance of 3/8” – ½”.

When welding PVC, a good finished weld will appear comparatively uniform with no brown or black discoloration. If insufficient heat has been applied, the rod will appear in its original form and can easily be pulled away from the base material. Small flow lines or waves on either side of the bead should be evident on a satisfactory weld.

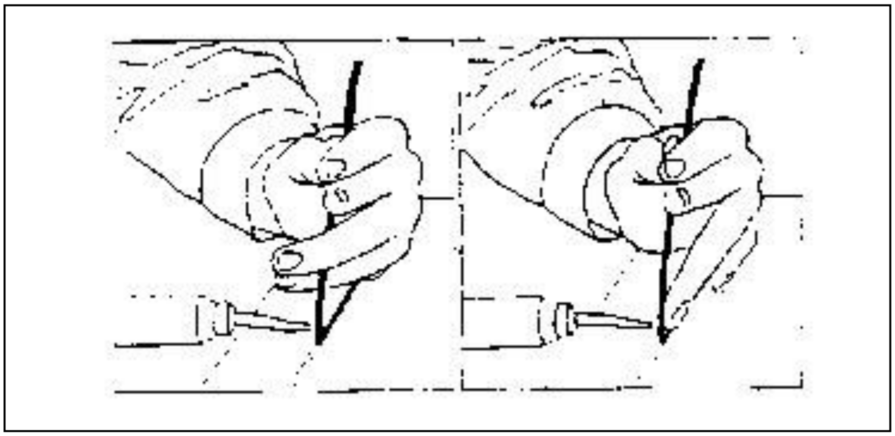


Figure 75: ***Methods of Re-positioning Grip on Welding Rod.***

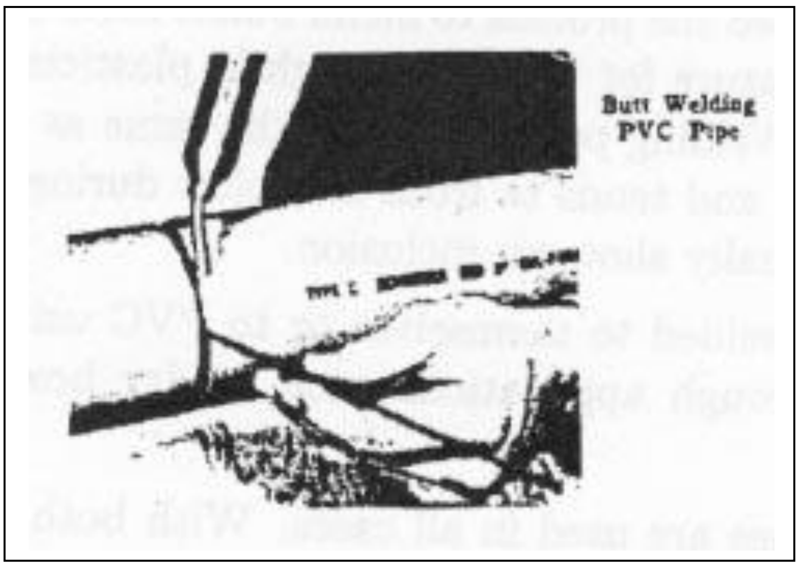


Figure 76: ***Butt Welding PVC Pipe.***

When welding heavy sections of material, multiple beads are welded in the joint, one on top of the other. Caution must be exercised when running these multiple beads so that the whole mass does not become overheated and produce a bad weld. When back welding a cemented pipe, be sure all cement at the joint is removed. When welding pipe, be sure that the 90-degree angle is maintained at all times between the rod and the base material. To decrease the number of welding runs when laying multiple beads, use larger size rod: 5/32" – 3/16". As a rule of thumb, a minimum number of beads should always be used. The finished weld should always overlap the beveled edge of the base material.

D.4 Instruction for Welding Individual Materials

The welding techniques outlined in the preceding sections apply to all the weldable thermoplastics. Each type of thermoplastic, however, has individual physical and chemical properties, which influence both the welding process and the finished product. Such factors as chemical resistance, recommended working temperatures, impact resistance; coefficient of expansion, structural rigidity, and notch sensitivity should be considered before selecting the material. Information and recommendations on special applications can be made from the product manufacturer. Laramy Products Company will furnish information on special welding techniques and equipment upon request. General information on welding the more commonly used thermoplastics is given below.

Polyvinyl Chloride: Three factors influence the welding of PVC used (normal or high impact), the amount of plasticizer present in the material and the quality of the welding rod. There are two types of rigid (un-plasticized) PVC available: Type 1, normal impact, and type II, high impact. Type II PVC is modified with rubber to increase impact resistance. However, welding temperatures and rates must be lower than those used for type I, to avoid scorching.

Plasticizers are liquids or compounds added during extrusion to make thermoplastics more flexible. Plasticizers are not used in rigid PVC pipe, fittings or sheets, but are used in some welding rods to improve the welding quality. Under normal conditions, a 10% plasticized rod gives better performance and strength, while the un-plasticized rod does not.

Acrylic ("Plexiglas"): One of the first thermoplastics to be welded was when the Germans, during World War 2, used the process to mend bullet holes in airplane canopies. Acrylic requires a high temperature for welding, a slightly plasticized rod, and may be welded with compressed air. Welding procedures are the same as for PVC. Acrylic is susceptible to stress cracking and tends to froth and blow during welding. The flow lines on a finished weld will usually show air inclusion.

Most acrylics can be welded to themselves or to PVC using a PVC rod. This makes them ideal for see-through applications such as dry boxes when clear PVC is not available.

PVC welding techniques are used in all cases. With both acrylics and PVC, extreme care should be taken to avoid charring the material. Welding is accomplished with air at 400 degrees – 450 degrees F for PVC rod and 500 degrees - 550 degrees F for acrylic rod. The rod is normally held at 90 degrees to the weld seam with 2-3 pounds of pressure exerted on it.

High Temperature PVC: Chlorinated PVC is best welded by hand at the present time to obtain maximum strengths of 80-100%. Since a high temperature is required to melt CPVC, a rod-feeding device is recommended to maintain a constant 3-pound rod pressure without undue discomfort to the fingers from the heat.

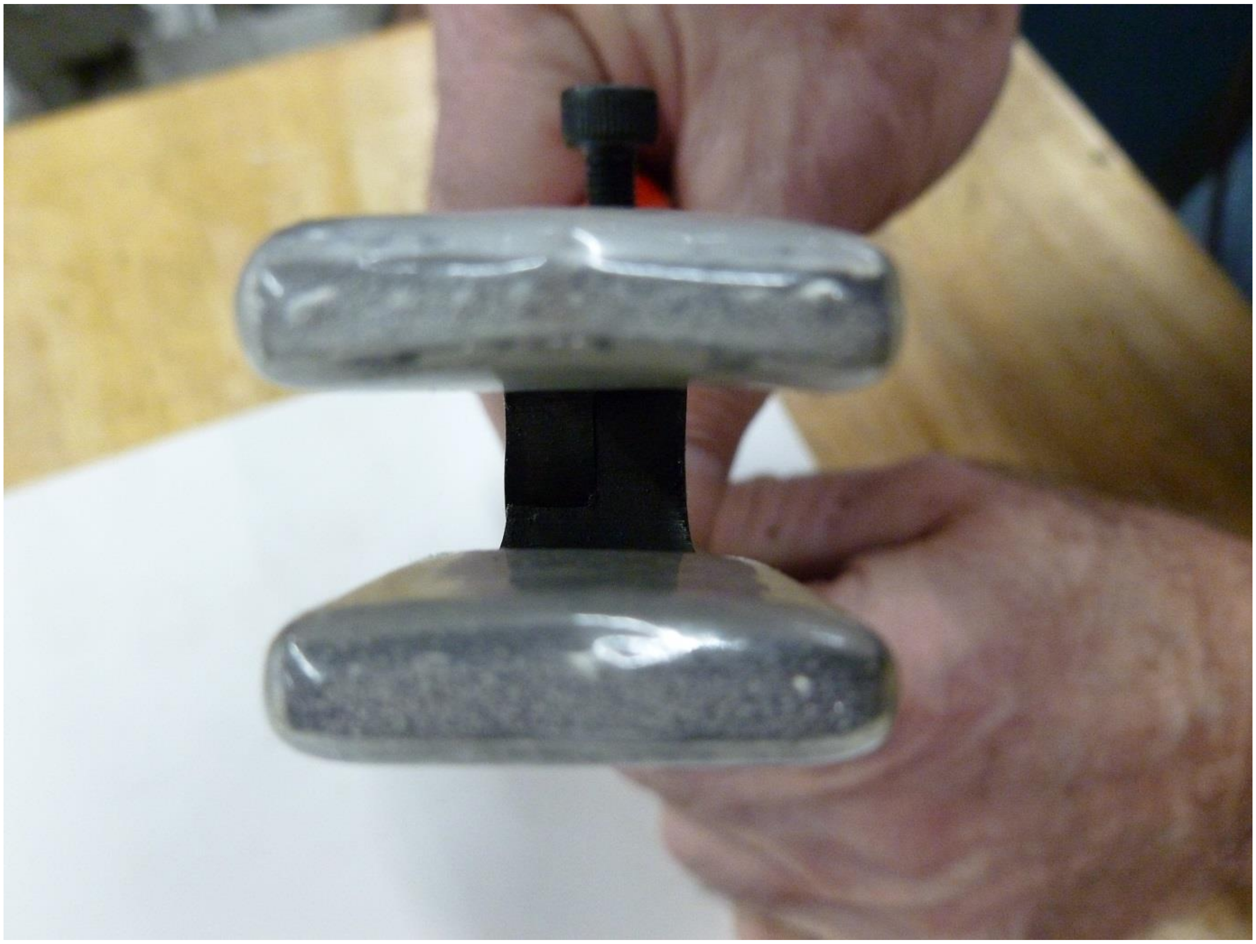
On typical sheet and pipe seams, the edges should be cleaned and beveled prior to tack welding. The finish weld is accomplished with 600-650 degrees F of air temperature at the tip with 3-4 psi gas pressure. The 90-degree welding angle should be maintained. When welding, the rod will soften and a slight bulge will occur just above the weld. The rod should not be allowed to melt further and care should be taken to prevent scorching of the rod or the work through excessive heat.

ABS: Acrylonitrile/butadiene/styrene has excellent forming properties for most applications. ABS is available in normal and high temperature types, either of which can be hand or speed welded if desired. Conventional ABS should be welded with air on an inert gas at approximately 350-400 degrees F, high temperatures ABS at 500-550 degrees F.

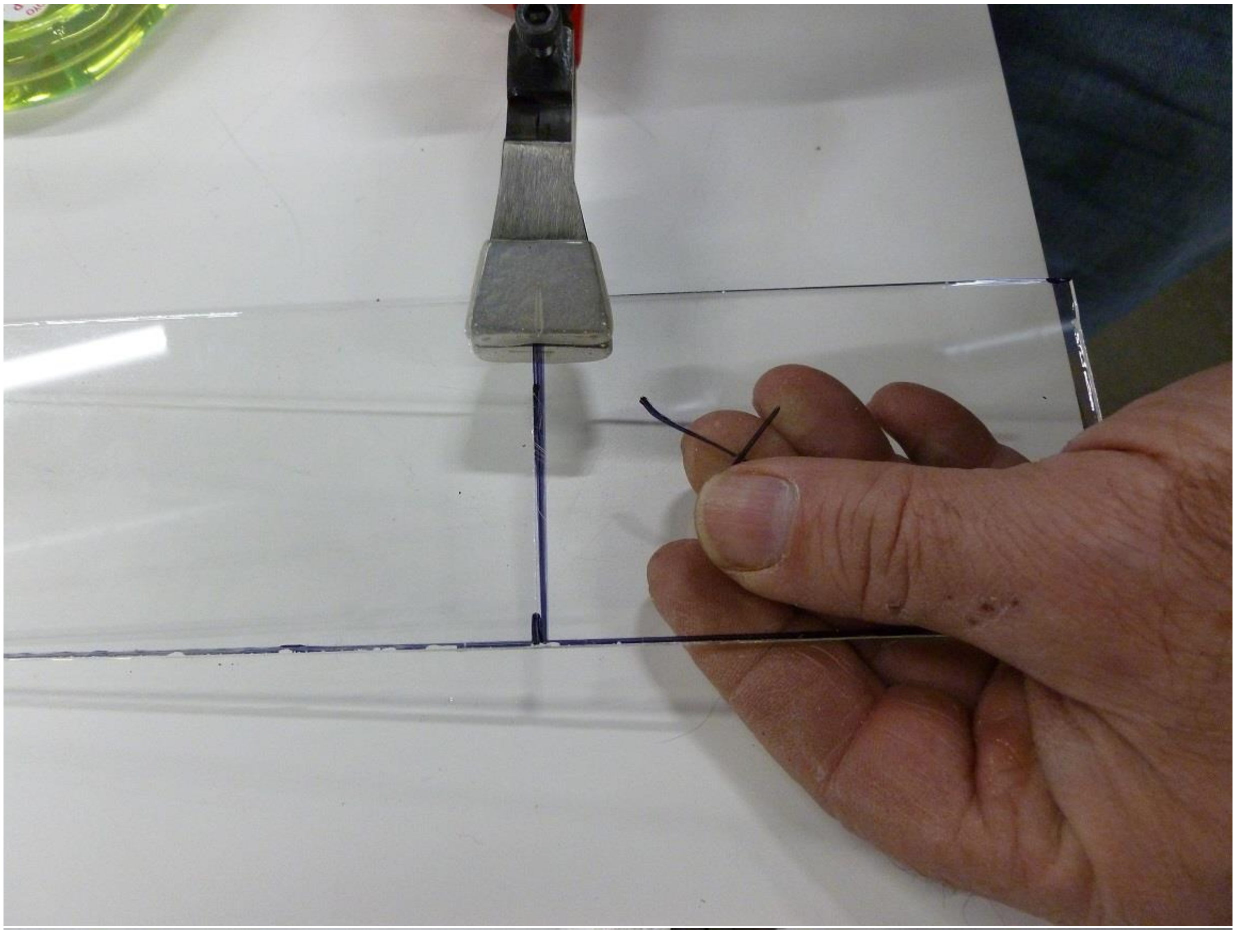
Appendix E: Glass cutting   A topic we will cover in nonmetallic techniques is glass cutting. Glass is an important engineering material. Glass is reasonably strong (though brittle), chemically inert, quite transparent (with the right ingredients and surface treatment) and harder than most cutting tools. It is this last quality

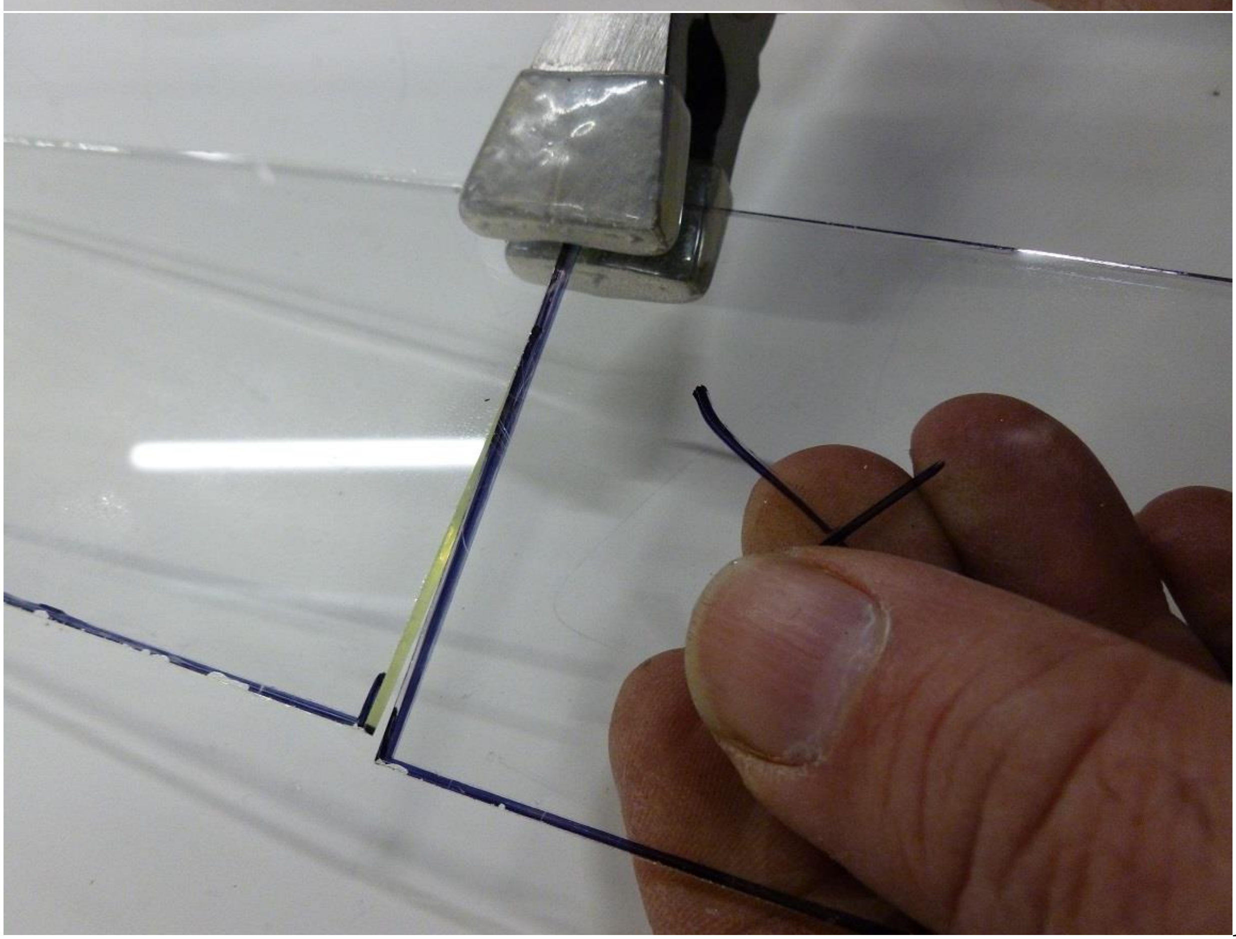
(combined with its brittle nature) that makes it awkward to use most ordinary cutting methods upon it.
Glass is not a conventional solid; structurally it is a really viscous liquid. We are going to take advantage of this property. Our glass cutting tool is rather similar to the ones you can buy in a hardware store. It has a sharp little wheel made of tungsten carbide (instead of the easily dulled hard steel wheel you get in the aforementioned hardware store).

When we press this wheel down onto the surface of the glass sheet it will score the surface of the glass (tungsten carbide being one of the few readily available materials harder than the glass). When we apply force across the score line it is rather like popping a bubble. There is a “pop” and we have two pieces of glass (if you have done it right).
One way to apply the force (and a very easily controlled one) is to use a pair of running pliers. They are a bit different from the pliers in your tool box. If you look at the jaws of the running pliers end on you will note that the jaws do not come together in a straight line.



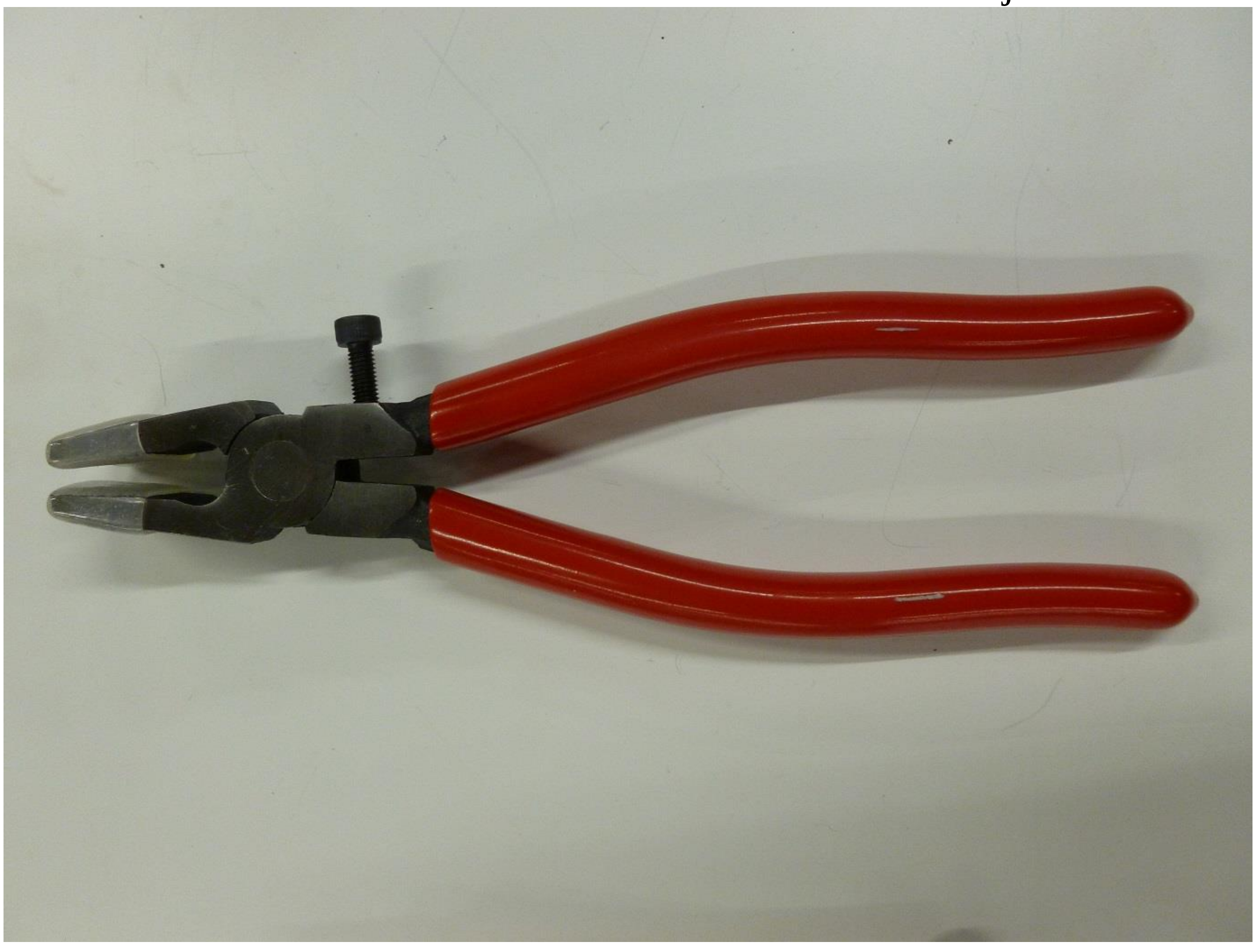
Instead they form a gentle arc. If you put the concave jaw on top of the glass with the center of the jaw (note center mark on the jaws) aligned with your score line, then bring up the convex jaw on the opposite side of the glass (without the score line) it is like snapping a stick across your knee.





As

mentioned previously glass is brittle. The score line weakens the glass and it cracks along this line. We are not really cutting the glass but cracking it in a (hopefully) controlled fashion. You will note in the picture of the running pliers another feature not normally seen on conventional pliers. There is an adjusting screw that enables the user to set the distance between the jaws.

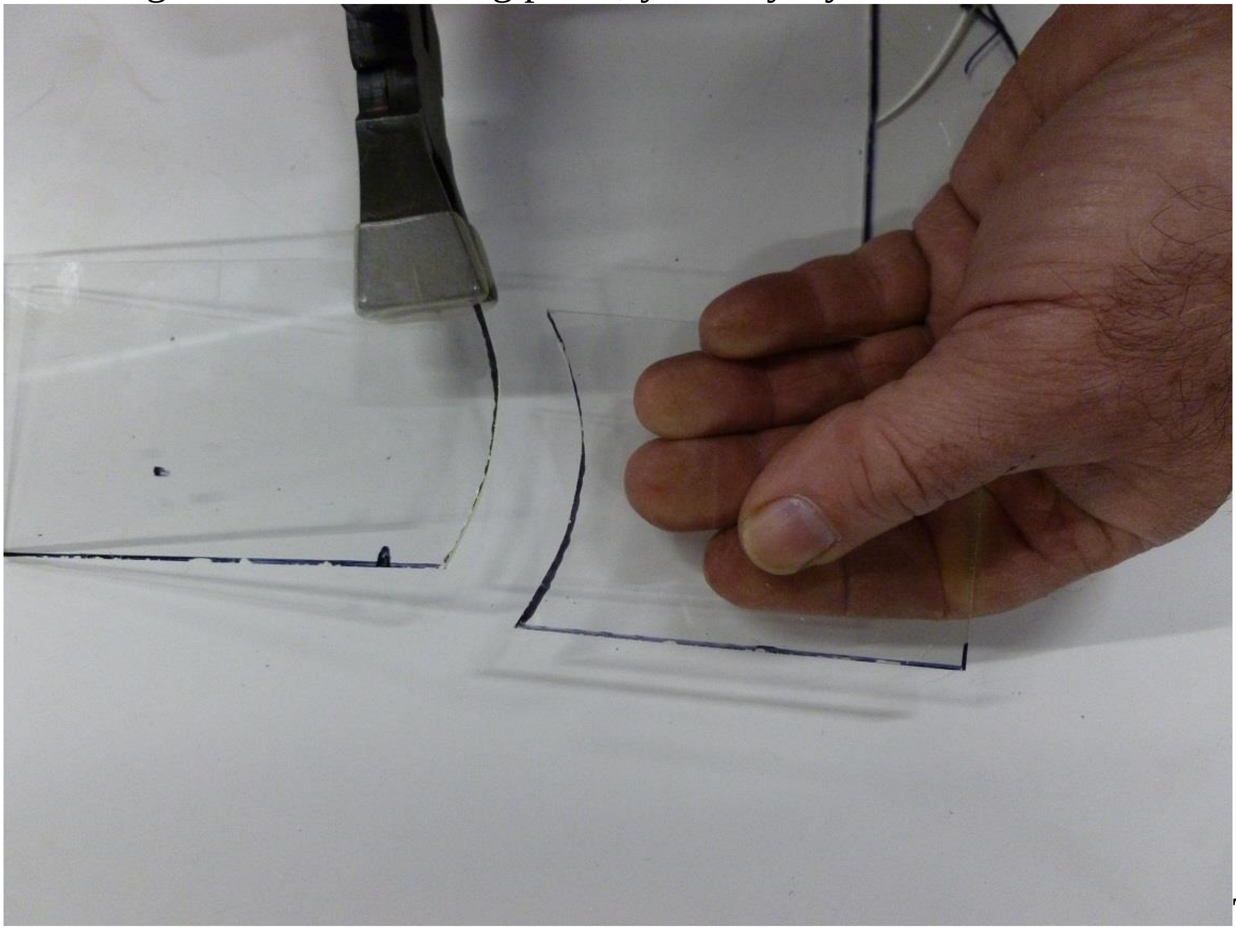


To use this gently close the jaws of the pliers upon the sheet of glass, then gradually tighten the screw until you “feel” it make contact with the opposite jaw. Open the pliers and loosen the screw about a quarter of a turn. The jaws are now adjusted to apply force onto the glass but only for a very short distance. It is just enough to “pop” the glass. It is a very controlled application of force.

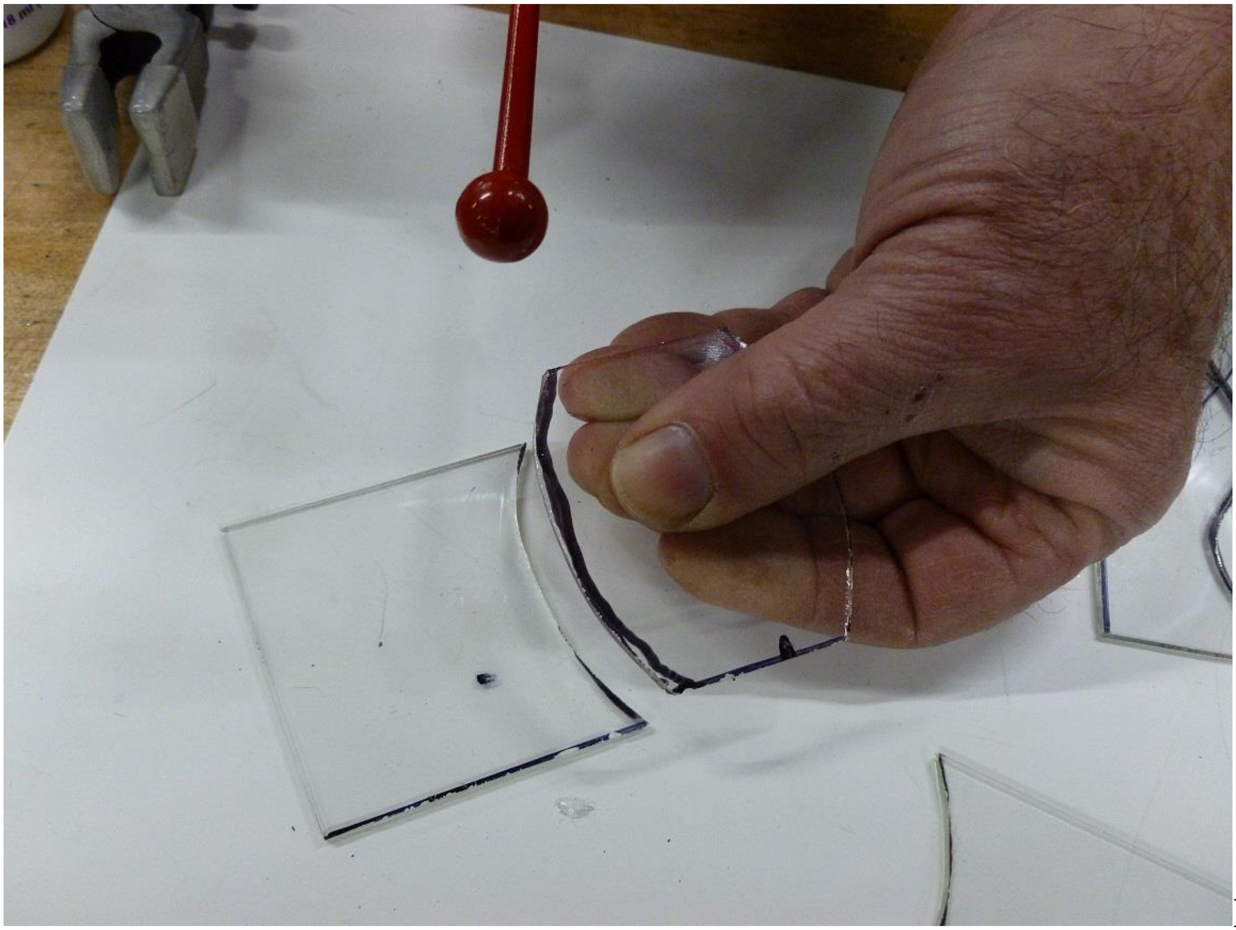
Another method of applying this force is to simply place the score line across the edge of a table and apply pressure. It will generally work but is not as easy to control and it will require more force.

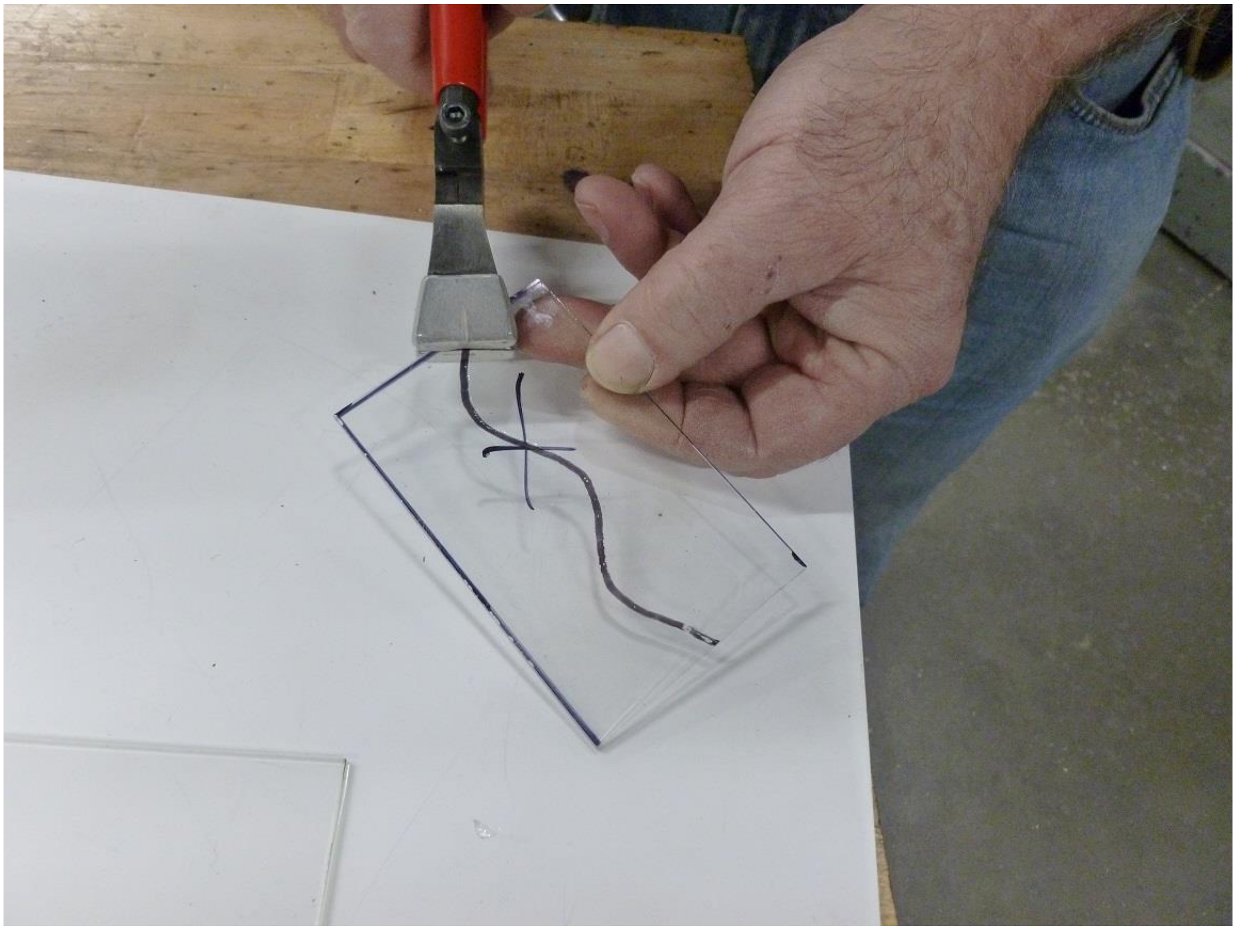
A third method is to tap one side of the glass while holding the portion on the other of the score line steady. That is what the ball on one end of the hardware glass cutter is for. Sometimes this technique has advantages particularly when cutting curves. Curved “cuts” in glass are tricky and not recommended if you do not have any extra pieces of glass.

Referring back to our running pliers, you may try this with an arc



The tapping method will also work with care

If you get more ambitious, see below it will work with care.



If you look closely at the glass cutter


you will see that the handle is hollow and contains a reservoir filled with light cutting oil. The oil lubricates the carbide cutting wheel and the surface tension of the light film of oil left on the score line prevents most small chips from flying around when the line “pops”.
Any small chips or broken pieces of glass are SHARP. Try not to handle them if you can avoid it. The best way to deal with them is to use a small broom and dust pan to sweep them up and deposit them (along with any other broken glass) in a yellow bin marked “Broken glass”.

Appendix H: Glossary

Brittle: Easily broken, cracked, or snapped.

Center drill: A short, very rigid drill, which is used to make a starting hole in metal. Because of the shape of this drill it will not flex (which would cause the tip to wander) when drilling.

Continuously variable: (also **infinitely-variable**) Able to take on any value within some working range.

Facing Off: Using the lathe to cut the end of the work piece, perpendicular to the axis of rotation.

Knurl: A checkered texturing pattern sometimes put on pieces of metal. Squeezing the metal between the wheels of a knurling tool forms it. This deforms the surface of the work piece, forming hills and valleys.

Layout: The process of transferring the important dimensions from a machine diagram to a piece of stock, prior to machining.

Scribe: 1) To mark a line by scratching with a pointed instrument. 2) The instrument used for scratching a line.

Spacer: (also **spacer block**) – A block of material used to raise the height of the work piece of clamp.

Stock: The raw material, which will be machined into a finished object.

Swarf: Debris (small chips, shavings), which is produced as a result of machining.

Turning: Machining by use of the lathe.

CNC is an acronym for “Computer Numerical Control”. In its most basic form it is like a series of X,Y, and sometimes Z coordinates which control the movements of an inkjet, (think printer) a router, a plasma jet, a water jet, a milling tool, a lathe tool, an extrusion nozzle (for a 3D printer) or a laser.

All of the devices have something in common. A computer program directs the device to go somewhere where it either removes material, possibly doing along the way, or deposits material. The operator may either write these co-ordinates manually or (more likely) use a computer program to generate them from a CAD file. In addition there are extra bits of code that start the spindle of the machine running (and in which direction) or turn it off, turn on or off coolant flow(for cutting operations), tell the machine how fast to execute operations, and what to use as the co-ordinate origin for the operations. One of the good points of this type of equipment is that a part will be done the same whether it is the first part or the 100,000th part. As an example of commonality, we have at RPI a CNC plasma cutter, an Abrasive Water Jet cutter and several laser cutters. To begin generating the code for any of then you make a CAD drawing. Be certain that it is two dimensional only (do not show the thickness). This is because all the moves for material removal are two dimensional and the program will become confused between the layers of a 3D drawing. Whatever CAD system you use we next save it as a DXF file. Why DXF? Some of the equipment will use the DXF directly to internally generate their G codes (short for go code). The rest will be imported into Master Cam and we will use one of its post processors to generate the appropriate G code. Speaking of confusing the machines, try to be certain that you do not have lines sitting on top of each other. Be sure you delete all duplicate entities as the computer will not be able to choose between them or, with the laser cutter, may try to cut them all. Since we are referring to the laser cutter, be aware that different colors of line will result in the lines being cut in a different order. For the laser cutter in the student shop the order is black first then red, green, yellow, blue etc. The operator must also set the relative speed and power represented by each color also. What you set it at depends on the material, its thickness and whether you are cutting or engraving. Be aware that not all materials can be safely laser cut. Some materials may give off fumes that are toxic, corrosive or both. Always ask if you are not certain (perhaps ever if you are). The laser cutter in the student shop (and all of our other lasers for that matter) will NOT cut sheet metal. The plasma cutter and water jet cutter will do so. The plasma cutter will only cut metals. The water jet cutter will cut almost anything.

The above has been a rather sketchy intro to CNC with an emphasis on the 2 D processes. Partly because the 2D work is easier and you get into less trouble if you make a mistake and partly because they are harder to crash. For the last reason alone we are much readier to let you use them.

With CNC lathes and mills, the machines will move wherever you tell them to. The equipment does not care if the command was not what you really intended or if you forgot a decimal point. It just moves. If there are solid objects in the way (including possibly you) it just moves. We are not so casual

Appendix G : Inside a Plasma Cutter

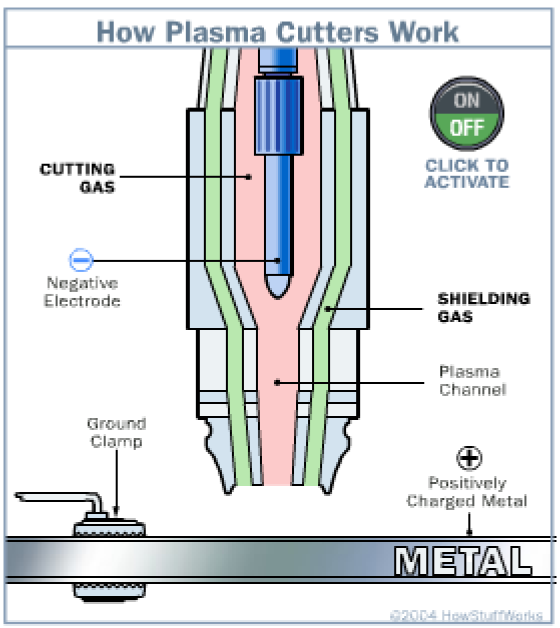
Plasma cutters come in all shapes and sizes. There are monstrous plasma cutters that use robotic arms to make precise incisions. There are also compact, handheld units that you might find in a handyman's shop. Regardless of size, all plasma cutters function on the same principle and are constructed around roughly the same design.

Photo courtesy
Torchmate CNC Cutting Systems

Inside a plasma cutter: The electrode is at the center, and the nozzle is just below it. The orange piece is the swirl ring, which causes the plasma to turn rapidly as it passes.

Plasma cutters work by sending a **pressurized gas**, such as nitrogen, argon, or oxygen, through a **small channel**. In the center of this channel, you'll find a **negatively charged electrode**. When you apply power to the negative electrode, and you touch the tip of the nozzle to the metal, the connection creates a **circuit**. A powerful **spark** is generated between the electrode and the metal. As the inert gas passes through the channel, the spark **heats the gas** until it reaches the fourth state of matter. This reaction creates a stream of directed plasma, approximately 30,000 F (16,649 C) and moving at 20,000 feet per second (6,096 m/sec), that reduces metal to molten slag.

The plasma itself conducts electrical current. The cycle of creating the arc is continuous as long as power is supplied to the electrode and the plasma stays in contact with the metal that is being cut. In order to ensure this contact, protect the cut from oxidation and regulate the unpredictable nature of plasma, the cutter nozzle has a second set of channels. These channels release a constant flow of **shielding gas** around the cutting area. The pressure of this gas flow effectively controls the radius of the plasma beam.



Appendix H **Acurite Manufacturing of Cannon Base and Train cab rev.1 8/2017**SY

These Steps Must Be Done First: Please note BOTH E-STOPS on machine

- With Quill handle move quill negative Z (down) approximately 2 inches

(**Remove quill handle**),

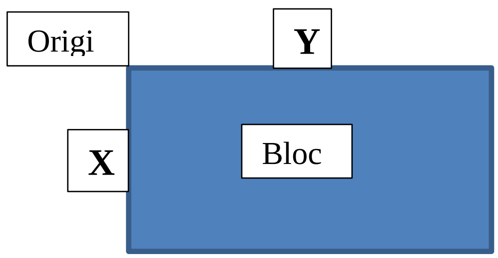
- Make sure table is around center (X, Y axes), not against stops.
- Turn Controller On (back of control box),
- Home machine (Find Home). Table and Quill will move

Cannon Base Block Setup (Train Cab also uses this set up for operation 1 and 2):

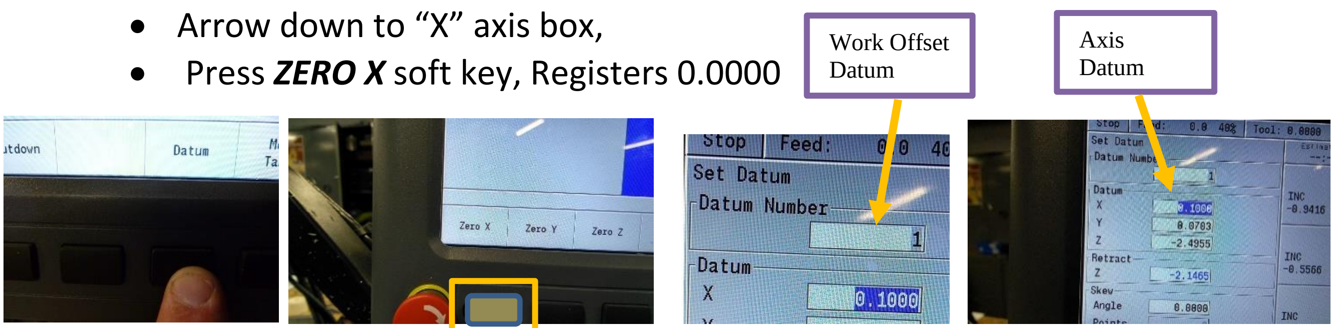
USE:

1-1/4" parallels,
Vise Jaw Stop on fixed jaw,
Edge finder in R8 collet,
1/2" 3 flute end mill (1" length of cut and extended 2-1/4") mounted in ER32 holder and collet

- Using Edge finder in R8 collet (1000-1300 rpm), locate "X Origin" upper left corner of "Pre-Milled to Length Block" (#10-32 holes must be drilled also).



- In Datum Screen, enter 1 in Datum Number box (work offset),
- Arrow down to "X" axis box,
- Press ***ZERO X*** soft key, Registers 0.0000



- Raise quill/edge finder + Z to clear part,
- Using hand wheel move table .100" (or radius of edge finder) towards part,
- Press ***ZERO X*** soft key again,
- Press ***USE*** hard key to set, this sets origin for that axis.
- Repeat above steps for setting "Y Axis" origin.
- Turn off spindle and remove collet and edge finder.

- For “Z Axis”, A Return Plane Height needs to be set, install tool to be used, position tip of tool 1 inch above tallest point on part using quill and knee (enough for negative Z travel for cutting and enough to retract tool above block). Quill has 5” total travel.
  (For Train: stand block up on end, on top of parallels. This will be the tallest point. See additional step for Train below).
- Press ***Datum*** soft key, Press ***ZERO Z*** soft key***,*** press ***USE*** hard key***.***

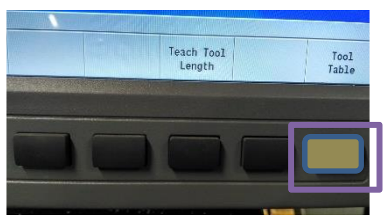
<u>**FROM THIS POINT ON DO NOT USE KNEE**</u> TO ADJUST Z HEIGHT

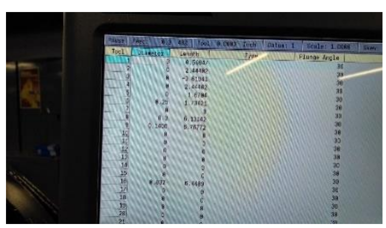
<u>**Setting Tool Length Offset(s):**</u>

- Tool # 1 used for Cannon Base and Train

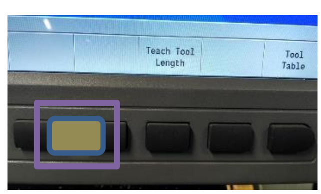
- Using quill carefully Lower tool down to top of stock. Do not jam into material as tool will chip. Shim stock can be used as a feeler gage.

- Press ***Tool*** hard key 

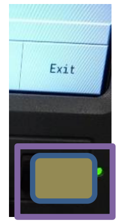
- Push ***Tool Table*** soft key, 

- Tool Table displays 

- use arrows to Highlight Length in Tool 1 column

- Press ***Teach Tool Length*** soft key, 

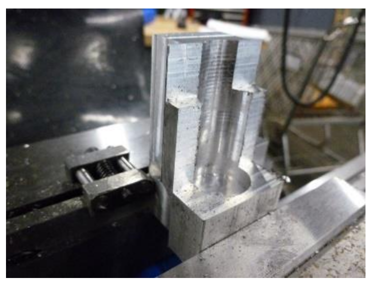
number will change setting the tool length to the Z origin of the part.

- Pressing ***Exit*** soft key finishes this process. 

- Press **Cancel** hard key 

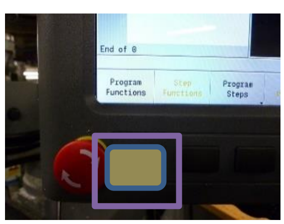
Train Additional Steps:

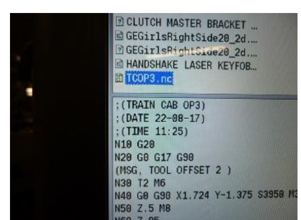
- Stand block up on end, on top of parallels, wheel opening against stop repeating Steps A thru F except ***use Tool 2 offsets***.



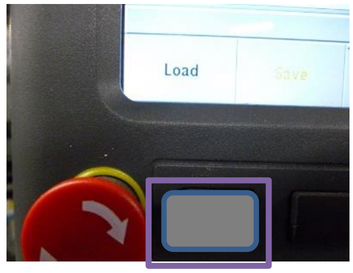
Loading and Running the Program: (Either Cannonbase.nc or Train Cab Combi.nc)

- Insert flash drive in front of control panel,
- Press ***DRO PGM*** hard key, ***Program Functions*** soft key, using ***Arrow Keys***, select program on drive,



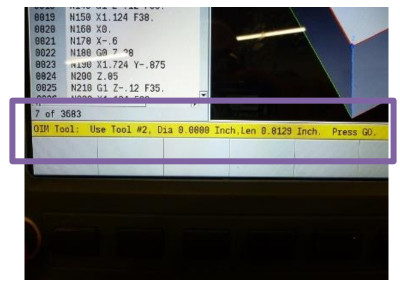



- Press ***Select*** soft key, then ***Enter*** hard key, 

- Press ***Load*** soft key 

- Press Run Options, Look Ahead should be highlighted, if not push soft key under Look Ahead,

- Turn Spindle on to 3850 RPM **(THIS IS MANUALLY TURNED ON AND OFF)**

- ***Press Go*** hard key   and follow messages.

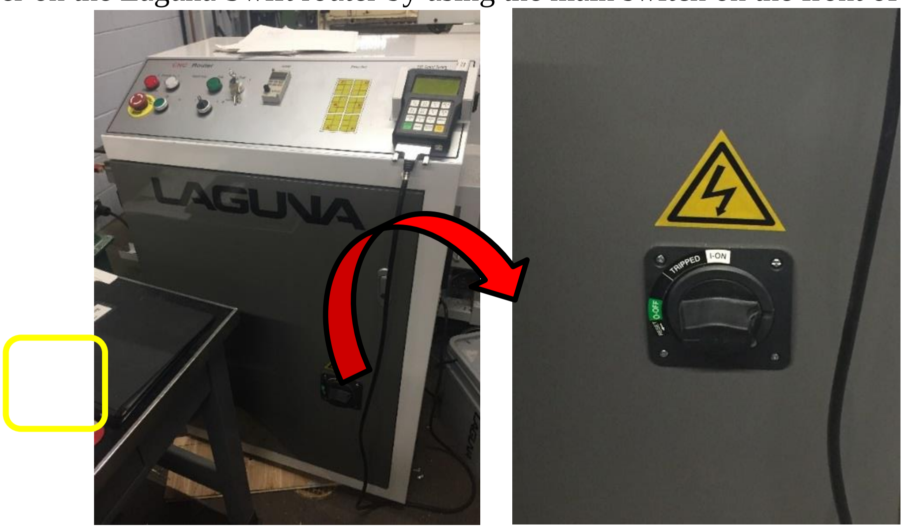
- Variable feed control should be mid-way 

(counter clockwise slows rapid and feed rates down, clockwise speeds them up)

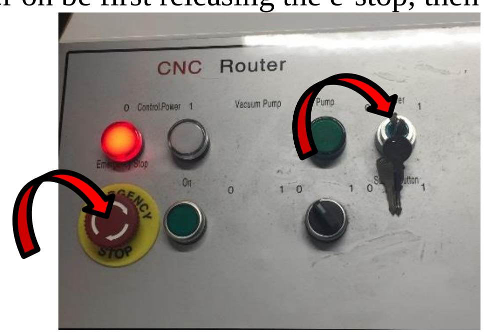
To STOP machine movement ***Press STOP*** hard key 
(Keep finger over this button while running)

Appendix I **This is a quick start up guide; ask for help if you are not sure how this machine or any other machines work. Additional information is available in the Laguna Manual**

Power on the Laguna Swift router by using the main switch on the front of the console

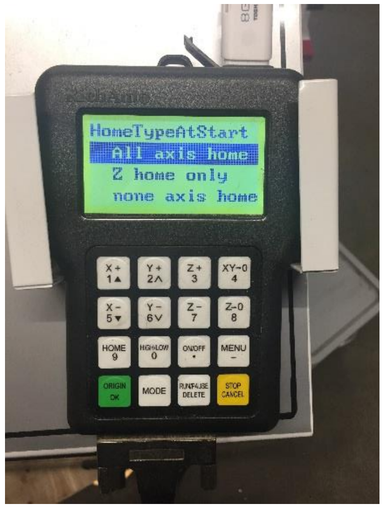


Turn the router on be first releasing the e-stop, then turning the key on



Turn on the controller power, followed by the water pump (check water level)
The spindle is a watercooled spindle and will be damaged if ran without the pump on

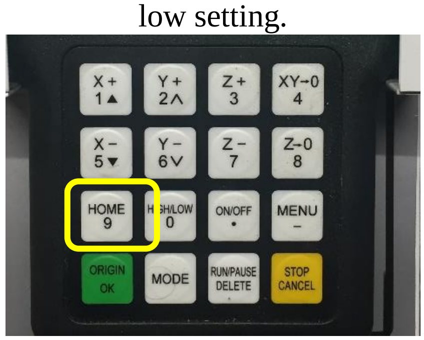
Once the machine is fully on, this promt will appear



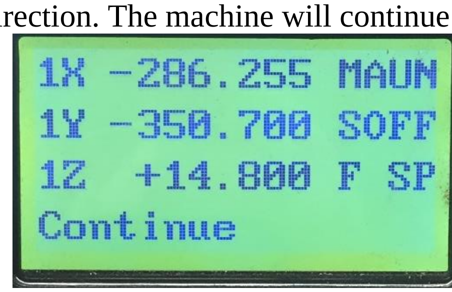
Make sure there is nothing between the spindle and the corner of the machine and press origin okay (green button). The machine will home.

Manually moving the spindle head:

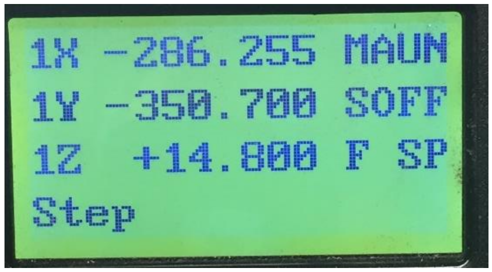
Jogging speed. There are two speeds at which the router can be manually jogged, high and low. Press the high and low button to toggle between the two. The high setting is about 4 times the speed of the low setting.



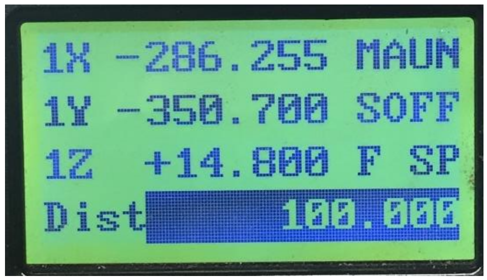
Continuous mode. Press the mode button until continuous is displayed. Hold the X+/-, Y+/-, or Z+/- to move the machine in the desired direction. The machine will continue to move until the button is released.



Step mode. Press the mode button until step is displayed. Each time the button is pressed the router will move a set distance (0.5mm for high setting, 0.1mm for low setting)



Distance Mode. Press the mode button until distance is displayed. This mode allows you to enter a distance you wish to move the machine. Enter the desired distance and press the coordinate in which you want the machine to do. For example type in 100 (mm) and press the X+ to move the machine 100mm in the X+ direction.



Setting the work zero:

The coordinate system for the part is determined on how the program is developed. Make sure to understand where your X0. And Y0. is located in relation to your stock. Move the spindle to where you want to set your origin point. Make sure the controller is displaying “machine coordinates.” If they are not being displayed press the HIGH/LOW and MENU button together. This will toggle between machine and origin (work) coordinates.

Setting Z Origin Point (tool touch off)

To touch off a tool load the tool into the spindle. Jog the tool down using the Z- button in low jog speed, until the tool is right above your work piece. Switch to step mode, on the low setting. This will move the spindle down 0.1mm (0.004”) at a time. Step down using the Z- button while rotating the tool by hand. Once you feel resistance from the tool stop stepping down. This is your Z0.. Press the Z-0/8 button to set the zero point. If greater accuracy is needed repeat above using a set distance of 0.0254mm (0.0001”) with the distance mode instead of the step mode.

Loading a program into the machine.

Load the program onto the USB that is with the router

Put the USB into the top of the controller

Press the RUN PAUSE/DELETE button. Select the U disk

Use the arrow keys to select the desired file, to load into the controller

Adjust settings as needed.

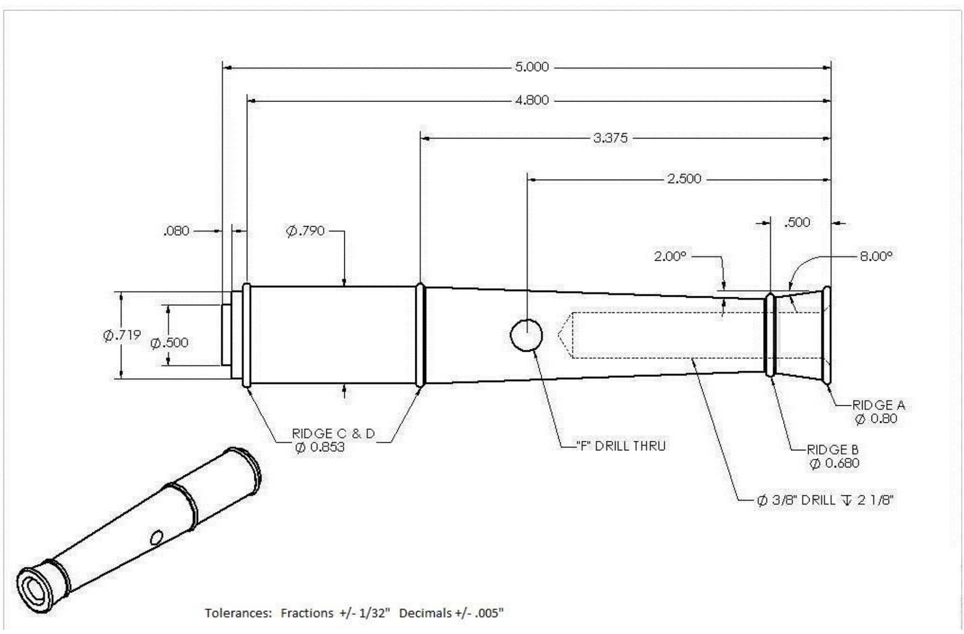
Press the OK button. Once the code is downloaded the machine will start.
Feed is set through the program and as a percentage by changing the SpdScale.
Spindle speed is set by changing F using the arrow keys. The spindle typically runs between 100 and 400. Need spindle conversion

Fitting the router bit into the spindle,
Select a router bit and the relevant collet

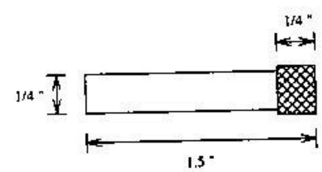
Fit the collet into the collet nut. Press the collet into the spindle nut until it snaps into place.
If the collet is not seated correctly it will damage the collet once put into the machine

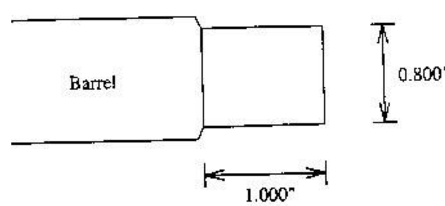
Fit the collet and spindle nut onto the spindle by hand.
Insert the tool bit into the collet making sure that the flute of the tool is outside of the collet. Tighten the collet with the provided wrenches.
It is important to not overtighten the collet as it will damage the collet.
Reference the Laguna manual or the instructor on what type of bit to use.

Appendix J A brief outline of Cannon and Train Fabrication

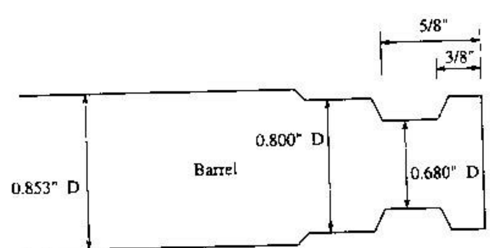


)ost



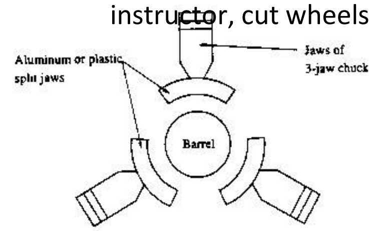


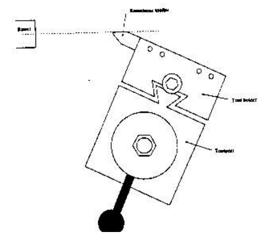
Directions: (this is an outline see material, beginning on p 52, for details)

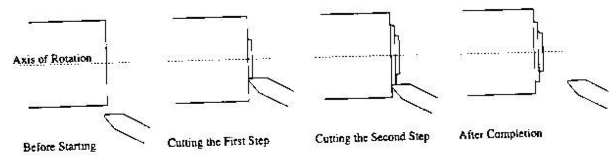


1. Cross drill barrel with F drill bit
2. Faceoff barrel
3. Axial Drill 2 1/8” deep with 3/8” drill bit
4. Turn outside diameter down to .853”
5. Turn muzzle diameter down to .800”, 1” from muzzle end
6. Cut .680 diameter ¼” wide from 3/8” to 5/8”
7. Cut Ridge A to .800” diameter
8. Cut Ridge B to .68” diameter
9. Cut Ridges C,D to .853” diameter
10. Turn barrel diameter between ridges C and D down to .790”
11. Set up lathe to make 2 and 8 degree tapers on barrel
12. Polish barrel if so desired
    a. See page 47
13. Cut off remaining stock with band saw, 5 1/8” from muzzle end
14. Cut wheels. See instructions ➔ After, finish breech end:
15. Install 3 jaw chuck in the lathe, line with aluminum or plastic jaws
16. Angle the tool. Move the round nose tool in 1/16”, and turn a step 0.5” in diameter. Move the tool in 1/16” again, and turn a step 0.719” in diameter. These dimensions are not critical. If you wish, you may deviate from the design here. **Be creative.**
17. Trunnion: Obtain ¼” diameter stock, 3 jack chuck installed
    18. Face off end of pin
19. Scribe line 1.5” from faced off end
20. Face off other end of pin, test fit in base
21. Knurl one end to expand diameter

1. Use remaining 7/8” of cut off stock in collet, about ½” in
2. Faceoff end
3. Center drill the stub with #6 twist drill (.204”)
4. Sand with rough emery paper to remove any Dykem (blue) or dirt
5. Knurl wheels if so desired
6. Obtain a parting tool and wheel gage from instructor, cut wheels

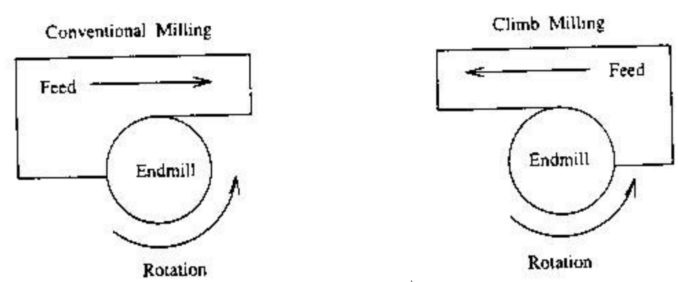


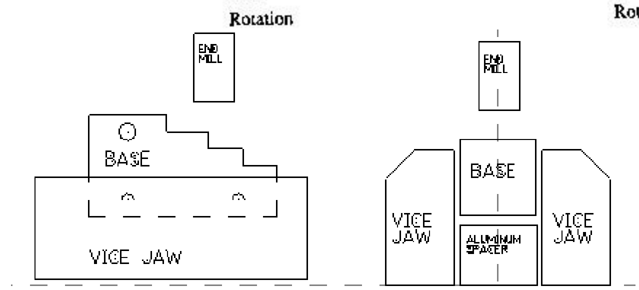


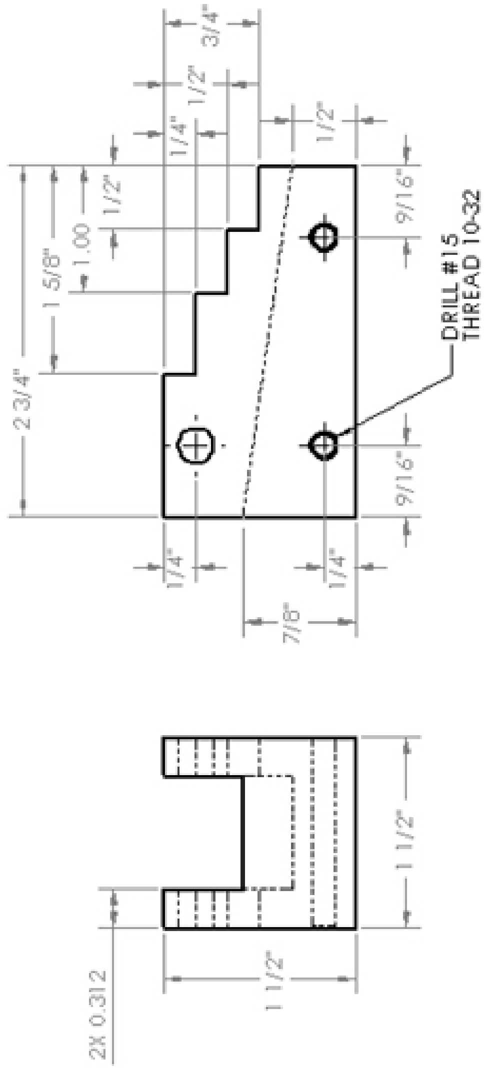


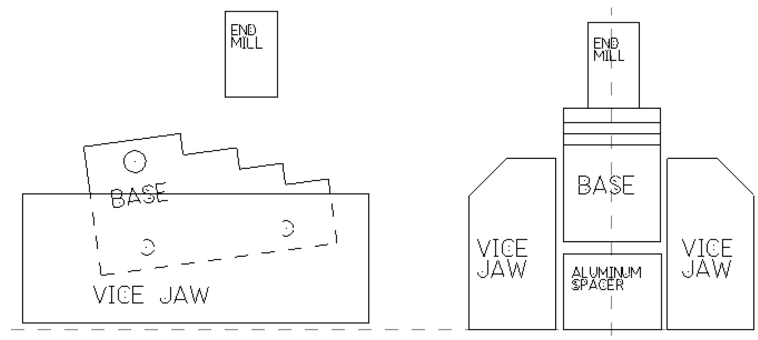
Directions: (this is an outline, see material starting on p 73 for details)

1. Mark block with Dykem (blue) (see page 54)
2. Use horizontal miller to trim to 2 ¾” length
3. Drill holes, tap holes
4. Cut off excess material with band saw
5. Install 7/8” 2flute end mill, check for 1800 RPM
6. Mill the stepped section up to scribed lines using conventional milling, .050” at a time
7. Mount the blocks next to each other in the vise such that the diagonal lines are even with the top of the vise.
8. Mark the bottom of the slot on one end of the blocks. Cut down the center (remove handle to the vertical control)
9. Mill each side to proper dimensions, using climb milling for the last cut for a nice finish (last cut should be .005” deep)
10. Polish the base (see page 78)
11. Assemble cannon

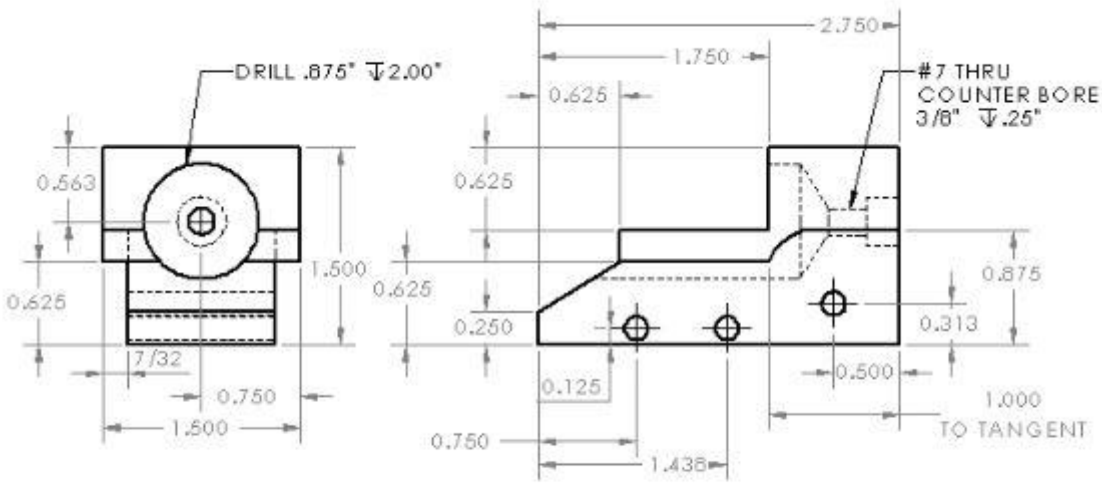


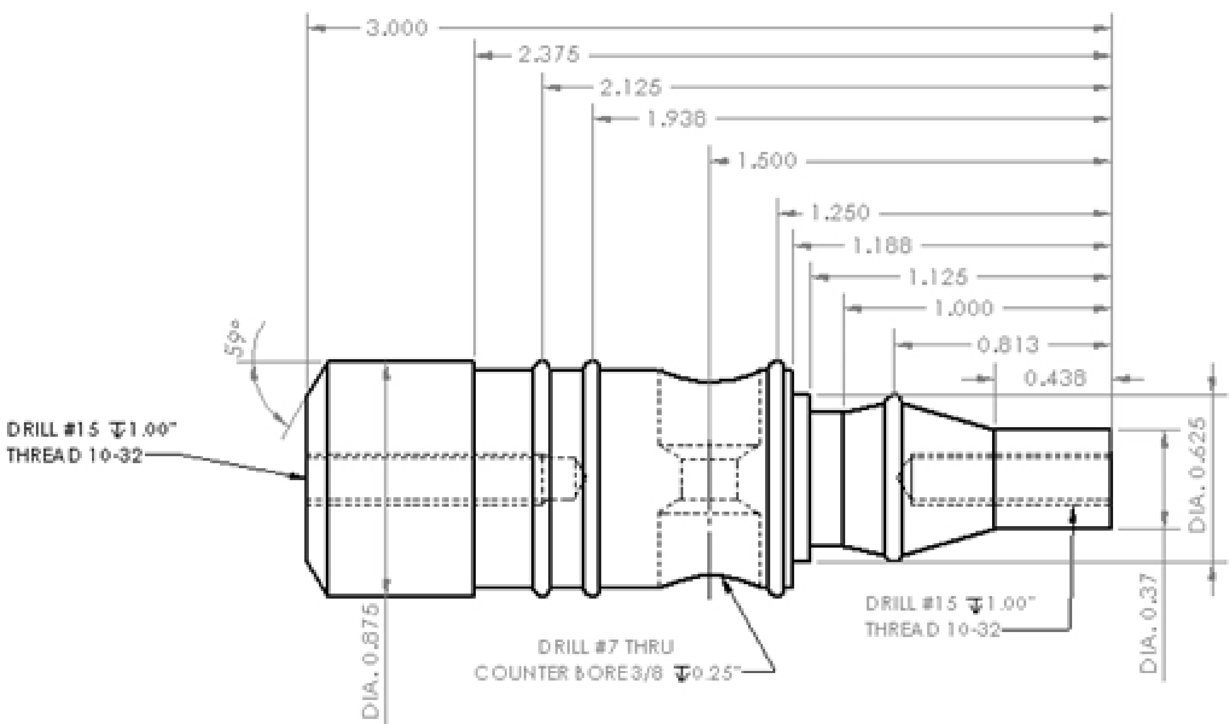



3/4"
1/2"
1/4"
1/2"
9/16"
1/2"
1.00
1 5/8"
2 3/4"
DRILL #15
THREAD 10-32
9/16"
1/4"
1/4"
7/8"
1 1/2"
1 1/2"
2X 0.312


END
MILL
BASE
VICE JAW
END
MILL
BASE
VICE
JAW
ALUMINUM
SPACER
VICE
JAW

Tolerances   fractions +- 1/32”   decimals +-.005”





References

[1] Charles Edgin, *General Welding*, John Wiley and Sons, New York City, NY, 1982.

[2] R.E. Green, editor, *Machinery's Handbook*, Industrial Press, Inc., New York City, NY 25th edition, 1996.

[3] Laramy Products Company, Inc., Lyndonville, VT, *Making Even Better Plastic Welds*, 1993.

[4] South Bend Lathe Works, South Bend, Indiana, *How to Grind Lathe Tool Cutter Bits*, 1938, Bullet No. 35.

Revised by Scott Yerbury 8/27/15