MANUFACTURING PROCESSES 4–5

MANUFACTURING PROCESSES 4-5

LAMNGEUN VIRASAK

Open Oregon Educational Resources

Contents

Part VIII. Chapter 8: CNC

Introduction

This textbook provides an introduction to the important area of manufacturing processes. This text will explain the hows, whys, and whens of various machining operations, set-ups, and procedures. Throughout this text, you will learn how machine tools operate, and when to use one particular machine instead of another. It is organized for students who plan to enter the manufacturing technology field and for those who wish to develop the skills, techniques, and knowledge essential for advancement in this occupational cluster. The organization and contents of this text focus primarily on theory and practice.

The machining processes and technology sections in this textbook cover such machine tools as surface grinders, bandsaws, drill presses, milling machines, and the engine lathe. Additionally, the importance of Computer Numerical Control (CNC) in the operation of the most machine tools is explained, and its role in automated manufacturing is explored thoroughly.

The Machinist is a skilled worker who uses blueprint drawings, hand tools, precision measuring tools, grinders, lathes, milling machines, and other specialized machine tools to shape and finish metal and nonmetal parts. Machinists must have a good understanding of the following basic and advanced machining practices and technologies.

- Proficiency in safely operating machine tools of various types.
- Knowledge of the working properties of metals and nonmetals.
- Academic skill.

Preface

The primary purpose of this book is to provide an open source textbook that covers most machine tool manufacturing process courses. The material in this textbook was obtained from a variety of sources. All sources are cited in the reference section at the end of each chapter.

Manufacturing and workshop practices have become an important part in the industrial environment to produce products for the service of mankind. Knowledge of manufacturing practices is highly essential for all machinists familiarizing themselves with modern concepts of manufacturing technologies. The requirement is to provide theoretical and practical knowledge of manufacturing processes and workshop technology to all machinist students. Therefore, an attempt has been made throughout this textbook to present both the theoretical and practical knowledge of these subjects.

This text book covers most of the syllabus of Manufacturing Processes 4 and 5. While preparing the manuscript of this textbook, the examination requirements of machinist students have been kept in mind. This book is written in very simple language so that even the average student can easily grasp the subject matter. Some comparisons have been given in tabular form and stress has been given on figures for the better understanding of tools, equipment, machines, and manufacturing setups used in various manufacturing shops.

Table of Contents

PART I

Chapter 1: Milling Machines

1

Milling Machines

Description

The milling machine is one of the most versatile machines in the shop. Usually they are used to mill flat surfaces, but they can also be used to machine irregular surfaces. Additionally, the milling machine can be used to drill, bore, cut gears, and produce slots into a workpiece.

The milling machine uses a multi-toothed cutter to remove metal from moving stock. There is also a quill feed lever on the mill head to feed the spindle up and down. The bed can also be manually fed in the X, Y, and Z axes. Best practices are to adjust the Z axis first, then Y, then X.

When an axis is properly positioned and is no longer to be fed, use the gib locks to lock it in place.

It is common for milling machines to have a power feed on one or more axes. Normally, a forward/reverse lever and speed control knob is provided to control the power feed. A power feed can produce a better surface finish than manual feeding because it is smoother. On long cuts, a power feed can reduce operator fatigue.

Safety

The following procedures are suggested for the safe operation of a milling machine.

1. Have someone assist you when placing a heavy machine attachment like a rotary table, dividing head, or vise.
2. Always refer to speed and feed tables.
3. Always use cutting tools that are sharp and in good condition.
4. Seat the workpiece against parallel bars or the bottom of the vice using a soft hammer or mallet. Check that the work is firmly held and mounted squarely.
5. Remove the wrench after tightening the vice.
6. Most operations require a FORWARD spindle direction. There may be a few exceptions.
7. Make sure there is enough clearance for all moving parts before starting a cut.
8. Make sure to apply only the amount of feed that is necessary to form a clean chip.
9. Before a drill bit breaks through the backside of the material, ease up on the drilling pressure.
10. Evenly apply and and maintain cutting fluids to prevent morphing.
11. Withdraw drill bits frequently when drilling a deep hole. This helps to clear out the chips that may become trapped within the hole.
12. Do not reach near, over, or around a rotating cutter.
13. Do not attempt to clean the machine or part when the spindle is in motion.
14. Stop the machine before attempting to make adjustments or measurements.
15. Use caution when using compressed air to remove chips and shavings. They flying particle may injure you, or those around you.
16. Use a shield or guard for protection against chips.
17. Remove drill bits from the spindle before cleaning to prevent injury.
18. Clean drill bits using a small brush or compressed air.
19. Properly store arbors, milling cutters, collets, adapters, etc., after using them. They can be damaged if not properly stored.
20. Make sure the machine is turned off and clean before leaving the workspace.

2

Unit 1: Tramming the Head

Objective

After completing this unit, you should be able to:

- Describe how to tram the mill head.
- Explain how to indicate the vise.
- Explain the use of spring collets.
- Describe the difference between climb vs. conventional milling.
- Explain how to use an edge finder.
- Describe how to set the quick change gearbox correctly.
- Describe how to square the stock.
- Describe face milling.
- Describe advanced workholding.

Tools For Tramming

A dial indicator is a precision tool used to measure minute amounts of deflection between two surfaces.

When tramming, a dial indicator attached to the chuck is used to determine the orientation of the mill head to the mill table. The same wrench used to tighten and loosen the quill can be used to adjust the various bolts on the mill head.

Dial indicator used for tramming the head.

Tramming the Mill Head

Tramming ensures that the mill head is perpendicular to the mill table's X and Y axis. This process ensures that cutting tools and the milling surfaces are perpendicular to the table. Proper tramming also prevents irregular patterns from forming when milling.

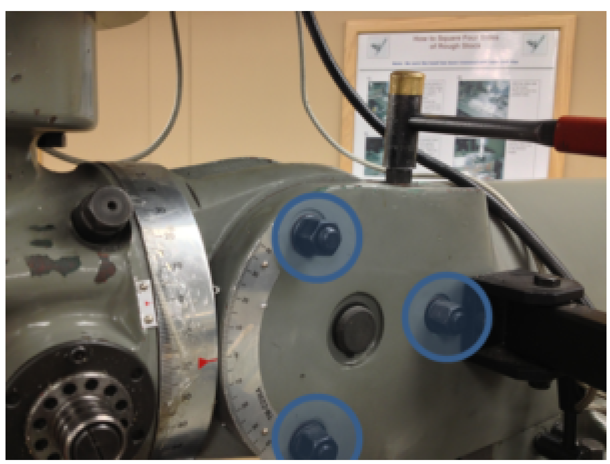
A dial indicator attached to the spindle for precise mill head alignment.

A vertical mill's head is able to tilt from front to back and side to side. Occasionally these adjustments can drift. The mill head should be checked and adjusted periodically, ensuring that the spindle is perpendicular to the table.

1. Remove the vice from the milling table.
2. Attach a dial indicator to the spindle and offset the dial six inches from the spindle's axis. Make sure the indicator probe is facing down.
3. Raise the mill table so that when it contacts the indicator, the indicator reads between 0.005 inches to 0.010 inches. This reading is called the preload.
4. Position the dial indicator so that it is visible, then set the bezel to zero.
5. Hand-turn the spindle while watching the indicator.
6. If the reading on the dial indicator stays at zero, the spindle is aligned.
7. If the reading is not zero, continue tramming the head as shown below.

Tramming Process for the X-Axis

1. To tram around the x-axis (the left-to-right direction of the mill bench when facing the front of the mill), loosen the six bolts (three on each side of the mill) using the mill wrench.



Location of the bolts to be loosened to allow the head to rotate about the X-axis.

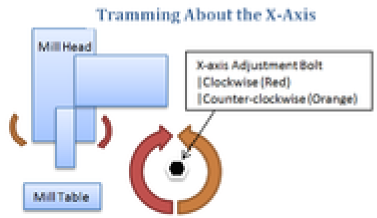
1. After loosening the bolts, re-tighten them by hand plus a ¼ of a turn using the mill wrench.
2. The adjustment bolt that moves the mill head up and down around the x-axis is located at the back of the mill.

Adjustment bolt used to position the mill head vertically around the X-axis.

1. Two protractors are used to indicate general alignment. The larger protractor on the mill head has a red indicating arrow that should align with the zero marker on the curved protractor on the body of the mill. This only provides a general guide, the dial indicator reading is required for precise alignment.
2. Position the dial indicator to the rear of the table. Zero the dial indicator (preloaded at 0.005″ to 0.010″). Be sure to measure on a pristine surface of the mill table. It may be necessary to shift the table to avoid the gaps that are in the table.

Dial indicating around the mill head X-axis.

1. With the dial zeroed and the spindle in neutral, rotate the spindle so that the dial indicator is now on the front of the table, ideally a 180 degree turn. Be sure to grab the clamp that is attached to the spindle (to avoid altering the dial's vertical configuration).
2. Note the direction that the dial rotates to determine the direction that the mill head needs to travel. A clockwise movement requires that the mill head will need to be adjusted up, while a counter-clockwise reading requires that the mill head will need to be adjusted downward.

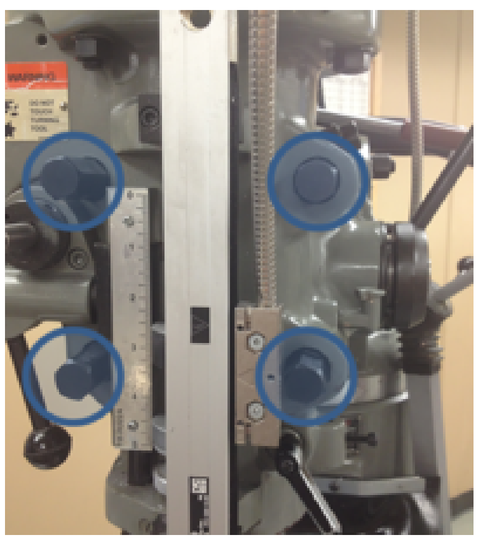


Mill head adjustment about the X-axis.

1. The diagram above shows how movement of the adjustment bolt correlates to movement in the mill head. Once confident in the correct direction the adjustment bolt needs to be turned, adjust the mill head so that ½ the difference between the back and front measurements is reached. For example, if the rear reading is zero and the front reading is 0.010″, adjust the mill head so that the dial reads 0.005″ closer to zero.
2. After the first adjustment is complete, again zero the dial indicator. It is recommended to zero off the same position to avoid confusion, however, it is not necessary. Continue the adjustment process until the difference between the front and the rear is no greater than 0.002 inches.
3. Once satisfied with the readings, begin re-tightening the bolts that were loosened, tightening them evenly in rotation to prevent change in the alignment. Recheck the measurement between the front and the rear to ensure that the mill head did not move significantly from tightening.

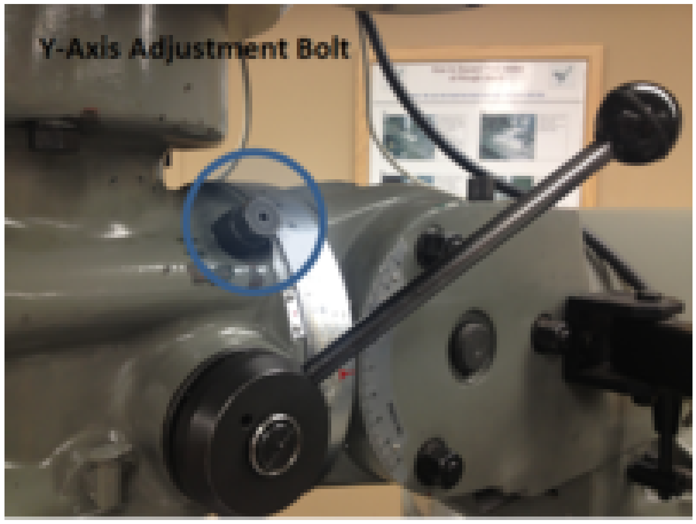
Tramming Process for the Y-Axis

1. To begin tramming about the y-axis, there are four bolts on the front of the mill that need to be loosened to allow movement of the mill head. The bolts should be loosened, then re-tightened to just beyond hand-tight (about ¼ turn past hand-tight with the appropriate wrench).



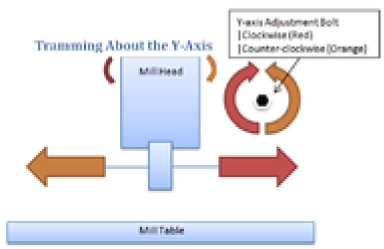
Location of the bolts to be loosened to allow the head to rotate about the Y-axis.

1. The adjustment bolt to move the mill head left and right about the y-axis is shown in the figure below. By twisting this bolt clockwise and counter-clockwise the mill head will move accordingly.



Adjustment bolt used to position the mill head around the Y-axis.

1. The indicating arrow on the protractors for tramming around the y-axis is located on a standalone plate that is in contact with the vertical protractor. This indicating arrow and the zero on the vertical protractor can be used to estimate a starting point for tramming.



Mill head adjustment about the Y-axis.

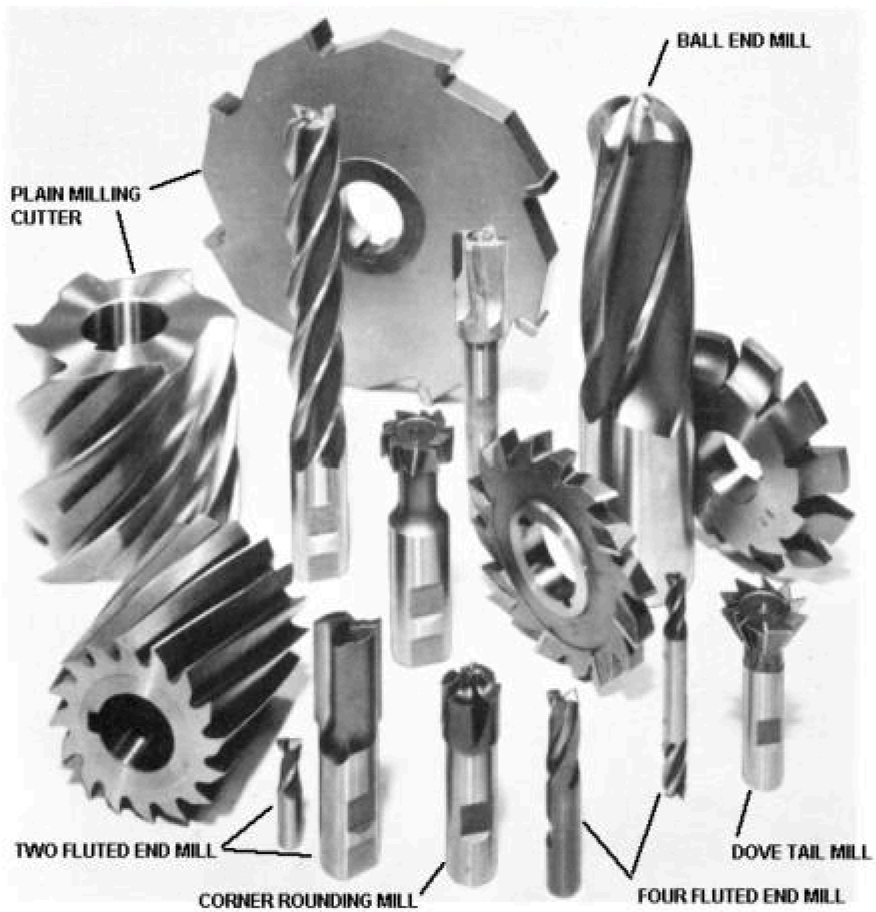
1. The figure above shows how the adjustment bolt for tramming about the y-axis affects the mill head. Use the same process as described for tramming about the x-axis, however, use locations left and right of the mill head as your reference points in contrast to the front and the rear as done previously.
2. Once the adjustments are complete, tighten the bolts on the head of the mill and re-check the measurements about the x-axis and the y-axis. It is possible that the tram in either direction may have been altered by the re-tightening of the bolts. Ensure that all measurements are within 0.002 inches. If the measurements are not within tolerance, the tramming process will have to be redone.

Indicating the Vise

1. Most workpieces are held in a vice that is clamped to the table.
2. It is important to line the vice up with the feed axes on the machine in order to machine features that are aligned with the stock's edges.
3. Fix the vice on the bed by using T-bolts and secure it snugly, while still allowing adjustment to the vice.
4. Install a dial indicator in the machine's spindle with the probe facing away from the operator.
5. Bring the spindle down then position the table's bed until the fixed jaw on the vice is touching the indicator. Continue until the indicator has registered half of a revolution.
6. Set the dial indicator's bezel to zero.
7. Run the indicator across the vice's face with the cross feed.
8. The indicator will stay at zero if the vice is squared.

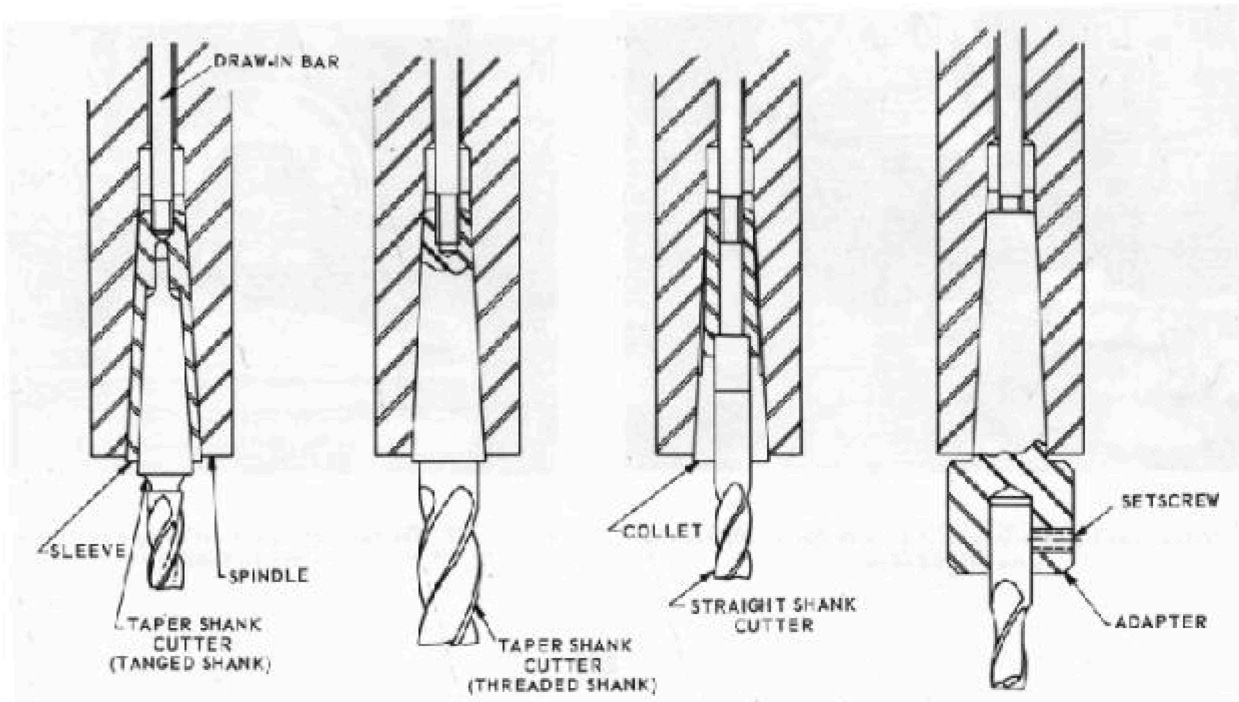
9. If the indicator does not stay at zero, realign the vice by lightly tapping with a soft hammer until the indicator reads half of its previous value.
10. Repeat the process until the dial indicator shows zero through a complete travel from one side of the vice to the other.
11. Fasten the T-bolts securely, while not changing the orientation of the vice. Recheck the alignment of the vice.

Types of Milling Cutters



An assortment of milling cutters.

1. Milling cutters that have solid shafts are usually used in vertical mills.
2. Milling cutters that have keyed holes are usually used in horizontal mills.
3. End mills are used to cut pockets, keyways, and slots.
4. Two fluted end mills can be used to plunge into a workpiece like a drill.
5. 2 and 3 flutes are generally for aluminum, 4 flutes is better for stainless steel. More flutes are better cutting, but come at a higher price.
6. End mills with more than two flutes should not be plunged into the work.
7. Fillets can be produced with ball end mills.
8. Multiple features like round edges can be made by formed milling cutters.



Methods of retaining an end mill.

Spring Collets

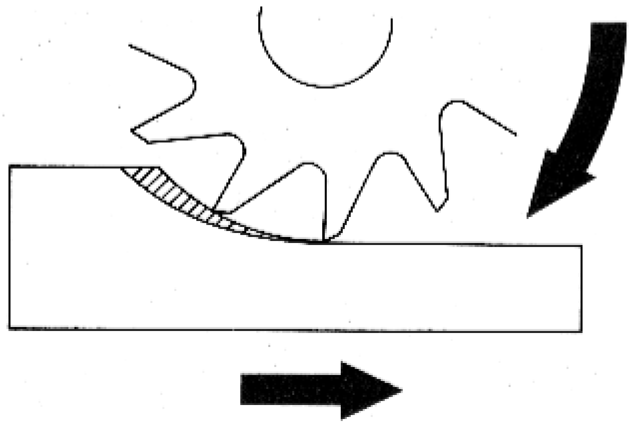
1. If a tool needs to be removed, lock the quill at the highest position.
2. Next, loosen the drawbar with a wrench while using the brake.
3. Make sure that the threads of the draw bar remain engaged in the collet. If they are not engaged, the cutter will fall and potentially be damaged when the collet is released from the spindle.
4. To release the collet from the spindle, tap on the end of the draw bar.
5. Finally, unscrew the drawbar off of the collet.
6. To install a different cutter, place the cutter in a collet that fits the shank.
7. Insert the collet into the spindle while making sure that the keyway aligns properly with the key in the spindle.
8. Begin threading the draw bar into the collet while holding the cutter with one hand. Afterwards, use a wrench to tighten the drawbar while engaging the brake.

Climb vs. Conventional Milling

It is important to know the difference between conventional and climb milling. Using the wrong procedure may result in broken cutters and scrapped workpieces.

Conventional Milling

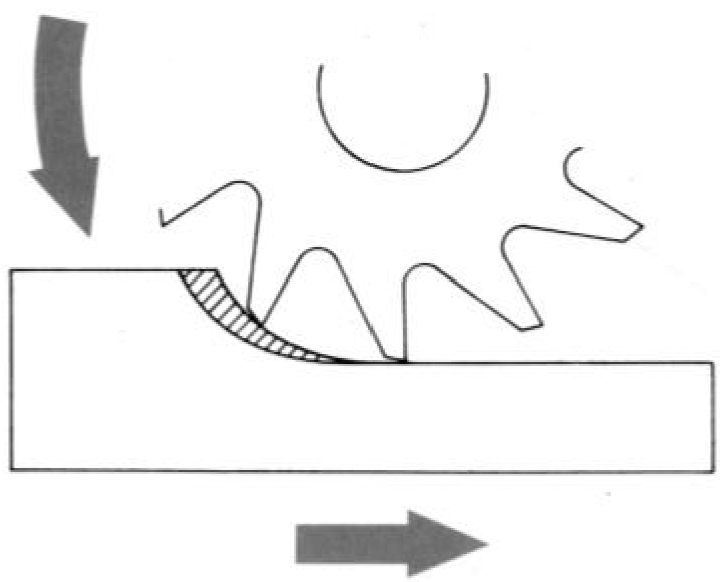
1. The workpiece is fed against the rotation of the cutter.
2. Conventional milling is usually preferred for roughing cuts.
3. Conventional milling requires less force than climb milling.
4. Does not require a backlash eliminator and tight table gibs.
5. Recommended when machining castings and hot-rolled steel.
6. Also recommended when there is a hard surface that has resulted from scale or sand.



Shown above: Conventional Milling

Climb Milling

1. The workpiece is fed with the rotation of the cutter.
2. This method results in a better finish. Chips are not carried into the workpiece, thus not damaging the finish.
3. Fixtures cost less. Climb milling forces the workpiece down, so simple holding devices can be utilized.
4. The chip thickness tends to get smaller the closer it is to an edge, so there is a less chance of an edge breaking, especially with brittle materials.
5. Increases tool life. The tool life can be increased by up to 50% due to chips piling up behind the tool.
6. Chips can be removed easier since the chips fall behind the cutter.
7. Reduces the power needed by 20%. This is due to the use a of a higher rake angle cutter.
8. Not recommended if the workpiece cannot be held securely or if the machine cannot support high forces.
9. Cannot be used to machine castings and hot-rolled steel.
10. This method may pull the workpiece into the cutter and away from the holding device, resulting in broken cutters and scrapped workpieces.



Shown above: Climb Milling

Setting Spindle Speed

1. Spindle speed changes depending on the geometry of the drive train.
2. A hand crank can be used to adjust the spindle speed on newer machines.
3. To change the speed, the spindle has to be rotating.
4. The speed (in RPM) is shown on the dial indicator.
5. There are two scales on the dial indicator for the low and high ranges.
6. A lever is used to change the machine's range.
7. Occasionally, slight rotation of the spindle is necessary for the gears to mate correctly.

Using an Edge Finder

1. The edges of a workpiece must be located before doing mill work that requires great accuracy. An edge finder helps in finding the edges.
2. 800-1200 spindle rpm is recommended.
3. To use an edge finder, slightly offset the two halves so they wobble as they spin.
4. Slowly move the workpiece towards the edge finder.
5. The edge finder will center itself, then suddenly lose concentricity.
6. The digital readout tells you the position of the spindle.
7. The diameter of the edge finder is 0.200″. So adding or subtracting half of that (0.100″) will be the tool center.
8. If centering on the top left, add 0.100″ to the X-axis and subtract 0.100″ from the Y-axis. If centering on the top right, subtract 0.100″ from the X-axis and subtract 0.100″ from the Y-axis.
9. Part Reference Zero is when the bit is zeroed on the X and X axes.
10. A pointed edge finder is a lot easier, but not as precise. Only use a pointed edge finder if precision is not necessary.

Using the Micrometer Dials

1. Most manual feeds on a milling machine have micrometer dial indicators.
2. If the length of the feed is known, the dial indicator should be set to that number (thousandths of an inch).
3. To free the dial indicator, rotate the locking ring counterclockwise. Set the dial and re-tighten.
4. Before setting the dial indicator, ensure that the table-driving mechanism backlash is taken up.
5. It is common for newer machines to have digital readouts, which are preferable because they directly measure table position. When using a digital readout, backlash concerns are negated.

Squaring Stock

1. When making a square corner, vertically orient a completed edge in the vice and clamp it lightly to the part.
2. Place machinist's square against the completed edge and the base of the vice.
3. Align the workpiece with the square by tapping it lightly with a rubber mallet.
4. Firmly clamp the vice.
5. The top edge of the part is ready to be milled.

Face Milling

1. It is frequently necessary to mill a flat surface on a large workpiece. This is done best using a facing cutter.
2. A cutter that is about an inch wider than the workpiece should be selected in order to finish the facing in one pass.

Shown above: Face milling

Milling Slots

1. Square slots can be cut using end mills.
2. In one pass, slots can be created to within two one-thousandths of an inch.
3. Use an end mill that is smaller than the desired slot for more accuracy.
4. Measure the slot and make a second pass to open the slot to the desired dimension.
5. The depth of cut should not exceed the cutter diameter.

Advanced Workholding

1. Use a v-block to secure round stock in a vice. It can be used both horizontally and vertically.
2. Clamping round stock in a v-block usually damages the stock.
    - 2.1 Collet blocks are made to hold round workpieces.
    - 2.1 To mill features at 90 degree increments, use a square collet block.
    - 2.1 To mill features at 60 degree increments, use a hexagonal block.
3. It is easiest to set up stock when the features are perpendicular or parallel to the edges of the workpiece. It is more difficult to set up a workpiece when features are not parallel or perpendicular to the edges. Sometimes, an angle plate can be used to mill stock at any desired angle.
4. Parts that don't fit well in a vise can be directly secured to the table with hold-down clamps.
5. Use parallels to create a gap between the work and bed.
6. Slightly tilt the clamps down into the work.
7. Rotary tables can be put on the bed to make circular features.
    - 7.1 Rotary tables allow rotation of the workpiece.
    - 7.1 Use a dial indicator to precisely control the angle of rotation.
8. Use a ball for irregularly shaped workpieces. Make sure to only take a small cuts to avoid throwing the workpiece out of the vice.

UNIT TEST

1. What tool is used for tramming the head?
2. Explain the process for the X-axis tramming.
3. Explain the process for the Y-axis tramming.
4. What is the purpose of indicating the vise?
5. Name three types of milling cutters.
6. Explain how a spring collet works.
7. What is the difference between conventional and climb milling?
8. Describe briefly how a rotary table may be centered with the vertical mill spindle.
9. Describe briefly how to set spindle speed on the milling machine.

10. What tool is used for milling large workpiece surfaces?

3

Unit 2: Speeds, Feeds, and Tapping

Objective

After completing this unit, you should be able to:

- Identify and select vertical milling machine setups and operations for a variety of machining tasks.
- Select a proper cutting speed for different types of materials.
- Calculate cutting speeds and feeds for end milling operations.
- Explain how to correctly set up for power feed tapping.

Cutting Speed

Cutting speed is defined as the speed at the outside edge of the tool as it is cutting. This is also known as surface speed. Surface speed, surface footage, and surface area are all directly related. If two tools of different sizes are turning at the same revolutions per minute (RPM), the larger tool has a greater surface speed. Surface speed is measured in surface feet per minute (SFM). All cutting tools work on the surface footage principle. Cutting speeds depend primarily on the kind of material you are cutting and the kind of cutting tool you are using. The hardness of the work material has a great deal to do with the recommended cutting speed. The harder the work material, the slower the cutting speed. The softer the work material, the faster the recommended cutting speed (See Figure 1).

Steel Iron Aluminum Lead

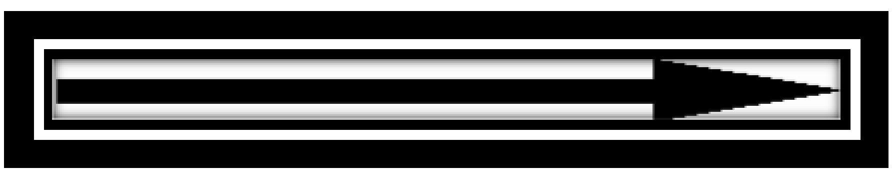


Figure 1: Increasing Cutting Speed Based on work material hardness

The hardness of the cutting tool material will also have a great deal to do with the recommended cutting speed. The harder the drill, the faster the cutting speed. The softer the drill, the slower the recommended cutting speed (See Figure 2).

Carbon Steel High Speed Steel Carbide

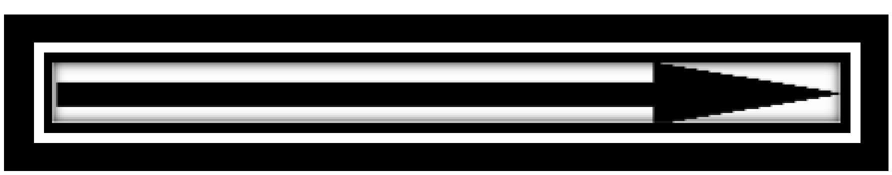


Figure 2: Increasing Cutting Speed Based on Cutting tool hardness

Table 1: Cutting Speeds for Material Types

Type of Material	Cutting Speed (SFM)
Low Carbon Steel	40-140
Medium Carbon Steel	70-120
High Carbon Steel	65-100
Free-machining Steel	100-150
Stainless Steel, C1 302, 304	60
Stainless Steel, C1 310, 316	70
Stainless Steel, C1 410	100
Stainless Steel, C1 416	140
Stainless Steel, C1 17-4, pH	50
Alloy Steel, SAE 4130, 4140	70
Alloy Steel, SAE 4030	90
Tool Steel	40-70
Cast Iron–Regular	80-120
Cast Iron–Hard	5-30
Gray Cast Iron	50-80
Aluminum Alloys	300-400
Nickel Alloy, Monel 400	40-60
Nickel Alloy, Monel K500	30-60
Nickel Alloy, Inconel	5-10
Cobalt Base Alloys	5-10
Titanium Alloy	20-60
Unalloyed Titanium	35-55
Copper	100-500
Bronze–Regular	90-150
Bronze–Hard	30-70
Zirconium	70-90
Brass and Aluminum	200-350
Silicon Free Non-Metallics	100-300
Silicon Containing Non-Metallics	30-70

Spindle Speed

Once the SFM for a given material and tool is determined, the spindle can be calculated since this value is dependent on cutting speed and tool diameter.

$$RPM = (CS \times 4) / D$$

Where:

- RPM = Revolutions per minute.
- CS = Cutter speed in SFM.

- D = Tool Diameter in inches.

Milling Feed

The feed (milling machine feed) can be defined as the distance in inches per minute that the work moves into the cutter.

On the milling machines we have here at LBCC, the feed is independent of the spindle speed. This is a good arrangement and it permits faster feeds for larger, slowly rotating cutters.

The feed rate used on a milling machine depends on the following factors:

1. The depth and width of cut.
2. The type of cutter.
3. The sharpness of the cutter.
4. The workpiece material.
5. The strength and uniformity of the workpiece.
6. The finish required.
7. The accuracy required.
8. The power and rigidity of the machine, the holding device, and the tooling setup.

Feed per Tooth

Feed per tooth, is the amount of material that should be removed by each tooth of the cutter as it revolves and advances into the work.

As the work advances into the cutter, each tooth of the cutter advances into the work an equal amount producing chips of equal thickness.

This chip thickness or feed per tooth, along with the number of teeth in the cutter, form the basis for determining the rate of feed.

The ideal feed rate for milling is measured in inches per minute (IPM) and is calculated by this formula:

$$IPM = F \times N \times RPM$$

Where:

- IPM = feed rate in inches per minute
- F = feed per tooth
- N = number of teeth
- RPM = revolutions per minute

For Example:

Feeds for end mills used in vertical milling machines range from .001 to .002 in. feed per tooth for very small diameter cutters on steel work material to .010 in. feed per tooth for large cutters in aluminum workpieces. Since the cutting speed for mild steel is 90, the RPM for a 3/8" high-speed, two flute end mill is

$$RPM = CS \times 4 \;/\; D = 90 \times 4 \;/\; (3/8) = 360 \;/ .375 = 960\ RPM$$

To calculate the feed rate, we will select .002 inches per tooth

$$IPM = F \times N \times RPM = .002 \times 2 \times 960 = 3.84\ IPM$$

Machine Feed

The machine movement that causes a cutting tool to cut into or along the surface of a workpiece is called feed.

The amount of feed is usually measured in thousandths of an inch in metal cutting.

Feeds are expressed in slightly different ways on various types of machines.

Drilling machines that have power feeds are designed to advance the drill a given amount for each revolution of the spindle. If we set the machine to feed at .006” the machine will feed .006” for every revolution of the spindle. This is expressed as (IPR) inches per revolution

Tapping Procedures

Good Practices:

Using Tap Guides

Tap guides are an integral part in making a usable and straight thread. When using the lathe or the mill, the tap is already straight and centered. When manually aligning a tap, be careful, as a 90° tap guide is much more accurate than the human eye.

Using Oil

When drilling and tapping, it is crucial to use oil. It keeps the bits from squealing, makes the cut smoother, cleans out the chips, and keeps the drill and stock from overheating.

Pecking

Pecking helps ensure that bits don’t overheat and break when using them to drill or tap. Peck drilling involves drilling partway through a part, then retracting it to remove chips, simultaneously allowing the piece to cool. Rotating the handle a full turn then back a half turn is common practice. Whenever the bit or tap is backed out, remove as many chips as possible and add oil to the surface between the drill or tap and the workpiece.

Hand Tapping Procedure

1. **Select a drill size from the chart.**

When choosing a tap size, this chart is the first place to look.

Screw Size	Major Diameter	Threads Per Inch	Minor Diameter	Tap Drill				Clearance Drill			
				75% Thread for Aluminum, Brass, & Plastics		50% Thread for Steel, Stainless, & Iron		Close Fit		Free Fit	
				Drill Size	Decimal Equiv.	Drill Size	Decimal Equiv.	Drill Size	Decimal Equiv.	Drill Size	Decimal Equiv.
0	.0600	80	.0447	3/64	.0469	55	.0520	52	.0635	50	.0700
1	.0730	64	.0538	53	.0595	1/16	.0625	48	.0760	46	.0810
		72	.0560	53	.0595	52	.0635				
2	.0860	56	.0641	50	.0700	49	.0730	43	.0890	41	.0960
		64	.0668	50	.0700	48	.0760				
3	.0990	48	.0734	47	.0785	44	.0860	37	.1040	35	.1100
		56	.0771	45	.0820	43	.0890				
4	.1120	40	.0813	43	.0890	41	.0960	32	.1160	30	.1285
		48	.0864	42	.0935	40	.0980				
5	.125	40	.0943	38	.1015	7/64	.1094	30	.1285	29	.1360
		44	.0971	37	.1040	35	.1100				
6	.138	32	.0997	36	.1065	32	.1160	27	.1440	25	.1495
		40	.1073	33	.1130	31	.1200				
8	.1640	32	.1257	29	.1360	27	.1440	18	.1695	16	.1770
		36	.1299	29	.1360	26	.1470				
10	.1900	24	.1389	25	.1495	20	.1610	9	.1960	7	.2010
		32	.1517	21	.1590	18	.1695				
12	.2160	24	.1649	16	.1770	12	.1890	2	.2210	1	.2280
		28	.1722	14	.1820	10	.1935				
		32	.1777	13	.1850	9	.1960				
1/4	.2500	20	.1887	7	.2010	7/32	.2188	F	.2570	H	.2660
		28	.2062	3	.2130	1	.2280				
		32	.2117	7/32	.2188	1	.2280				
5/16	.3125	18	.2443	F	.2570	J	.2770	P	.3230	Q	.3320
		24	.2614	I	.2720	9/32	.2812				
		32	.2742	9/32	.2812	L	.2900				
3/8	.3750	16	.2983	5/16	.3125	Q	.3320	W	.3860	X	.3970
		24	.3239	Q	.3320	S	.3480				
		32	.3367	11/32	.3438	T	.3580				
7/16	.4375	14	.3499	U	.3680	25/64	.3906	29/64	.4531	15/32	.4687
		20	.3762	25/64	.3906	13/32	.4062				
		28	.3937	Y	.4040	Z	.4130				
1/2	.5000	13	.4056	27/64	.4219	29/64	.4531	33/64	.5156	17/32	.5312
		20	.4387	29/64	.4531	15/32	.4688				
		28	.4562	15/32	.4688	15/32	.4688				

9/16	.5625	12	.4603	31/64	.4844	33/64	.5156	37/64	.5781	19/32	.5938
		18	.4943	33/64	.5156	17/32	.5312				
		24	.5114	33/64	.5156	17/32	.5312				
5/8	.6250	11	.5135	17/32	.5312	9/16	.5625	41/64	.6406	21/32	.6562
		18	.5568	37/64	.5781	19/32	.5938				
		24	.5739	37/64	.5781	19/32	.5938				
11/16	.6875	24	.6364	41/64	.6406	21/32	.6562	45/64	.7031	23/32	.6562
3/4	.7500	10	.6273	21/32	.6562	11/16	.6875	49/64	.7656	25/32	.7812
		16	.6733	11/16	.6875	45/64	.7031				
		20	.6887	45/64	.7031	23/32	.7188				
13/16	.8125	20	.7512	49/64	.7656	25/32	.7812	53/64	.8281	27/32	.8438
7/8	.8750	9	.7387	49/64	.7656	51/64	.7969	57/64	.8906	29/32	.9062
		14	.7874	13/16	.8125	53/64	.8281				
		20	.8137	53/64	.8281	27/32	.8438				
15/16	.9375	20	.8762	57/64	.8906	29/32	.9062	61/64	.9531	31/32	.9688
1	1.000	8	.8466	7/8	.8750	59/64	.9219	1-1/64	.0156	1-1/32	1.0313

1. **If necessary, add a chamfer to the hole before tapping.**
Chamfers and countersinks are additional features that are sometimes desired for screws. For best results, the speed of the spindle should be between 150 and 250 rpm.
2. **Get a tap guide.**
The hole is now ready to tap. To do this, use the taps and guide blocks near the manual mills. The guide blocks will have several holes for different sized taps. Select the one closest to the size of the tap being used and place it over the drilled hole.
3. **Tap the threads.**
Peck tap using the tap wrenches. Apply gentle pressure while turning the wrench a complete turn in, then a half-turn out. Peck tap to the desired depth.
4. **Complete the tap.**
If the tap does not go any further or the desired depth has been reached, release pressure on the tap; it has likely bottomed out. Remove the tap from the hole.Applying any more pressure is likely to break the tap. The smaller the tap, the more likely it is to break.

Power Feed Tapping Procedure (Vertical Mill)

1. Power feed tapping is similar to hand tapping. Instead of tapping by hand, however, use the vertical mill to tap the workpiece.
2. Before starting the machine, change the mill to low gear.
3. Release the quill lock and move the quill to the lowest it can go. This ensures that there is sufficient space to tap to the desired depth.
4. Turn the spindle on FORWARD and set the spindle speed to 60 RPM.
5. Feed the tap down. When the tap grabs the stock, it will automatically feed itself into the hole.
6. When the desired depth has been reached, quickly flip the spindle direction switch from forward to reverse. This will reverse the direction of the tap and remove it from the hole. Reversing the direction in one fluid motion will prevent damage to the tapped hole and the tap.
7. Turn off the machine.
8. Clean the tapped hole, tap, and power feed machine before leaving.

UNIT TEST

1. Explain cutting speeds for harder and softer materials.

2. What is the cutting speed for Tool Steel and Aluminum?
3. Calculate the RPM for a ½ in. diameter HSS end mill to machine aluminum.
4. Calculate the feed rate for a three-flute tool. Use the RPM from Question 3.
5. Calculate the RPM for a ¾ in. diameter HSS end mill to machine bronze.
6. Calculate the feed rate for two-flute ½ in. diameter carbide end mill to machine low-carbon steel.
7. What is the purpose of pecking when using them to drill or tap?
8. Select a proper drill size for 5/16 – 24 tap.
9. Why are cutting fluids used?
10. Describe the difference between hand and power feed tapping.

4

Unit 3: Sine Bar

Objective

After completing this unit, you should be able to:

- Understand the principle of the sine bar.
- Explain how to use a sine bar correctly.
- Understand slip gauge blocks and wringing.
- Calculate gauge block height.

The Sine Bar

A sine bar is used in conjunction with slip gauge blocks for precise angular measurement. A sine bar is used either to measure an angle very accurately or face locate any work to a given angle. Sine bars are made from a high chromium corrosion resistant steel, and is hardened, precision ground, and stabilized.

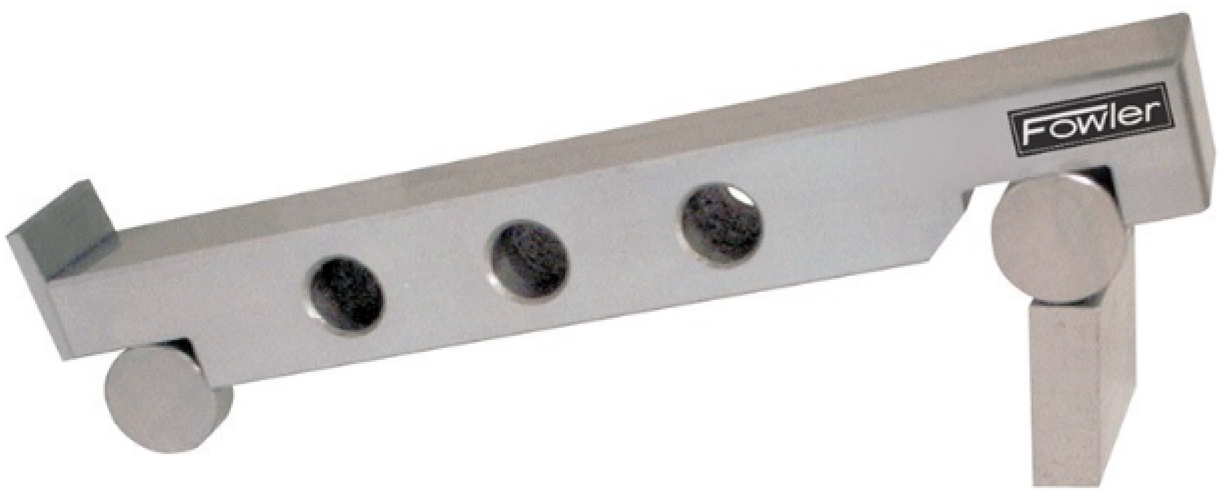


Figure 1. The Sine Bar

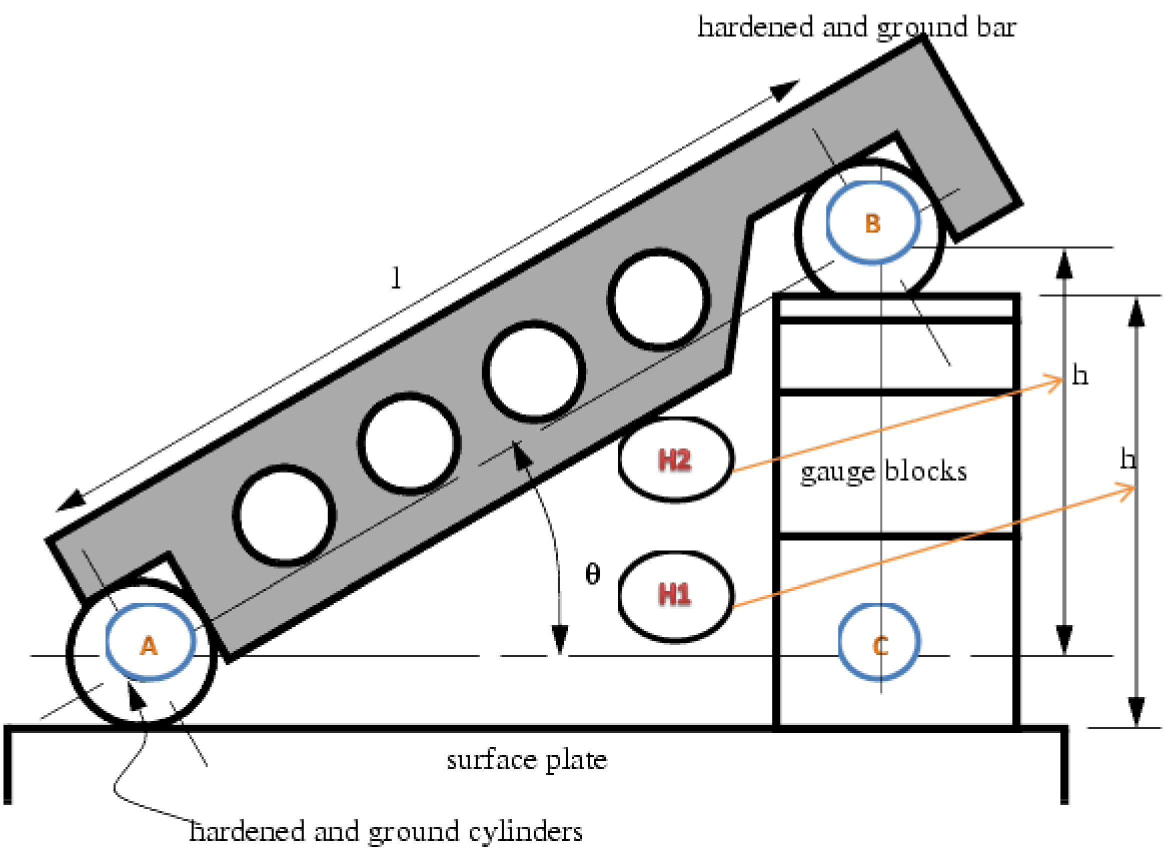
Two cylinders of equal diameter are placed at the ends of the bar. The axes of these two cylinders are mutually parallel to each other, and are also parallel to, and at equal distance from, the upper surface of the sine bar. Accuracy up to 0.01mm/m of length of the sine bar can be obtained.

A sine bar is generally used with slip gauge blocks. The sine bar forms the hypotenuse of a right triangle, while the slip gauge blocks form the opposite side. The height of the slip gauge block is found by multiplying the sine of the desired angle by the length of the sine bar: **$H = L * \sin(\theta)$**.

For example, to find the gauge block height for a **13°** angle with a **5.000″** sine bar, multiply the **sin(13°)** by **5.000″**: **$H = 5.000'' * \sin(13^\circ)$**. Slip gauge blocks stacked to a height of **1.124″** would then be used elevate the sine bar to the desired angle of **13°**.

Sine Bar Principles

- The application of trigonometry applies to sine bar usage.
- A surface plate, sine bar, and slip gauges are used for the precise formation of an angle.
- It is possible to set up any angle ϴ by using the standard length of side **AB**, and calculating the height of side **BC** using **BC = AB * sin(ϴ)**.
- The angle ϴ is given by ϴ **= asin(BC/AB)**.
- Figure 1 shows a typical sine bar set up on a surface plate with slip gauge blocks of the required height **BC** to form a desired angle ϴ.



l = distance between centres of ground cylinders (typically 5" or 10")
h = height of the gauge blocks
θ = the angle of the plate

$$\boxed{\theta = \operatorname{asin}\left(\frac{h}{l}\right)}$$

Figure 2: Forming an Angle with a Sine Bar and Gauge Blocks

Wringing

The term wringing refers to a condition of intimate and complete contact by tight adhesion between measuring faces. Wringing is done by hand by sliding and twisting motions. One gauge is placed perpendicular to other using standard gauging pressure then a rotary motion is applied until the blocks are lined up. In this way air is expelled from between the gauge faces causing the blocks to adhere. This adherence is caused partially by molecular attraction and partially by atmospheric pressure. Similarly, for separating slip gauges, a combined sliding and twisting motion should be used.

1. To set an angle on any sine bar, you must first determine the center distance of the sine bar (C), the angle you wish to set (A) and whether the angle is in degrees-minutes-seconds or decimal degrees.
2. Next, enter that information in the appropriate input areas below. Use a decimal point for the separator, whether the angle is in degrees-minutes-seconds or decimal degrees.
3. Hit the 'Calculate' button and then assemble a stack of gauge blocks (G) to equal the size that is returned. The units of the stack will match the units of the center distance (i.e., If you enter the center distance as 5 for a 5 inch sine plate, the gage block stack will also be in inches.).
4. Place these slip gauges blocks under the gauge block roll of the sine device and the desired angle is set.
5. Tighten the locking mechanism on those devices that have one and you're ready to go.

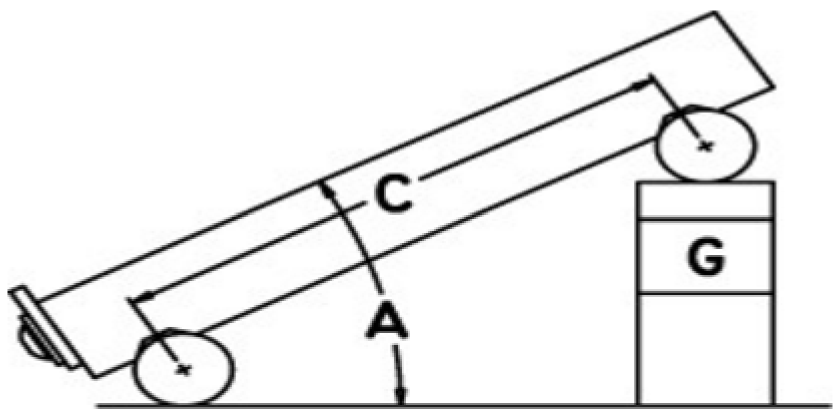


Figure 3: Uses the formula: G = C * Sin(A)

If you just want to set an angle with a sine bar and stack of blocks, then take the sine of the desired angle on your calculator and multiply the result by the distance between the centers of the cylinders in the sine bar. Assemble a stack of blocks equal to this value and put it under one of the cylinders.

Sine Bar Set-Up Calculation

To calculate the gauge block's height needed to set-up a sine bar to a specific angle all you have to do is take the SIN of the angle and multiply it by the sine bar length. The length of the sine bar is the distance between the centers of the sine bar gauge pins.

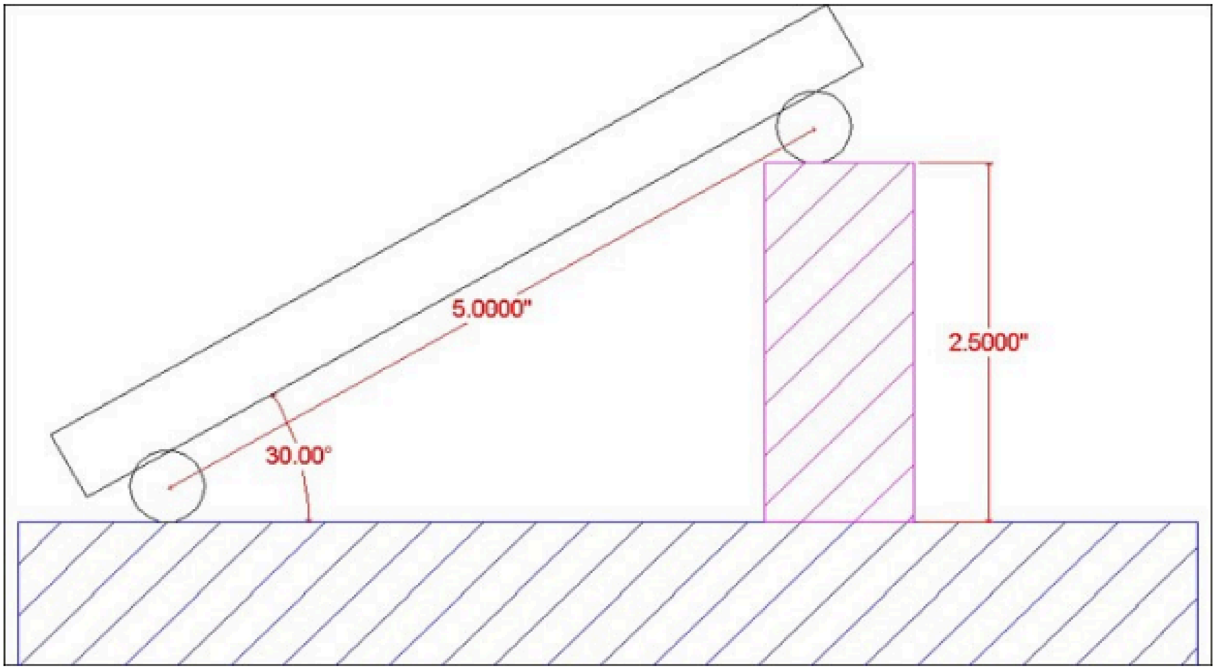


Figure 4. Sine Bar

Example:
Set up a 5.0" sine bar or sine plate to 30°
SIN (30°) = 0.5000
0.5000 x 5.0″ (sine Bar Length) = 2.5000″
Round 2.5000″ to 4 Decimal Places = 2.5000″ Gage Block Height.

Table 1 Common Angles and heights for a 5-inch sine bar:

Angle	Height
5°	0.4358″
10°	0.8682″
15°	1.2941″
20°	1.7101″
25°	2.1131″
30°	2.5000″
35°	2.8679″
40°	3.2139″
45°	3.5355″
50°	3.8302″
55°	4.0958″
60°	4.3301″

Sine Bar Usage

To measure a known angle or locate any work to a given angle:

1. Always use a perfectly flat and clean surface plate.
2. Place one roller on the surface plate and the other roller on the slip gauge block stack of height **H**.
3. Let the sine bar be set to an angle ϴ.
4. Then sin(ϴ) = H/L, where L is the distance between the center.
5. Thus knowing ϴ, H can be found and any work can be set out at this angle as the top face of the sine bar is inclined at angle ϴ to the surface plate.
6. For better result both rollers must placed on slip gauge block of height H1 and H2 respectively. See above figure,
7. ??? ? = **(?? − ??) / L**

UNIT TEST

1. Describe the use of the sine bar.
2. Calculate the required sine bar elevation for angle of 37˚.
3. A 5.00" sine bar is elevated 1.50". Calculate the angle.
4. Determine the elevation for 30˚ using 5.00" sine bar.
5. Determine the elevation for 42˚ using 5.00" sine bar.
6. A 5.00" sine bar is elevated 1.25". What angle is established?
7. What gauge block stack would establish an angle of 35˚ using a 5.00" sine bar?

5

Unit 4: Offset Boring Head

OBJECTIVE

After completing this unit, you should be able to:

- Identify offset Boring head
- Explain how to correct set up for Rotary Table.

Offset Boring Head

The offset boring is an attachment that fits the milling machine spindle and permits most drilled holes to have a better finish and greater diameter accuracy. Offset boring head are used to create large hole when tolerance do not allow for a drill bit or do not have a large enough drill or reamer. A offset boring head can be used to enlarge hole, or adjust hole centerline in certain instances.

Safety:

Be sure all set screws are tight before operation. Be sure offset boring head has a clearance to fit into hole when boring. Remove Allen wrench before turning the mill one. Double check mill speed before operation.

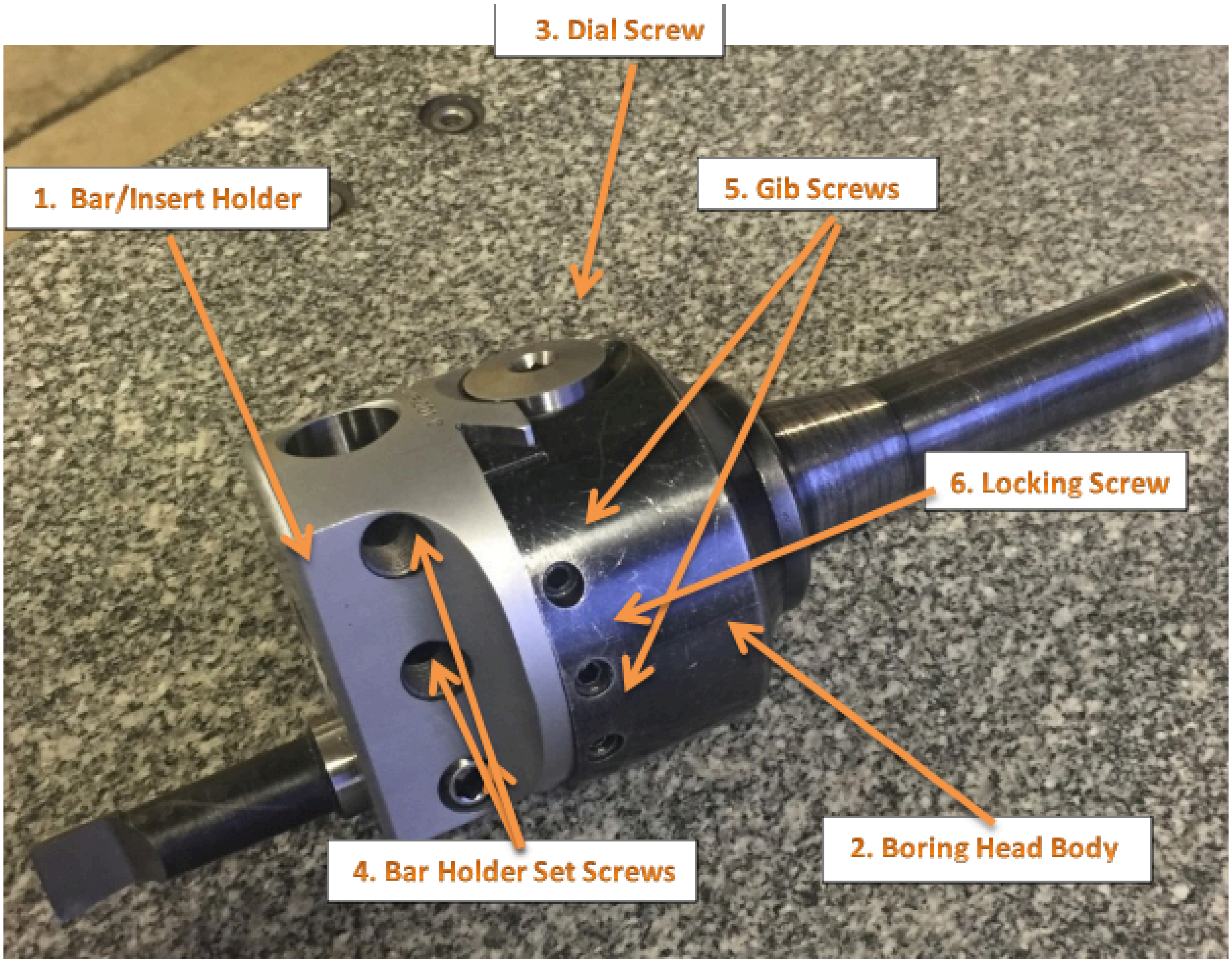


Figure 1. Offset Boring Head

OFFSET BORING HEAD AND TOOLS

Figure 1. shows an offset boring head. Note that the boring bar can be adjusted at a right angle axis. This feature makes it possible to position the boring cutter accurately to bore holes of varying diameters.

This adjustment is more convenient than adjusting the cutter in the boring bar holder or changing the boring bar. Another advantage of the offset boring head is the fact that graduated micrometer collar allows the tool to be moved accurately a specified amount usually in increments of (0.001) without the use of dial indicator or other measuring device.

Offset Boring Head

A Boring Heads have three major components:

- boring head body
- bar holder/insert holder
- dial screw

The boring head body has a black oxide finish for rust prevention. The bar holder or insert holder (#1) has been satin chromed for wear resistance. The dial screw (#3) has been precision ground to give accurate movement of the bar holder/insert holder in the dove tail slide. The gib tension has been preset at the factory. The two gib screws (#5) should not be loosened to make size adjustments. These screws are for adjusting the gib pressure only and are filled with red wax to prevent accidental adjustment. The locking screw (#6) is the only screw used for making size changes to the boring head.

Diameter Adjustment

To adjust the diameter of an Allied Criterion standard boring head:

1. Loosen the locking screw (#6).

2. Turn the dial screw (#3) clockwise to increase the diameter and counterclockwise to decrease the diameter.

3. Tighten the locking screw (#6). Adjusting Standard Boring Heads

Procedure:

1. Set up and carefully align the work parallel to the table travel.
2. Align the center of the Milling Machine spindle with the reference point on the work.
3. Spot the location of hole with a center drill or spotting tool.
4. Drilled hole over ½ inch, Be sure offset boring head has a clearance to fit into hole when boring.
5. Install bore head into Milling Machine.
6. Install boring bar and tighten set screw and loosen lock screw and adjust boring bar to hole edge.
7. Recheck the work alignment, as well as the alignment of the spindle with the reference point, to make sure it has not shifted. If any error is evident, it will necessary to repeat procedure 6 before processing.
8. Adjust Milling Machine speed for hole size and material.
9. Engage worm feed on Mill. Bring quill to material. Pull handle out to engage power feed. When at desired depth push hand back to disengage feed and then turn off Mill. Remove boring head from hole.
10. Finish bore hole to the required size.

NOTE: Repeat Procedures 6-9 until hole is desired size.

Rotary Table

A rotary table can be used to make arcs and circles. For example, the circular T-slot in the swivel base for a vise can be made using a rotary table. Rotary tables can also be used for indexing, where a workpiece must be rotated an exact amount between operations. You can make gears on a milling machine using a rotary table. Dividing plates make indexing with a rotary table easier.

Rotary tables are most commonly mounted "flat", with the table rotating around a vertical axis, in the same plane as the cutter of a vertical milling machine. An alternate setup is to mount the rotary table on its end (or mount it "flat" on a 90° angle plate), so that it rotates about a horizontal axis. In this configuration a tailstock can also be used, thus holding the workpiece "between centers."

With the table mounted on a secondary table, the workpiece is accurately centered on the rotary table's axis, which in turn is centered on the cutting tool's axis. All three axes are thus coaxial. From this point, the secondary table can be offset in either the X or Y direction to set the cutter the desired distance from the workpiece's center. This allows concentric machining operations on the workpiece. Placing the workpiece eccentrically a set distance from the center permits more complex curves to be cut. As with other setups on a vertical mill, the milling operation can be either drilling a series of concentric, and possibly equidistant holes, or face or end milling either circular or semicircular shapes and contours.

A rotary table can be used:

- To machine spanner flats on a bolt
- To drill equidistant holes on a circular flange
- To cut a round piece with a protruding tang
- To create large-diameter holes, via milling in a circular toolpath, on small milling machines that don't have the power to drive large twist drills (>0.500″/>13 mm)
- To mill helixes
- To cut complex curves (with proper setup)
- To cut straight lines at any angle
- To cut arcs
- With the addition of a compound table on top of the rotary table, the user can move the center of rotation to anywhere on the part being cut. This enables an arc to be cut at any place on the part.
- To cut circular pieces

Setting Up a Rotary Table

When using a rotary table on a Milling Machine, whether to mill an arc or drill holes in some circular pattern, there are two things that must be done to set up the workpiece. First, the workpiece must be centered on the rotary table. Second, the rotary table must be centered under the spindle. Then the mill table can be moved some appropriate distance and you can start cutting.

You could center the table under the spindle first, by indicating off the hole in the center of the table. Then you could mount the workpiece on the table and indicate off the workpiece. There are two problems with this approach. First, you are assuming that the hole in the table is true and centered. That may or may not be true. Second, this approach risks a sort of accumulation of errors, as you're measuring from two different features (the rotary table's hole and some feature on the workpiece). First center the workpiece on the rotary table, and then center the rotary table under the spindle.

To center the workpiece on the rotary table, spin the rotary table and watch for deflection of the indicator pointer. Adjust the position of the mill table(X and Y) as required, until the needle no longer deflects.

You dial in a rotary table by placing a dial test indicator in a chuck or collet in the spindle, which is then rotated by hand with the indicator tip in contact with the hole of the rotary table. If your machine can be taken out of gear, it helps to do so, so the spindle swings freely. It's obviously easier to use a drill chuck than a collet, too, so you have something that you can turn easily. Make your adjustments using the saddle and table hand wheels.

Once you have center located (the indicator will read the same as you rotate the spindle, it's a very good idea to set both of your dials at "0", instead of marking some random location. Make sure you have backlash set properly, too. Set the dial is reading in a positive direction so it's easy to count off any changes, and you never have to remember which way you had chosen to set backlash. I also always mark the table and saddle with a wax pencil so I know where center is located. That tells you when to stop turning the handle when "0" comes around if you want to get the table back to center to load another part.

Once you have located center of the table and have set dials and locked the table and saddle, you usually have some feature on your part that you desire to be centered. In some cases it may be a hole, in others it may be the outside edge of the circular part. In a case like either of these, it's common practice to use the same indicator and swing it inside the hole or the perimeter of the part. The perimeter may require you to get around clamps, which can usually be accomplished by using the quill to move the indicator up far enough to clear them. When you dial in parts to a table that has already been located, you tap the part around, you do not make adjustments with the saddle or table handles. Tap the part after you've snugged up the clamps slightly, so it doesn't move about wildly. You can achieve virtually perfect location that way, certainly as close as the machine is capable of working.

After the workpiece is centered on the rotary table, you now turn the spindle by hand, so the indicator tip sweeps the inside of the hole. Adjust the position of the mill table as required until no needle deflection is noted.

Setting up your Rotary Table

How to center the spindle over the center of the rotary table. Here are some of the methods to use.

To Center the Rotary Table with the Vertical Mill Spindle

Follow The following procedure:

1. Square the vertical head with the machine table.

2. Mount the rotary table on the milling machine table.

3. Place a test plug in the center hole of the rotary table.

4. Mount an dial indicator in the milling machine spindle.

5. With the dial indicator just clearing the top of the test plug, rotate the machine spindle by hand and approximately align the plug with the spindle.

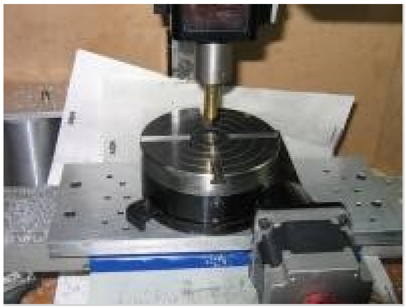
6. Bring the dial indicator into contact with the diameter of the plug, and rotate the spindle by hand.

7. Adjust the machine table by the longitudinal(X) and crossfeed(Y) handles until the dial indicator registers no movement.

8. Lock the milling machine table and saddle, and recheck the alignment.

9. Readjust if necessary.

A way to setup your rotary table



Rough Position

Made a 3/8″ piece of brass and put a 60 degree point on it. It Sh

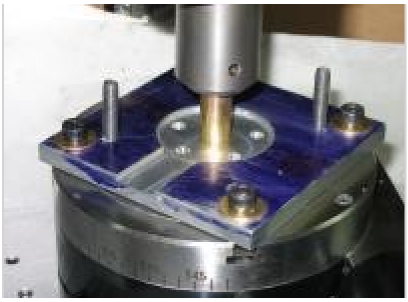
Visual Position

To perform a visual position. Your eye is pretty good and judgi

Indicate

To get a really accurate, to dial indicate in the rotary table. In t
table. I then run the table through 360 degrees of rotation watch
The true center will be half way between the two readings.

For the final adjusting for centering that on the same side of the
in the same direction when doing the center adjustment. If on



Lineup Jig

To locate a jig or workpiece on the rotary table. I start off wi

Indicate Jig

Centering the jig or workpiece over the center of the rotary table.

To Center a Workpiece with the Rotary Table

Often it is necessary to perform a rotary table operation on several identical workpieces, each having a machined hole in the center. To quickly align each workpiece, a special plug can be made to fit the center hole of the workpiece and the hole in the rotary table. Once the machine spindle has been aligned with the rotary table, each succeeding piece can be aligned quickly and accurately by placing it over the plug.

If there are only a few pieces, which would not justify the manufacture of a special plug, or if the workpiece does not have a hole through it center, the following method can be used to center the workpiece on the rotary table.

1. Align the rotary table with the vertical mill head spindle.

2. Lightly clamp the workpiece on the rotary table in the center. Do not move the longitudinal(X) or crossfeed(Y) feed handles.

3. Disengage the rotary table worm mechanism.

4. Mount an dial indicator in the milling machine spindle or milling machine table, depending upon the workpiece.

5. Bring the dial indicator into contact with the surface to be indicated, and revolve the rotary table by hand.

6. With a soft metal bar, tap the workpiece(away from the indicator movement) until no movement is registered on the indicator in a complete revolution of the rotary table.

7. Clamp the workpiece tightly, and recheck the accuracy of the setup.

Radius Milling

To mill the end on the workpiece to a certain radius or to machine circular slots having a definite radius, following procedure below should be followed.

1. Align the vertical milling machine at 90* to the table.

2. Mount an dial indicator in the milling machine spindle.

3. Mount rotary table on the milling machine table.

4. Center the rotary table with the machine spindle using a test plug in the table and a dial indicator on the spindle.

5. Set the longitudinal(X)feed dial and the crossfeed(Y) dial to zero.

6. Mount the workpiece on the rotary table, aligning the center of the radial cuts with the center of the table. A special arbor may be used for this. Another method is to align the center of the radial cut with a wiggler mounted in the machine spindle.

7. Move either the crossfeed or the longitudinal feed(whichever is more convenient) an amount equal to the radius required.

8. Lock both the table and the saddle.

9. Mount the proper end mill.

10. Set the correct speed(RPM).

11. Rotate the workpiece, using the rotary table feed handwheel, to the starting point of the cut.

12. Set the depth of the cut and machine the radius to the size indicated on the drawing, using hand or power feed.

UNIT TEST

1. When is an offset boring head used?

2. Name three major components of Boring Heads.

3. Why is the locking screw tightened after tool slide adjustments have been made.

4. Why does the tool slide have multiple holes to hold boring tools?

5. What determines the cutting speed in boring?

6. For what purpose may a rotary table be used?

7. What is the purpose of the hole in the center of a rotary table?

8. Describe briefly how a rotary table may be centered with a vertical mill spindle.

9. Describe briefly how a single workpiece would be centered on a rotary table.

10. Explain how a large radius may be cut using a rotary table.

Chapter Attribution Information

This chapter was derived from the following sources.

PART II

Chapter 2: Lathe Machines

6

Chapter 2: Lathe Machine

Unit 1: The Engine Lathe

OBJECTIVE

After completing this unit, you should be able to:

- Identify the most important parts of the Lathe and their functions.
- Understand the Lathe safety rules. • Describe setup a cutting tool for machining.
- Describe mount workpiece in the lathe.
- Explain how to install cutting tool.
- Describe the positioning the tool.
- Describe how to centering the workpiece and tailstock center.

Description

The lathe is a very versatile and important machine to know how to operate. This machine rotates a cylindrical object against a tool that the individual controls. The lathe is the forerunner of all machine tools. The work is held and rotated on its axis while the cutting tool is advanced along the line of a desired cut. The lathe is one of the most versatile machine tools used in industry. With suitable attachments, the lather may be used for turning, tapering, form turning, screw cutting, facing, dulling, boring, spinning, grinding, polishing operation. Cutting operations are performed with a cutting tool fed either parallel or at right angles to the axis of the work. The cutting tool may also be fed at an angle, relative to the axis of the work, for machining taper and angles. On a lathe, the tailstock does not rotate. Instead, the spindle that holds the stock rotates. Collets, centers, three jaw chucks, and other work-holding attachments can all be held in spindle. The tailstock can hold tools for drilling, threading, reaming, or cutting tapers. Additionally, it can support the end of the workpiece using a center and can be adjusted to adapt to different workpiece lengths.

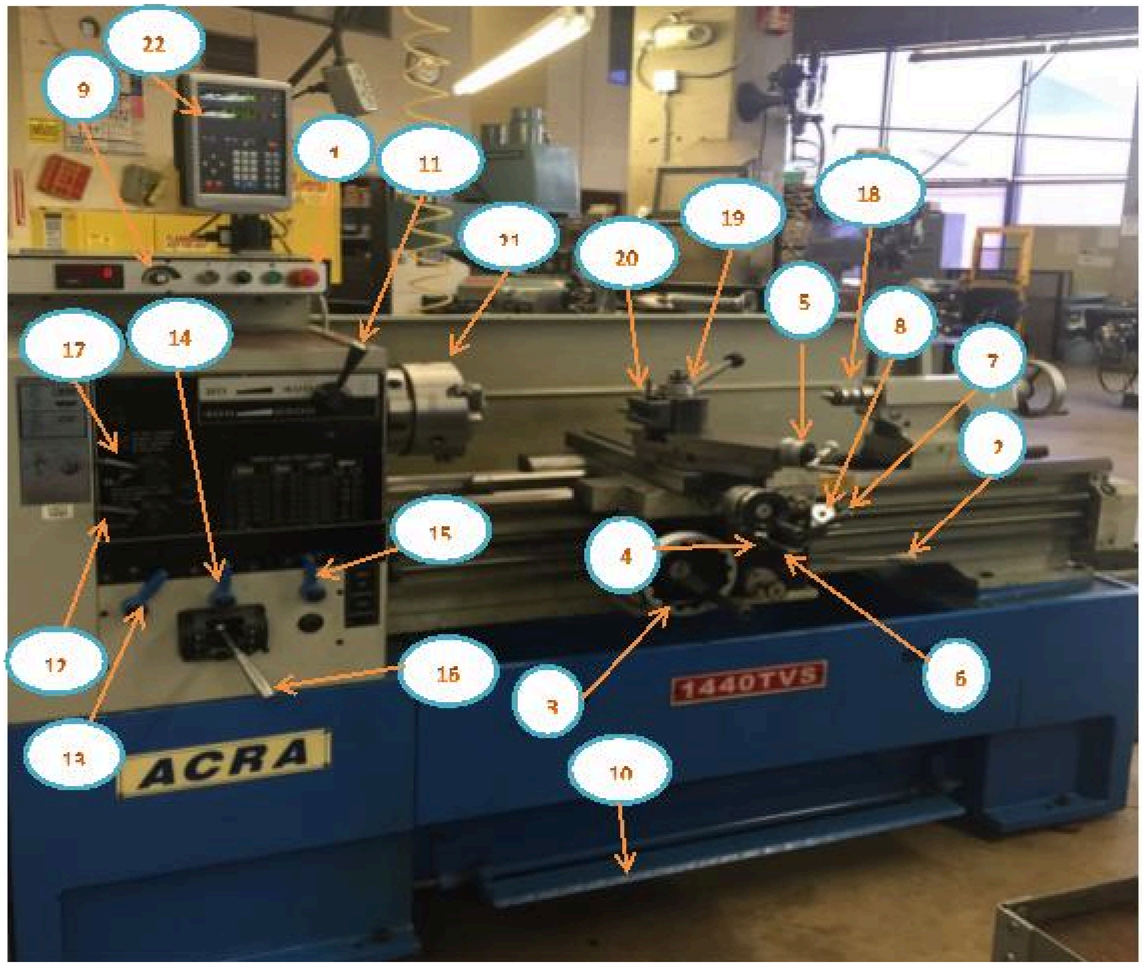


Figure 1. Parts of a lathe

1. Power On/Off
2. Spindle Forward/Reverse (flip handle up or down)
3. Carriage Handwheel 4. Cross Feed Handwheel
5. Compound Feed Handwheel
6. Carriage/Cross Feed Engage
7. Threading Half Nut
8. Threading Dial
9. Spindle Speed
10. Brake
11. Spindle High/Low Range
12. Thread/Feed Reverse (push in/pull out)
13. Feed Ranges (A, B, C)
14. Feed Ranges (R, S, T)
15. Feed Ranges (V, W, X, Y, Z) – V and Z are settings for threading
16. Gear Box
17. Gear Box Low/High
18. Tailstock
19. Tool Post
20. Toolholder
21. Three – Jaw Chuck
22. DRO (Digital Read Out) Threading/Feed Selector (see item15)

Lathe Safety

As always we should be aware of safety requirements and attempt to observe safety rules in order to eliminate serious injury to ourselves or others.

Wear glasses, short sleeves, no tie, no rings, no trying to stop the work by hand. Stop the machine before trying to check the work. Don't know how it works? –"Don't run it." Don't use rags when the machine is running.

1. Remove the chuck key from the chuck immediately after use. Do not turn the lathe on if the chuck is still in the chuck key.
2. Turn the chuck or faceplate through by hand unless there are binding or clearance issues.
3. It is important that the chuck or faceplate is securely tightened onto the lathe's spindle.
4. Move the tool bit to a safe distance from the chuck, collet, or face plate when inserting or removing your part.
5. Place the tool post holder to the left of the compound slide. This will ensure that the compound slide will not run into the spindle or chuck attachments.
6. When installing and removing chucks, face plates, and centers, always be sure all mating surfaces are clean and free from burrs.
7. Make sure the tool bit is sharp and has correct clearance angles.
8. Clamp the tool bit as short as possible in the tool holder to prevent it from vibrating or breaking.
9. Evenly apply and maintain cutting fluids. This will prevent morphing.
10. Do not run a threaded spindle in reverse.
11. Never run the machine faster than the recommended speed for the specific material.
12. If a chuck or faceplate is jammed on the spindle nose, contact an instructor to remove it.
13. If any filing is done on work revolving in the lathe, file left handed to prevent slipping into the chuck.
14. Always stop the machine before taking measurements.
15. Stop the machine when removing long stringy chips. Remove them with a pair of pliers.
16. Make sure that the tailstock is locked in place and that the proper adjustments are made if the work is being turned between centers.
17. When turning between centers, avoid cutting completely through the piece.
18. Do not use rags while the machine is running.
19. Remove tools from the tool post and tailstock before cleaning.
20. Do not use compressed air to clean the lathe.
21. Use care when cleaning the lathe. The cutting tools are sharp, the chips are sharp, and the workpiece may be sharp.
22. Make sure the machine is turned off and clean before leaving the workspace. Always remove the chuck wrench after use, avoid horseplay, keep floor area clean. Use care when cleaning the lathe, the cutting tools are sharp, the chips are sharp, and the workpiece may be sharp.

Here are some questions which are important when running a lathe:

• **Why is proper Cutting Speed important?**

When set too high the tool breaks down quickly, time is lost replacing or reconditioning the tool. Too low of a CS results in low production.

Know:

• Depth of cut for Roughing.

• Depth of cut for Finishing.

Notice the largest roughing cuts range from .010 to .030 depending on the material being machined, and .002 to .012 for the finish feed for the different materials.

• Feedrate for Roughing cut

• Feedrate for Finishing cut

Notice the Feedrate for roughing cuts range from .005 to .020 depending on the material being machined, and .002 to .004 for the finish feed for the different materials.

Cutting Tool Terminology

There are many different tools that can be used for turning, facing, and parting operations on the lathe. Each tool is usually composed of carbide as a base material, but can include other compounds. This section covers the different appearances and uses of lathe cutting tools.

Figure A:depicts a standard turning tool to create a semi-square shoulder. If there is enough material behind the cutting edge, the tool can also be used for roughing.

Figure B:depicts a standard turning tool with a lead angle. This angle enables for heavy roughing cuts. Ititalso possible to turn the tool to create a sem square shoulder.

Figure C:nose has a very large radius, which helps with fine finishes on both light and heavy cuts. The tool can also be used to form a corner radius.

Figure D:depicts a rotated standard turning tool. Its nose leads the cutting edge to create light finishing cuts on the outside diameter and face of th
shoulder.

Figure E:depicts a form tool. Different forms can be ground into the tool, which will be reproduced onto the part.

Figure F:depicts a facing tool. This cutter is used to face the end of a workpiece to provide for a smooth, flat finish. If the stock has a hole in the ce
utilize a half-center to stabilize and support the workpiece.

Figure G:depicts a grooving or under-cutting tool. As shown, it is used to cut grooves into the workpiece. When there are proper clearances, the tool can cut deeply, or cut to the left or right.

Figure H:depicts a parting tool. Parting tools cut off the stock at a certain length. This tool requires a preformed blade and holder.

Figure I:depicts a 60° threading tool used to thread stock.

Figure I

To setup a Cutting Tool for Machining

• Move the toolpost to the left-hand side of the compound rest.
• Mount a toolholder in the toolpost so that the set screw in the toolholder is about 1 inch beyond the toolpost.
• Insert the proper cutting tool into the toolholder, having the tool extend .500 inch beyond the toolholder.
• Set the cutting tool point to center height. Check it with straight rule or tailstock.
• Tighten the toolpost securely to prevent it from moving during a cut

Figure 2: Toolpostand Toolholder

To Mount Workpiece in Lathe

• Check that the line center is running true. If it is not running true, remove the center, clean all surfaces, and replace the center. Check again for trueness.
• Clean the lathe center points and the center holes in the workpiece.
• Adjust the tailstock spindle until it projects about 3 inch beyond tailstock.
• Loosen the tailstock clamp nut or lever.
• Place the end of the workpiece in the chuck and slide the tailstock up until it supports the other end of the workpiece.
• Tighten the tailstock clamp nut or level.

Figure 3: Workpiece in Lathe

Installing a Cutting Tool

• Tool holders are used to hold lathe cutting tools.
• To install, clean the holder and tighten the bolts.
• The lathe's tool holder is attached to the tool post using a quick release lever.
• The tool post is attached to the machine with a T-bolt.

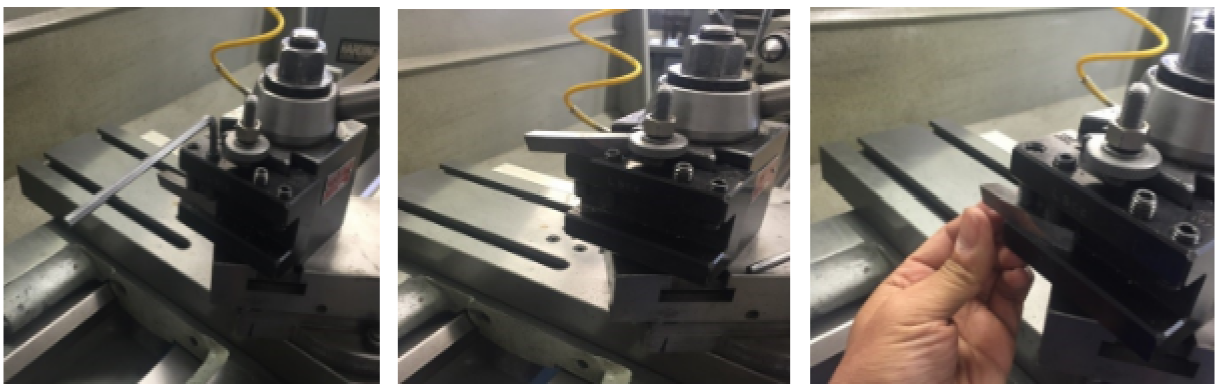


Figure 4: Installing a Cutting Tool

Positioning the Tool

To reposition the cutting tool, move the cross slide and lathe saddle by hand. Power feeds are also available. Exact procedures are dependent on the machine. The compound provides a third axis of motion, and its angle can be altered to cut tapers at any angle.

1. Loosen the bolts that keep the compound attached to the saddle.
2. Swivel the compound to the correct angle, using the dial indicator located at the compound's base.
3. Tighten the bolts again.
4. The cutter can be hand fed along the chosen angle. The compound does not have a power feed.
5. If needed, use two hands for a smoother feed rate. This will make a fine finish.
6. Both the compound and cross slide have micrometer dials, but the saddle lacks one.
7. If more accuracy is needed when positioning the saddle, use a dial indicator that is attached to the saddle. Dial indicators press against stops.

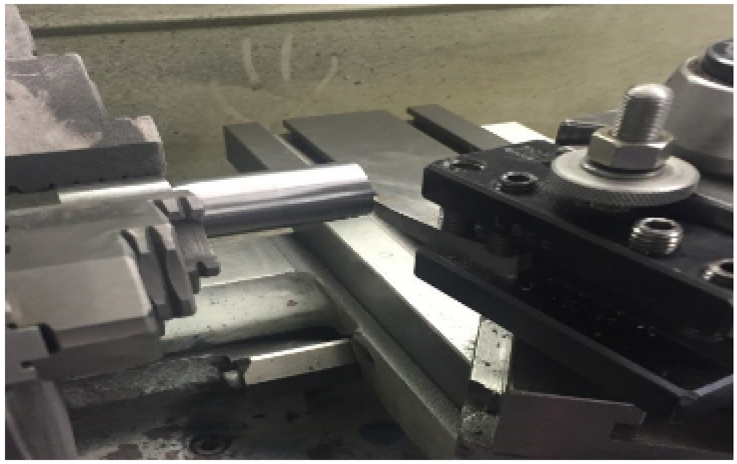
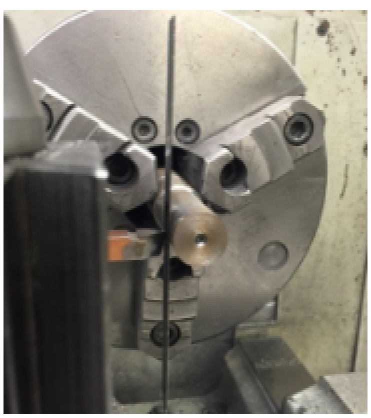


Figure 5: Positioning the Tool

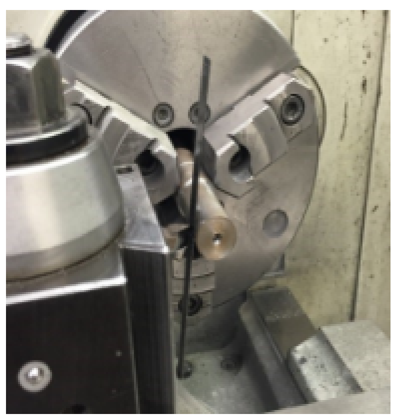
Centering the Workpiece

Steel Rule

1. Place the steel rule between the stock and the tool.
2. The tool is centered when the rule is vertical.
3. The tool is high when the rule is lean forward.
4. The tool is low when the rule is lean backward.



Tool is Centered

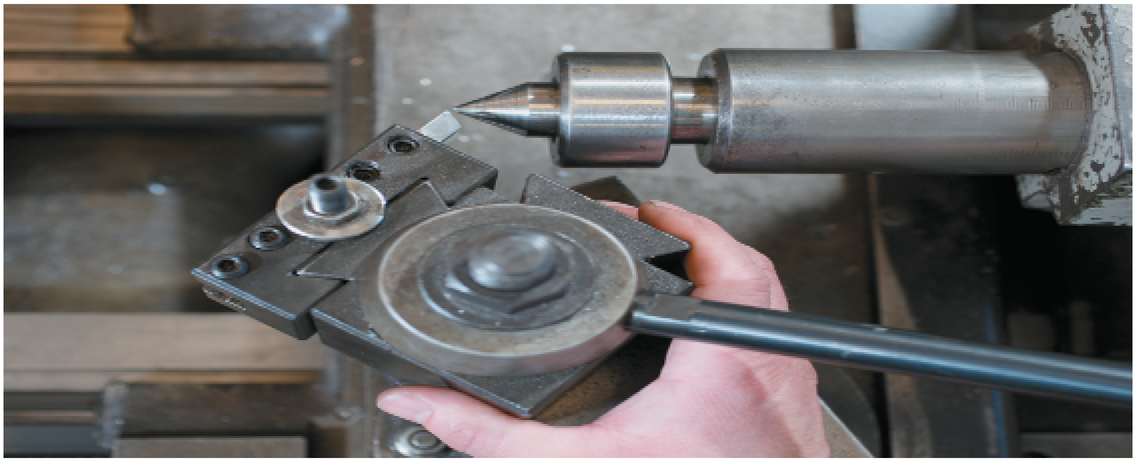


Tool is High

Tool is Low

Tailstock Center

1. Reference the center of the tailstock when setting the tool.
2. Position the tip of the tool with the tailstock center.



Setting the tool to the center of the workpiece using the tailstock center.

UNIT TEST

1. Please list the ten most important parts of the Lathe.
2. Please list five Lathe safety guidelines.
3. Why is cutting speed important?
4. What is a Toolholder?
5. Where do you mount a Toolholder?
6. How far do you extend the cutting tool in the Toolholder?
7. Please list three different cutting tools.
8. Please describe the positioning of the tool.
9. Explain how to center the workpiece.
10. What are the two way to center the workpiece?

7

Unit 2: Speed and Feed

OBJECTIVE

After completing this unit, you should be able to:

- Describe the Speed, Feed, and Depth of cut.
- Determine the RPM for different materials and diameters.
- Describe the federate for turning.
- Describe the setting speed.
- Describe the setting feed.

To operate any machine efficiently, the machinist must learn the importance of cutting speeds and feeds. A lot of time can be lost if the machines are not set at the proper speed and feeds for the workpiece.

In order to eliminate this time loss, we can, and should, use recommended metal-removal rates that have been researched and tested by steel and cutting-tool manufactures. We can find these cutting speeds and metal removal rates in our appendix or in the Machinery's Handbook.

We can control the feed on an engine lathe by using the change gears in the quick-change gearbox. Our textbook recommends whenever possible, only two cuts should be taken to bring a diameter to size: a roughing cut and a finishing cut.

It has been my experience to take at least three cuts. One to remove excess material quickly: the rough cut, one cut to establish finish and to allow for tool pressure, and one to finish the cut.

If you were cutting thread all day long: day in and day out. You might set the lathe up for only two cuts. One cut to remove all but .002 or .003 of material and the last cut to hold size and finish. This is done all the time in some shops today.

Have you noticed that when you take a very small cut on the lathe .001 to .002 that the finish is usually poor, and that on the rough cut you made prior to this very light cut, the finish was good? The reason for this is: some tool pressure is desirable when making finish cuts.

IPM = Inches Per Minute

RPM = Revolutions Per Minute

Feed = IPM

#T = Number of teeth in cutter

Feed/Tooth = Chip load per tooth allowed for material

Chip/Tooth = Feed per tooth allowed for material

Feed Rate = ChipTooth × #T × RPM

Example: Material = Aluminum 3" Cutter, 5 Teeth Chip Load = 0.018 per tooth RPM = 3000 IPS = 0.018 × 5 × 3000 = 270 Inches Per Minute

Speed, Feed, and Depth of Cut

1. Cutting speed is defined as the speed (usually in feet per minute) of a tool when it is cutting the work.
2. Feed rate is defined as tool's distance travelled during one spindle revolution.
3. Feed rate and cutting speed determine the rate of material removal, power requirements, and surface finish.
4. Feed rate and cutting speed are mostly determined by the material that's being cut. In addition, the deepness of the cut, size and condition of the lathe, and rigidity of the lathe should still be considered.
5. Roughing cuts (0.01 in. to 0.03 in. depth of cut) for most aluminum alloys run at a feedrate of .005 inches per minute (IPM) to 0.02 IPM while finishing cuts (0.002 in. to 0.012 in. depth of cut) run at 0.002 IPM to 0.004 IPM.

6. As the softness of the material decreases, the cutting speed increases. Additionally, as the cutting tool material becomes stronger, the cutting speed increases.

7. Remember, for each thousandth depth of cut, the diameter of the stock is reduced by two thousandths.

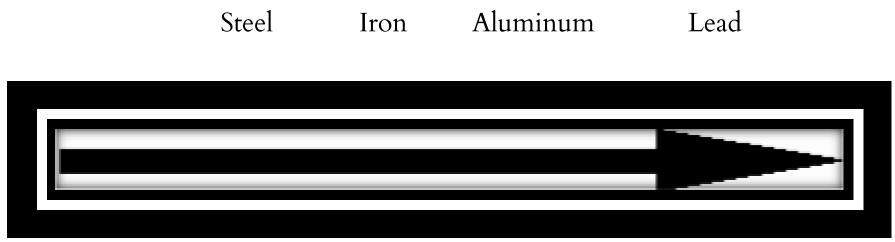


Figure 1: Increasing Cutting Speed Based on work material hardness

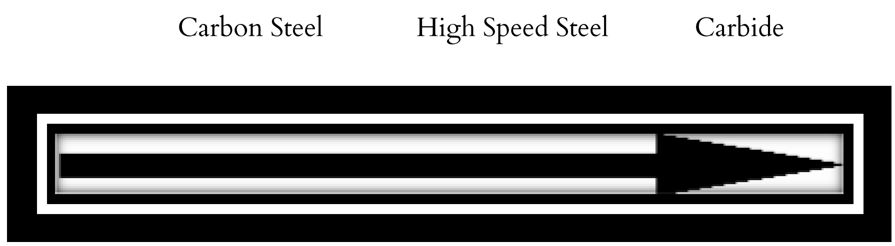


Figure 2: Increasing Cutting Speed Based on Cutting tool hardness

Cutting Speed (V)	$= \dfrac{\pi \times D \times S}{1{,}000}$
Spindle Speed (S)	$= V \div \pi \div D \times 1{,}000$
Feed (F)	$= S \times f \times N$
feed per Tooth (f)	$= \dfrac{F}{S \times N}$

V = Cutting Speed
π = The Circular Constant
D = Diameter
S = Spindle Speed
F = Feed
f = Feed per Tooth
N = Number of Fluites

Cutting Speeds:

A lathe work cutting speed may be defined as the rate at which a point on the work circumference travels past the cutting tool. Cutting speed is always expressed in meters per minute (m/min) or in feet per minute (ft/min.) industry demands that machining operations be performed as quickly as possible; therefore current cutting speeds must be used for the type of material being cut. If a cutting speed is too high, the cutting tool edge breaks down rapidly, resulting in time lost recondition the tool. With too slow a cutting speed, time will be lost for the machining operation, resulting in low production rates. Based on research and testing by steel and cutting tool manufacturers, see lathe cutting speed table below. The cutting speeds for high speed steel listed below are recommended for efficient metal removal rates. These speeds may be varied slightly to shift factors such as the condition of the machine, the type of work material and sand or hard spots in the metal. The RPM at which the lathe should be set for cutting metals is as follows:

To determine the RPM of the lathe while performing procedures on it:

Formula: RPM = (CuttingSpeed x 4) / Diameter

We first must find what the recommended cutting speed is for the material we are going to machine.

Learn to use the Machinery's Handbook and other related sources to obtain the information you need.

EXAMPLE: How fast should a 3/8 inch drill be turning when drilling mild steel?

From our recommended cutting speed from our class handouts, use a cutting speed of 100 for mild steel.

(100 x 4) / .375 = 1066 RPM

What would the RPM be if we were turning a .375 diameter workpiece made out of mild steel on the lathe?

RPM = 100 X4 / 1.00 = 400 RPM

Recommended Cutting Speeds for Six Materials in RPM

Cutting Tool	Mild Steel	Carbon Steel Annealed	Aluminum	Soft Brass	Cast Iron	Annealed Stainless
HSS	100	80	250 to 350	175	100	80 to 100
Carbide	300	200	750 to 1000	500	250	200 to 250

These charts are for HSS tools. If using carbide, the rates may be increased.

Lathe Feed:

The feed of a lathe is the distance the cutting tool advances along the length of the work for every revolution of the spindle. For example, if the lathe is set for a .020 inch feed, the cutting tool will travel the length of the work .020 inch for every complete turn that work makes. The feed of a lathe is dependent upon the speed of the lead screw or feed rod. The speed is controlled by the change gears in the quick change gearbox.

Whenever possible, only two cut should be taken to bring a diameter cut. Since the purpose of a rough cut is to remove excess material quickly and surface finish is not too important. A coarse feed should be used. The finishing cut is used to bring the diameter to size and produce a good surface finish and therefore a fine feed should be used.

The recommended feeds for cutting various materials when using a high speed steel cutting tools listed in table below. For general purpose machining a .005 – .020 inch feed for roughing and a .012 to .004 inch feed for finishing is recommended.

To select the proper feed rate for drilling, you must consider several factors.

1. Depth of hole – chip removal
2. Material type – machinability
3. Coolant – flood, mist, brush
4. Size of drill
5. How strong is the setup?
6. Hole finish and accuracy

Feed Rates for Turning:

For general purpose machining, use a recommended feed rate of .005 – .020 inches per revolution for roughing and a .002 – .004 inches per revolution for finishing.

Feeds for Various Materials (using HSS cutting tool)

Material	Roughing Cut (IPR)	Finishing Cut (IPR)
Mild steel	.005 - .020	.002 - .004
Tool steel	.005 - .020	.002 - .004
Cast Iron	.005 - .020	.002 - .004
Brass	.005 - .020	.002 - .004
Aluminum	.005 - .020	.002 - .004

Setting speeds on a lathe:

The lathes are designed to operate at various spindle speeds for machining of different materials. There speeds are measured in RPM (revolutions per minute) and are changed by the cone pulleys or gear levels. One a belt-driven lathe, various speeds are obtained by changing the flat belt and the back gear drive. One the geared-head lathe speeds are changed by moving the speed levers into proper positions according to the RPM chart fastened to the lathe machine (mostly on headstock). While shifting the lever positions, place one hand on the faceplate or chuck, and form the face plate slowly by hand. This will enable the levers for engage the gear teeth without clashing. Never change speeds when the lathe is running on lathers equipped with variable speed drivers, the speed is changed by turning a dial of handle while he machine is running.

Setting feeds:

The feed of on lathe, or the distance the carriage will travel in on revolution of the spindle, depends on the speed of the feed rod or lead screw. This is controlled by the change gears in the quick-change gearbox. This quick change gearbox obtains its drive from the head stock spindle through the end gear train. A feeds and thread chart mounted on the front of the quick-change gearbox indicates the various feeds and metric pitches or thread per inch which may be obtained by setting levers to the positions indicated.

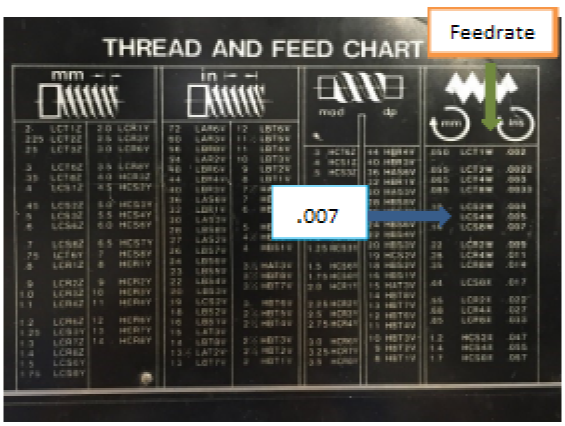


Figure 5: Thread and Feedrate Chart

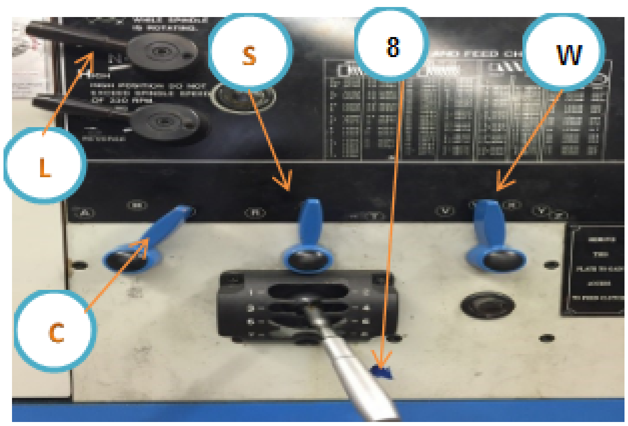


Figure 6: Quick Change Gearbox

To set the feedrate for Acura Lathe:

Example:

1. Select the desired feedrate on the chart (See Figure 2)
2. Select federate of .007 – LCS8W (See Figure 2)
3. L = Select High/Low lever (See Figure 3)
4. C = Select Feed Ranges and change to C on this lever (See Figure 3)
5. S = Select Feed Ranges and change to S on this lever (See Figure 3)
6. 8 = Select Gear Box and change to 8 on this lever (See Figure 3)

7.W = Select Feed Ranges and change to W on this lever (See Figure 3) Before turning on the lathe, be sure all levers are fully engaged by turning the headstock spindle by hand, and see that the feed rod turns.

UNIT TEST

1. What is IMP and RPM?
2. What is the formula for Feedrate?
3. What would the RPM be if we were turning a 1.00" diameter workpiece made out of mild steel, using HSS cutting tool?
4. What would the RPM be if we were turning a 1.00" diameter workpiece made out of mild steel, using Carbide cutting tool?
5. The cutting speed for carbon steel and the workpiece diameter to be faced is 6.00". Find the correct RPM.
6. A center drill has a 1/8" drill point. Find the correct RPM to use carbon steel.

7. If the cutting speed of aluminum is 300 sfm and the workpiece diameter is 4.00", What is the RPM?
8. What is roughing and finishing federate for aluminum?
9. Please set the roughing cut feederate from figure 5.
10. Please set the finishing cut feederate from figure 5.

Unit 3: Chucks

OBJECTIVE

After completing this unit, you should be able to:

• Describe different type chucks.

Chucks:

Some work pieces, because of their size and shape, cannot be held and machined between lathe centers. Lather chucks are used extensively for holding work for machining operations. The most commonly used lathe chucks are the three jaw universal, four jaw independent, and the collects chuck.

Three-jaw universal chuck:

Three-jaw universal chuck is used to hold round and hexagonal work. It grasps the work quickly and within a few hundredths of a millimeters or thousandths of an inch of accuracy, because the three jaws move simultaneously when adjusted by the chuck wrench. This simultaneous motion is cause by a scroll plate into which all here jaws fit. Three jaws chucks are made in various sizes from 1/8-16 inch in diameter. They are usually provided with two sets of jaws, one for outside chucking and the other for inside chucking.

Figure 1: Three-jaw universal chuck

Four-jaw independent chuck:

This four- jaw independent chuck has four jaws; each of which can be adjusted independently by a chance wrench. They are used to held wound, square, hexagonal, and irregular-shaped workpieces. The jaws can be reversed to hold work by the inside diameter.

Figure 2: Four-jaw independent chuck

Collect chuck:

The collect chuck is the most accurate chuck and is used for high precision work and small tools. Spring collects are available to hold round, square, or hexagon shaped workpieces. A adaptor is filled into the taper of the headstock spindle, and a hollow draw bar having an internal thread is instead in the opposite end of the headstock spindled. As the hand wheel and draw bar is revolved, it draws the collet into the tapered adaptor, causing the collet to tighten on the workpieces.

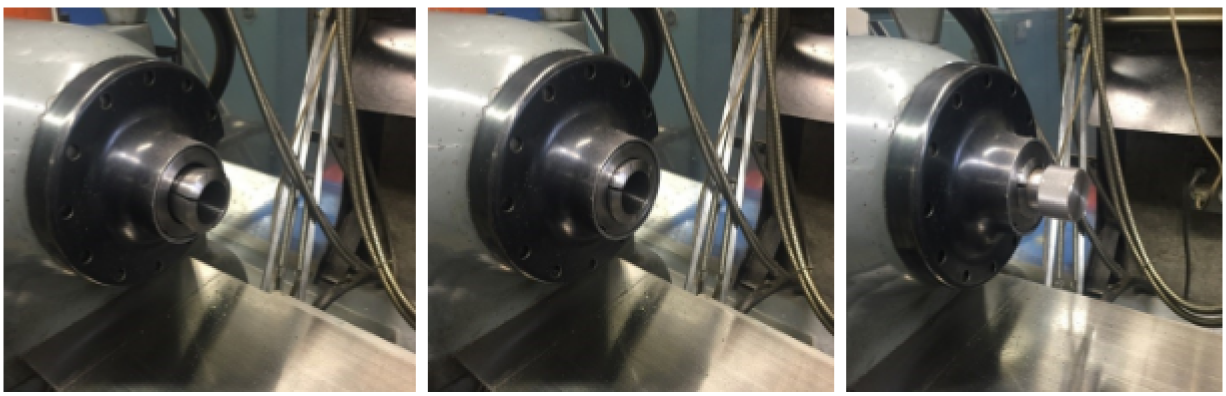


Figure 3: Collect chuck

The Jacob collect chuck has a wider range the spring collect chuck. Instead of a draw bar, it incorporates an impact-tightening hand wheel to close the collect on the workpiece. A set of II rubber flex collects, each capable of a range of almost 1/8 in, makes it possible to hold a wide range of work diameter. When the hand wheel is turned clockwise, the rubber flex collect is forced into a taper, causing it to tighten on the workpiece. When the hand wheel is turned counterclockwise, the collect opens and releases the workpiece.

Magnetic chucks:

A magnetic chucks are used to hold iron or steel parts that are too thin, or that may be damaged if held in a conventional chuck. These chucks are fitted to an adaptor mounted on the headstock spindle. Work id held lightly for aligning purposes by turning the chuck wrench approximately ¼. After the work has been turned

Faceplates:

A faceplates are used to hold work that is too large or of such a shape that it cannot be held in a chuck or between centers. Faceplates are equipped with several slots to permit the use of bolts to secure the work, so that the axis of the workpiece may be aligned with the lathe centers. When work is mounted off – center, a counterbalance should be fastened to the faceplate to prevent imbalance and the resultant vibrations when the lathe is in operation.

UNIT TEST

1. What are the most commonly used lathe chucks? Name three.

2. Describe three-jaw universal chuck.
3. Describe four-jaw universal chuck.
4. Describe Collect chuck.
5. Describe Jacobs collect chuck chuck.
6. Describe Magnetic chucks chuck.
7. Describe Faceplates

9

Unit 4: Turning

OBJECTIVE

After completing this unit, you should be able to:

- Describe the rough and finishing turning.
- Describe the turning shoulder.
- Describe the facing cut.
- Explain how to set up for center/spot drill.
- Explain how to set up for boring.
- Explain how to set up for knurling.
- Correctly set up a workpiece for parting/grooving.
- Determine the taper calculation.
- Correctly set up workpiece in a 4-jaw chuck.

Workpiece is generally machined on a lathe for two reasons: to cut it to size and to produce a true diameter. Work that must be cut to size and have the same diameter along the entire length of the workpiece involves the operation of parallel turning. Many factors determine the amount of materials which can be removed on a lathe. A diameter should be cut to size in two cuts: an roughing cut and finishing cut.

To have the same diameter at each end of the workpiece, the lathe centers must be in line.

To set an accurate depth of cut

Procedure:

1. Set the compound rest at 30 degrees.
2. Attach a roughing or finishing tool. Use a right-handed turning tool if feeding the saddle in the direction of the headstock.
3. Move the tool post to the left hand side of the compound rest and set the tool bit to right height center.
4. Set the lathe to the correct speed and feed for the diameter and type of material being cut.
5. Start the lathe and take a light cut about .005 inch and .250 inch long at the right hand end of the workpiece.
6. Stop the lathe, but do not move the crossfeed screw handle.
7. Move the cutting tool to the end of the workpiece (to the right side) by turning the carriage hand wheel.
8. Measure the work and calculate the amount of material to be removed.
9. Turn the graduated collar half the amount of material to be removed. For example, if .060 inch to be removed, the graduated collar should be turned in .030 inch, since the cut is taken off the circumference of the workpiece.
10. **Remember**, for each thousandth depth of cut, the diameter of the stock is reduced by two thousandths.

Rough Turning

The operation of rough turning is used to remove as much metal as possible in the shortest length of time. Accuracy and surface finish are not important in this operation. Therefore, max depth of .030 inch and a .020 to .030 inch feed is recommended. Workpiece is generally rough turned to within about .030 inch of the finished size in a few cuts as possible.

Procedure:

1. Set the lathe to the correct speed and feedrate for the type and size of the material being cut.

2. Adjust the quick change gear box for a .010 to .030 inch feed, depending on the depth of cut and condition of the machine.
3. For Example: .010
4. Move the tool holder to the left hand side of the compound rest and set the tool bit to right height to center.
5. Tighten the tool post securely to prevent the toolholder from moving during the machining operation.
6. Take a light trial cut at the right hand end of the workpiece for about .250 inch length.
7. Measure the workpiece and adjust the tool bit for the proper depth of cut.
8. Cut along for about .250 inch, stop the lathe and check the diameter for size. The diameter should be about .030 inch over the finish side.
9. Re-adjust the depth of cut, if necessary.

Finish Turning

Finish turning on a lathe, which follows rough turning , produces a smooth surface finish, and cuts the workpiece to an accurate size. Factors such as the condition of the cutting tool bit, the rigidity of the machine and workpiece and the lathe speed and feedrate, may affect the type of surface finish produced.

Procedure:

1. Check to see if the cutting edge of the tool bit is free from nicks, burns, etc. It is good practice to home the cutting edge before you take a finish cut.
2. Set the lathe to the recommended speed and feedrate. The feed rate used depends upon the surface finish required.
3. Take a light trial cut about .250 inch long at the right hand end of the work to produce a true diameter, set the cutting tool bit to the diameter and set the graduated collar to the right diameter.
4. Stop the lathe, measure the diameter.
5. Set the depth of cut for half the amount of material to be removed.
6. Cut along for .250 inch, stop the lathe and check the diameter.
7. Re-adjust the depth of cut, if necessary and finish turn the diameter. In order to produce the truest diameter possible, finish turn workpiece to the required size. Should it be necessary to finish a diameter by filming or polishing, never leave more than .002 to .003 inch for this operation.

Turning to a Shoulder

When turning more than one diameter on a workpiece. The change in diameter or step, is known as shoulder.

Three common types of shoulder:

1. Square
2. Filleted corner
3. Angular of Tapered

Procedure:

1. With a workpiece mounted in a lathe, lay out the shoulder position from the finished end of the workpiece. In case of filleted shoulders, all sufficient length to permit the proper radius to be formed on the finished shoulder.
2. Place the point of the tool bit at this mark and cut a small groove around the circumference to mark off the length.
3. With a turning tool bit, rough and finish turn the workpiece about .063 inch of the required length.
4. Set up an end facing tool. Chalk the small diameter of the workpiece, and bring the cutting tool up until it just removes the chalk mark.
5. Note the reading on the graduated collar of the cross feed handle.
6. Face square the shoulder, cutting to the line using hand feed.
7. For successive cuts, return the cross feed handle to the same graduated collar setting.

If a filleted corner is required, a tool bit having the same radius is used for finishing the shoulder. Angular or chamfered edges may be obtained by setting the cutting edge of the tool bit to the desired angle of chamfer and feeding it against the shoulder, or by setting the compound rest to the desired angle.

Facing

Workpieces to be machined are generally cut a little longer than required, and faced to the right length. Facing is an operation of machining the ends of a workpiece square with its axis. To produce a flat, square surface when facing, the lathe might be true.

The purpose of facing are:

- To provide a true, flat surface, square with the axis of the workpieces.
- To provide and accurate surface from which to take measurements.
- To cut the workpieces to the required length.

Figure 1. Facing Operation

Procedure:

1. Move the tool post to the left-hand side of the compound rest, and set the right hand facing tool bit to the right height of the lathe center point. The compound rest may be set at 30 degrees for accurate end facing.
2. Mount the workpiece in the chuck to face. Use a line center in the tail stuck or straight ruler if needed for true.
3. Insert a facing tool.
4. Position the tool slightly off from the part.
5. Set the facing tool bit pointing left at a 15-20 degree angle. The point of the tool bit must be closest to the workpiece and space must be left along the side.
6. Set the lathe to the correct speed and feed for the diameter and type of material being cut.
7. Before turning the machine on, turn the spindle by hand to make sure parts do not interfere with spindle rotation.
8. Start the lathe and bring the tool bit ad close to the lathe center as possible.
9. Move the carriage to the left, using the handwheel, until the small cut is started.
10. Feed the cutting tool bit inwards to the center by turning the cross feed handle. If the power feed cross feed is used for feeding the cutting tool, the carriage should be locked in position.
11. Repeat procedure 6,7 and 8 until the workpiece is cut to the correct length. 12. There will be a sharp edge on the workpiece after facing, which should be broken with a file.

To spot a workpiece

Spotting Tool bit is used to make a shallow, v-shaped hole in the center of the workpiece. Provides a guide for the drill to follow. A hole can be spotted quickly and fairly accurately by using a center drill. A spotting tool bit should be used for extreme accuracy.

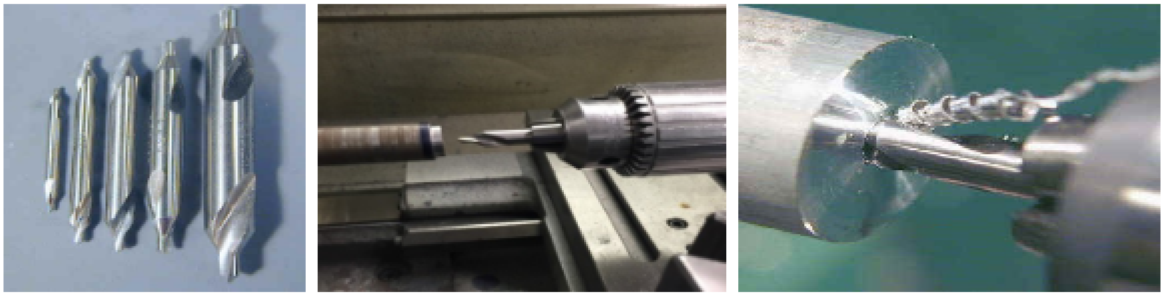


Figure 2: Center/Spot Tool

Procedure:
1. Mount workpiece true in a chuck.
2. Mount the drill chuck into the tailstock.
3. Ensure that the tang of the drill chuck is properly secured in the tailstock.
4. Move and lock the tailstock to the desired position.
5. Before turning the machine on, turn the spindle by hand to make sure parts do not interfere with spindle rotation.
6. Set the lathe to the proper speed for the type of material to be spot or center drill.
7. Start the hole using a center drill. 8. Spot the hole with a spotting or center drill tool bit.

Drilling

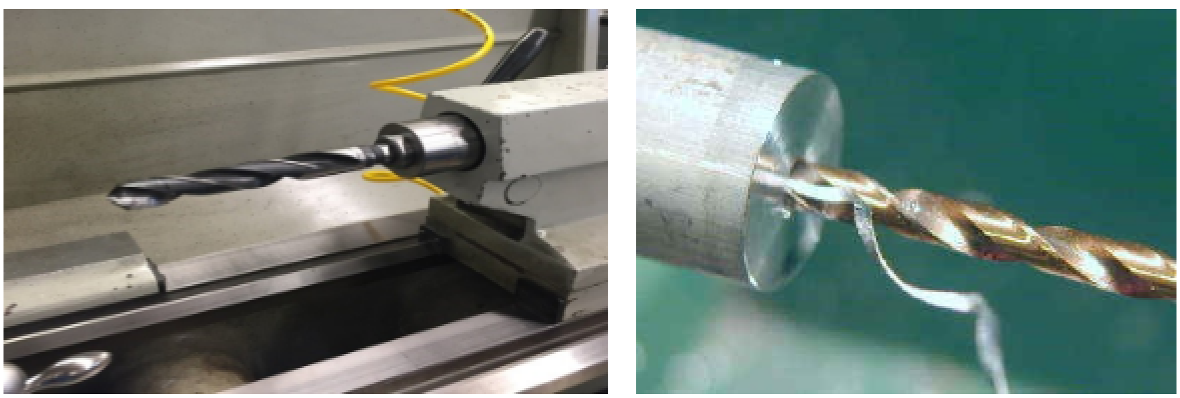


Figure 3. Drill

Procedure:
1. Mount the drill chuck into the tailstock.
2. Mount workpiece true in a chuck.
3. Check the tool stock center and make sure it is in line.
4. Ensure that the tang of the drill chuck is properly secured in the tailstock.
5. Move and lock the tailstock to the desired position.
6. Before turning the machine on, turn the spindle by hand to make sure parts do not interfere with spindle rotation.
7. Start the hole using a spotting or center drill tool bite.
8. When using a center drill, always use cutting fluid along with it.
9. A center drill doesn't cut as easily as a drill bit would, as it has shallow flutes for added stiffness.
10. Drill past the entirety of the taper to create a funnel to guide the bit in.
11. Mount the drill in the tailstock spindle, in a drill chuck or in a drill holder.
12. Set the lathe to the proper speed the type of material to be drilled.
13. Start the lathe and drill to the desired depth according to the blueprint drawing, applying cutting fluid.
14. To gauge the depth of the hole, use the graduations on the tailstock spindle, or use a steel rule to measure the depth.
15. Use the peck drill operation to remove the chips and measure the depth of the hole.

16. When drilling, take off at most one or two drill bit diameters worth of material before backing off, clearing chips, and reapplying cutting fluid.
17. If the drill bit squeaks against the stock, apply more cutting fluid.
18. To remove the drill chuck from the tailstock, draw it back by around a quarter turn more than it will easily go.
19. Use a pin to press the chuck out of the collet.

Boring

Boring is an operation to enlarge and finish holes accurately. Truing of a hole by removing material from internal surfaces with a single point tool bit cutter. Special diameter holes, for which no drills are available, can be produced by boring.

Boring utilizes a single point cutting tool to enlarge a hole. This operation provides for more accurate and concentric hole, as opposed to drilling.

Since the cutter extends from the machine from a boring bar, the tool is not as well-supported, which can result in chatter. The deeper the boring operation, the worse the chatter. To correct this:

1. Reduce the spindle speed.
2. Increase the feed.
3. Apply more cutting fluid.
4. Shorten the overhang of the boring bar.
5. Grind a smaller radius on the tool's nose.

Procedure:

1. Mount the workpiece in a chuck.
2. Face, spot and drill the hole on the workpiece.
3. Check to see if the boring bar has enough clearance.
    - If the hole is too small for the boring bar, the chips will jam while machining and move the bar off-center.
4. Make sure that the point of the boring tool is the only part of the cutter than contacts the inner surface of the workpiece.
5. If the angle does not provide sufficient end relief, replace the cutter with one that has a sharper angle.
6. Position the boring bar so the point of the cutter is positioned with the centerline of the stock.
7. A tool that is not placed in line with the center of the workpiece will drag along the surface of stock, even if there is a sufficient end relief angle.
8. Select a boring bar as large as possible and have it extend beyond the holder only enough to clear the depth of the hole to be bored.
9. Mount the holder and boring tool bar with the cutter tool bit on the left hand side of the tool post and revolving the workpiece.
10. Set the boring tool bit to center.
    - Note: Depending on the rigidity of the setup, the boring tool bit will have a tendency to spring downward as pressure is applied to the cutting edge. By setting the boring tool bit slightly above center, compensation has been made for the downward spring and the tool bit will actually be positioned on the exact center of the workpiece during machining operations.
11. Set the lathe to the proper cutting speed and feed. a. Note: For feedrate select a medium feed rate.
12. Apply lube to the hole before turning the machine on.
13. Turn the machine on and move the tool into the pre-drilled hole.
14. Start the lathe and slowly bring the boring tool until it touch the inside diameter of the hole.
15. Take a light cut (about .003 in.) and about -375 long.
16. Stop the lathe and measure the hole diameter, use a telescopic gauge or inside micrometer.
17. After measure the hole, determine the amount of material to be removed from the hole. Leave about .020 in a finish cut.
18. Start the lathe and take the roughing cut.
19. Feed the boring bar into the workpiece, taking off about .020 on each pass.

20. Bring the boring bar out once the desired depth has been reached.

21. Repeat steps 19 and 20 until the desired diameter of the inside hole has been attained.

22. After roughing cut is completed, stop the lathe and bring the boring tool bit out of the hole without moving the cross feed handle.

23. Set the depth of the finish cut and bore the hole to size. For a good surface finish, a fine feedrate is recommended.

24. On the last pass, stop at the desired depth and bring the cutter back towards the center of the stock. This will face the back of the hole.

25. Bring the boring bar out of the machine and stop the machine.

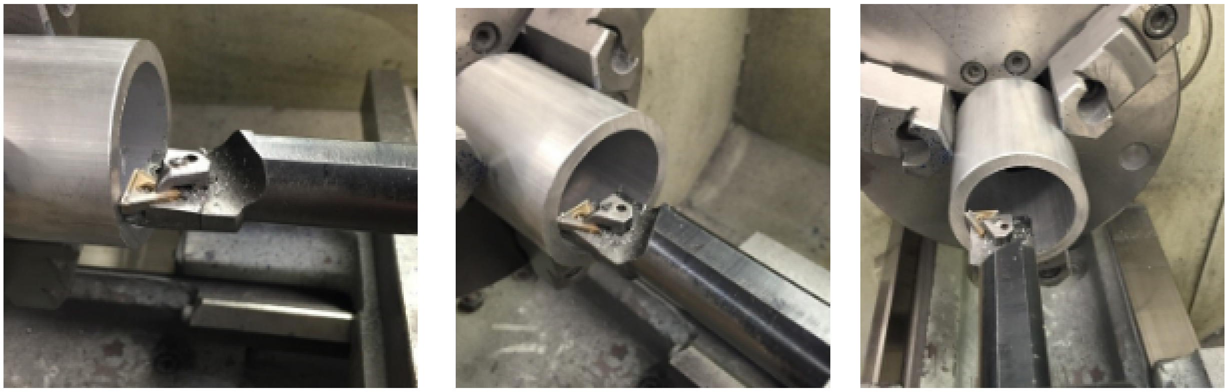


Figure 4. Boring on a lathe

Knurling

1. A knurl is a raised impression on the surface of the workpiece produced by two hardened rolls.

2. Knurls are usually one of two patterns: diamond or straight.

3. Common knurl patterns are fine, medium, or coarse.

4. The diamond pattern is formed by a right-hand and a left-hand helix mounted in a self-centering head.

5. Used to improve appearance of a part & provide a good gripping surface for levers and tool handles.

6. Common knurl patterns are fine, medium, or coarse.

7. The straight pattern, formed by two straight rolls, is used to increase the size of a part for press fits in light-duty applications.

8. Three basic types of knurling toolholders are used: the knuckle-joint holder, the revolving head holder, and the straddle holder.

9. Knurling works best on workpieces mounted between centers.

10. Knurls do not cut, but displace the metal with high pressure.

11. Lubrication is more important than cooling, so a cutting oil or lubricating oil is satisfactory.

12. Low speeds (about the same as for threading) and a feed of about .010 to .020 in. are used for knurling.

13. The knurls should be centered on the workpiece vertically & the knurl toolholder square with the work.

14. A knurl should be started in soft metal about half depth and the pattern checked.

15. Several passes may be required on a slender workpiece to complete a knurl because the tool tends to push it away from the knurl.

16. Knurls should be cleaned with a wire brush between passes.

Figure 5. Knurling

Procedure:

1. Mount the knurling tool into a tool holder and and adjust it to the exact centerline of the lathe spindle.
2. Position and secure the knurling tool 90 degrees to the surface of the knurled.
3. Move the lathe carriage by hand and locate the area on the workpiece to be knurled.
4. Rotate the knurling head to index to the correct set knurls.
5. Position the knurls to the right edge of work such that half of the knurl contacts the right edge of the workpiece.
6. Apply cutting oil to the work.
7. Turn the spindle to about 100 RPM and use the crossfeed handwheel to move the knurling tool into the work. This should be approximately 0.030 inches, or until knurls track and form a good pattern.
8. Engage the lathe power feed to move the carriage towards the headstock at a feedrate of 0.010 to 0.020 inches per revolution.
9. Apply oil as required and brush knurled area with a stiff brush to clean chips from knurl.
10. When the knurls reach the end of knurled area, reverse the carriage feed direction feed direction and feed knurls into the work another 0.005 to 0.010 inches.
11. Continue knurling back and forth until a sharp diamond develops.

Parting and Grooving on a Lathe

The purpose of parting and grooving:

There are times when you may want to cut a piece from the end of a workpiece, or you may want to cut a groove into a workpiece.

Grooving, commonly called recessing, undercutting, or necking, is often done at the end of a thread to permit full travel of the nut up to a shoulder or at the edge of a shoulder to ensure a proper fit of mating parts. There are three types of grooves: square, round, and u-shaped.

Rounded grooves are usually used where there is a strain on the part, and where a square corner would lead to fracturing of the metal.

To cut a Groove

Procedure:

1. Select a tool bit to the desired size and shape of the groove required.
2. Lay out the location of the groove.
3. Set the lathe to half the speed for turning.
4. Mount the workpiece in the lathe.
5. Set the tool bit to center height.
6. Slowly feed the tool bit into the workpiece using the cross feed handle.
7. Apply plenty of cutting oil to the point of the cutting tool. To ensure that the cutting will not blind in the groove. If chatter develops, reduce the spindle speed.

8. Stop the lathe and check the depth of groove.
9. Repeat procedures 6-7 until the work is cut to the correct depth.

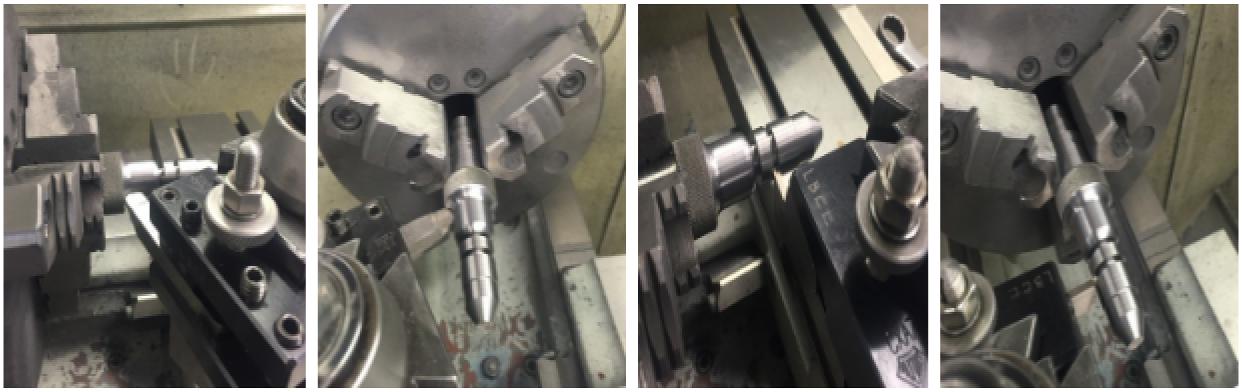


Figure 6. Cutting a Groove

Parting

Cut off tools, often called parting tools, are used for cutting workpiece. There are three types of parting tools. The parting tool consists of a straight holder, left hand offset and right hand offset inserted blade are the most commonly used.

There are two common problems in parting, chattering and hugging in. A chattering occurs when the tool is not held solidly enough, any looseness in the tool, holder, or any part of the lathe itself makes cutting off difficult, uneven, and often impossible. Hugging in means the tool tends to dig into the workpiece tends to climb over the top of the cutting edge. This usually breaks off the tool bit or wrecks the workpiece. Hugging in is usually caused when the parting tool is set too high or too low.

- Parting tools are narrower but deeper than turning tools. Parting tools are used to create narrows grooves and cut off parts of the stock.
- The tool holder should barely clear the workpiece when the parting tool is installed.
- Make sure the parting tool is perpendicular to the axis of rotation.
- Ensure the tip of the tool rests at the same height as the center of the stock. Holding the tool against the face of the part may help with this.
- Set the tool's height, lay it against the part's face, and lock the tool in place. Remember to apply cutting fluid, especially when making a deep cut.

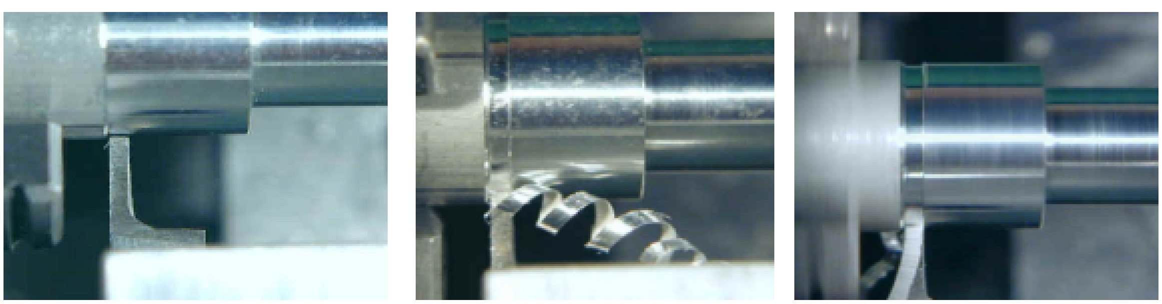


Figure 7. Parting

Procedure:

1. Mount the workpiece in the chuck with the part to be cut off as close to the chuck as possible.
2. Mount the parting tool on the left hand side of the compound rest with the cutting edge set on center.
3. Place the holder as close to the tool post as possible to prevent vibration and chatter.
4. Adjust the tool bit. The tool bit should extend from the holder a distance equal to little more than half the diameter of the workpiece. Adjust the revolution per minute (rpm) to about ⅔ the speed for turning.

5. Mark the location of the cut.

6. Move the cutting tool into position.

7. Start the lathe and slowly feed the parting tool into workpiece by hand. Grip the cross feed handle with both hands in order to feed steadily and uniformly. Apply plenty of cutting oil.

8. When the workpiece is about ¼ in, it is good practice to move the parting tool sideways slightly. This side motion cut a little wider to prevent the tool from jamming.

9. To avoid chatter, keep the tool cutting and apply cutting oil consistently during the operation. Feed slowly when the part is almost cut off.

10. Keep advancing the tool until it reaches the center of the workpiece. As you get close, the workpiece is suspendend by a thin stalk of metal.

11. The end of the workpiece that you cut off will generally have a pretty rough finish and a little stalk of metal protruding from the end. See figure 19 below.

12. The final step it to mount this piece in the chuck and make a facing cut to clean up the end. One problem with this step is that the chuck jaws can mar the finished workpiece. If you look carefully at figure 20 below you can actually see the imprint of the chuck jaws. To avoid this, you could wrap the workpiece in a thin strip of emory paper, or similar protective material, before clamping it.

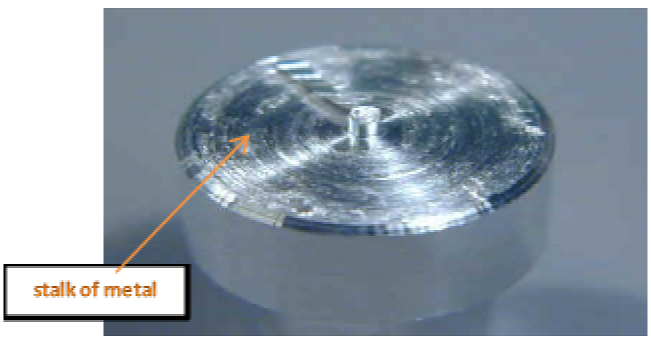


Figure 8. Workpiece Cutoff

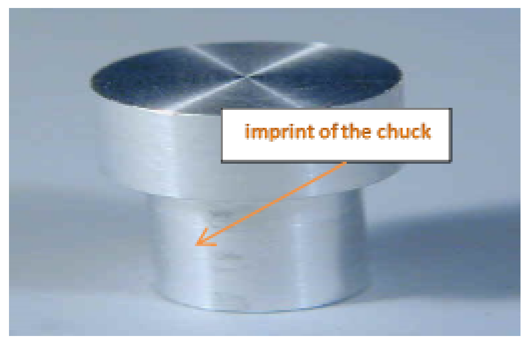


Figure 9. Finished Workpiece

Alignment of Lathe Centers

To produce a parallel diameter when machining work between centers, it is important that is, the two lathe centers must be in line with each other and running true with the centerline of the lathe. If the center are not aligned, the work being machined will be tapered.

There are three methods to align lathe centers:

1. By aligning the centerlines on the back of the tail stock with each other. This is only a visual check and therefore not for accurate.

2. The trial cut method, where a small cut is taken from each end of the work and the diameter are measured with a micrometer.

3. Align Centers using a Dial Indicator.

Method 1. To align centers by adjusting the tailstock.

Procedure:

1. Loosen the tailstock clamp not or lever.

2. Loosen one of the adjusting screw on the left or right side, depending upon the direction the tail stock must be moved. Tighten the other adjusting screw until the line on the top half of the tail stock aligns exactly with the line on the bottom half.

3. Tighten the loosened adjusting screw to lock both halves of the tailstock in place.

4. Lock the tailstock clamp nut or lever.

Method 2. To align center by the trail cut method.

Procedure:

1. Take a light cut about .010 to a true diameter, from section A at the tailstock end of .250 inch long.
2. Stop the feed and note the reading on the graduated collar of the cross feed handle.
3. Move the cutting tool close to the headstock end.
4. Bring the cutting tool close to the same collar setting as step 1 (Section A).
5. Return the cutting tool to the same collar setting as step 1. (Section A)
6. Cut a .250 length at Section B and then stop the lathe.
7. Measure both diameters with a micrometer.
8. If both diameters are not the same size, adjust the tailstock either toward or away from the cutting tool one-half the difference of the two readings.
9. Take another light cut at Section A and B. Measure these diameters and adjust the tailstock, if required.

Method 3. To Align Centers using a Dial Indicator.

Procedure:

1. Clean the lathe and work centers and mount the dial indicator.
2. Adjust the test bar snuggly between centers and tighten the tailstock spindle clamp.
3. Mount a dial indicator on the tool post or lathe carriage. Be sure that the indicator plunger is parallel to the lathe bed and that the contact point is set on center.
4. Adjust the cross slide so that the indicator registers about .025 inch at the tailstock end.
5. Move the carriage by hand so the test indicator registers on the diameter at the headstock end and note the test indicator reading.
6. If both test indicator readings are not the same. Adjust the tailstock by the adjusting screw until the indicator registers the same reading at both ends.

Taper Calculations

To calculate the taper per foot (tpf). It is necessary to know the length of the taper, large and small diameter.

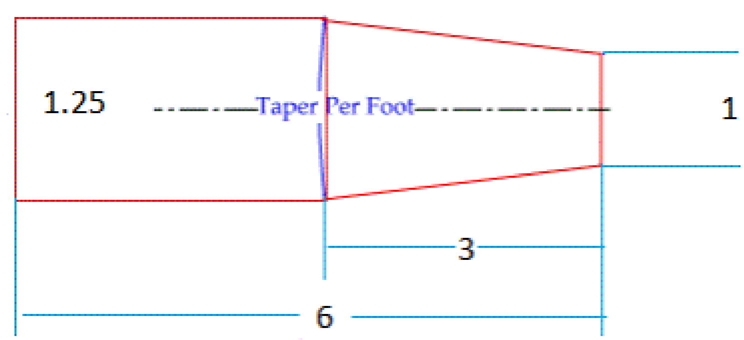


Figure 10. The main part of an inch taper

Formula:

Tpf = ((D-d) / length of taper) x 12

Example:

Tpf = ((1.25 – 1) / 3) x 12 = (.25 / 3) x 12 = 1 in.

Tailstock Offset Calculations

When calculating the tail stock offset, the taper per foot and total length of the workpiece must be known.

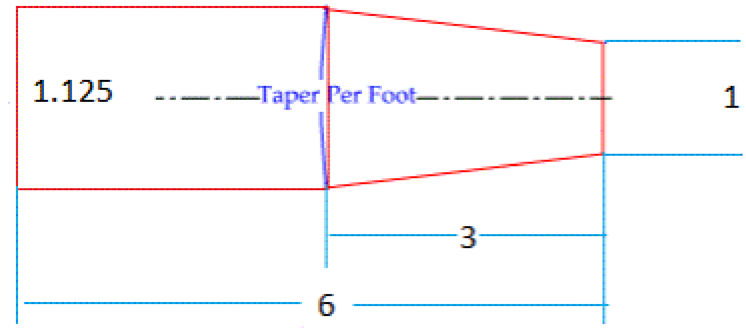


Figure 11. Dimension of a workpiece having a taper

Formula:

Tailstock offset = (tpf x total length of workpiece) / 24

Example:

1. Find tpf:

tpf = ((1.125 – 1) x 12) / 3 = (.125 x 12) / 3 = .50 in.

2. FInd the Tailstock offset:

Tailstock offset = (.5 x 6) / 24 = 3 / 24 = .125 in.

In some case where it is not necessary to find the taper per foot, the following simplified formula can be used.

Formula:

Tailstock Offset = (OL / TL) x ((D-d) / 2)

OL = Overall length of workpiece

TL = length of the tapered section

D = large diameter end

d = small diameter end

Example:

Tailstock Offset = (6 / 3) x ((1.125-1) / 2) = .125

Taper Turning

Using the compound rest to produce short or steep tapers. The tool bit must be fed in by hand, using the compound rest feed handle.

Cut a taper producer with Compound rest

Procedure:

1. Refer to the blueprint drawing for the amount of the taper required in degrees.
2. Loosen the compound rest lock screws.
3. Swivel the compound rest to the angle desired. (See first Picture)
4. Tighten the compound rest lock screws.
5. Adjust the tool bit on center and feed the cutting tool bit, using the compound rest feed screw.
6. Check the taper for size and fit.

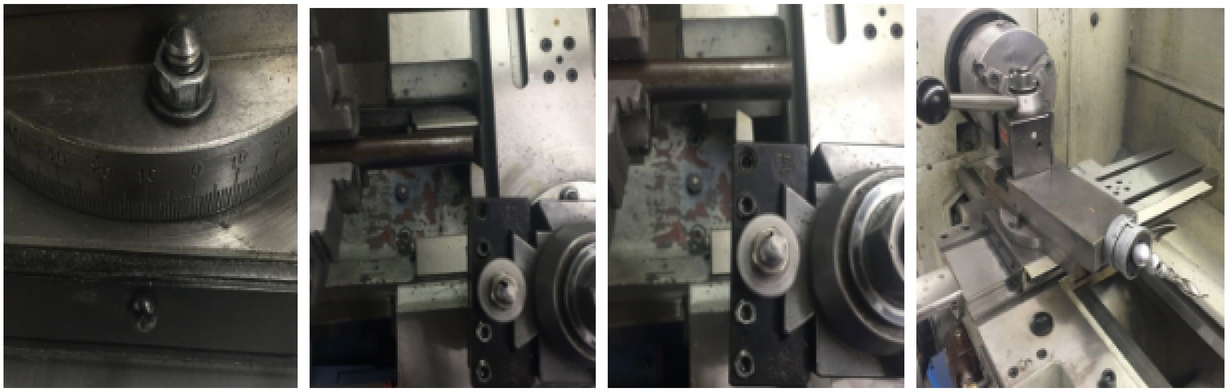


Figure 12. Taper Turning Operation

True workpiece in a 4-jaw chuck

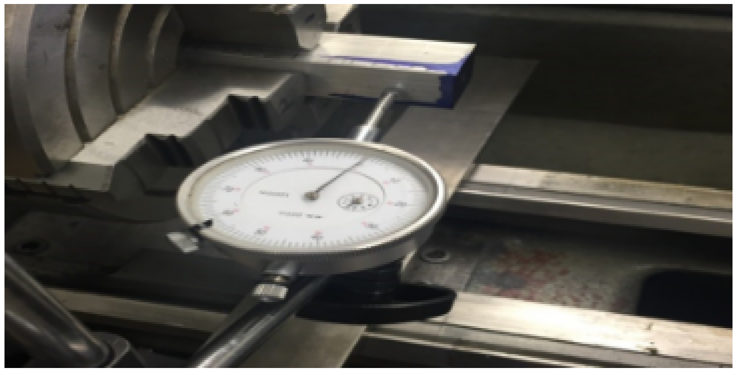
1. A dial or test indicator should be used whenever a machined diameter must be aligned to within a thousandths of an inch.

2. Procedure:

3. Insert the workpiece in the 4-jaw chuck and true it approximately, using either the chalk or surface gauge method.

4. Mount an indicator, in the tool post of the lathe.

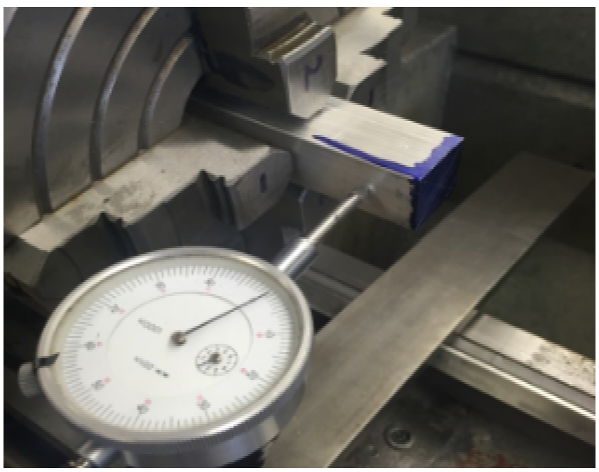
5. Set the indicator spindle in a horizontal position with the contact point set to the center height.

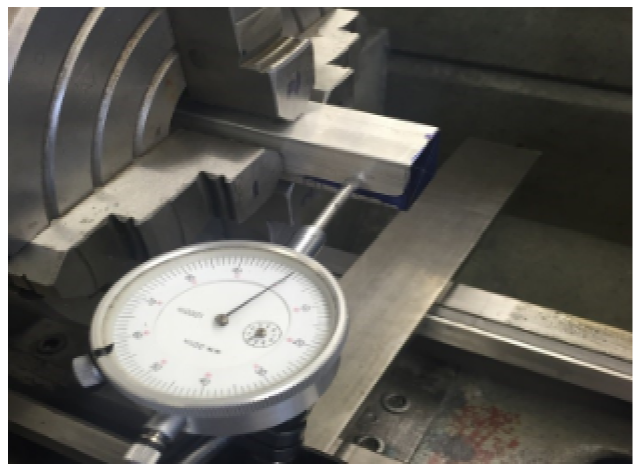


6. Bring the indicator point against the workpiece diameter so that it registers about .020 and rotate the lathe spindle by hand.

7. As you revolve the lathe, note the highest and lowest reading on the dial indicator.

8. Slightly loosen the chuck jaw at the lowest reading, and tighten the jaw at the high reading until the work is moved half the difference between the two indicator readings.

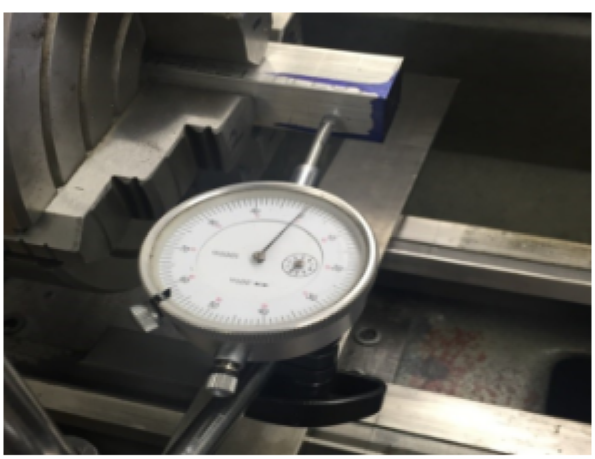


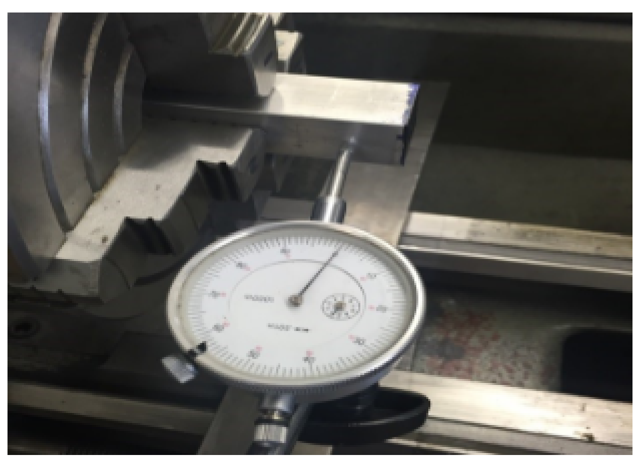


Side 1. Left and Right Side

9. Continue to adjust only these two opposite jaws until the indicator registers the at both jaws. Disregard the indicator readings on the work between these two jaws.

10. Adjust the other set of opposite jaws in the same manner until the indicator register the same at any point on the workpiece circumference.





Side 2. Left and Right Side

11. Tighten all jaws evenly to secure the workpiece firmly.
12. Rotate the lathe spindle by hand and recheck the indicator reading.

UNIT TEST

1. The compound rest is set at what angle?
2. Explain the different between rough and finish turning.
3. Should the point of the tool be set above, or at the center of the spindle axis when taking a facing cut?
4. What is the purpose of facing?
5. Why do we spot drill a workpiece?
6. What is the purpose of boring?
7. Name three types of parting tools.
8. Name three methods to align lathe centers.
9. Calculate the offset for the taper if D=2, d=1, OL=6, and TL=3. The formula is:

   Offset = (OL x (D-d)) / (2 x TL)
10. Please describe the producer for cut a taper.

10

Unit 5: Tapping

OBJECTIVE

After completing this unit, you should be able to:

- Describe the tapping procedure.
- Determine the RPM for tapping.
- Describe the filling and polishing.
- Describe the advanced workholding.

Tapping

Tapping is the process of cutting a thread inside a hole so that a cap screw or bolt can be threaded into the hole. Also, it is used to make thread on nuts.

Tapping can be done on the lathe by power feed or by hand. Regardless of the method, the hole must be drilled with the proper size tap drill and chamferred at the end.

Tapping Procedures

Good Practices

Using Tap Guides

Tap guides are an integral part in making a usable and straight tap. When using the lathe or the mill, the tap is already straight and centered. When manually aligning a tap, be careful, as a 90° tap guide is much more accurate than the human eye.

Using Oil

When drilling and tapping, it is crucial to use oil. It keeps the bits from squealing, makes the cut smoother, cleans out the chips, and keeps the drill and stock from overheating.

Pecking

Pecking helps ensure that bits don't overheat and break when using them to drill or tap. Peck drilling involves drilling partway through a part, then retracting it to remove chips, simultaneously allowing the piece to cool. Rotating the handle a full turn then back a half turn is common practice. Whenever the bit or tap is backed out, remove as many chips as possible and add oil to the surface between the drill or tap and the workpiece. Hand Tapping Procedure 1. Select drill size from chart. When choosing a tap size, this chart is the first place to look

Hand Tapping Procedure

1. Select drill size from chart.

When choosing a tap size, this chart is the first place to look.

Tap & Clearance Drill Sizes

Screw Size	Major Diameter	Threads Per Inch	Minor Diameter	Tap Drill				Clearance Drill			
				75% Thread for Aluminum, Brass, & Plastics		50% Thread for Steel, Stainless, & Iron		Close Fit		Free Fit	
				Drill Size	Decimal Equiv.	Drill Size	Decimal Equiv.	Drill Size	Decimal Equiv.	Drill Size	Decimal Equiv.
0	.0600	80	.0447	3/64	.0469	55	.0520	52	.0635	50	.0700
1	.0730	64	.0538	53	.0595	1/16	.0625	48	.0760	46	.0810
		72	.0560	53	.0595	52	.0635				
2	.0860	56	.0641	50	.0700	49	.0730	43	.0890	41	.0960
		64	.0668	50	.0700	48	.0760				
3	.0990	48	.0734	47	.0785	44	.0860	37	.1040	35	.1100
		56	.0771	45	.0820	43	.0890				
4	.1120	40	.0813	43	.0890	41	.0960	32	.1160	30	.1285
		48	.0864	42	.0935	40	.0980				
5	.125	40	.0943	38	.1015	7/64	.1094	30	.1285	29	.1360
		44	.0971	37	.1040	35	.1100				
6	.138	32	.0997	36	.1065	32	.1160	27	.1440	25	.1495
		40	.1073	33	.1130	31	.1200				
8	.1640	32	.1257	29	.1360	27	.1440	18	.1695	16	.1770
		36	.1299	29	.1360	26	.1470				
10	.1900	24	.1389	25	.1495	20	.1610	9	.1960	7	.2010
		32	.1517	21	.1590	18	.1695				
12	.2160	24	.1649	16	.1770	12	.1890	2	.2210	1	.2280
		28	.1722	14	.1820	10	.1935				
		32	.1777	13	.1850	9	.1960				
1/4	.2500	20	.1887	7	.2010	7/32	.2188	F	.2570	H	.2660
		28	.2062	3	.2130	1	.2280				
		32	.2117	7/32	.2188	1	.2280				
5/16	.3125	18	.2443	F	.2570	J	.2770	P	.3230	Q	.3320
		24	.2614	I	.2720	9/32	.2812				
		32	.2742	9/32	.2812	L	.2900				
3/8	.3750	16	.2983	5/16	.3125	Q	.3320	W	.3860	X	.3970
		24	.3239	Q	.3320	S	.3480				
		32	.3367	11/32	.3438	T	.3580				
7/16	.4375	14	.3499	U	.3680	25/64	.3906	29/64	.4531	15/32	.4687
		20	.3762	25/64	.3906	13/32	.4062				
		28	.3937	Y	.4040	Z	.4130				
1/2	.5000	13	.4056	27/64	.4219	29/64	.4531	33/64	.5156	17/32	.5312
		20	.4387	29/64	.4531	15/32	.4688				
		28	.4562	15/32	.4688	15/32	.4688				

9/16	.5625	12	.4603	31/64	.4844	33/64	.5156	37/64	.5781	19/32	.5938
		18	.4943	33/64	.5156	17/32	.5312				
		24	.5114	33/64	.5156	17/32	.5312				
5/8	.6250	11	.5135	17/32	.5312	9/16	.5625	41/64	.6406	21/32	.6562
		18	.5568	37/64	.5781	19/32	.5938				
		24	.5739	37/64	.5781	19/32	.5938				
11/16	.6875	24	.6364	41/64	.6406	21/32	.6562	45/64	.7031	23/32	.6562
3/4	.7500	10	.6273	21/32	.6562	11/16	.6875	49/64	.7656	25/32	.7812
		16	.6733	11/16	.6875	45/64	.7031				
		20	.6887	45/64	.7031	23/32	.7188				
13/16	.8125	20	.7512	49/64	.7656	25/32	.7812	53/64	.8281	27/32	.8438
7/8	.8750	9	.7387	49/64	.7656	51/64	.7969	57/64	.8906	29/32	.9062
		14	.7874	13/16	.8125	53/64	.8281				
		20	.8137	53/64	.8281	27/32	.8438				
15/16	.9375	20	.8762	57/64	.8906	29/32	.9062	61/64	.9531	31/32	.9688
1	1.000	8	.8466	7/8	.8750	59/64	.9219	1-1/64	.0156	1-1/32	1.0313

2. If necessary, add chamfer to the hole before tapping. Chamfers and countersinks are additional features that are sometimes desired for screws. For best results, the speed of the spindle should be between 150 and 250 rpm.

3. Get a tap guide. The hole is now ready to tap. To do this, use the taps and guide blocks near the manual mills. The guide blocks will have several holes for different sized taps. Select the one closest to the size of the tap being used and place it over the drilled hole.

4. Tap the block. Peck tap using the tap wrenches. Apply gentle pressure while turning the wrench a complete turn in, then a half-turn out. Peck tap to the desired depth.

5. Complete the tap. If the tap does not go any further or the desired depth has been reached, release pressure on the tap; it has likely bottomed out. Remove the tap from the hole.

Applying any more pressure is likely to break the tap. The smaller the tap, the more likely it is to break.

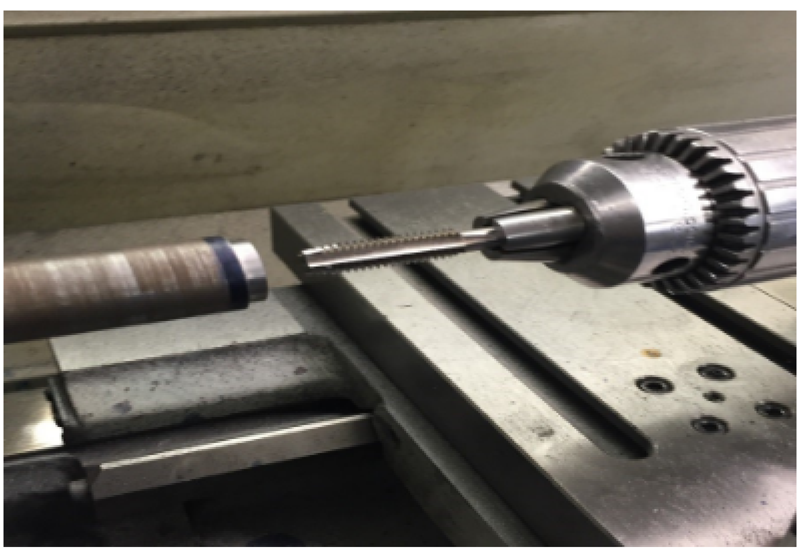


Figure 1. Tap

Tapping Procedure for Lathe

Procedure:

1. Mount the workpiece in the chuck.
2. Face and center drill.
3. Select the proper tap drill for the tap to be used.
4. Example: ¼ – 20 unc used # 7 drill.
5. Set the lathe to the proper speed and drill with the tap to the required depth. Use plenty cutting fluid.
6. Note: the workpiece will rotate when tapping using the lathe power. Use a very slow spindle speed. (40 to 60 rpm) and plenty of cutting fluid.
7. Chamfer the edge of the hole.

Filing in a Lathe

A workpiece should be filled in a lathe only to remove a small amount of stock, to remove burns or round off sharp corners. Workpiece should always be turned to about .002 to .003 inch of size, if the surface is to be filed. Hold the file handle in the left hand to avoid injury when filing on the lathe, so that the arms and hands can be kept clear of the revolving chuck.

Procedure:

1. Set the spindle speed to about twice that used for turning.
2. Mount the workpiece in the chuck, lubricate, and adjust the dead center in the workpiece.
3. Move the carriage as far to the right side as possible and remove the tool post (if needed)
4. Disengage the lead screw and feed rod.
5. Select right file to be used.
6. Start the lathe.
7. Grasp the file handle in the left hand and support the file point with the right hand finger.
8. Apply light pressure and push the file forward to its full length. Release pressure on the return stake.
9. Move the file about half the width of the file for each stroke and continue filing, using 30 to 40 strokes per minute until the surface is finished.

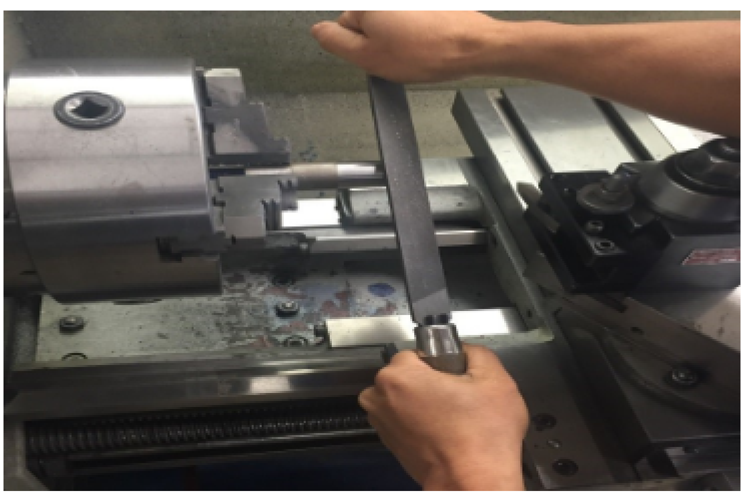


Figure 2. Filing

When filing in a lathe, the following safety should be observed.

- Roll up sleeves.
- Do not use a file without a properly fitted handle.
- Remove watches and rings.
- Do not apply too much pressure to the file.
- Clean the file frequently with a file brush. Rub a little chalk into the file teeth to prevent clogging and facilitate cleaning.

Polishing in a Lathe

After the workpiece has been filed, the finish may be improved by polishing with abrasive cloth.

Procedure:

1. Select the collect type and grade of abrasive cloth, for the finish desired, use a piece about 6 to 8 inch long and 1 inch wide.
2. Set the lathe to run on high speed (about 800-1000 rpm).
3. Disengage feed rod and lead screw.
4. Lubricate and adjust the dead center.
5. Start the lathe.
6. Hold the abrasive cloth on the workpiece.
7. With the right hand, press the cloth firmly on the work while tightly holding the other end of the abrasive cloth with the left hand.
8. Move the cloth slowly back and forth along the workpiece.

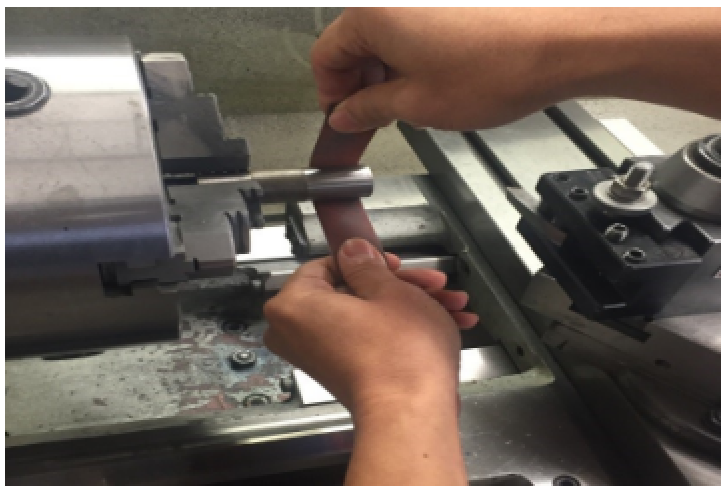


Figure 3. Polishing

When polishing in a Lathe, the following safety should be observed:

1. Roll up sleeves.
2. Tuck in any loose clothing

For normal finishes, use 80 to 100 grit abrasive cloth. For better finishes, use a finnier grit abrasive cloth.

Advanced Workholding

Some parts may be irregular, calling for specialized tools to hold them properly before being machined.

1. The part cannot be placed into a collet or chuck when cutting on the entire outside diameter of the stock.
2. Parts with holes through it should be pressed onto a lathe arbor (a tapered shaft) and then clamped onto the arbor rather than the part itself.
3. If the hole is too large, using a lathe arbor will not sufficiently support the piece. Instead, use the outside jaws to grasp the inside diameter of the part.
4. Parts with complex geometries may need to be attached onto a faceplate that will be further installed onto the spindle.

LATHE WORKHOLDING:

The following table provides a quick comparison of the strengths and weaknesses of the different means of holding the workpiece on a lathe:

Method	Precision	Repeatability	Convenience	Notes
Collets	High	High	High	Fast, high precision, high repeatability, grips well, unlikely to mar workpiece, grip spread over a wide area. Expensive chucks and collets. Handles limited lengths. Workpiece must be round andmust fit nearly exactly to the collet size.
3-Jaw Chuck With Soft Jaws	High	High	High	For larger workpieces, 3 jaw chucks withsoftjawsare the norm in the CNC world.
3-Jaw Self-Centering Chuck with Hard Jaws	Low	Low	High	Common, cheap, simple. Low precision, low repeatability if you remove the workpiece and have to put it back.
4-Jaw Chuck	High	High	Medium	Can be time consuming to individually adjust the jaws, but will result in high precision. Can hold pieces offset for turning cams or eccentrics. Can hold irregular shapes and square or rectangular stock.
6-Jaw Self-Centering Chuck	Medium	Medium	High	Best for thin wall work or to grip finished edges of workpiece. Obviously good for hex stock.
Faceplate Turning	Varies w/ Setup	Medium	Low	Great for irregular shapes. Involves clamps like a milling setup. May need counterweights to keep things balanced.
Turning Between Centers	High	High	Low	Great precision, allows part to be put back between centers with very high repeatability.
Constant Face Turning	High	High	High	The modern alternative to turning between centers. Instead of using lathe dogs, which are kind of a nuisance to set up, the constant face system uses hydraulic or other force to grip and drive the spindle end.
Expanding Arbors	High	High	High	These work from the inside out rather than the outside in but are otherwise much like collets.

Method describes the particular technique or tooling to be used.

Precision describes how precisely the workpiece will be held, or how close to concentrically it will run with the spindle before taking any cuts.

Repeatability describes how easy it is to take the workpiece out and then get it back in precisely again.

UNIT TEST

1. What drill size to be used for ½ -20 tap?
2. What is the purpose of chamfer?
3. What is the best RPM for tapping?
4. What spindle speed do we set for filing?

5. What is the purpose of polishing?
6. What is the best grit abrasive cloth for normal finishes?
7. What type of work is best suited to three-jaw chucks?
8. What are the special characteristics of the three-jaw chuck?
9. Explain the different between three-jaw chuck and 4-jaw chuck.
10. What are the advantages and disadvantages of a collect chuck?

11

Unit 6: Lathe Threading

OBJECTIVE

After completing this unit, you should be able to:

- Determine the infeed depth.
- Describe how to cut a correct thread.
- Explain how to calculate the pitch, depth, and minor diameter, width of flat.
- Describe how to set the correct rpm.
- Describe how to set the correct quick change gearbox.
- Describe how to set the correct compound rest.
- Describe how to set the correct tool bit.
- Describe how to set both compound and crossfeed on both dials to zero.
- Describe the threading operation.
- Describe the reaming.
- Describe how to grind a tool bit.

Lathe Threading

Thread cutting on the lathe is a process that produces a helical ridge of uniform section on the workpiece. This is performed by taking successive cuts with a threading toolbit the same shape as the thread form required.

Practice Exercise:

1. For this practice exercise for threading, you will need a piece of round material, turned to an outside tread Diameter.
2. Using either a parting tool or a specially ground tool, make an undercut for the tread equal to its single depth plus .005 inch.
3. The formula below will give you the single depth for undertaking unified threads:

$$d = P \times 0.750$$

Where d = Single Depth

P = Pitch

n = Number of threads per inch (TPI)

Infeed Depth = .75 / n

Thread Calculations

To cut a correct thread on the lathe, it is necessary first to make calculations so that the thread will have proper dimensions. The following diagrams and formulas will be helpful when calculating thread dimensions.

Example: Calculate the pitch, depth, minor diameter, and width of flat for a ¾–10 NC thread.

P = 1 / n = 1 / 10 = 0.100 in.

Depth = .7500 x Pitch = .7500 x .100 = .0750 in.

Minor Diameter = Major Diameter – (D + D) = .750 – (.075 + .075) = 0.600 in.

Width of Flat = P / 8 = (1 / 8) x (1/10) = .0125 in.

Procedure for threading:

1. Set the speed to about one quarter of the speed used for turning.
2. Set the quick change gearbox for the required pitch in threads. (Threads per inch)

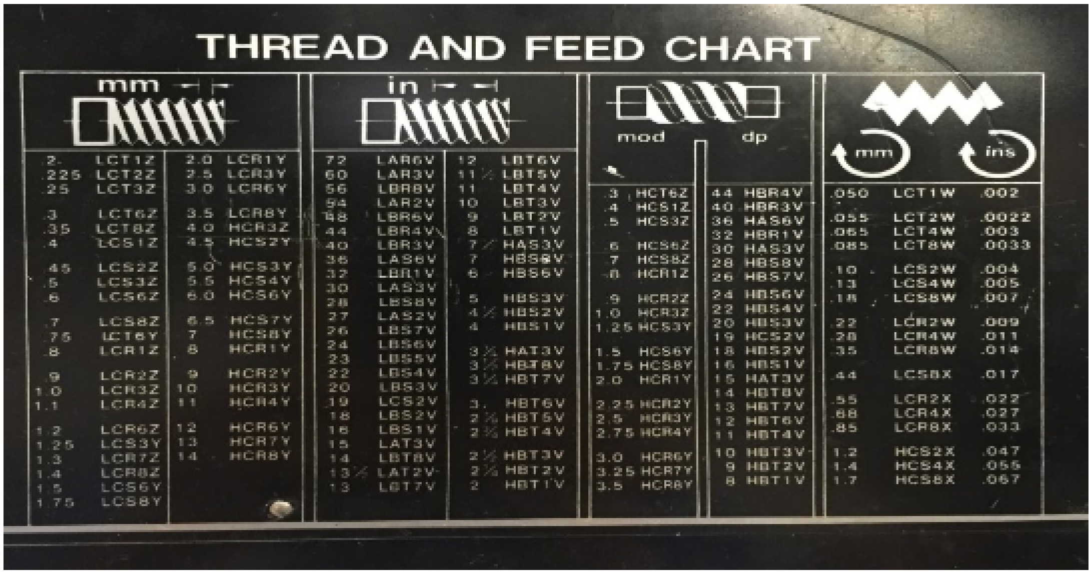


Figure 1. Thread and Feed Chart

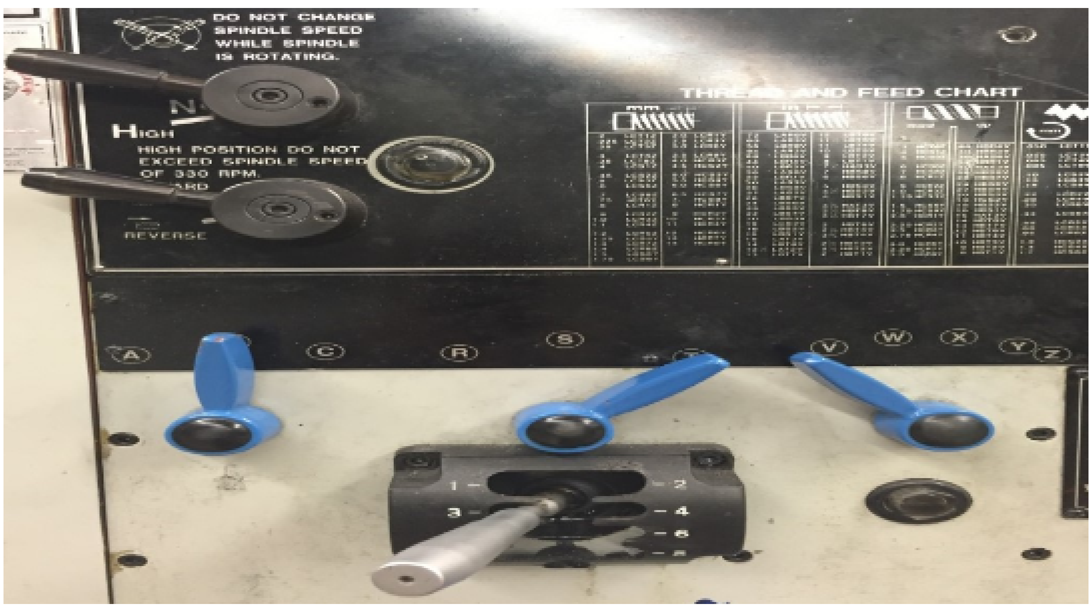


Figure 2. Setting Gearbox

3. Set the compound rest at 29 degrees to the right for right hand threads.

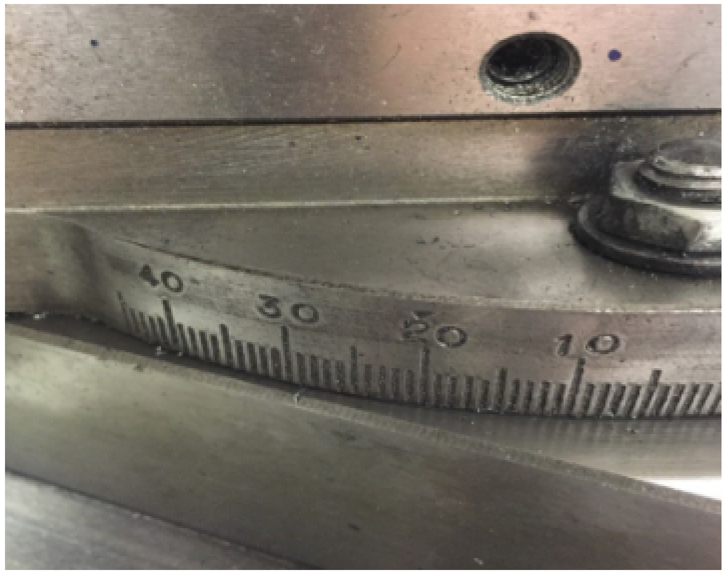


Figure 3. 29 Degrees

4. Install a 60 degree threading tool bit and set the height to the lathe center point.

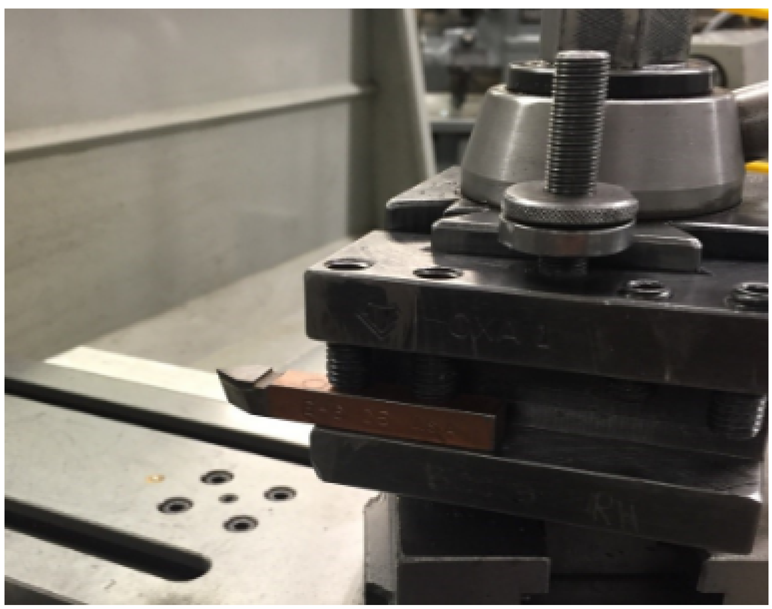


Figure 4. 60 Degree Threading Tool

5. Set the tool bit and a right angles to the work, using a thread gage.

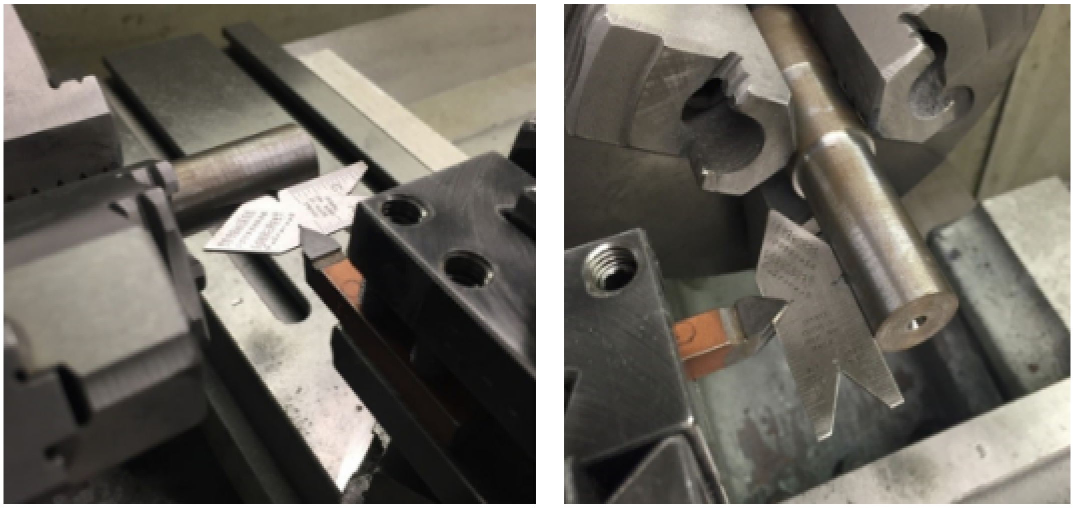


Figure 5. Using the Center gage to positionthe tool for machining Threads

6. Using a layout solution, coat the area to be threaded.

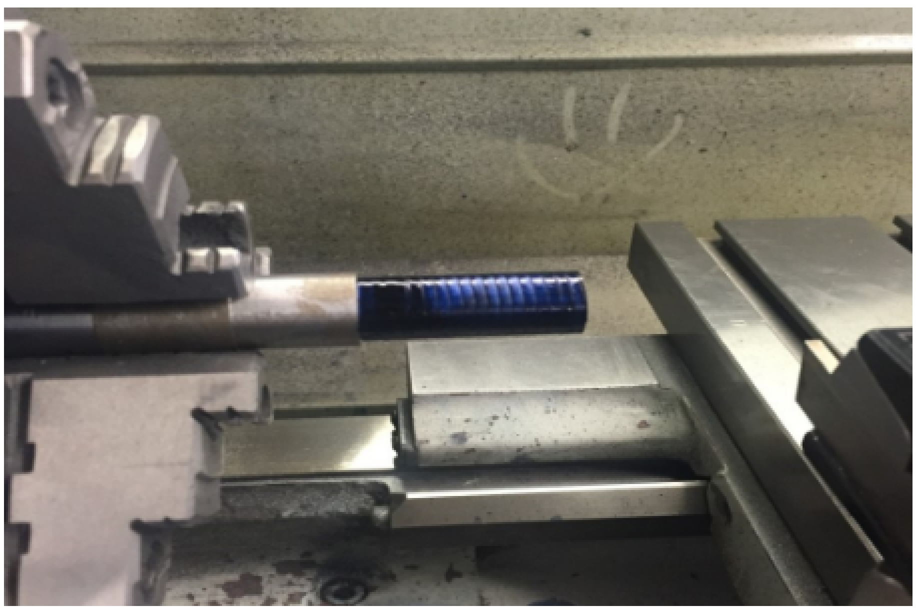


Figure 6. Layout

7. Move the threading tool up to the part using both the compound and the cross feed. Set the micrometer to zero on both dials.

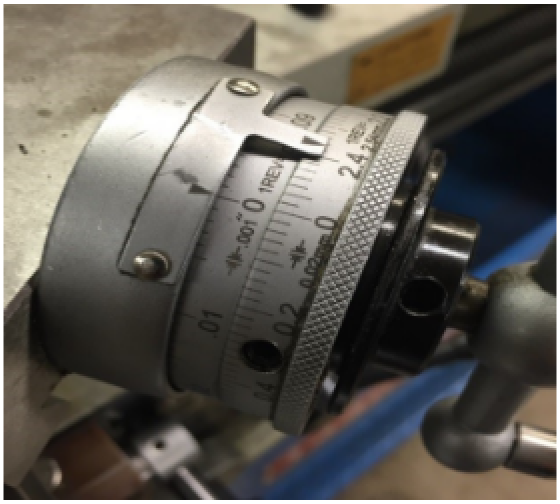


Figure 7. Compound

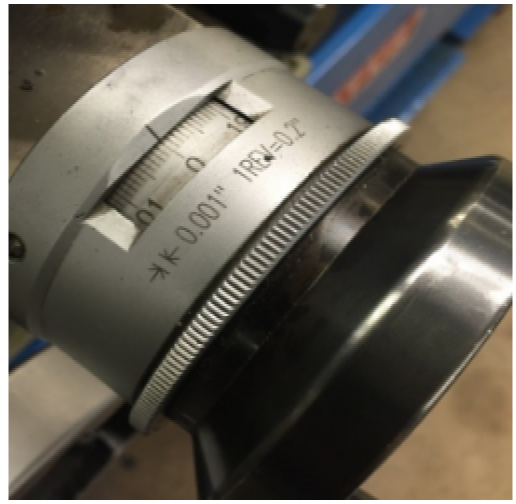


Figure 8. Cross Feed

8. Move cross feed to the back tool off the work, move carriage to the end of the part and reset the cross feed to zero.

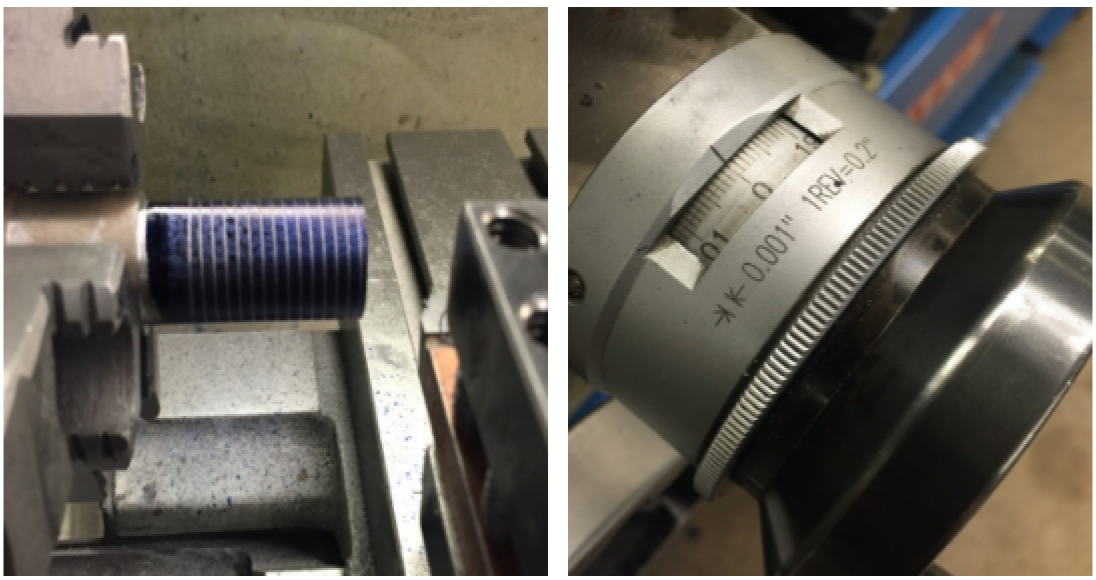


Figure 9. End of the part and Cross feed to Zero

9. Using only the compound micrometer, feed in .001 to .002 inch.

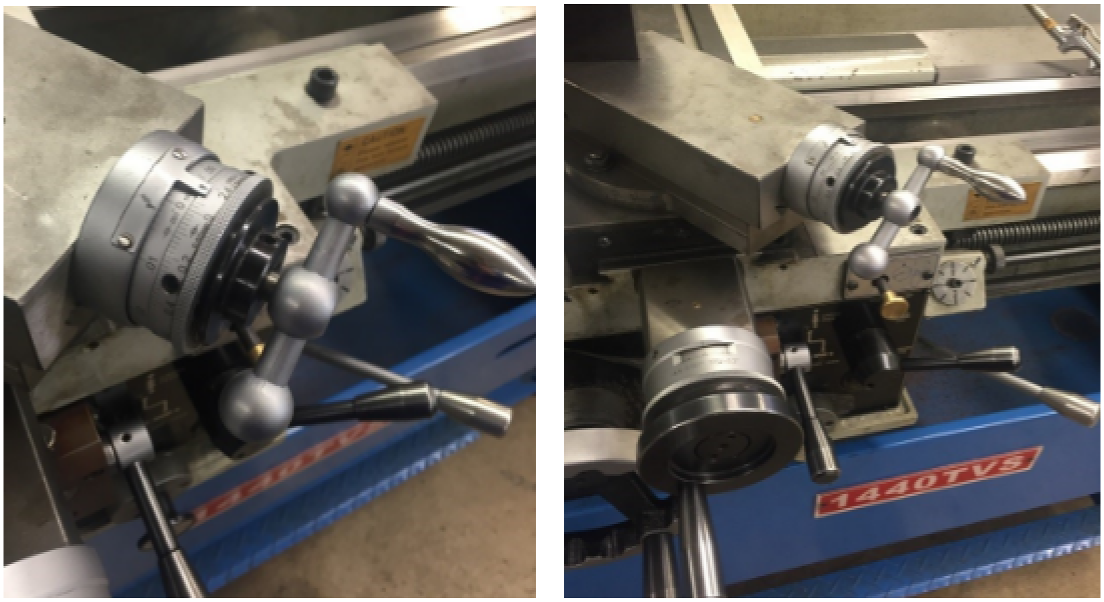


Figure 10: Compound feed in .002 inch

10. Turn on the lathe and engage the half nut.

Figure 11: On/Off Lever and Half Nut

11. Take a scratch cut on the part without cutting fluid. Disengage the half nut at the end of the cut, stop the lathe and back out the tool using the cross feed. Return the carriage to the starting position.

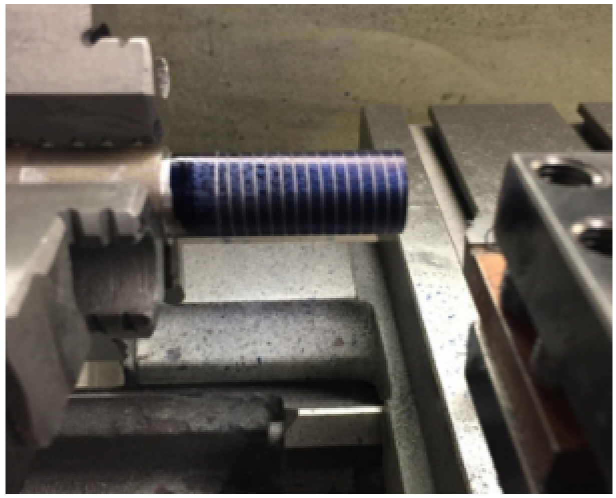


Figure 12. Starting Position

12. Using a screw pitch gage or a rule check the thread pitch. (Threads per inch)

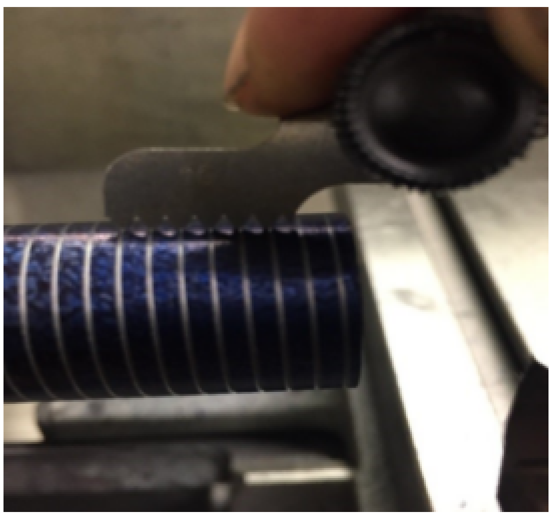


Figure 13. Screw Pitch Gage Figure 14. Screw Pitch Gage(10)

13. Feed the compound in .005 to .020 inch for the first pass using cutting oil. As you get near the final size, reduce the depth of cut to .001 to .002 inch.

14. Continue this process until the tool is within .010 inch of the finish depth.

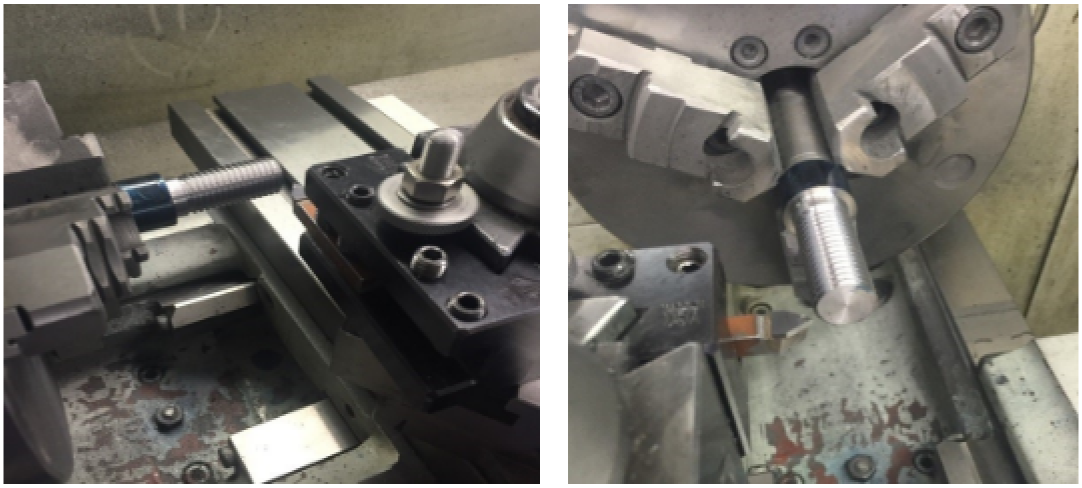


Figure 15. Threading operation

15. Check the size using a screw thread micrometer, thread gage, or using the three wire system.

Figure 16. Three wire measurement

16. Chamfer the end of the thread to protect it from damage.

Reaming

Reamers are used to finish drilled holes or bores quickly and accurately to a specified sized hole and to produce a good surface finish. Reaming may be performed after a hole has been drilled or bored to within 0.005 to 0.015 inch of the finished size since the reamer is not designed to remove much material.

The workpiece is mounted in a chuck at the headstock spindle and the reamer is supported by the tailstock.

The lathe speed for machine reaming should be approximately 1/2 that used for drilling.

Reaming with a Hand Reamer

The hole to be reamed by hand must be within 0.005 inch of the required finished size.

The workpiece is mounted to the headstock spindle in a chuck and the headstock spindle is locked after the workpiece is accurately setup. The hand reamer is mounted in an adjustable reamer wrench and supported with the tailstock center. As the wrench is revolved by hand, the hand reamer is fed into the hole simultaneously by turning the tailstock handwheel. Use plenty cutting fluid for reaming.

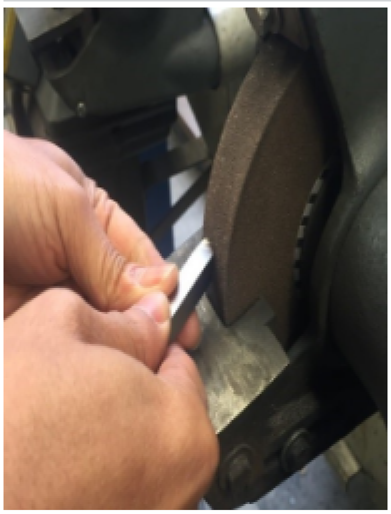
Reaming with a Machine Reamer

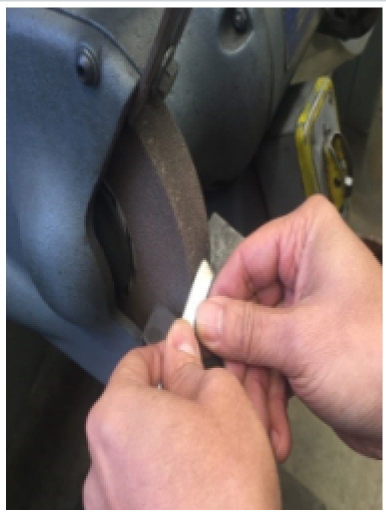
The hole to be reamed with a machine reamer must be drilled or bored to within 0.010 inch of the finished size so that the machine reamer will only have to remove the cutter bit marks. Use plenty cutting fluid for reaming.

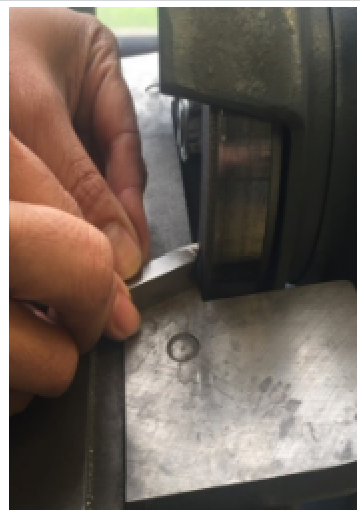
Grind a Lathe Tool bit

Procedure:

1. Grip the tool bit firmly while supporting the hand on the grinder tool set.
2. Hold the tool bit at the proper angle to grind the cutting edge angle. At the same, tilt the bottom of the tool bit in towards the wheel and grind 10 degrees side relief or clearance angle on the cutting edge. The cutting edge should be about .5 inches long and should be over about ¼ the width of the tool bit.
3. While grinding tool bit, move the tool bit back and forth across the face of the grinding wheel. This accelerates grinding and prevents grooving the wheel.
4. The tool bit must be cooled frequently during the grinding operation by dip into the water. Never overheat a tool bit.
5. Grind the end cutting angle so that it form an angle a little less than 90 degrees with the side cutting edge. Hold the tool so that the end cutting edge angle and end end relief angle of 15 degrees are ground at the same time.
6. Check the amount of end relief when the tool bit is in the tool holder.
7. Hold the top of the tool bit at about 45 degrees to the axis of the wheel and grind the side rake about 14 degrees.
8. Grind a slight radius on the point of the cutting tool, being sure to maintain the same front and side clearance angle.


Grind front


Grind side


Grind radius

Cutting tool Materials

Lathe tool bits are generally made of four materials:

1. High speed steel
2. Cast alloys
3. Cemented Carbides
4. Ceramics

The properties that each of these materials possess are different and the application of each depends on the material being machined and the condition of the machine.

Lathe tool bits should possess the following properties.

1. They should be hard.
2. They should be wear resistant.
3. They should be capable of standing up to high temperatures developed during the cutting operation.
4. They should be able to withstand shock during the cutting operation.

Cutting tool Nomenclature

Cutting tools used on a lathe are generally single pointed cutting tools and although the shape of the tool is changed for various applications. The same nomenclature applies to all cutting tools.

Procedure:

1. Base: the bottom surface of the tool shank.
2. Cutting Edge: the leading edge of the tool bit that does the cutting.
3. Face: the surface against which the chip bears as it is separated from the work.
4. Flank: The surface of the tool which is adjacent to and below the cutting edge.
5. Nose: the tip of the cutting tool formed by the junction of the cutting edge and the front face.
6. Nose radius: The radius to which the nose is ground. The size of the radius will affect the finish. For rough cut, a 1/16 inch nose radius used. For finish cut, a 1/16 to ⅛ inch nose radius is used.
7. Point: The end of the tool that has been ground for cutting purposes.
8. Shank: the body of the tool bit or the part held in the tool holder.
9. Lathe Tool bit Angles and Clearances

Proper performance of a tool bit depends on the clearance and rake angles which must be ground on the tool bit. Although these angles vary for different materials, the nomenclature is the same for all tool bits.

- Side cutting edge angle: The angle which the cutting edge forms with the side of the tool shank. This angle may be from 10 to 20 degrees depending on the material being cut. If angle is over 30 degrees, the tool will tend to chatter.
- End cutting edge angle. The angle formed by the end cutting edge and a line at right angle to the centerline of the tool bit. This angle may be from 5 to 30 degrees depending on the type of cut and finish desired. For roughing cuts an angle of 5 to 15 degrees, angle between 15 and 30 degrees are used for general purpose turning tools. The larger angle permits the cutting tool to be swivelled to the left when taking light cuts close to the dog or chuck, or when turning to a shoulder.
- Side Relief (clearance) angle: The angle ground on the flank of the tool below the cutting edge. This angle may be from 6 to 10 degrees. The side clearance on a tool bit permit the cutting tool to advance lengthwise into the rotating work and prevent the flank from rubbing against the workpiece.
- End Relief (clearance) angle: the angle ground below the nose of the tool bit which permits the cutting tool to be fed into the work. This angle may be 10 to 15 degrees for general purpose cut. This angle must be measured when the tool bit is held in the tool holder. The end relief angle varies with the hardness and type of material and type of cut being taken. The end relief angle is smaller for harder materials, to provide support under the cutting edge.
- Side Rake Angle: The angle at which the face is ground away from the cutting edge. This angle may be 14 degrees for general purpose tool bits. Side rake centers a keener cutting edge and allows the chip to flow away quickly. For softer materials, the side rake angle is generally increased.
- Back (Top) Rake: The backward slope of the tool face away from the nose. This angle may be about 20 degrees and is provide for in the tool holder. Back rake permits the chips to flow away from the point of the cutting tool.

UNIT TEST

1. What is pitch for ¼-20 tap?
2. To what angle must the compound be turned for Unified Thread?
3. Explain why you swivel the compound in Question 2.
4. What is the depth of thread for UNF ½-20 screw?
5. How would you make a left-hand thread? This is not covered in the reading—think it out?
6. What Tool bit do we use for cutting thread?
7. Please describe Center Gage.
8. What do we use to check the thread pitch(Thread Per Inch)?
9. The first and final pass, how much do we feed the compound in?
10. Name four material that use to make Tool bits.

Chapter Attribution Information

This chapter was derived from the following sources.

- **Lathe** derived from Lathe by the Massachusetts Institute of Technology, CC:BY-NC-SA 4.0.
- **Cutting Tool Terminology** derived from Lathe Cutting Tools – Cutting Tool Shapes by the Wisconsin Technical College, CC:BY-NC 4.0.
- **Cutting Tool Terminology** derived from Cutter Types (Lathe) by the University of Idaho, CC:BY-SA 3.0.
- **Centering** derived from [Manual Lathes Document]

PART III

Chapter 3: Drill Presses

12

Chapter 3: Drill Press

OBJECTIVE

After completing this unit, you should be able to:

- Identify Drill Press
- Understand the safety rules.
- Describe Tooling to be use.
- Describe Reaming a hole.
- Describe Drilling a hole procedure.
- Describe power feed and hand feed tapping procedure.
- Describe Dressing the Wheel procedures.

Description

Drilling machines, or drill presses, are primarily used to drill or enlarge a cylindrical hole in a workpiece or part. The chief operation performed on the drill press is drilling, but other possible operations include: reaming, countersinking, counterboring, and tapping.

The floor type drill press used in the Student Shop is a very common machine, found in both home and industrial workshops. This style drill press is composed of four major groups of assemblies: the head, table, column, and base.

The head contains the motor and variable speed mechanism used to drive the spindle. The spindle is housed within the quill, which can be moved up or down by either manual or automatic feed. The table is mounted on the column, and is used to support the workpiece. The table may be raised or lowered on the column, depending upon the machining needs. The column is the backbone of the drill press. The head and base are clamped to it, and it serves as a guide for the table. The cast-iron base is the supporting member of the entire structure.

Safety

1. Be familiar with the location of the start and stop switches.
2. The drill press table should be cleared of miscellaneous tools and materials.
3. Ensure that all drill bits are sharpened and chucks are in working condition. Any dull drill bits, battered tangs or sockets should not be used.
4. Never attempt to remove scraps from the table by hand. Use brushes or other proper tools.
5. Never attempt to perform maintenance on the machine without the power cord unplugged.
6. Never insert a chuck key into the chuck until the machine has been turned off and stopped completely.
7. Belts and pulleys should be guarded at all times. If any are frayed, immediately report to the instructor for replacement.
8. All workpieces should be secured by a vise or clamp before starting the machining.

9. If the workpiece moves while in the vise or clamp:
   - Do not attempt to hold the workpiece in place by hand.
   - Do not try to tighten the vise or clamp while the machine is turned on.
   - Turn the power off and wait for the machine to stop completely before re-tightening the vise or clamp.
10. Use the proper speed settings and drill type for the material to be machined.
11. When mounting a drill bit, it should be to the full depth and centered in the chuck.
12. Eliminate the possibility of the drill bit hitting the table by using a clearance block and by adjusting the feed stroke.
13. Always feed the bit slowly into the workpiece. If the hole to be drilled is deep, draw the bit back often to remove shavings.
14. Before leaving the drill press for any amount of time, the power should be turned off and machine should be at a complete stop.
15. In any unsafe condition or movement is observed on the drill press, report it to the instructor immediately.
16. Leave the drill press cleaned and tidy at all times.

Procedures

Successful operation of the drill press requires the operator to be familiar with the machine and the desired operation. The following are some good observations to follow when drilling a hole:

1. Prior to drilling a hole, locate the hole by drawing two crossing lines. Use a center punch to make an indentation for the drill point to aid the drill in starting the hole.
2. Select the proper drill bit according to the size needed.
3. Select an appropriate size center drill.
4. Select a cutting fluid.
5. Properly secure the workpiece to the table.
6. Select the correct RPM for the drill bit. Take into account: size of bit, material, and depth of hole to be drilled.
7. Use an interrupted feed, called peck drilling, to break up the chips being produced.
8. Pilot holes should be used on holes larger than 3/8" dia. Holes are to be enlarged in no more than 1/4" increments.
9. Clean the drill press and surrounding area when finished.

*** Hard and fast rules are not always practical for every operation performed in a drill press, since many factors can influence the speed and feed at which a material can be worked. The above suggestions, combined with knowledge of the tool being used, will provide a reasonable guideline for the operator using a drill press.

Tooling

Twist drills- A twist drill is a pointed cutting tool used for making cylindrical holes in the workpiece. It has helical flutes along its length for clearing chips from the holes. Twist drills are the most common used today, but there are many other styles with different purposes. A twist drill is composed of three major parts: a shank, body, and point. The shank is the part of the drill bit held in the spindle of the drill press. The drill press' power is transferred through the shank. Shanks are either one of two styles, straight or tapered. Straight shank drills are held in a friction chuck. Slippage between the drill bit and the chuck is often a problem, especially for larger drills. When using drill bits larger than 1/2" dia., tapered shank drill bits are often used. These provide greater torque with less slippage than straight shank drill bits. The body, as described above, generally has two flutes to clear chips. These flutes are not cutting edges and should not be used for side cutting as an end mill. The point of the drill bit does all of the cutting action, which produces the cut chips. The point is ground on the end of the drill bit.

Holes produced by twist drill bits are generally oversize by as much as up to 1% of the bit's dia. The accuracy of the hole is dependent on the following factors: size of the bit, accuracy of the bit's point, accuracy of the chuck, accuracy and rigidity of the spindle, rigidity of the press, and rigidity of the workpiece in its setup. All holes to be drilled should be started with a centerpunch, centerdrill, or both.

Twist Drill Formats

1. Number sizes: #80 (.0135") to #1 (.228")

2. Letter sizes: A (.234") to Z (.413")

3. Fractional sizes: 1/64" (.0156") upwards by 64ths/inch

Reaming a Hole

A reamer is a precision cutting tool designed to finish a hole to a specific dia. Since drill bits produce slightly oversized holes, reamers are used where precision tolerances are required, .001". Reamers have little if no cutting action on their ends, so a pilot hole is required as a preoperation to reaming. Some general guidelines for using reamers are:

- Drill a pilot hole that is a bit smaller

- When starting a hole, drill it a bit undersized.

- Drive the reamer at a slow, constant speed.

1. The cutting speed for reaming should be about 1/3 of the speed used for drilling operation of the same material.

2. Before reaming, leave about .010" of material on holes up to 1/2", and about .020" of material on larger holes.

3. Never rotate a reamer in the reverse direction.

4. Use the proper cutting fluid for the material.

5. Remove the reamer from the hole occasionally while cutting to clear chips, which can cause galling on the surface of the precision hole.

6. Never stop the machine with the reamer in the hole.

7. Clean and return the reamer to its proper storage place.

Countersinks – Countersinking is an operation in which a cone-shaped enlargement is cut at the top of a hole to form a recess below the surface. A conical cutting tool is used to produce this chamfer. When countersinking, the cutter must be properly aligned with the existing hole, and should be rotated about 1/3 the cutting speed of the drilling operation for the hole. Countersinking is useful in removing burrs from edges of holes, as well as providing a flush fit for flat-headed fasteners.

Counterbores – Counterboring is the process of cylindrically enlarging a hole part way along its length. A counterbore cutter is similar to a drill bit in that it has a shank and fluted body, but instead of a point, it has a smaller diameter pilot portion. The pilot fits into a pre-drilled hole, and guides the counterbore. Therefore the counterbore must be aligned with the original hole, so the pilot will follow the hole properly. Counterbores are used to accommodate studs, bolts, or socket head capscrews where a flush surface application is required.

Tapping – A tap is a tool used to cut internal threads in a cylindrical hole. A tap is fluted like a drill, but the flutes actually perform the cutting operation. The flutes extend the length of the threaded section and also serve to remove the chips being produced. The most common taps used are:

1. The starting or tapered tap. This tap is used to start threads. At least the first six threads of this tap are tapered before the full diameter of the thread is reached.

2. The plug tap. This is the general use tap, and is used to cut threads after the taper tap has been used and removed. Three to five of its first threads are tapered. This is the last tap used if the hole extends all the way through the workpiece.

Cutting fluids should always be used when tapping holes. It is also recommended

to advance the tap one full turn and the reverse it 1/4 turn to break the chip being formed.

Always use a tap handle, not pliers or a crescent wrench to turn the tap. They can damage

the tap, and the unequal torque provided can cause a thread to be cut poorly.

Drilling a Hole

1. A center drill should be used to aid with the drilling.

2. A center drill has short flutes and a thick shaft. Therefore, it is very stiff and will not wander. Since a center drill doesn't cut as easily as a drill bit, use cutting oil.

3. The hole is ready to be cut with a drill bit now.

4. It is recommended to use a smaller pilot hole before drilling the final one if the hole is large. This increases the accuracy of the hole and allow the bits to last longer.

5. If the hole is deeper than the diameter of the hole, use cutting liquid and back off occasionally.

6. The spindle speed should be reduced as drill size is increased.

7. When drilling a through hole, make sure the bit will not drill into the table after drilling through the work.

8. Set a depth stop on the quill to reach a desired depth of the hole.

Deburring a Hole

Often times the top edge of a hole will be clean, but the bottom edge will have some burrs. To remove the burrs, run a deburring tool in the hole around the edge with medium pressure. Repeat this process until the edges are no longer sharp.

Power Feed Tapping Procedure

1. Power feed tapping is similar to hand tapping. Instead of tapping by hand, however, use the drill press to tap the workpiece.

2. Before starting the machine, change the drill press to low gear.

3. Release the quill lock and move the quill to the lowest it can go. This ensures that there is sufficient space to tap to the desired depth.

4. Turn the spindle on FORWARD and set the spindle speed to 60 RPM.

5. Feed the tap down. When the tap grabs the stock, it will automatically feed itself into the hole.

6. When the desired depth has been reached, quickly flip the spindle direction switch from forward to reverse. This will reverse the direction of the tap and remove it from the hole. Reversing the direction in one fluid motion will prevent damage to the tapped hole and the tap.

7. Turn off the machine.

8. Clean the tapped hole, tap, and power feed machine before leaving.

Hand Feed Tapping Procedure

1. Ensure correct tap size for the drilled hole. If the size is off, the tap might break in the hole.
2. Place a center finder into the chuck and align the quill over the hole.
3. Fix a tapered guide to the chuck.
4. Position the tap and apply gentle pressure with the quill while turning the tap.
5. For every quarter turn of thread cut, it is wise to back the tap up slightly.

UNIT TEST

1. What is the chief operation performed on the drill press?

2. Please lists other possible operations performed on the drill press?

3. The drill press is composed of four major, Please lists them.

4. Please name three major parts of a twist drill.

5. All holes to be drilled should be started with What?

6. Name three types of twist drill formats.

7. What is the cutting speed for reaming ?

8. Before reaming, How much of material to leave on holes up to 1/2”, and on larger holes?

9. Explain the different between countersinks and counterbores.

10. Explain the Power Feed Tapping vs. Hand Feed Tapping.

Chapter Attribution Information

This chapter was derived from the following sources.

- **Drill Press** derived from Mechanical Engineering Tools by the Massachusetts Institute of Technology, CC:BY-NC-SA 4.0.
- **Reaming** derived from Reaming with a Reamer by CORE-Materials, CC:BY-NC-ND.

PART IV

Chapter 4: Bandsaws

13

Chapter 4: Bandsaw

OBJECTIVE

After completing this unit, you should be able to:

- Identify Bandsaw.
- Understand the safety rules.
- Describe operation of the Horizontal Bandsaw.
- Describe operation of the Vertical Bandsaw.
- Describe the Chop Saw.
- Explain saw blades selection.
- Describe the Tooth set.
- Explain Vise loading.
- Describe lubrication.

Bandsaw

There are two types of band saws available in the market – one is the horizontal band saw and the other is vertical band saw. Band saws have become fairly common in any machine shop and require no special skills to use. However, considering the nature of work involved, it is important that you familiarize yourself with the equipment and follow a few simple steps when using a band saw. Here are some simple instructions on how to safely use vertical band saws.

Step 1: Safety

Before handling any sort of power tool it is important to wear safety goggles, gloves and any other relevant safety gear. Try to minimize loose clothing as it could potentially get caught in the saw blades.

Step 2: Know your Machine

Most band saw machines come with variable speeds but if yours is only one speed that is not a cause of concern. Your power switch and speed indicators are usually located on the left side of the machine if you are standing in front of the machine. The transmission shift lever and the variable speed control will be located at the back of the machine. The tilt table at the front allows you to move the object you are cutting with ease. The air blower at the top of the blade makes sure any particles are blown away from you and not towards you.

Step 3: Measurements

Mark your measurements on the object you need to cut. Make sure that the sizes that you are trying to cut are able to fit through the machine. This is more important for any contour sawing as opposed to straight line cutting. When cutting straight lines, make sure the width of the object does not increase the distance between the blade and the column of the machine. If you are cutting a contour, make sure the object can pass through the gap between the column and blade in all directions. If that is not the case, simple cutoff any excess object that you can before using the machine.

Step 4: Set Speed

Depending on what type of material you are cutting, the speed of the saw will vary accordingly. The general rule of thumb is to use fast speed for softer materials and relatively slower speed for harder materials. Once you switch on the machine, wait a few seconds as it powers up and settles at its working speed.

Step 5: Feeding

Once you have marked the object and set the speed, you are ready to feed the object through the machine. Depending on what type of machine you are using, the object can be fed manually or using the powered feeder. Before feeding the object and even before turning the machine on, check which side the teeth of the blade are facing. This is the side you will be feeding the object from.

If you are manually feeding the object make sure you keep your hands out of the way of the blade and if you are using the powered feeder, make sure you are not in a position to get caught in any of the moving part of the machine. Firmly grab the object, align the cutting line with the blade, clear your hands from the path of the blade and push the object into the line of the band saw blades. Once you have cut through the object, remove the articles from the machine and turn the machine off.

Safety

1. Know where the start and stop switches are located.
2. Make sure that the blade is adjusted correctly and that the doors are closed before using the machine.
3. Use the right blade for the thickness of the material being cut. There should be at least three teeth for the thickness of the material.
4. Never run the machine faster than the recommended speed for the specific material.
5. Make sure that the saw blade is sharp enough to cut the material.
6. Adjust all guards in place before operation. The upper guide/guard assembly should be placed within ¼ of an inch of the workpiece.
7. Make sure the workpiece is flat on the table before starting the cut.
8. Do not start cutting until the blade has attained its full speed.
9. Maintain a safe distance between your hands and the blade.
10. Use the appropriate amount of force when cutting a piece.
11. When pushing irregular or small stock, use a board or push stick.
12. Be mindful of thin pieces jamming the slot or hitting the end of the slot in the insert.
13. If blade binding occurs, turn the machine off by unplugging the power cord and wait until it stops fully before attempting to remove the blade from the workpiece. Blade binding is when the saw blade gets stuck in the work piece.
14. Never make adjustments until the machine has fully stopped.
15. In the event of a broken band, unplug and keep away from the machine until it comes to a complete stop. Contact the instructor immediately.
16. Remove excess chips using brushes or rags after stopping the machine in order to prevent large quantities of chips from accumulating.
17. Make sure the machine is turned off and clean before leaving the workspace.

Horizontal Bandsaw

Adjusting the Vice

Loading Vice

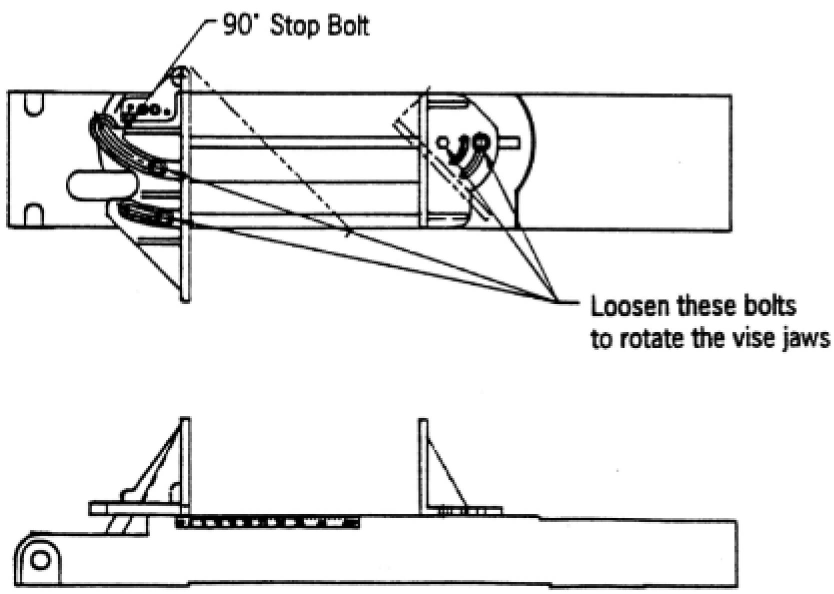
1. Turning the handle to the left will loosen the vice. Turning it to the right will tighten it.

2. The vice will be movable by hand if it is not clamped down. Some force may be needed to move the vice, and if it is sticking, slightly loosening the handle should resolve the problem.
3. The workpiece should be secured in a manner in which it will not pop out during the cutting process.
4. Make sure that the workpiece is being cut by multiple blade teeth, not just one.

Rotating Vice

1. If the desired cut is not a 90 degree angle, the vice's angle can be adjusted by up to 45 degrees.

2. To change the angle, lift the cutting head and adjust the bolts as shown on the image below.



3. Before cutting, tighten the bolts and restore the jaws to their original position.
4. When the vice is rotated by a full 45 degrees, the maximum size for the stock becomes 8" round and 8" square.

Procedure

1. Lift the handle and lock the machine in place.

2. Mount the stock inside the vice and tighten it.
3. Do not cut thin, vertical pieces, as they can damage the blade.
4. Do not cut large, flat pieces on the horizontal bandsaw. Cut these pieces with the vertical bandsaw.
5. Adjust the blade holders so that they clear the stock.
6. Turn the coolant or cutting oil on if necessary.
7. Place the emergency stop button in the extended position.
8. Turn the machine off by pressing the green button.

9. Adjust the speed of the blade to suit your needs.

10. Slowly turn the vertical feed lever and change the speed as needed.Ask your instructor about head speed. If the speed is too low, the blade can become dull. However, if the speed is too high, it may damage the blade. The blade should be doing all of the work.

11. The machine will stop automatically after cutting the workpiece.

12. Return the head speed to the default and clean the machine.

Vertical Bandsaw

Setup

1. Make sure that the power is off and adjust the height of the guidepost to ¼ of an inch above the top face of the stock by loosening the guidepost lock. When finished, remember to tighten the lock.

2. Make sure that there is a push stick available.

Procedure

1. Turn the machine on and wait for it to reach maximum speed.

2. Cut the stock, making sure to keep fingers at least 4 inches away from the blade.
3. When cutting small objects, use a push stick.
4. Make relief cuts when cutting curves or intricate cuts. This will lower the amount of blade stain. It will also remove the need to back out of a cut.
5. Use a tail man when re-sawing or cutting long pieces.
6. When ripping bevels keep the fence on the low side of the blade.
7. If the blade makes clicking sounds, turn the machine off immediately. The sound signifies a breaking blade. Notify the instructor immediately.
8. Turn off the machine and stop the blade using the foot brake when finished cutting. Do not leave the band saw without stopping the blade completely.

Post-Job Procedure

1. Clean up any sawdust and pieces using a broom and dustpan.

2. Clean off the saw and check for damage. If any is seen, report to the instructor immediately.

Chop Saw Procedure

1. Material should be laid on the table and flush against the fence.

2. Angle the table to the desired orientation and secure the pivot.

3. Do not turn the saw on in this step. Line up the blade to the desired cutting length.

4. Bring the saw up completely. Do not suddenly let go of the saw.

5. Securely hold the stock to the table and fence. Ensure that your hand is at least 6 inches away from the blade during the cut.

6. While the saw is at maximum height, hold the handle firmly and press and hold the switch.

7. Once the blade has reached full speed, slowly lower the blade through the stock.

8. After the cut is complete, turn off the saw by releasing the switch.

9. Slowly raise the saw back to its original position after the blade has stopped completely.

There are different types of chop saws, with each differing slightly from the others.

Compound chop saws tilt and pivot on the vertical axis. They can cut angles on both the side and the top of the workpiece.

Sliding compound chop saws also tilt and pivot on the vertical axis, but can also slide on linear rails. This allows the saw to make longer cuts.

Saw Blades Selection

Choosing the right blade for the material to be cut plays an important role in cost effective

band sawing. Here are some guidelines to help you make the right decision.

Blade Terminology

A clear understanding of blade terminology can help avoid confusion when discussing

cutting problems.

1. Blade Back: The body of the blade not including tooth portion.

2. Thickness: The dimension from side to side on the blade.

3. Width: The nominal dimension of a saw blade as measured from the tip of the tooth to the back of the band.

4. Set: The bending of teeth to right or left to allow clearance of the back of the blade through the cut.

Kerf: Amount of material removed by the cut of the blade.

5. Tooth Pitch: The distance from the tip of one tooth to the tip of the next tooth.

6. TPI: The number of teeth per inch as measured from gullet to gullet.

7. Gullet: The curved area at the base of the tooth. The tooth tip to the bottom of the gullet is the gullet depth.

8. Tooth Face: The surface of the tooth on which the chip is formed.

9. Tooth Rake Angle: The angle of the tooth face measured with respect to a line perpendicular to the cutting direction of the saw.

Tooth Form

The shape of the tooth's cutting edge affects how efficiently the blade can cut through a piece of material while considering such factors as blade life, noise level, smoothness of cut and chip carrying capacity.

Variable Positive: Variable tooth spacing and gullet capacity of this design reduces noise and vibration, while allowing faster cutting rates, long blade life and smooth cuts.

Variable: A design with benefits similar to the variable positive form for use at slower cutting rates.

Standard: A good general purpose blade design for a wide range of applications.

Skip: The wide gullet design makes this blade suited for non-metallic applications such as wood, cork, plastics and composition materials.

Hook: Similar in design to the Skip form, this high raker blade can be used for materials which produce a discontinuous chip (such as cast iron), as well as for non-metallic materials.

Tooth Set

The number of teeth and the angle at which they are offset is referred to as "tooth set." Tooth set affects cutting efficiency and chip carrying ability.

Raker: 3 tooth sequence with a uniform set angle (Left, Right, Straight). Modified Raker: 5 or 7 tooth sequence with a uniform set angle for greater cutting efficiency and smoother surface finish (Left, Right, Left, Right, Straight). The order of set teeth can vary by product.

Vari-Raker: The tooth sequence is dependent on the tooth pitch and product family. Typically Vari-Raker set provides quiet, efficient cutting and a smooth finish with less burr.

Alternate: Every tooth is set in an alternating sequence. Used for quick removal of material when finish is not critical.

Wavy: Groups of teeth set to each side within the overall set pattern. The teeth have varying amounts of set in a controlled pattern. Wavy set is typically used with fine pitch products to reduce noise, vibration and burr when cutting thin, interrupted applications.

Vari-Set: The tooth height / set pattern varies with product family and pitch. The teeth have varying set magnitudes and set angles, providing for quieter operation with reduced vibration. Vari-Set is efficient for difficult-to-cut materials and larger cross sections.

Single Level Set: The blade geometry has a single tooth height dimension. Setting this geometry requires bending each tooth at the same position with the same amount of bend on each tooth.

Dual Level Set: This blade geometry has variable tooth height dimensions. Setting this product requires bending each tooth to variable heights and set magnitudes in order to achieve multiple cutting planes.

Vise Loading

The position in which material is placed in the vise can have a significant impact on the cost per cut. Often, loading smaller bundles can mean greater sawing efficiency.

All machines have a stated loading capacity, but the practical level is usually lower, 1/2 to 1/3 as much, depending on the material being cut (harder materials are best cut at 1/3 rated capacity).

When it comes to cutting odd-shaped material, such as angles, I-beams, channel, and tubing, the main point is to arrange the materials in such a way that the blade cuts through as uniform a width as possible throughout the entire distance of cut.

The following diagrams suggest some cost-effective ways of loading and fixturing. Be sure, regardless of the arrangement selected, that the work can be firmly secured to avoid damage to the machine or injury to the operator.

Lubrication

Lubrication is essential for long blade life and economical cutting. Properly applied to the shear zone, lubricant substantially reduces heat and produces good chip flow up the face of the tooth. Without lubrication, excessive friction can produce heat high enough to weld the chip to the tooth. This slows down the cutting action, requires more energy to shear the material and can cause tooth chipping or stripping which can destroy the blade.

Follow the lubrication manufacturer's instructions regarding mixing and dispensing of lubricant. Keep a properly mixed supply of replenishing fluid on hand. Never add water only to the machine sump. A fluid mixture with too high a water-to-fluid ratio will not lubricate properly and may cause rapid tooth wear and blade failure. Use a refractometer, and inspect the fluid visually to be sure it is clean. Also, make sure the lubrication delivery system is properly aimed, so that the lubricant flows at exactly the right point.

How to select your band saw blades:

The following information needs to be specified when a band saw blade is ordered:

For Example:	Product Name	Length x Width x Thickness	Teeth Per Inch
	CONTESTOR GT	16' x 1-1/4" x .042"	¾TPI

These steps are a guide to selecting the appropriate product for each application:

Step 1: Analyze the sawing application Machine:
For most situations, knowing the blade dimensions (length x width x thickness) is all that is necessary.

Material: Find out the following characteristics of the material to be cut.

• Grade • Hardness (if heat treated or hardened)

• Shape • Size

• Is the material to be stacked (bundled) or cut one at a time?

Other Customer Needs: The specifics of the application should be considered.

• Production or utility/general purpose sawing operation?

• What is more important, fast cutting or tool life?

• Is material finish important?

Step 2: Determine which product to use Use the charts below.

• Find the material to be cut.

• Read the chart to find which blade is recommended.

Step 3: Determine the proper number of teeth per inch (TPI)

Use the tooth selection chart (See chart below).

• If having difficulty choosing between two pitches, the finer of the two will generally give

better performance.

• When compromise is necessary, choose the correct TPI first.

Step 4: Order Sawing Fluids and Lubricants for better performance and longer

life on any blade.

Step 5: Install the blade and fluid

Step 6: Break in the blade properly

Step 7: Run the blade at the correct speed and feed rate.
Refer to the chart below.

BAND SPEED AND RECOMMEND CUTTING RATES FOR BI-METAL SAWING APPLICATIONS

Material Size	Up to 1"		From 1" to 3"		From 3" to 6"		Over 6"	
Suggested tooth pitch	10/14, 8/12		8/12, 6/10, 5/8		5/8, 4/6, 3/4		3/4, 2/3, 1.5/1.9, 1.1/1.4	
	Blade Speed	Cutting Rate	Blade Speed	Cutting Rate	Blade Speed	Cutting Rate	Blade Speed	Cutting Rate
	(SFPM)	(SIPM)	(SFPM)	(SIPM)	(SFPM)	(SIPM)	(SFPM)	(SIPM)
Carbon Steel:								
1008-1013	250	8-10	275	9-12	280	12-15	250	9-12
1015-1018	250	8-10	275	9-12	250	12-15	230	12-15
1048-1065	200	5-7	200	5-7	175	5-10	150	5-10
1065-1095	200	4-6	200	5-7	150	6-8	120	6-8
Free Machining Steels								
1108-1111	300	9-11	330	12-14	275	13-15	220	11-14
1112-1113	300	8-11	330	11-13	275	12-15	220	12-15
1115-1132	300	7-10	330	10-13	275	13-16	220	11-14
1137-1151	275	6-8	250	8-10	250	8-11	200	7-10
1212-1213	300	8-10	320	11-13	300	13-15	255	11-14
Manganese Steels:								
1320-1330	250	5-7	250	5-8	200	8-11	175	7-10
1335-1345	250	5-7	225	5-7	200	7-9	175	5-8
Nickel Steels:								
2317-	270	4-5	270	4-6	250	5-7	230	4-6
2330-2345	220	2-3	220	3-5	190	3-5	170	3-5
2512-2517	200	2-3	200	3-5	160	4-6	150	4-6
Nickel Chrome Steels:								
3115-3130	260	4-6	260	5-7	230	5-7	225	5-7
3135-3150	220	4-6	200	4-7	180	5-7	160	4-6
Molybdenum Steels:								
4017-4024	300	3-5	270	4-7	250	6-8	220	5-8
4032-4042	300	3-5	270	4-7	250	6-8	230	5-8
4047-4068	250	3-5	220	4-6	200	5-7	180	3-5
Chrome Moly Steels:								
4130-4140	280	4-6	250	5-8	250	8-10	220	6-8
4142-4150	230	3-5	200	4-6	200	5-7	170	4-6
Nickel Chrome Moly steels:								

4317-4320	250	3-5	225	4-6	200	5-7	170	4-6
4337-4340	230	3-4	200	4-5	200	4-6	170	4-5
8615-8627	250	4-5	230	6-7	230	6-8	200	6-7
8630-8645	250	3-5	230	4-6	230	5-7	180	4-6
8647-8660	220	2-4	200	3-5	200	4-6	150	3-5
8715-8750	250	3-5	220	4-6	220	5-7	180	4-6
9310-9317	200	1-3	160	2-3	160	2-4	150	2-3
9437-9445	250	4-5	230	4-5	230	5-6	180	4-5
9747-9763	250	4-5	230	3-5	200	4-6	180	3-5
9840-9850	240	4-5	220	4-6	200	5-7	180	4-6
Nickel Moly Steels:								
4608-4621	250	3-5	220	5-6	220	6-7	200	5-6
4640-	220	3-5	200	4-6	200	5-7	170	4-6
4812-4820	200	3-5	180	3-5	180	4-6	160	4-5
Chrome Steels:								
5045-5046	280	4-6	250	5-7	250	8-10	200	7-8
5120-5135	280	4-6	250	6-7	240	7-8	180	5-8
5140-5160	250	3-5	230	4-6	230	5-7	200	4-6
50100-52100	180	2-4	160	3-5	150	4-6	100	3-5
Chrome Vanadium steels:								
6117-6210	225	4-5	225	5-7	200	6-8	170	5-7
6145-6152	225	3-4	200	4-5	200	5-6	150	4-5
Silicon Steels:								
9255-9260	200	2-4	180	3-5	180	3-5	150	3-5
9261-9262	200	1-3	160	2-3	160	2-4	150	2-3

High Speed Tool Steels:								
T-1, T-2	130	1-2	110	2-3	100	2-4	90	2-3
T-4, T-5	110	1-2	100	1-2	90	2-3	80	1-2
T-6, T-8	110	1-2	100	1-2	80	1-2	70	1-2
T-15	80	1	80	1	70	1	50	1
M-1	150	1-3	140	2-4	130	3-5	110	2-4
M-2, M-3	120	1-2	110	2-3	100	3-4	80	2-3
M-4, M-10	100	1-2	90	1-2	80	1-3	60	1-2
Die Steels:								
A-2	210	2-3	200	3-4	190	3-4	180	2-3
D-2, D-3	110	1-2	100	1-2	90	1-2	80	1-2
D-7	90	1	80	1	70	1	70	1
O-1, O-2	240	3-4	210	4-5	190	5-6	170	4-5
O-6	230	3-4	200	4-6	180	5-7	150	4-6

Hot Work Steels:								
H-12, H-13, H-21	150	2-4	125	3-5	125	2-4	125	2-4
H-22, H-24, H-25	150	1-3	125	1-3	125	1-3	125	1-3
Shock ResistingTool Steels:								
S-1	220	2-4	180	3-5	165	3-5	150	2-4
S-2, S-5	170	1-3	150	2-4	120	2-4	100	1-3
Special Purpose Tool Steels:								
L-6	200	2-4	180	3-5	170	3-5	150	2-4
L-7	200	2-4	180	3-5	150	3-5	100	2-4
Stainless Steels:								
201, 202, 302, 304	120	2-4	100	2-4	100	2-4	100	1-3
303, 303F	140	2-4	120	2-4	100	3-5	100	2-4
308, 309, 310, 330	90	1	70	1	60	2	60	1
314, 316, 317	90	1	80	1	70	2	60	1
321, 347	130	1-3	110	1-3	100	2-4	80	1-3
410, 420, 420F	150	1-3	130	1-3	120	2-4	100	1-3
416, 430F	200	3-5	180	4-6	170	5-7	150	4-6
430, 446	100	1-3	90	2-4	80	2-4	80	1-3
440 A.B.C	120	1-3	100	1-3	90	2-4	70	1-3
A-7	100	1-3	100	1-3	120	2-4	100	1-3
17-4PH, 17-7PH	100	2-3	90	2-4	80	3-4	80	2-3

Beryllium Copper:								
BHN-100-120	350	4-6	300	5-7	275	6-8	225	5-7
BHN-220-250	250	2-4	225	3-5	200	4-6	175	3-5
BHN-310-340	200	1-2	160	1-2	140	2-3	100	1-2
Nickel Based Alloys:								
Monel	100	1-2	100	1-2	80	1-2	60	1
R Monel	140	2-3	140	2-4	125	2-4	75	2-3
Inconel	110	1-2	100	1-3	80	1-3	80	1-2
Inconel X	90	1	80	1	70	1	60	1
HastelloyA	120	1-2	100	1-2	85	1-2	75	1-2
HastelloyB	110	0-1	100	1-2	90	1-2	75	0-1

HastelloyC	100	0-1	90	0-1	70	0-1	60	0-1
Rene 41	90	1	90	1	90	1-2	90	1-2
Udimit	100	1	90	1-2	90	0-1	90	1-2
Waspalloy	90	1	90	1-2	90	1-2	90	1-2
Titanium	100	1-2	100	2-3	100	2-3	100	2-3
Titanium Alloys:								
TI-4AL-4MO alpha beta alloy	100	0-1	90	0-1	80	0-1	70	0-1
TI-14OA 2CR-2MO	100	0-1	90	0-1	80	0-1	60	0-1
TI-150A	100	0-1	90	0-1	80	0-1	60	0-1
MST-6AL-4V	100	0-1	90	0-1	80	0-1	60	0-1
99% pure titanium	100	0-1	90	0-1	80	0-1	60	0-1

UNIT TEST:

1. Name two types of band saws.

2. Lists five important step when using a band saw.

3. Please explain Kerf.

4. What is a Tooth Pitch?

5. Please define TPI.

6. Please explain Variable Positive.

7. What is a tooth set?

8. Please lists and describe five tooth sets.

9. Lists three reason why we use Lubrication.

10. When ordered a bandsaw blade, what information needs to be specified?Chapter Attribution Information

This chapter was derived from the following sources.

- **Horizontal Bandsaw Procedure** derived from Horizontal Band Saw Operation by the University of Idaho, CC:BY-SA 3.0
- **Vertical Bandsaw Procedure** derived from [blank] by the [blank], CC:BY-SA 3.0
- **Chop Saw Procedure** derived from Woodshop Red Safety and Basic Usage by novaLABS, CC:BY-SA 3.0.

PART V

Chapter 5: Surface Grinders

Chapter 5: Surface Grinder

OBJECTIVE

After completing this unit, you should be able to:

- Identify Surface Grinder.
- Identify Procedures.
- Describe Dressing the Wheel procedures.
- Describe the Ring Test.
- Describe replacing the Grinding Wheel.
- Describe procedure select the grinding wheel.
- List principal abrasives with their general areas of best use.
- List principal bond with the types of application where they are most used.
- Identify by type number and name , from unmarked sketches, or from actual wheels.
- Interpret wheel shape and size markings together with five basic symbols of a wheel specification into description of the grinding wheel.
- Given several standard , common grinding jobs, recommend the appropriate abrasive, approximate grit size, grade, and bond.

The Surface Grinder is mainly used in the finishing process. It is a very precise tool which uses a stationary, abrasive, rotating wheel to shave or finish a metallic surface which is held in place by a vise. This vise, which is part of a table, or carriage is moved back and forth under the abrasive wheel. The surface grinder can cut steel in pieces no bigger than 18" long by 6" high by 8" wide. The table of the grinder is also magnetic, which aids in holding the material still. These magnets can be toggled by means of a lever located on the front side of the grinder. This instrument has a maximum cut of .005 of an inch, and a minimum cut of .005 of an inch. The movement of the grinder can be an automatic, back and forth motion, or manually moved as required.

Safety Precautions

Besides regular machine shop safety rules, these are some tips on how to use this machine safely:

- Always wear safety glasses as this machine may send shavings in all directions.
- Always wait for the wheel to reach maximum speed before using it, as there may be
- If you have long hair, you should keep it tied back, so that it does not get caught in the machine.
- Never strike the wheel against the material as this could cause faults in the wheel, which may result in a loss of integrity and it may fly apart.
- Always make sure that the guard is in place over the grinding wheel, as this protects the user from the shavings that are removed from the material.
- Always make sure the material is securely fastened in place.
- Always make sure the magnetic table is clean before placing material on it, as shavings may scratch your material or even cause the material to slide wheel you are using the grinder.
- Ensure that the grinder has a start/stop button within easy reach of the operator.
- Check the grinding wheel before mounting it. Make sure it is properly maintained and in good working order.
- Follow the manufacturer's instructions for mounting grinding wheels.
- Keep face of the wheel evenly dressed.
- Ensure that the wheel guard covers at least one half of the grinding wheel.
- File off any burrs on the surface of work that is placed on the magnetic chuck.

- Clean the magnetic chuck with a cloth and then wipe with the palm of your hand.
- Place a piece of paper slightly larger than workpiece in the center of chuck.
- Position work on the paper and turn on the power to the magnetic chuck.
- Check that the magnetic chuck has been turned on by trying to remove work from the chuck.
- Check that the wheel clears the work before starting the grinder.
- Run a new grinding wheel for about one minute before engaging the wheel into the work.
- Wait for the wheel to reach maximum speed before using it as there may be unseen faults in the wheel.
- Stand to one side of the wheel before starting the grinder.
- Turn off coolant before stopping the wheel to avoid creating an out-of-balance condition.
- Keep the working surface clear of scraps, tools and materials.
- Keep the floor around the grinder clean and free of oil and grease.
- Use an appropriate ventilation exhaust system to reduce inhalation of dusts, debris, and coolant mists. Exhaust systems must be designed and maintained appropriately.
- Follow lockout procedures when performing maintenance work.

Procedure for Use

- The first step in using the surface grinder, is to make sure that the material you wish to shape can be used in the grinder. Soft materials such as aluminum or brass will clop up the abrasive wheel and stop it from performing effectively, and it will then have to be cleaned. This process is explained in the Maintenance section. The maximum size of a material that the grinder can machine is 18" long by 8" wide by 6" high.
- The next step is to make sure the material is secured. This is done by use of a vice, and then by engaging the magnetic clamp. Once the material is secure, it must be manually positioned under the abrasive wheel. This is done by turning the longitude and latitude wheels located on the front of the grinder. The abrasive wheel itself can be moved slightly to get the material in the perfect position.
- Then the machine may be started. It should reach maximum speed before you try to use it for the safety reasons. If the wheel is working properly, manually used when very precise work needs to be done.

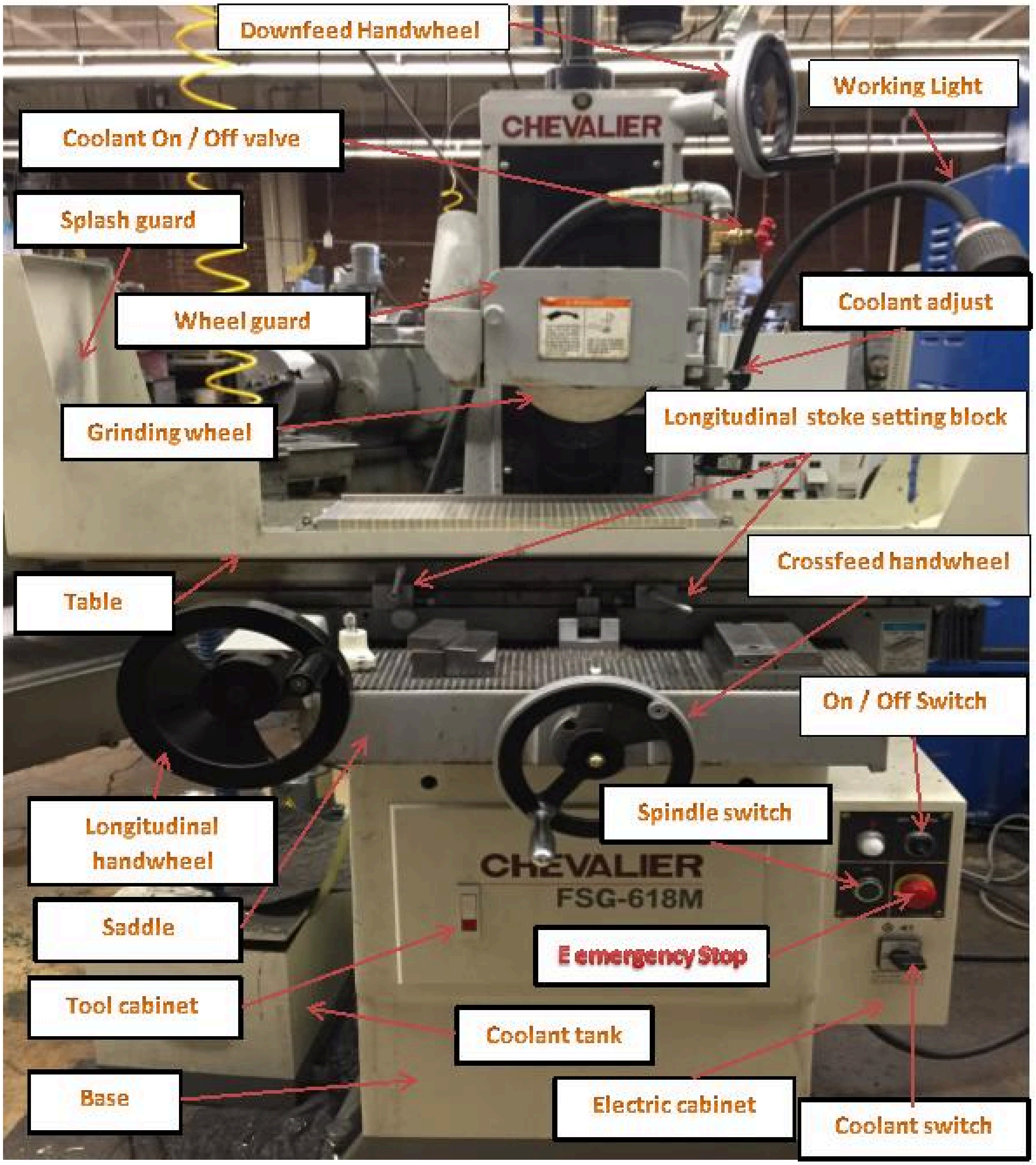


Figure 1. Chevalier Surface Grinder

Dressing the Wheel

1. Place the diamond wheel dresser onto the bed.
2. Keep the diamond dresser ¼ of an inch to the left of the center of the wheel.
3. Lock the dresser onto the bed by turning the magnetic chuck on.
4. Turn on the machine power by turning the switch to the "ON" position. Then press the green button to start the spindle.
5. Move the grinding wheel down using the vertical table handwheel until it barely makes contact with the dresser.
6. Turn the machine off after making contact with the dresser.

7. Turn the machine on again. While the wheel is spinning, lower the grinding wheel down in the Z direction until it makes a small plume of dust.
8. Once the small plume of dust has been made, make one pass back and forward along the Y-axis. Stop the machine when the dresser has made on pass back and forward.
9. When stopping the machine, make sure that the dresser is about ½ inches away from the wheel.
10. Check the wheel to see if it is clean. If not, repeat steps 8 and 9.

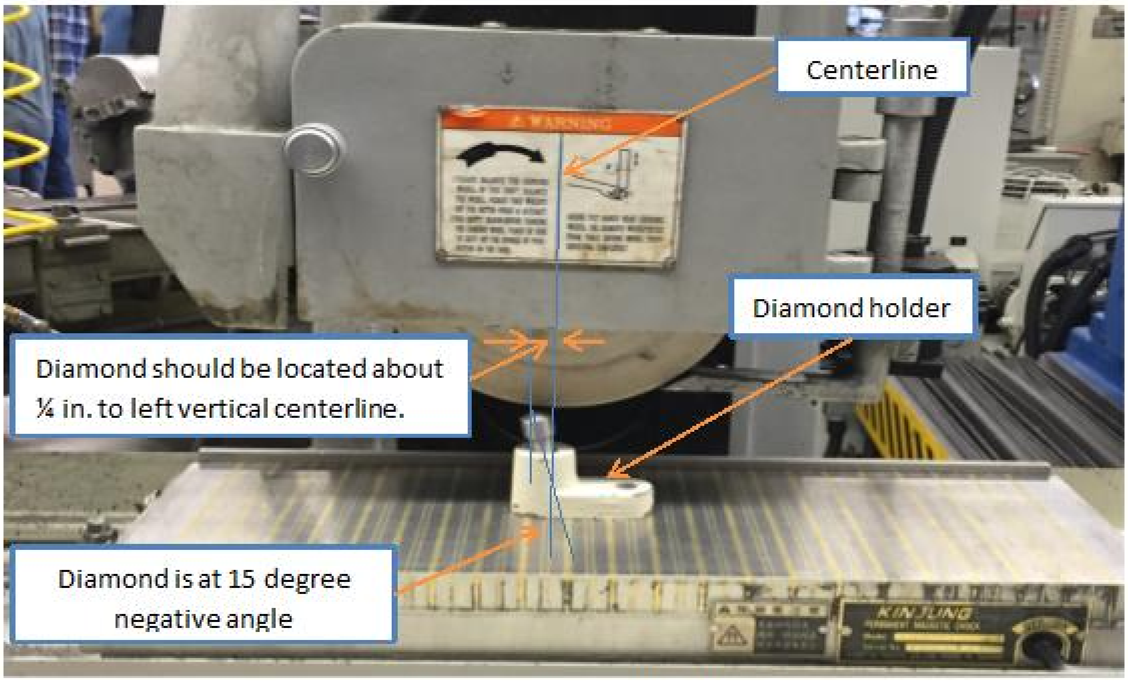


Figure 2. Dressing the wheel

Ring Test

Grinding wheels must be inspected and "ring-tested" before they are mounted to ensure that they are free from cracks or other defects. Wheels should be tapped gently with a light, nonmetallic instrument. A stable and undamaged wheel will give a clear metallic tone or "ring."

Performing the ring test:

Make sure the wheel is dry and free of sawdust or other material that could deaden the sound of the ring.

You will need a hard plastic or hard wood object, such as the handle of a screwdriver or other tool, to conduct the test. Use a wood mallet for heavier tools. Do not use metal objects.

1. Suspend the wheel on a pin or a shaft that fits through the hole so that it will be easy to turn, but do not mouth the wheel on the grinder. If the wheel is too large to suspend, stand it on a clean, hard surface.
2. Imagine a vertical plumb line up the center of the wheel.
3. Tap the wheel about 45 degrees on each side of the vertical line, about one or two inches from the wheel's edge. (Large wheels may tapped on the edge rather than the side of the wheel.)
4. Turn the wheel 180 degrees so that the bottom of the wheel is now on top.
5. Tap the wheel about 45 degrees on each side of the vertical line again.
6. The wheel passes the test if it gives a clear metallic tone when tapped at all four points. If the wheel sounds dead at any of the four points, it is cracked. Do not use it.

Replacing the Grinding Wheel

1. Open the wheel case. If the wheel case is very tight, this may require a pair of brace wrench, wrench and a rubber mallet.
2. Remove the metal plate on top by loosening the screws that are holding it to the wheel case.

Figure 3. Remove metal plate and wheel case

3. Behind the wheel, on the spindle, there is a hole. Insert the brace wrench on the right side into the back of the spindle. The brace wrench should be able to fit into the hole.

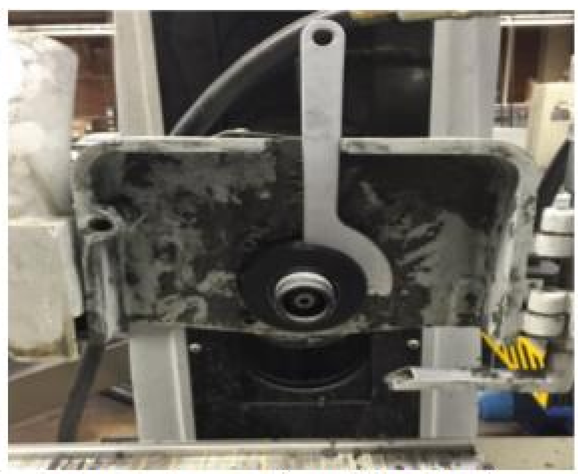


Figure 4. Brace wrench into hole Figure 5. Remove the grinding wheel

4. Insert the wrench into the two holes in the front of the wheel. When loosening the wheel from the wheel spindle, turning right will loosen and turning left will tighten.

5. Hit the wishbone-shaped wrench with a rubber mallet to loosen the wheel.

6. To put a new grinding wheel on, reverse the procedure. Turning the wishbone-shaped wrench to the left will tighten it. When installing the wheel, make sure that the wrench is on the left side, not on the right side. Turn the wishbone-shaped wrench by hand, and when no longer possible, use the rubber mallet.

7. Remove the wrench from the back of the spindle.

8. Screw the plate back on top of the wheel case.

9. Close the wheel case, and tighten the knob.

Grinding Procedure

1. Ensure the proper wheel for the stock is being used. There are different grinding wheels for aluminum, stainless steel, and titanium.
2. Clean the bed before placing the workpiece onto it. This will prevent interference with the magnetic chuck.
3. Place magnetic parallels around the workpiece to ensure the workpiece does not shift during grinding.
4. Turn the magnetic chuck on to secure the pieces onto the bed.
5. Adjust the bed and saddle position to center the stock below the wheel.
6. Lower the wheel an inch above the workpiece.
7. Take a piece of paper and place it between the wheel and the stock. Move the paper back and forth while simultaneously lowering the wheel until the paper is no longer able to move to zero the z-axis. See figure 1.
8. Zero the z-axis of the workpiece by setting the dial on downfeed handwheel to 0 inches. See figure 2.
9. Lock the table Longitudinal stoke setting block so that there is about an inch of overtravel at each end of the table stroke.
10. Adjust the table position so the wheel sits about an inch to the right of the workpiece.
11. Lower the wheel to the desired depth of grinding. There should be a maximum downfeed of 0.001 inch per pass.
12. Ensure the wheel is not in contact with the workpiece before turning the main power on. Press the green button to turn the spindle on and turn the coolant switch on.
13. Grind the stock by making passes left to right along the x-axis.
14. Once the first strip of the workpiece has been sufficiently ground, turn the y-axis handwheel half a turn clockwise.
15. Grind another strip of the workpiece from left to right along the x-axis.
16. Repeat until the workpiece is fully ground, then repeat all of the previous steps for the other side.


Figure 6. Setting the z axis Figure 7. Setting downfeed

Grinding Wheel

Select the grinding wheel:

Keep in mind that a grinding wheel is a form of cutting tool, and except in the case of wheel for general purpose grinding, the abrasive, grit size, grade and structure, bond type should be selected to fit the particular job on which the wheel is to be used, just as a cutter, drill or tap is selected for its specific job.

To select the grinding wheel, there are eight factors which affect the choice of the grinding wheel specifications. There are:

1. Grinding wheel manufactures instruction.

2. Material to be ground and its hardness.

3. Amount of stock to be removed and finish required.

4. Are of grinding contact.

5. Severity of the grinding operation.

6. Wheel speed.

7. Feed rate

8. Operating technique.

Suggestions:

1. First consider the material to be ground and its hardness. These effect the choice of abrasive, grift size, and grade or hardness of the wheel.

- Aluminum oxide are best for steels, while Silicon carbide abrasives are better suited to grinding cast iron, nonferrous metals and nonmetallic materials.
- A relatively fine grit size works best on taking heavier cuts can be used advantageously on soft and ductile materials that are readily penetrated.
- The hardness of the material to be ground also affects choice of the wheel grade or hardness. A harder grade

can be used on soft, easily penetrated materials than on hard materials which naturally tend to dull the wheel faster. The softer grades release the dull grains more readily to present new, sharp grains to the work.

2. Second factor, in selecting a wheel in the amount of stock to be removed and the finish required. These affect the choice of grift size and bond as follows:

- A relatively coarse grit size is selected for rapid stock removal without regard for finish as rough grinding; a fine grift should be used where a high finish is desired.
- Vitrified bonded wheels are generally used where a commercial finish satisfactory. The organic bonds, resinoid, rubber and shellac, produce the highest finish.

3. The area of grinding contact between the wheel and the work affects the choice of grift size and grade.

- A coarse grift is required when the contact area is relatively large, as in surface grinding with cup wheels, cylinders or segments, to provide adequate chip clearance between the abrasive grains. As area of contact becomes smaller and the unite pressure tending to break down the wheel face becomes greater, finer grit wheels should be used.
- As to the grade or hardness, on large area of contact a soft grade will provide normal breakdown of the wheel, insuring continuous, free-cutting action. A harder grade, on the other hand, is needed to stand up under the increasingly higher unit pressure as the area of contact becomes smaller.

4. The severity of the grinding operation affects the choice of abrasive and grade.

- A tough abrasive like 4A Aluminum Oxide should be used for rough, heavy duty grinding of steel.
- The milder abrasives like 32 and 38 Aluminum Oxide are best for lighter precision grinding operations on steels and semisteels, while the intermediate 57 and 19 Aluminum Oxide abrasives are used for precision and semiprecision grinding of both mild and hard steels.
- The severity of the grinding operation also influences the choice of grade. Hard grade provide durable wheels for rough grinding such as snagging, while medium and softer grade wheels can be used for precision type operations which are less severe on the wheel.

5. The speed at which the grinding wheel is to be operated often dictates the type of bond.

- Vitrified bonded wheels should not be used at speeds over 6,500 s.f.p.m. With few exceptions, when the speed exceeds this figure, resinoid, rubber or shellac bonded wheels should be used. Note, the safe operating speed shown on the tag, wheel or blotter must never be exceeded.

6. Feed rate

- The higher the feed rate, the greater the grinding pressure is. If the grinding speed of workpiece must be increased, the feed rate will be increased, then the wear of the wheel will be faster. Therefore a harder grinding wheel is required.

- A standard wheel marking system is used for the identifying five major factors in grinding wheel selection:
  - Type of abrasive
  - Grit size
  - Grade or hardness
  - Structure
  - Bond

First Symbol: Type of Abrasive

A wheel marked A 60-J8V indicates the following:

A – Fused aluminum oxide

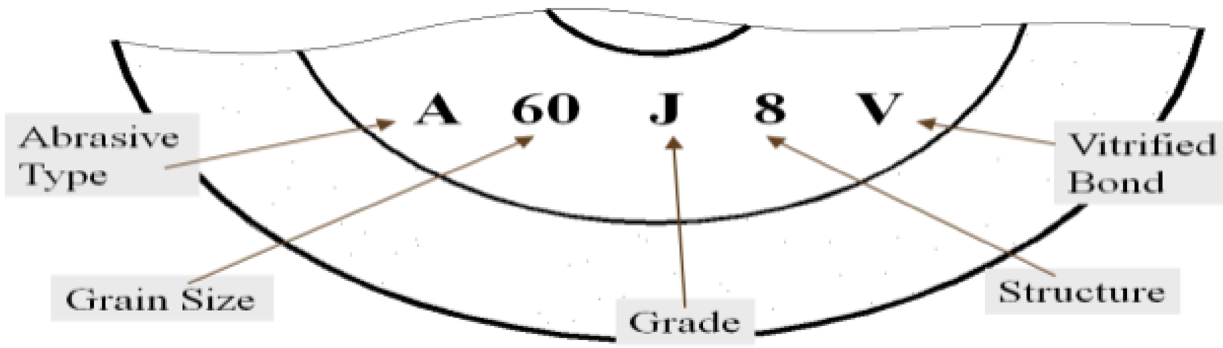


Figure 8: Grinding Wheel Marking

Second Symbol: Grit Size

The following scale can be used to determine grit:

4 36 46 60 100 120 240 500
Coarse *Medium* *Fine*

Third Symbol: Grade of Hardness

- Hardness grade is a measure of bond strength of the grinding wheel.
  - Bond material holds abrasive grains together in the wheel.
  - The stronger the bond, the harder the wheel.
- Hardness grade is a measure of bond strength of the grinding wheel.

A to G are softer.
H to P are more medium grades.
R to Z are harder.

Fourth Symbol: Structure

- Structure, the spacing of the abrasive grains in the wheel is indicated by numbers.

1 is a dense structure.
8 is a more medium structure.
15 is an open structure.

Fifth Symbol: Bond

- Bond is identified by letter according to the following:
  - V – Vitrified
  - B – Resinoid

- R – Rubber
- E – Shellac
- M – Metal

Standard grinding wheel marking example:

1- A – 305 X 25 X 127 WA 46 K 8 V 7N 2000m/min

FROM(WHEEL TYPE): 1(Straight-plain)

FACE: A

SIZE: Dia. (D) X Width(W) X Bore(H)

ABRASIVE TYPE: WA (See Figure 2)

GRAIN SIZE: 46 (See Figure 2)

GRADE: K (See Figure 2)

STRUCTURE: 8 (See Figure 2)

BONE TYPE: V (See Figure 2)

MAKER CODE: 7N

MAX. RPM: 2000m/min.

WA		60	K	7	V
ABRASIVES		GRIT SIZE	GRADE	STRUCTURE	BOND TYPE
A	Regular Aluminium Oxide	10 Coarse	A Soft	1 Dense	V : Vitrified
WA	White Aluminium Oxide	12	B	2	B : Resinoid
19A	Mixture of A&WA	14	C	3	R : Rubber
FA	Semi-friable Aluminium Oxide	16	D	4	O : MgO
PA,RA	Pink Aluminium Oxide	20	E	5	E : Epoxy
SA(HA)	Single Crystal Aluminium Oxide	24	F	6	
23A	Mixtue of A$SA	30	G	7 To	
AZ	Zirconium Oxide	36	H	8	
C	Black Silicon Carbide	46	I	9	
GC	Green Silicon Carbide	54	J	10	
RC	Mixture of C&GC	60 To	K To	11	
		80	L	12	
		100	M	13	
		120	N	14 Open	
		150	O		
		180	P		
		220	Q		
		280	R		
		320	S		
		400	T		
		500	U		
		600	V		
		800	W		
		1000	X		
		1200 Fine	Y		
			Z Hard		

Figure 9: Grinding Wheel Selection Chart

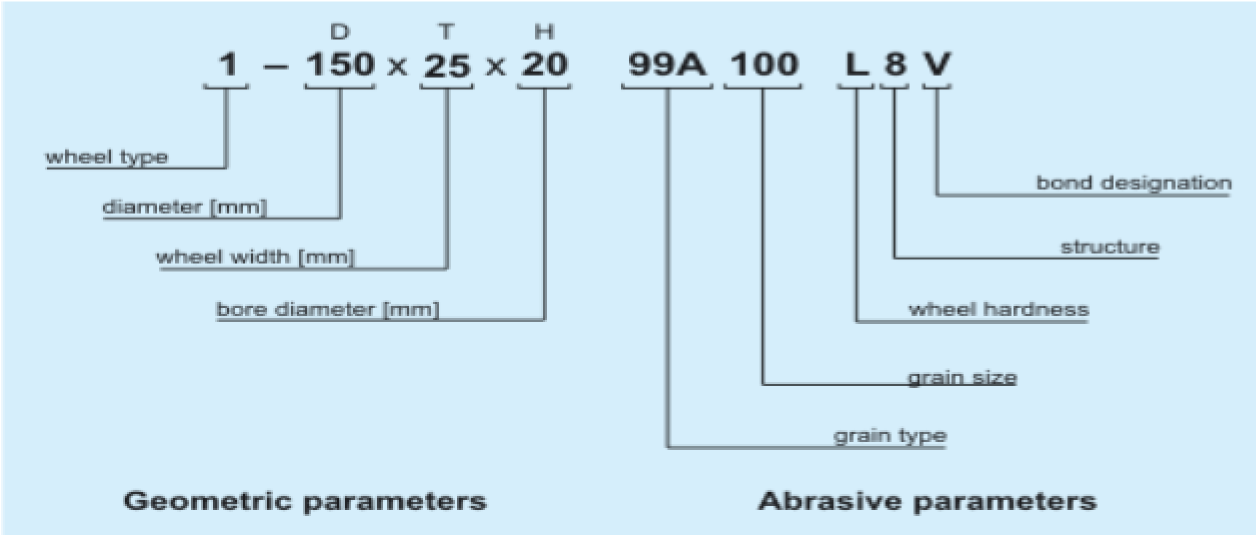


Figure 10: Grinding Wheel Selection Chart

UNITS TEST

1. Please list five Safety Precautions.

2. Please list five main parts of the surface grinders.

3. What is a diamond wheel dresser?

4. When Dressing the Wheel how far Diamond dresser should be located to the left of the center of the wheel?

5. What is a Ring Test?

6. How do you Performing the ring test?

7. When select the grinding wheel, there are eight factors which affect the choice of the grinding wheel specifications. Please list five out of eight factors.

8. Aluminum oxide grinding wheel are best for what?

9. A standard wheel marking system is used for the identifying factors in grinding wheel selection. Please all five major factors?

10. A wheel marked WA 80-L9B, Please indicates the following.

Chapter Attribution Information

This chapter was derived from the following sources.

Grinding and Buffing derived from Mechanical Engineering Tools by the Massachusetts Institute of Technology, CC:BY-NC-SA 4.0.

PART VI

Chapter 6: Heat Treating

15

Chapter 6: Heat Treating

OBJECTIVE

After completing this unit, you should be able to:

- Correctly harden a piece of tool steel and evaluate your work.
- Correctly temper the hardened piece of the tool steel and evaluate your work.
- Describe the proper heat treating procedures for other tool steels.

Safety

The following procedures are suggested for a safe heat treating operation.

1. Wear heat-resistant protective clothing, gloves, safety glasses, and a face shield to prevent exposure to hot oils, which can burn skin.
2. Before lighting the furnace, make sure that air switches, exhaust fans, automatic shut-off valves, and other safety precautions are in place.
3. Make sure that there is enough coolant for the job. Coolant will absorb heat given off by the metal as it is cooling, but if there is insufficient coolant, the metal will not cool at the optimal speed.
4. Make sure that there is sufficient ventilation in the quenching areas in order to maintain desired oil mist levels.
5. When lighting the furnace, obey the instructions that have been provided by the manufacturer.
6. During the process of lighting an oil or gas-fired furnace, do NOT stand directly in front of it.
7. Make sure that the quenching oil is not contaminated by water. Explosions can be results of moisture coming into contact with the quenching oil.
8. Before taking materials out of the liquid carburizing pot, make sure that the tongs are not wet and that they are the correct tongs for the job.
9. Make sure that an appropriate fungicide or bacterial inhibitor has been mixed into the quenching liquid.
10. When quench tanks are not being used, always cover them.
11. Use a nonflammable absorbent to clean leaks and oil spills. This should be done immediately.
12. If possible, keep tools, baskets, jigs, and work areas free from oil contamination.
13. Before breaks and before moving on to the next task, wash your hands thoroughly.
14. If any skin trouble is shown or suspected, report to your instructor and get medical help.
15. Fumes from the molten carburizing salt bath should not be inhaled, because carbon monoxide is a product of the carburizing process.
16. Make sure there is good ventilation in the work area.
17. Be on the lookout for contamination from pieces of carburized metal.
18. Do not take oil-soaked clothes or equipment to areas where there are food or beverages.
19. Do not take food or beverages where oils are either being used or stored.

Procedure

The first important thing to know when heat treating a steel is its hardening temperature. Many steels, especially the common tool steels, have a well established temperature range for hardening. O-1 happens to have a hardening temperature of 1450 – 1500 degrees Fahrenheit.

To begin the process:

1. Safety first. Heat treating temperatures are very hot. Dress properly for the job and keep the area around the furnace clean so that there is no risk of slipping or stumbling. Also, preheat the tongs before grasping the heated sample part.
2. Preheat the furnace to 1200 degrees Fahrenheit.
3. When the furnace has reached 1200 degrees Fahrenheit, place the sample part into the furnace. Place the sample part into the center of the oven to help ensure even heating. Close and wait.
4. Once the sample part is placed in the furnace, heat it to 1500 degrees Fahrenheit. Upon reaching this temperature, immediately begin timing the soak for 15 minutes to an hour (soak times will very depending on steel thickness).

Table 1: Approximate Soaking Time for Hardening, Annealing and Normalizing Steel

Thickness Of Metal (inches)	Time of heating to required Temperature (hr)	Soaking time (hr)
up to 1/8	.06 to .12	.12 to .25
1/8 to 1/4	.12 to .25	.12 to .25
1/4 to 1/2	.25 to .50	.25 to .50
1/2 to 3/4	.50 to .75	.25 to .50
3/4 to 1	.75 to 1.25	.50 to .75
1 to 2	1.25 to 1.75	.50 to .75
2 to 3	1.75 to 2.25	.75to 1.0
3 to 4	2.25to 2.75	1 to 1.25
4 to 5	2.75 to 3.50	1 to 1.25
5 to 8	3.50 to 3.75	1 to 1.50

Soak time is the amount of time the steel is held at the desired temperature, which is in this case 1500 degrees Fahrenheit.

5. When the soak time is complete, very quickly but carefully take the sample out with tongs. Place the sample part into a tank of oil for quenching. Move the sample part around as much as possible while it is quenching.
6. Once the sample part has been quenched down to around 125 degrees Fahrenheit, begin the tempering process. To temper the sample part it must be placed into the furnace at 375 degrees Fahrenheit. Allow it to soak for 2 hours, then remove the sample part and allow it to cool to room temperature. The sample part should now be approximately at a hardness of 60 RC.

Austenitize and Air-Cool:

1. This heat treatment is usually done by the manufacturer, which results in it being called the as-received condition
2. To reach this state, a process called normalizing (also called the thermal history) is done. Normalizing 1045 steel usually consists of these steps:
    2.1 Austenitize: Place the steel in the furnace at 1562°F in the austenite range, and keep it there for an

hour until the metal has reached its equilibrium temperature and corresponding solid solution structure.
2.2 Air-cool: Take the steel out of the furnace and let it air-cool to room temperature.

Austenitize and Furnace-Cool (Annealing):

1. This process is also referred to as annealing. During annealing, the steel goes through the following temperature histories:
   1.1 Austenitize: Place the steel in the furnace at 1562°F in the austenite range, and keep it there for an hour until the metal has reached its equilibrium temperature and corresponding solid solution structure.
   1.2 Furnace-Cool: cool the steel slowly in the furnace. Allow the temperature to drop from 1562°F to 1292°F over a ten hour period.
   1.3 Air-cool: Take the steel out of the furnace and let it air-cool to room temperature.

Austenitize and Quench:

1. Austenitize: Place the steel in the furnace at 1562°F in the austenite range, and keep it there for an hour until the metal has reached its equilibrium temperature and corresponding solid solution structure.
2. Quench: quickly remove the steel from the furnace, plunge it into a large container of water at room temperature, and stir vigorously. When using 1045 steel, room temperature water is used as the quenching medium.

Quench: Rapidly remove material from furnace, plunge it into a large reservoir of water at ambient temperature, and **stir vigorously.**

For 1045 steel, the quenching medium is water at room temperature (for other steels, other quenching media such as oil or brine are used).

4. Austenitize, Quench, and Temper:

1. Austenitize: Place the steel in the furnace at 1562°F in the austenite range, and keep it there for an hour until the metal has reached its equilibrium temperature.
2. Quench: Quickly remove the steel from the furnace, plunge it into a large container of water at room temperature, and stir vigorously.
3. Temper:
   3.1 Bring the steel to the tempering temperature and hold it there for about 2 hours.
   3.2 There is a range of different tempering temperatures. For 1045 steel the range is from 392 to 932°F.
   3.3 The different temperatures lead to differences in mechanical properties.
   3.4 Lower temperatures give higher yield strength but lower toughness and ductility.
   3.5 Higher temperatures give lower strength but increase toughness and ductility.
4. Air-cool: Take the steel out of the furnace and let it air-cool to room temperature.

16

Unit 2: Hardness Testing

OBJECTIVE

After completing this unit, you should be able to:

- Perform a Rockwell Test
- Perform a Brinell Test

Beyond verifying our in shop heat treatment, testing hardness is sometime necessary for production work as well. Even though it's bad planning, occasionally a job arrives at our machine shop with an unknown alloy or maybe its composition is known but the hardness isn't. It is possible to use a file to roughly test the machinability of that metal, but the best way to select cutter types, speed, and feeds is a true hardness measurement.

Brinell: Testing hardness by reading the diameter of a ball penetrator mark.

Rockwell: Testing hardness by reading a penetrator depth.

The Rockwell Hardness Test

The Rockwell is a widely accepted method for both soft and hard metals. This system gauges malleability by measuring the depth a pointed probe known shape and size will penetrate into the material given an exact amount of force upon it. Due to Rockwell's range it is the most popular test in tooling and small production shops and training labs.

Rockwell Numbers:

There are several different scales within the Rockwell system. We'll use the Rockwell C scale, correctly used on the hardened steel. The C scale can be said to start at 0(annealed Steel) and run up to 68, harder than a HSS tool bit, near that of a carbide tool. It is symbolized by a larger R with the scale subscript.

$$\mathbf{R_C}$$

The two step Rockwell test:

Step 1. Calibrate Load

The test object is set upon the lower anvil such that it's stable and won't move when pressed down from above. Next, a cone-shaped diamond penetrator is brought into contact then driven into metal to a predetermined of 20lbs. That cause the conical point to sinks into the metal from 0.003 to 0.006 inch. This is the initial calibration load. At that time, a large dial indicator is rotated to read zero.

Step 2.

Test Load Then with the calibration pressure upon the penetrator and indicator set to zero, a second addition 20lbs test load is added. As the diamond sink farther, its added depth is translated to dial, but in an inverse relationship. The deeper the diamond penetrates, the softer the metal tests, therefore the lower number that must appear on the dial face. Inversely, when the point can't go very deep, the metal is hard and registers higher on the dial face.

The Rockwell Method

The Rockwell method measures the permanent depth of indentation produced by a force/load on an indenter.

1. Prepare the sample.

2. Place the test sample on the anvil.
3. A preliminary test force (commonly referred to as preload or minor load) is applied to a sample using a diamond indenter.
4. This load represents the zero or reference position that breaks through the surface to reduce the effects of surface finish. After the preload, an additional load, called the major load, is applied to reach the total required test load.
5. This force is held for a predetermined amount of time (dwell time: 10-15 seconds) to allow for elastic recovery.
6. This major load is then released and the final position is measured against the position derived from the preload, the indentation depth variance between the preload value, and the major load value. This distance is converted to a hardness number.

The Brinell Hardness Test

The Brinell Hardness test is very similar to the Rockwell system in that a penetrator is forced into the sample, however, here the measured gauge is the diameter of the dent made by penetration of a hard steel ball of known size, into the work-piece surface. Hardened tool steel balls are used for testing softer material, while a carbide penetrator ball is used to test harder metals.

Due to the upper hardness, limiting factor of Brinell ball, this test it correctly used as a test of soft to medium hard metals.

The Brinell scale numbers:

The scale runs from 160 for annealed steel up to approximately 700 for very hard steel.

The Brinell hardness test is an alternative way to test the hardness of metals and alloys.

1. Prepare the sample.
2. Place the test sample on the anvil.
3. Move the indenter down into position on the part surface.
4. A minor load is applied and a zero reference position is established.
5. The major load is applied for a specified time period (10 to 15 seconds) beyond zero.
6. The major load is released, leaving the minor load applied.
7. Follow the process to determine the Brinell hardness of an aluminum sample.
    7.1 Press the indenter into the sample using an accurately controlled test force.
    7.2 Maintain the force for a specific dwell time (usually 10 to 15 seconds).
    7.3 After the dwell time is complete, remove the indenter, leaving a round indent in the sample.
    7.4 The size of the indent is determined optically by measuring two diagonals of the round indent using either a portable microscope or one that is integrated with the load application device.
    7.5 The Brinell hardness number is a function of the test force divided by the curved surface area of the indent. The indentation is considered to be spherical, with a radius equal to half the diameter of the ball. The average of the two diagonals is used in the following formula to calculate the Brinell hardness.

BHN=F2D(D-D2- d2)

Unit Test:

1. Please lists five heat treating safety.
2. What is first important thing to know when heat treating a steel?
3. What is the soak time for 1 to 2” Thickness Of Metal?
4. Please explain Soak time.
5. After the soak time is complete, what is the next step?
6. To temper the sample part it must be placed into the furnace at what temperature?
7. Please explain Austenitize and Quench.
8. What is a Air-cool?
9. Please explain The Rockwell Method.
10. Please explain The Brinell Hardness Test.

Chapter Attribution Information

This chapter was derived from the following sources.

- **Heat Treating** derived from Heat Treatment of Plain Carbon and LowAlloy Steels by the Massachusetts Institute of Technology, CC:BY-NC-SA 4.0.
- **Brinell Hardness Test Equation** derived from Brinell Hardness by CORE-Materials Resource Finder, CC:BY.

PART VII

Chapter 7: Lean Manufacturing

17

Chapter 7: Lean Manufacturing

OBJECTIVE

After completing this unit, you should be able to:

- Apply 5S in any Machine shop.
- Describe Kaizen Concept.
- Describe Implementing Lean Manufacturing.

Lean 5S:

"5S" is a method of workplace organization that consists of five words: Sort, Set in order, Shine, Standardize, and Sustain. All of these words begin with the letter S. These five components describe how to store items and maintain the new order. When making decisions, employees discuss standardization, which will make the work process clear among the workers. By doing this, each employee will feel ownership of the process.

Phase 0: Safety

It is often assumed that a properly executed 5S program will improve workplace safety, but this is false. Safety is not an option; it's a priority.

Phase 1: Sort

Review all items in the workplace, keeping only what is needed.

Phase 2: Straighten

Everything should have a place and be in place. Items should be divided and labeled. Everything should be arranged thoughtfully. Employees should not have to bend over repetitively. Place equipment near where it is used. This step is a part of why lean 5s is not considered "standardized cleanup".

Phase 3: Shine

Make sure that the workplace is clean and neat. By doing this, it will be easier to be aware of where things are and where they should be. After working, clean the workspace and return everything to its former position. Keeping the workplace clean should be integrated into the daily routine.

Phase 4: Standardize

Standardize work procedures and make them consistent. Every worker should be aware of what their responsibilities are when following the first three steps.

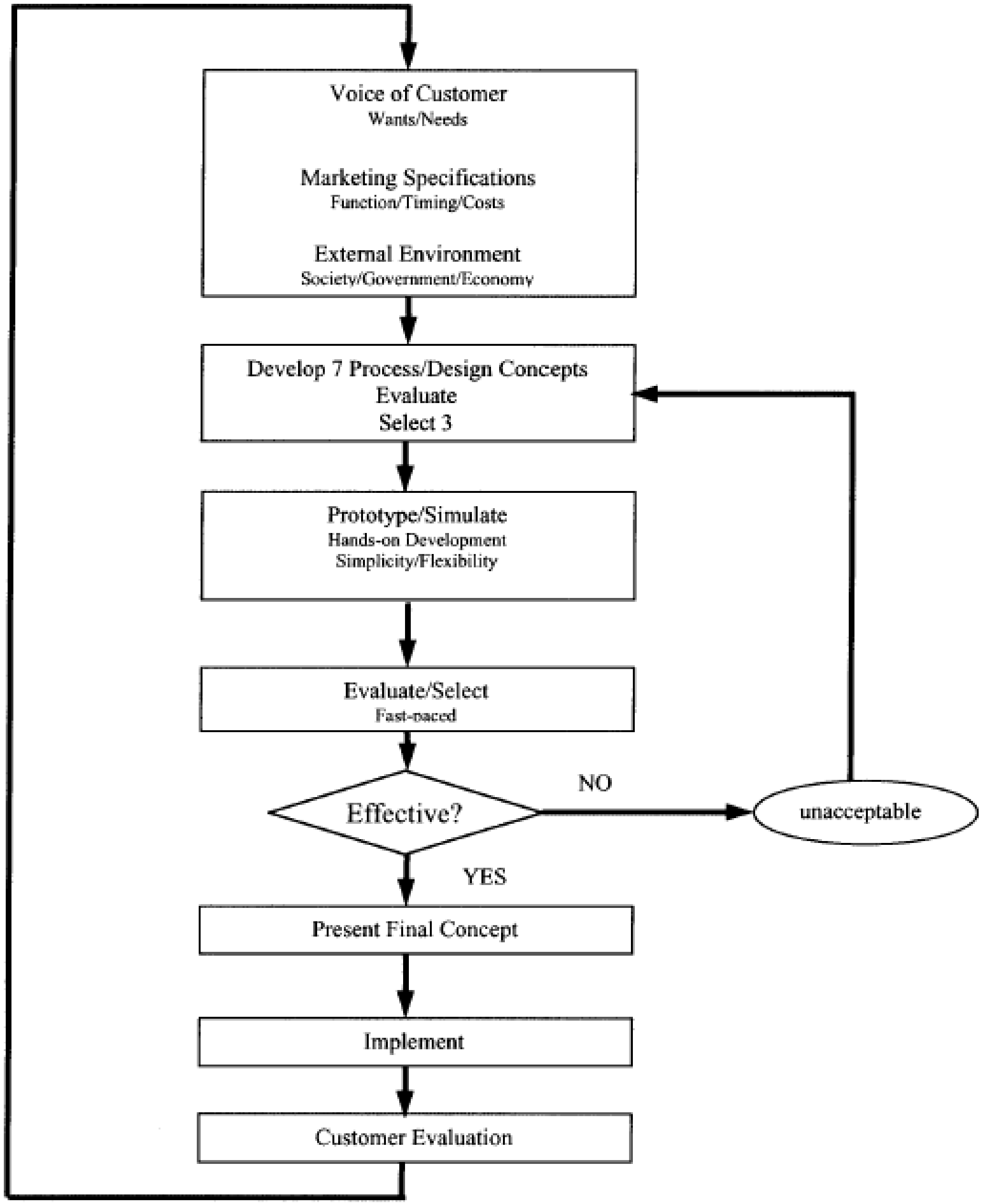
Phase 5: Sustain

Assess and maintain the standards. The aforementioned steps should become the new norm in operation. Do not gradually revert to the old ways. When taking part of the new procedure, think of ways to improve. Review the first four steps when new tools or output requirements are presented.

Kaizen

While the lean 5S process focuses on the removal of waste, Kaizen focuses on the practice of continuous improvement. Like lean 5S, Kaizen identifies three main aspects of the workplace: **M**uda (wastes), **M**ura (inconsistencies) and **M**uri (strain on people & machines). However, the Kaizen step-by-step process is more extensive that the lean 5S process.

The Kaizen process overview:



1. Identify a problem.
2. Form a team.
3. Gather information from internal and external customers, and determine goals for the project.
4. Review the current situation or process.
5. Brainstorm and consider seven possible alternatives.
6. Decide the three best alternatives of the seven.
7. Simulate and evaluate these alternatives before implementation.
8. Present the idea and suggestions to managers.
9. Physically implement the Kaizen results and take account of the effects.

Lean manufacturing improves as time goes one, so it is important to continue education about maintaining standards. It is crucial to change the standards and train workers when presented with new equipment or rules.

Lean

Think of a maintenance department as serving internal customers: the various departments and workers in the company.

Lean is different from the traditional western, mass production model that relies on economies of scale to create profits. The more you make the cheaper the product will become, the greater the potential profit margin. It is based on predictions of customer needs, or creating customer needs. It has difficulty dealing with unusual changes in demand.

Lean production responds to proven customer demand. Pull processing – the customer pulls production. In a mass system the producer pushes product onto the market, push processing.

Building a long-term culture that focuses on improvement.

Respect for workers better trained and educated, more flexible

Lean is a philosophy that focuses on the following:

1. Meeting customer needs
2. Continuous, gradual improvement
3. Making continuously better products
4. Valuing the input of workers
5. Taking the long term view
6. Eliminating mistakes
7. Eliminating waste

Wastes: using too many resources (materials, time, energy, space, money, human resources, poor instructions)

Wastes:

1. Overproduction
2. Defects
3. Unnecessary processing
4. Waiting (wasting time)
5. Wasting human time and talent
6. Too many steps or moving aroundExcessive transportation
7. Excessive inventory

Lean production includes working with suppliers, sub contractors, and sellers to stream line the whole process.

The goal is that production would flow smoothly avoiding costly starts and stops.

The idea is called just in time "produce only what is needed, when it is needed, and only in the quantity needed." Production process must be flexible and fast.

Inventory = just what you need

In mass production = just in case. Extra supplies and products are stored just in case they are needed.

Terminology:

Process simplification – a process outside of the flow of production

Defects – the mass production system does inspection at the end of production to catch defects before they are shipped. The problem is that the resources have already been "spent" to make the waste product" Try to prevent problems immediately, as they happen, then prevent them. Inspection during production, at each stage of production.

Safety – hurt time is waste time

Information – need the right information at the right time (too much, too little, too late)

Principles:

Poka-yoke – mistake proof determining the cause of problems and then removing the cause to prevent further errors

Judgment errors – finding problems after the process

Informative inspections – analyzing data from inspections during the process

Source inspections – inspection before the process begins to prevent errors.

MEAN LEAN

One of the terms applied to a simply cost cutting, job cutting interpretation of Lean is Mean Lean. Often modern manager think they are doing lean without understanding the importance of workers and long term relationships.

Reliability Centered Maintenance

Reliablity centered maintenance is a system for designing a cost effective maintenance program. It can be a detailed complex, computer, statistically driven, but at its basics it is fairly simple. Its ideas can be applied to designing and operating a PM system, and can also guide your learning as you do maintenance, troubleshooting, repair and energy work.

These are core principles of RCM. These nine fundamental concepts are:

- Failures happen.
- Not all failures have the same probability
- Not all failures have the same consequences
- Simple components wear out, complex systems break down
- Good maintenance provides required functionality for lowest practicable cost
- Maintenance can only achieve inherent design reliability of the equipment
- Unnecessary maintenance takes resources away from necessary maintenance
- Good maintenance programs undergo continuous improvement.

Maintenance consists of all actions taken to ensure that components, equipment, and systems provide their intended functions when required.

An RCM system is based on answering the following questions:

1. What are the functions and desired standards of performance of the equipment?
2. In what ways can it fail to fulfil its functions? (Which are the most likely failures? How likely is each type of failure? Will the failures be obvious? Can it be a partial failure?)
3. What causes each failure?
4. What happens when each failure occurs? (What is the risk, danger etc.?)
5. In what way does each failure matter? What are the consequences of a full or partial failure?
6. What can be done to predict or prevent each failure? What will it cost to predict or prevent each failure?
7. What should be done if a suitable proactive task cannot be found (default actions) (no task might be available, or it might be too costly for the risk)?

Equipment is studied in the context of where when and how it is being used

All maintenance actions can be classified into one of the following categories:

- Corrective Maintenance – Restore lost or degraded function
- Preventive Maintenance – Minimizes opportunity for function to fail
- Alterative Maintenance – Eliminate unsatisfactory condition by changing system design or use

Within the category of preventive maintenance all tasks accomplished can be described as belonging to one of five (5) major task types:

- Condition Directed – Renew life based on measured condition compared to a standard
- Time Directed – Renew life regardless of condition
- Failure Finding – Determine whether failure has occurred
- Servicing – Add/replenish consumables
- Lubrication – Oil, grease or otherwise lubricate

We do maintenance because we believe that hardware reliability degrades with age, but that we can do something to restore or maintain the original reliability that pays for itself.

RCM is reliability-centered. Its objective is to maintain the inherent reliability of the system or equipment design, recognizing that changes in inherent reliability may be achieved only through design changes. We must understand that the equipment or system must be studied in the situation in which it is working.

Implementing Lean Manufacturing

Analyze each step in the original process before making change

Lean manufacturing main focuses is on cost reduction and increases in turnover and eliminating activities that do not add value to the manufacturing process. Basically what lean manufacturing does is help companies to achieve targeted production, as well as other things, by introducing tools and techniques that are easy to apply and maintain. What these tools and techniques are doing is reducing and eliminating waste, things that are not needed in the manufacturing process.

Manufacturing engineers set out to use the six-sigma DMAIC (Design, Measure, Analyze, Improve, Control) methodology—in conjunction with lean manufacturing—to meet customer requirements related to the production of tubes.

Manufacturing engineers were charged with designing a new process layout of the tube production line. The objectives for project were including:

- Improved quality
- Decreased scrap
- Delivery to the point of use
- Smaller lot sizes
- Implementation of a pull system
- Better feedback
- Increased production
- Individual Responsibility
- Decreased WIP
- Dine flexibility

Before making changes, the team analyze each step in the original layout of the tube production line process.

1. There try to understand the original state process, identify the problem area, unnecessary step and non value added.

2. After mapping the process, the lean team collected data from the Material Review Board (MRB) bench to measure and analyze major types of defects . To better understand the process, the team also did a time study for 20 days period production run.

In the original state, the tube line consisted of one operator and four operations, separated into two stations by a large table using a push system. The table acted as a separator between the second and third operation.

The first problem discovered was the line's unbalanced . The first station was used about 70% of the time. Operators at the second station were spending a lot of their time waiting between cycle times. By combining stations one and two, room for improvement became evident with respect to individual responsibility, control of inventory by the operator, and immediate feedback when a problem occurred. The time study and the department layout reflect these findings.

A second problem was recognized. Because of the process flow, the production rate did not allow the production schedule to be met with two stations. Because operators lost track of machine cycles, machines were waiting for operator attention. Operators also tried to push parts through the first station—the bottleneck operation in the process—and then continued to manufacture the parts at the last two operations. Typically, long runs of WIP built up, and quality problems were not caught until a lot number of defective pieces were produced.

The original state data were taken from the last 20 days before the change. The teams analyze each step in the original and making changes. The findings of the time study on the original process provided the basis for reducing cycle time, balancing the line, designing the using Just In Time kanbans and scheduling, improve quality, decrease lot size and WIP , and improve flow. The new process data were taken starting one month after implementation. This delay gave the machine operators an opportunity to train and get to with the new process layout system.

With the U shaped cell design; The parts meet all the customer requirement. Table in the original process was removed ,almost eliminating WIP. With the reducing WIP and increasing production.

Some of the concepts used to improve the process included total employee involvement (TEI), smaller lot sizes, scheduling, point of use inventory, and improved layout. All employees and supervisors in the department were involved in all phases of the project. Their ideas and suggestions were incorporated in the planning and implementation process to gain wider acceptance of the changes to the process. Smaller lot sizes were introduced to minimize the number of parts produced before defects were detected. Kanbans were introduced (in the form of material handling racks) to control WIP and to implement a pull system. And the cell layout decreased travel between operations.

Operators were authorized to stop the line when problems arose. In the original-state , the operators were still continue running parts when a operation was down. With kanban

control, the layout eliminated the ability to store WIP, requiring the operator to shut down the entire line. The cell layout provides excellent opportunities for improving communication between operators about problems and adjustments, to achieve better quality.

Day-to-day inspection of the original-state process the operators spent a lot of time either waiting for material-handling person, or performing as a material handling. With the U-shaped cell, delivery to the point of use is more better for the operator. The operator places boxes of raw material on six moveable roller carts, where it's easily to get. The six boxes are enough to last a 24-hr period.

To reduce setup times, tools needed for machine repair and adjustments are located in the cell. The screws are not standardized; tools are set up in order of increasing size to quickly identify the proper tool.

For three months the process was monitored to verify that it was in control. Comparison of time studies from the original-state and the implemented layout demonstrated an increase in production from 300 to 514 finished products per shift. The new layout eliminated double handling between the second and third operations, as well as at the packing step. It also reduced throughout time by making it easier to cycle all four operations in a pull-system order. Customer demand was met by two shifts, which reduced the labor cost.

The results of the redesign are as follows:

- WIP decreased by 97%
- Production increased 72%
- Scrap was reduced by 43%
- Machine utilization increased by 50%
- Labor utilization increased by 25%
- Labor costs were reduced by 33%
- Sigma level increased from 2.6 to 2.8

This project yielded reduced labor and scrap costs, and allowed the organization to do a better job of making deliveries on time, while allowing a smaller finished-goods inventory. Daily production numbers and single-part cycle time served as a benchmark for monitoring progress towards the goal. Although the sigma level increase , the 43% reduction in defects, 97% reduction in WIP, and production increase of 72% contributed to the project objective.

Implementing lean is a never ending process; this is what continuous improvement is all

about. When you get one aspect of lean implemented, it can always be improved. Don't get hung up on it, but don't let things slip back to the starting point. There will always be time to go back and refine some of the processes.

Before Lean Manufacturing was implemented at Nypro Oregon Inc., we would operate using traditional manufacturing. Traditional manufacturing consists of producing all of a given product for the marketplace so as to never let the equipment idle. These goods them need to be warehoused or shipped out to a customer who may not be ready for them. If more is produced than can be sold, the products will be sold at a deep discount (often a loss) or simply scrapped. This can add up to an enormous amount waste. After implementing Lean Manufacturing concepts, our company uses just in time. Just in time refers to producing and delivering good in the amount required when the customer requires it and not before. In lean Manufacturing, the manufacture only produces what the customer wants, when they want it. This often a much more cost effective way of manufacturing when compared to high priced, high volume equipment.

Unit Test:

1. What is 5S?
2. Please Explain each "S" of the 5S.
3. Please Explain Kaizen concept.
4. What is the Pull processing?
5. What is the Poka-yoke?

6. What is the six-sigma DMAIC?
7. What is the objectives for a new process layout of the tube production line?
8. Before making changes, The Manufacturing engineers team do what first?
9. Please lists the results of the redesign.
10. The key to implementing lean new idea or concept is to do what?

CHAPTER ATTRIBUTION INFORMATION

This chapter was derived from the following sources.

- **Lean 5S** derived from Lean Manufacturing by various authors, CC:BY-SA 3.0.
- **Kaizen** derived from A Kaizen Based Approach for Cellular Manufacturing System Design: A Case Study by VirginiaTech, CC:BY-SA 4.0.
- **Kaizen (image)** derived from A Kaizen Based Approach for Cellular Manufacturing System Design: A Case Study by VirginiaTech, CC:BY-SA 4.0.

PART VIII

Chapter 8: CNC

18

Chapter 8: CNC

Unit 1: Introduction to CNC

What is CNC? CNC is a Computer Numerical Control. CNC is the automation of machine tools that are operated by precisely programmed commands encoded and played by a computer as opposed to controlled manually via handwheels or levers.

In modern CNC systems, end-to-end component design is highly automated using Computer-Aided Design (CAD) and Computer-Aided Manufacturing (CAM) programs. The series of steps needed to produce any part is highly automated and produces a part that closely matches the original CAD design.

In the CNC machines the role of the operators is minimized. The operator has to merely feed the program of instructions in the computer, load the required tools in the machine, and rest of the work is done by the computer automatically. The computer directs the machine tool to perform various machining operations as per the program of instructions fed by the operator.

The CNC technology can be applied to wide variety of operations like drafting, assembly, inspection, sheet metal working, etc. But it is more prominently used for various metal machining processes like turning, drilling, milling, shaping, etc. Due to the CNC, all the machining operations can be performed at the fast rate resulting in bulk manufacturing becoming quite cheaper.

How It Works

The CNC machine comprises of the computer in which the program is fed for cutting of the metal of the job as per the requirements. All the cutting processes that are to be carried out and all the final dimensions are fed into the computer via the program. The computer thus knows what exactly is to be done and carries out all the cutting processes. CNC machine works like the Robot, which has to be fed with the program and it follows all your instructions.

You don't have to worry about the accuracy of the job; all the CNC machines are designed to meet very close accuracies. In fact, these days for most of the precision jobs CNC machine is compulsory. When your job is finished, you don't even have to remove it, the machine does that for you and it picks up the next job on its own. This way your machine can keep on doing the fabrication works all the 24 hours of the day without the need of much monitoring, of course you will have to feed it with the program initially and supply the required raw material.

Since the earliest days of production manufacturing, ways have been sought to increase dimensional accuracy as well as speed of production. Simply put, numerical control is a method of automatically operating a manufacturing machine based on a code of letter, numbers, and special characters. As they developed, application of digital computers control of manufacturing equipment was realized. Computers were soon used to provide direct control of machine tools. The integrated circuit led to small computers used to control individual machines, and the computer numerical control (CNC) era was born. This Computer Numerical Control era has become so sophisticated it is the preferred method of almost every phase of

precision manufacturing, particularly machining. Precision dimensional requirements, mainstay of the machining processes, are ideal candidates for use of computer control systems. Computer numerical control now appears in many other types of manufacturing processes. A distinct advantage of computer control of machine tools is rapid, high-precision positioning of workpiece and cutting tools.

Today, manual machine tools have been largely replaced by Computer numerical Control(CNC) machine tools. The machine tool are controlled electronically rather than by hand. CNC machine tools can produce the same part over and over again with very little variation. Modern CNC machines can position cutting tools and workpieces at traverse feed rates of several hundred inches per minute, to an accuracy of .0001". Once programming is complete and tooling is set up, they can run day or night, week after week, without getting tried, with only routine service and cutting tool maintenance. These are obvious advantage over manual machine tools, which need a great deal of human interaction in order to do anything. Cutting feed rates and spindle speeds may be optimized through program instructions. Modern CNC machine tools have turret or belt toolholders and some can hold more than 150 tools. Tool change take less than 15 seconds.

Computer Numerical Control machine are highly productive. They are also expensive to purchase, set up, and maintain. However, the productivity advantage can easily offset this cost if their use is properly managed. A most important advantage of CNC is ability to program the machine to do different jobs. Tool selection and changing under program control is extremely productive, with little time wasted applying a tool to the job.

A program developed to accomplish a given task may be used for a short production run of one, or a few parts. The machine may then be set up for a new job and used for long production runs of hundreds or thousands of production units. It can be interrupted, used for the original job or another new job, and quickly returned to the long production run. This makes the CNC machine tool extremely versatile and productive. Computer-aided design(CAD), has become the preferred method of product design & development. The connection between CAD & CNC was logical. A computer part design can go directly to program used to develop CNC machine control information. A CNC manufacturing machine can then make the part. The computer is extremely useful for assisting the CNC programmer in developing a program to manufacture a specific part. Computer-aided manufacturing, or CAM, systems are now the industry standard for programming. When CAD, CAM & CNC are blended, the greatest capability emerges, producing parts extremely difficult or impossible to make by manual methods.

CNC motion is based on the Cartesian coordinate system. A CNC machine cannot be successfully operated without an understanding of the how coordinate systems are defined in CNC machine and how the systems work together.

To fully understand numerical control programming you must understand axes and coordinates. Think of a part that you would have make. You could describe it to someone else by its geometry. For example, the part you have make is a 5 inch by 8 inch rectangle. All parts can be described in this fashion. Any point on the machined part, such as a pocket to be cut or a hole to be drilled, can be described in term of its position. The system that allows us to do this, called the Cartesian Coordinate or rectangular coordinate system.

Unit 2: CNC Machine Tool Programmable Axes and Position Dimensioning Systems

OBJECTIVE

After completing this unit, you should be able to:

- Understand the cartesian coordinate system.
- Understand the Cartesian coordinates of the plane.
- Understand the Cartesian coordinates of three-dimensional space.
- Understand the four Quadrants.
- Explain the difference between polar and rectangular coordinated.
- Identify the programmable axes on a CNC machining.

THE CARTESIAN COORDINATE SYSTEM

Cartesian coordinates allow one to specify the location of a point in the plane, or in three-dimensional space. The Cartesian coordinates or rectangular coordinates system of a point are a pair of numbers (in two-dimensions) or a triplet of numbers (in three-dimensions) that specified signed distances from the coordinate axis. First we must understand a coordinate system to define our directions and relative position. A system used to define points in space by establishing directions(axis) and a reference position(origin). A coordinate system can be rectangular or polar.

Just as points on the line can be placed in one to one correspondence with the real number line, so points in plane can be placed in one to one correspondence with pairs of real number line by using two coordinate lines. To do this, we construct two perpendicular coordinate line that intersect at their origins; for convenience. Assign a set of equally space graduations to the x and y axes starting at the origin and going in both directions, left and right (x axis) and up and down (y axis) point along each axis may be established. We make one of the number lines vertical with its positive direction upward and negative direction downward. The other number lines horizontal with its positive direction to the right and negative direction to the left. The two number lines are called coordinate axes; the horizontal line is the x axis, the vertical line is the y axis, and the coordinate axes together form the Cartesian coordinate system or a rectangular coordinate system. The point of intersection of the coordinate axes is denoted by O and is the origin of the coordinate system. See Figure 1.

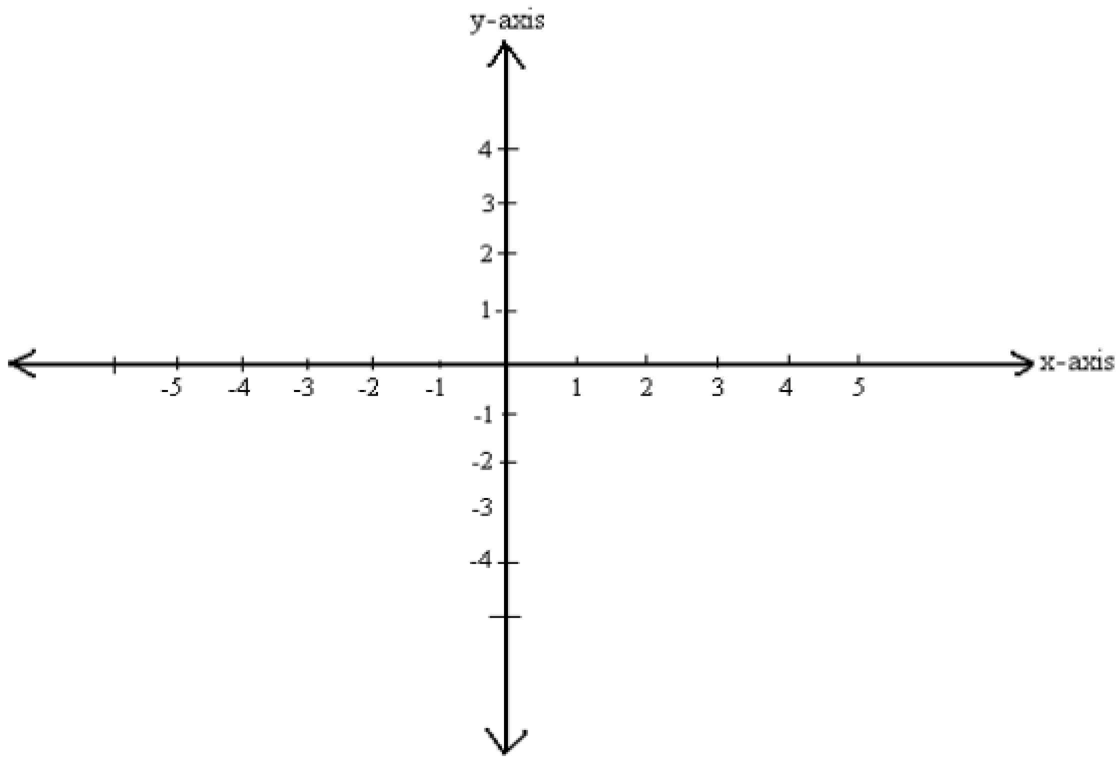


Figure 1

It is basically, Two Real Number Lines Put Together, one going left-right, and the other going up-down. The horizontal line is called x-axis and the vertical line is called y-axis.

The Origin

The point (0,0) is given the special name "The Origin", and is sometimes given the letter "O".

Real Number Line

The basis of this system is the real number line marked at equal intervals. The axis is labeled (X, Y or Z). One point on the line is designated as the Origin. Numbers on one side of the line are marked as positive and those to the other side marked negative. See Figure 2.

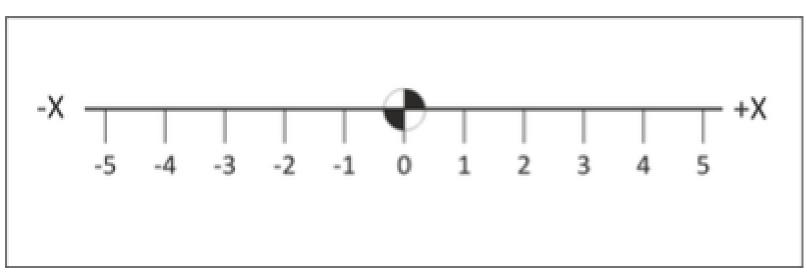


Figure 2. X-axis number line

Cartesian coordinates of the plane

A plane in which a rectangular coordinate system has been introduced is a coordinate plane or an x-y-plane. We will now show how to establish a one to one correspondence between points in a coordinate plane and pairs of real number. If A is a point in a coordinate plane, then we draw two lines through A, one perpendicular to the x-axis and one perpendicular to the y-axis. If the first line intersects the x-axis at the point with coordinate x and the second line intersects the y-axis at the point with coordinate y, then we associate the pair (x,y) with the A(See Figure 2). The number a is the x-coordinate or abscissa of P and the number b is the y-coordinate or ordinate of p; we say that A is the point with coordinate (x,y) and denote the point by A(x,y). The point (0,0) is given the special name "The Origin", and is sometimes given the letter "O".

Abscissa and Ordinate:

The words "Abscissa" and "Ordinate" … they are just the x and y values:

- Abscissa: the horizontal ("x") value in a pair of coordinates: how far along the point is.
- Ordinate: the vertical ("y") value in a pair of coordinates: how far up or down the point is.

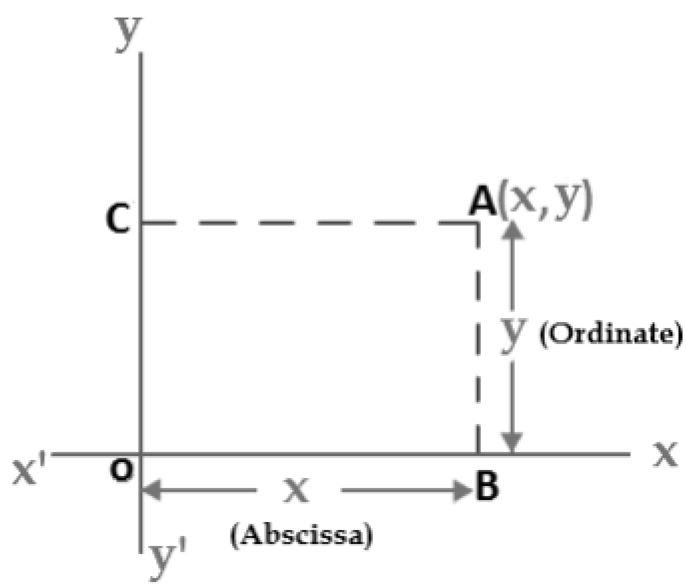


Figure 3

Negative Values of X and Y:

The Real Number Line, you can also have negative values.

Negative: start at zero and head in the opposite direction; See Figure 4

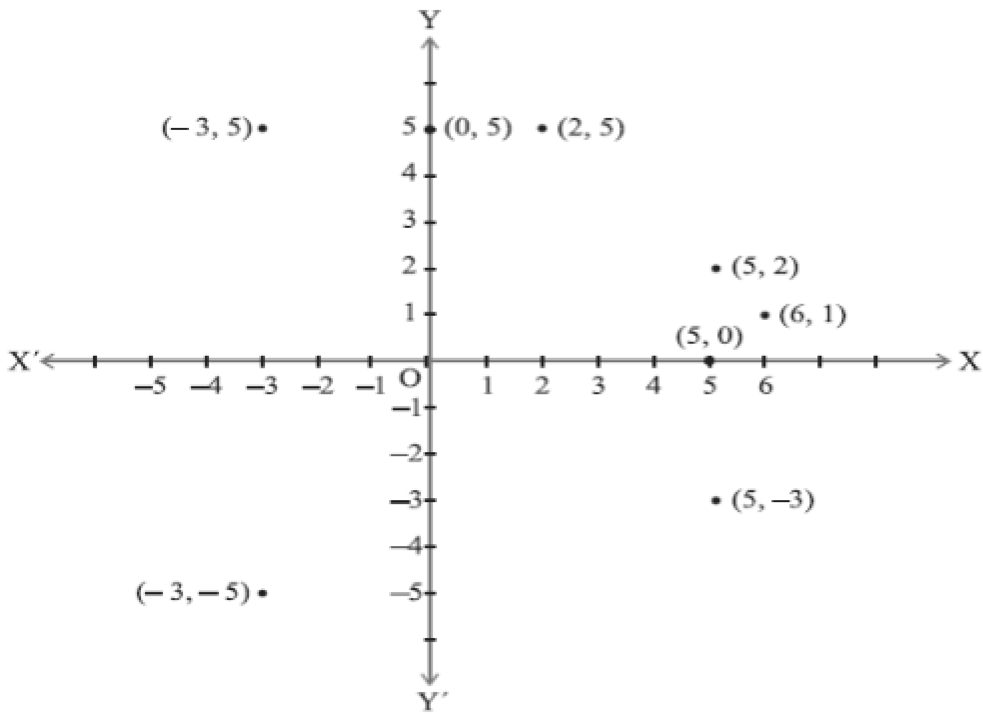


Figure 4

So, for a negative number:

- go left for x
- go down for y

For example (-3,5) means:
go left along the x axis 3 then go up 5 in the y-axis. (Quadrant II x is negative ,y is positive)
And (-3,-5) means:
go left along the x axis 3 then go down 5 in the y-axis. (Quadrant III x is negative ,y is negative)
Using Cartesian Coordinates, mark a point on a graph by how far along and how far up it is; See figure 5. The point (12,5) is 12 units along the x-axis, and 5 units up on the y-axis.

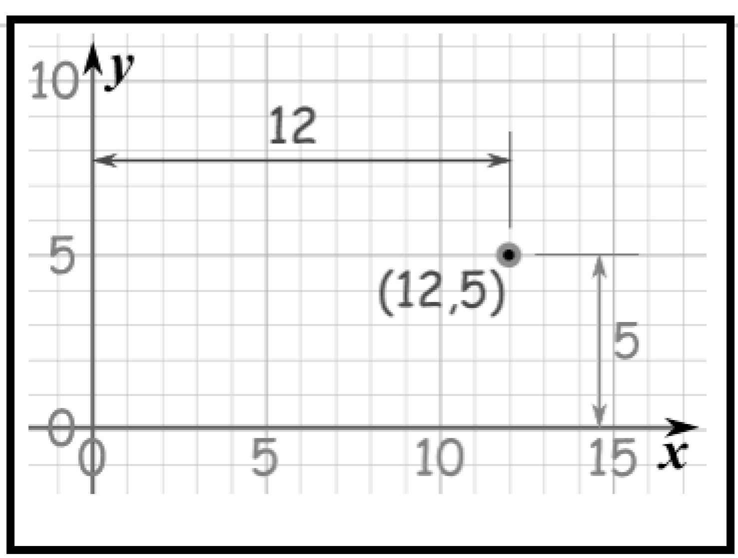
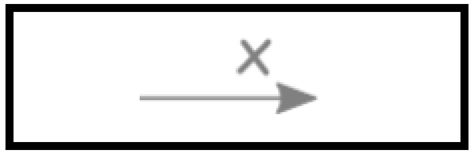
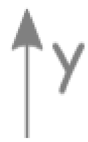


Figure 5

X and Y Axis:



The horizontal line is called x-axis and the vertical line is called y-axis; both line runs throughzero(Origin, (0,0)).



The horizontal line is called x-axis and the vertical line is called y-axis; both line runs throughzero(Origin, (0,0)).
Put them together on a graph …See figure 6

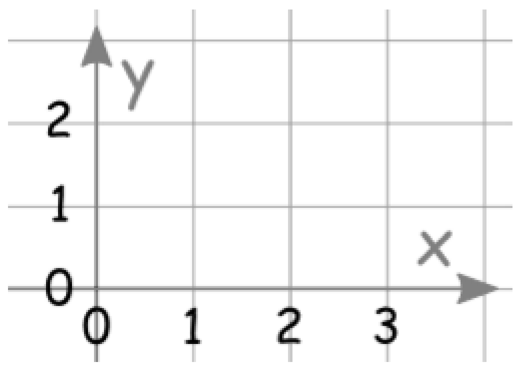
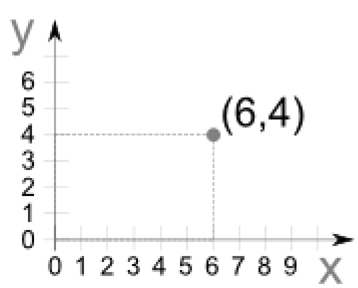


Figure 6

It is basically, a set of two Real Number lines.
Axis: The reference line from which distances are measured.

Example:



Point (6,4) is

Go along the x direction 6 units then go up 4 units up in the y direction then "plot the dot".

And you can remember which axis is which by:

The coordinates are always written in a certain order:

- the horizontal distance first,
- then the vertical distance.

Ordered pair:

The numbers are separated by a comma, and parentheses are put around the whole thing like this: (7,4)

Example: (7,4) means 7 units to the right(x-axis), and 4 units up(y-axis)

Cartesian coordinates of three-dimensional space

In three-dimensional space(xyz space), oriented at right angles to the xy-plane. The z axis, passes through the origin of the xy-plane. Coordinates are determined according to the east-west for x-axis north-south for y-axis, and up-down for the z-axis displacements from the origin. The Cartesian coordinate system is based on three mutually perpendicular coordinate axes: the x-axis, the y-axis, and the z-axis, See Figure 6 below. The three axes intersect at the point called the origin. You can imagine the origin being the point where the walls in the corner of a room meet the floor. The x-axis is the horizontal line along which the wall to your left and the floor intersect. The y-axis is the horizontal line along which the wall to your right and the floor intersect. The z-axis is the vertical line along which the walls intersect. The parts of the lines that you see while standing in the room are the positive portion of each of the axes. The negative part of these axes would be the continuations of the lines outside of the room.

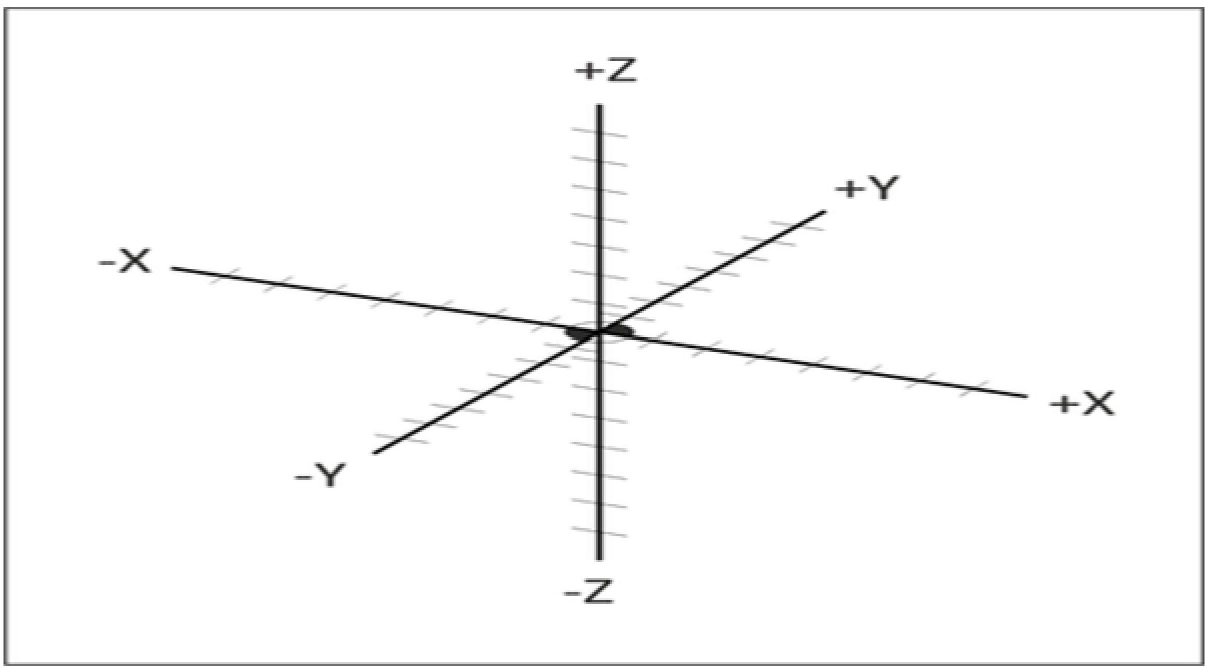


Figure 7. 3D Cartesian Coordinate System

Three-dimensional Cartesian coordinate axes. A representation of the three axes of the three-dimensional Cartesian coordinate system. The positive x-axis, positive y-axis, and positive z-axis are the sides labeled by x, y and z. The origin is the intersection of all the axes. The branch of each axis on the opposite side of the origin (the unlabeled side) is the negative part.

When dealing with 3-dimensional motion, is to set up a suitable coordinate system. The most straight-forward type of coordinate system is called a Cartesian system. A Cartesian coordinate system consists of three mutually perpendicular axes, the X, Y, and Z-axes. By convention, the orientation of these axes is such that when the index finger, the middle finger, and the thumb of the right-hand are configured so as to be mutually perpendicular, the index finger, the middle finger, and the thumb can be aligned along the X, Y, and Z-axes, respectively. Such a coordinate system is termed right-handed. See Figure 7. The point of intersection of the three coordinate axes is termed the origin of the coordinate system.

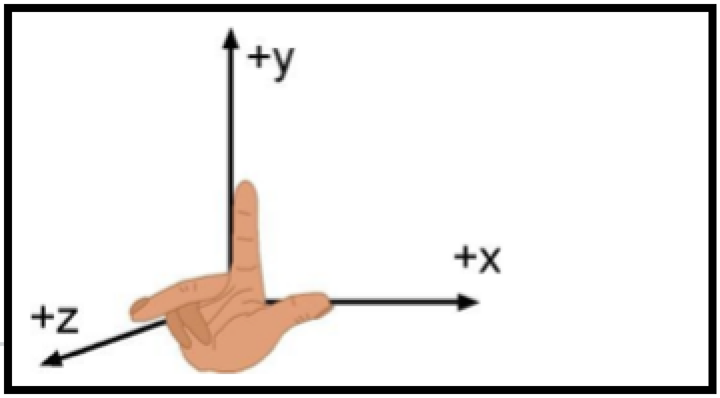


Figure 8. The Right Handed Cartesian System

The Cartesian coordinates of a point in three dimensions are a triplet of numbers (x,y,z). The three numbers, or coordinates, specify the signed distance from the origin along the x, y, and z-axes, respectively. They can be visualized by forming the box with edges parallel to the coordinate axis and opposite corners at the origin and the given point.

The points may now be defined in a three dimensional volume of space. This permits to define points in three dimensions from the origin. The Cartesian coordinates (x,y,z) of a point in three-dimensions specify the signed distance from the origin along the x, y, and z-axes, respectively. Z-axis points become the third entry when defining coordinate locations.

Given the above corner-of-room analogy, we could form the Cartesian coordinates of the point at the top of your head, as follows. Imagine that you are five meters tall the z-axis, and that you walk two meters from the origin along the x-axis, then turn left and walk parallel to the y-axis four meters into the room. The Cartesian coordinates of the point at the top of your head would be (2,4,5).

For example, a notation of (2,4,5) corresponds to the value of X2, Y4, and Z5. See Figure 8.

3 Dimensions

Cartesian coordinates can be used for locating points in 3 dimensions as in this example:

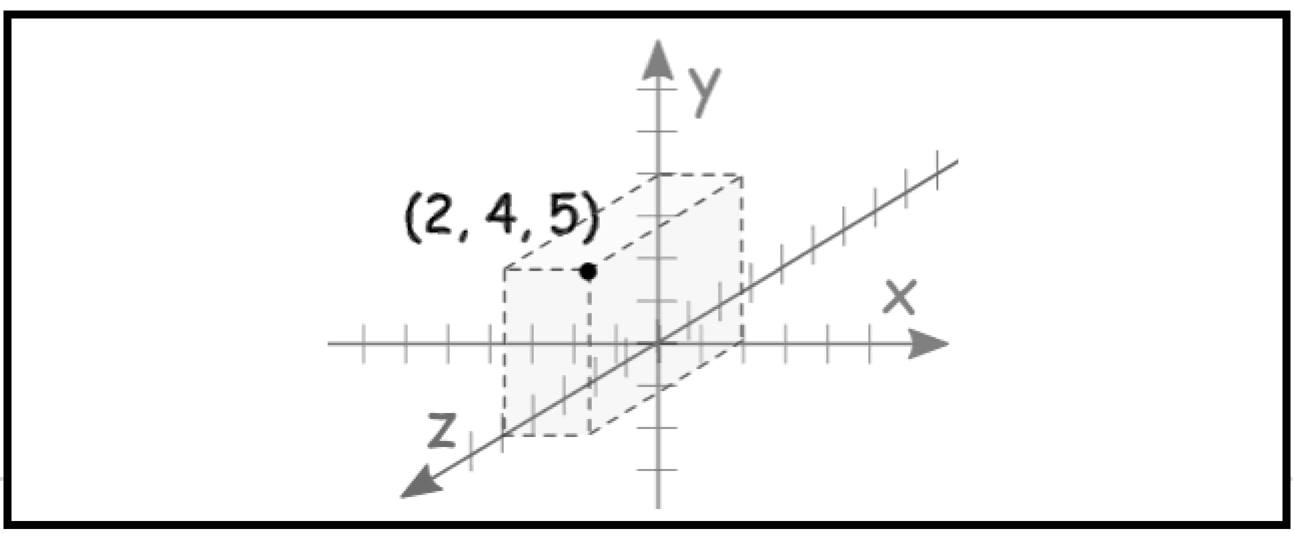


Figure 9. The point (2, 4, 5) is shown in three-dimensional Cartesian coordinates.

Quadrants

The coordinate axes divide the plane into four parts, called quadrants (See Figure 9). The quadrants are number counterclockwise, starting from the upper right, labeled I, II, III and IV with axes designations as shown in the illustration below.

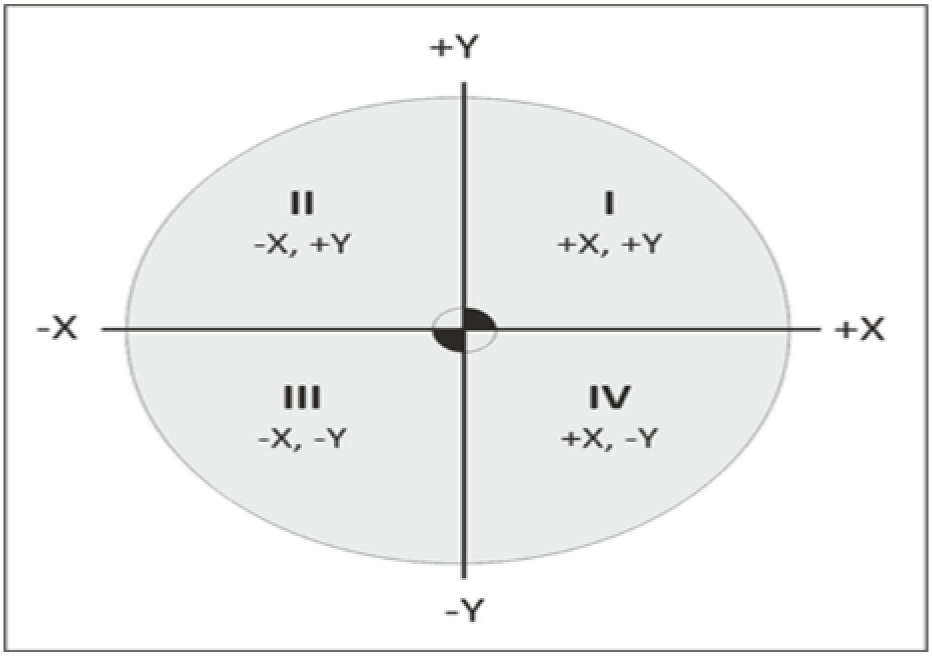


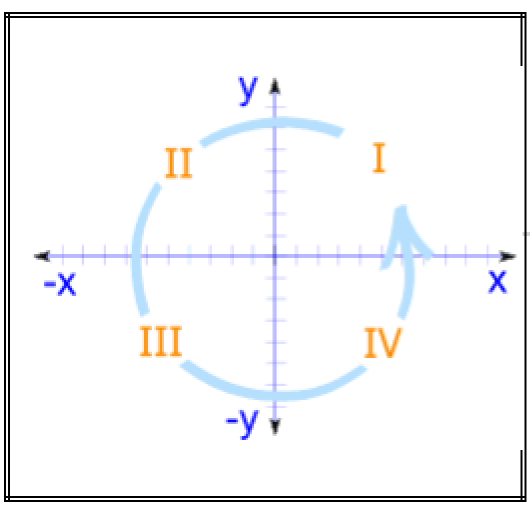
Figure 10

Four Quadrants:

When we include negative values, the x and y axes divide the space up into 4 pieces:

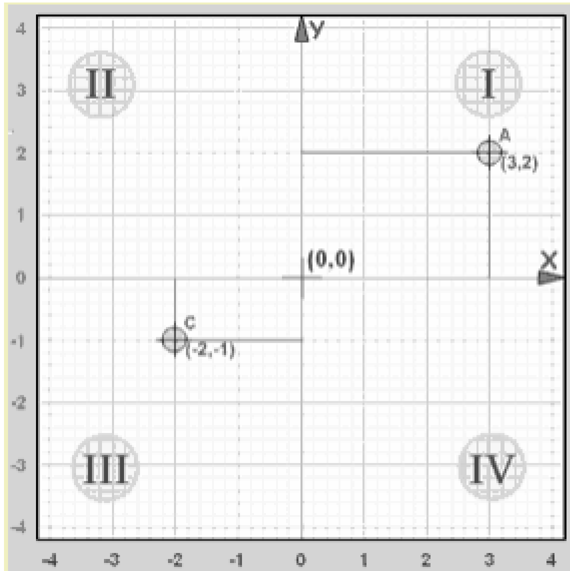
Quadrants I, II, III and IV

(They are numbered in a counterclockwise direction)



In Quadrant I : both x and y are positive
In Quadrant II : x is negative (y is still positive)

In Quadrant III : both x and y are negative
In Quadrant IV : x is positive again, while y is negative



Quadrant	X (Horizontal)	Y (Vertical)
I	Positive	Positive
II	Negative	Positive
III	Negative	Negative
IV	Positive	Negative

Example: The point "A" (3,2) is 3 units along the x-axis, and 2 units up the y-axis.

Both x and y are positive, so that point is in "Quadrant I"

Example: The point "C" (-2,-1) is 2 units along the x-axis in the negative direction, and 1 unit down the y-axis in the negative direction.

Both x and y are negative, so that point is in "Quadrant III"

Dimensions: 1, 2, 3 and more …

1. The Real Number Line can only go:
    - left-right
    - so any position needs just one number
2. Cartesian coordinates can go:
    - left-right, and
    - up-down
    - so any position needs two numbers
3. 3 dimensions

- left-right,
- up-down, and
- forward-backward

UNIT TEST

1. What is CNC?

2. Describe the cartesian coordinate system.

3. What is The Origin?

4. The Horizontal line is called what?

5. The Vertical line is called what?

6. Describe the real number line.

7. Explain Abscissa and Ordinate.

8. What are the representation of the three axes of the three dimensional cartesian coordinate system.

9. The coordinate axes divide the plane into four parts, is called what?

10. In Quadrant IV, the X axes and the Y axes is what?

20

Unit 3: Vertical Milling Center Machine Motion

OBJECTIVE

After completing this unit, you should be able to:

- Understand the Vertical Milling Center Machine Motion.
- Understand the Machine Home Position.
- Understand the CNC Machine Coordinates.
- Understand the Work Coordinate System.
- Understand the Machine and Tool Offsets.
- Set Tool Length Offset for each tool.

VMC Machine Motion

CNC machines use a 3D Cartesian coordinate system. Figure 10. shows a typical Vertical Milling Center (VMC). Parts to be machined is fastened to the machine table. This table moves in the XY-Plane. As the operator faces the machine, the X-Axis moves the table left-right. The Y-Axis moves the table forward-backward. The machine column grips and spins the tool. The column controls the Z-axis and moves up-down.

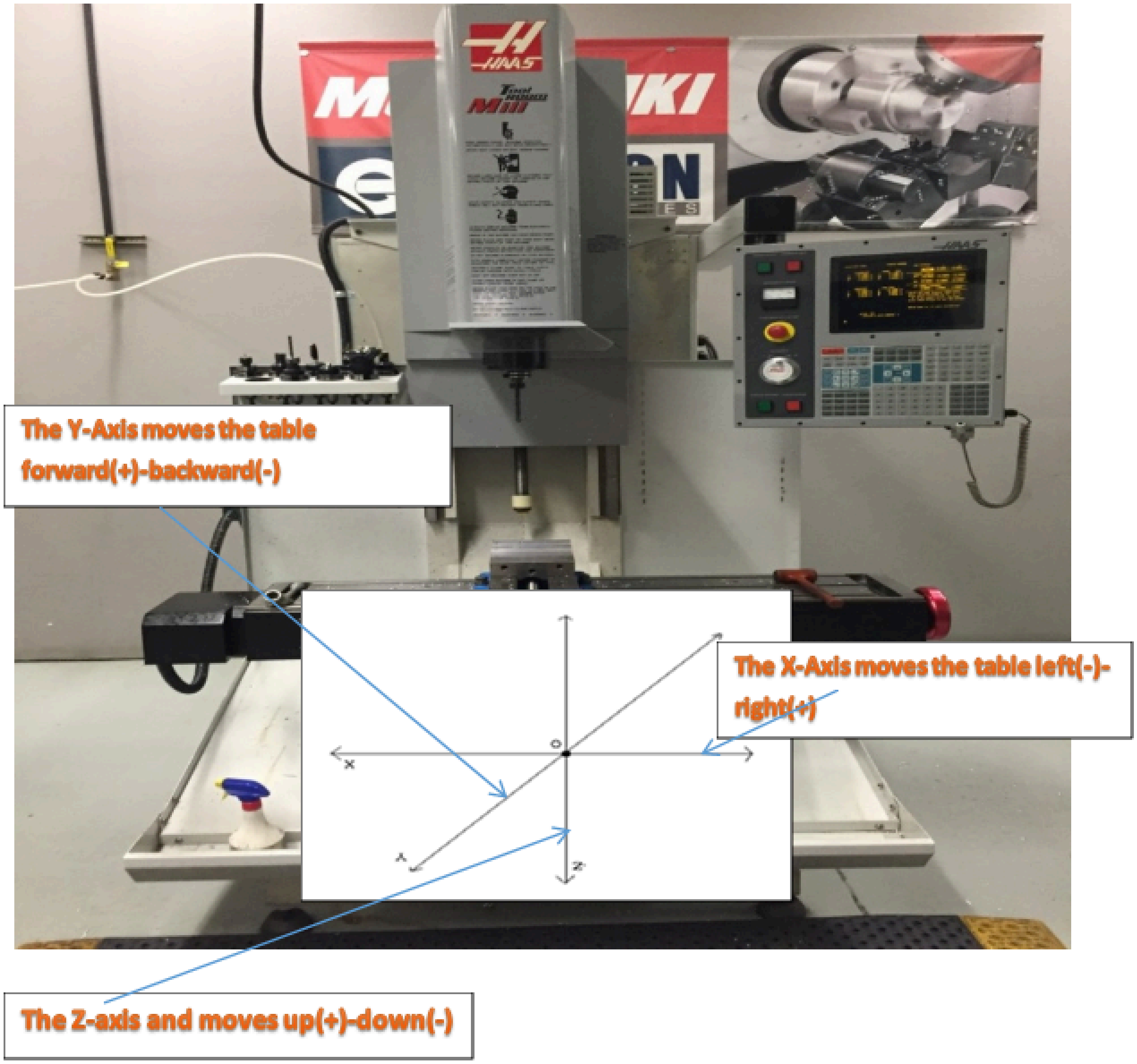


Figure 1.VMC Machine Motion

CNC Machine Coordinates

The CNC Machine Coordinate System is illustrated in Figure 11. The control point for the Machine Coordinate System is defined as the center-face of the machine spindle. The Origin point for the machine coordinate system is called Machine Home. This is the postion of the center-face of the machine spindle when the Z-axis is fully retracted and the table is moved to its limits near the back-left corner.

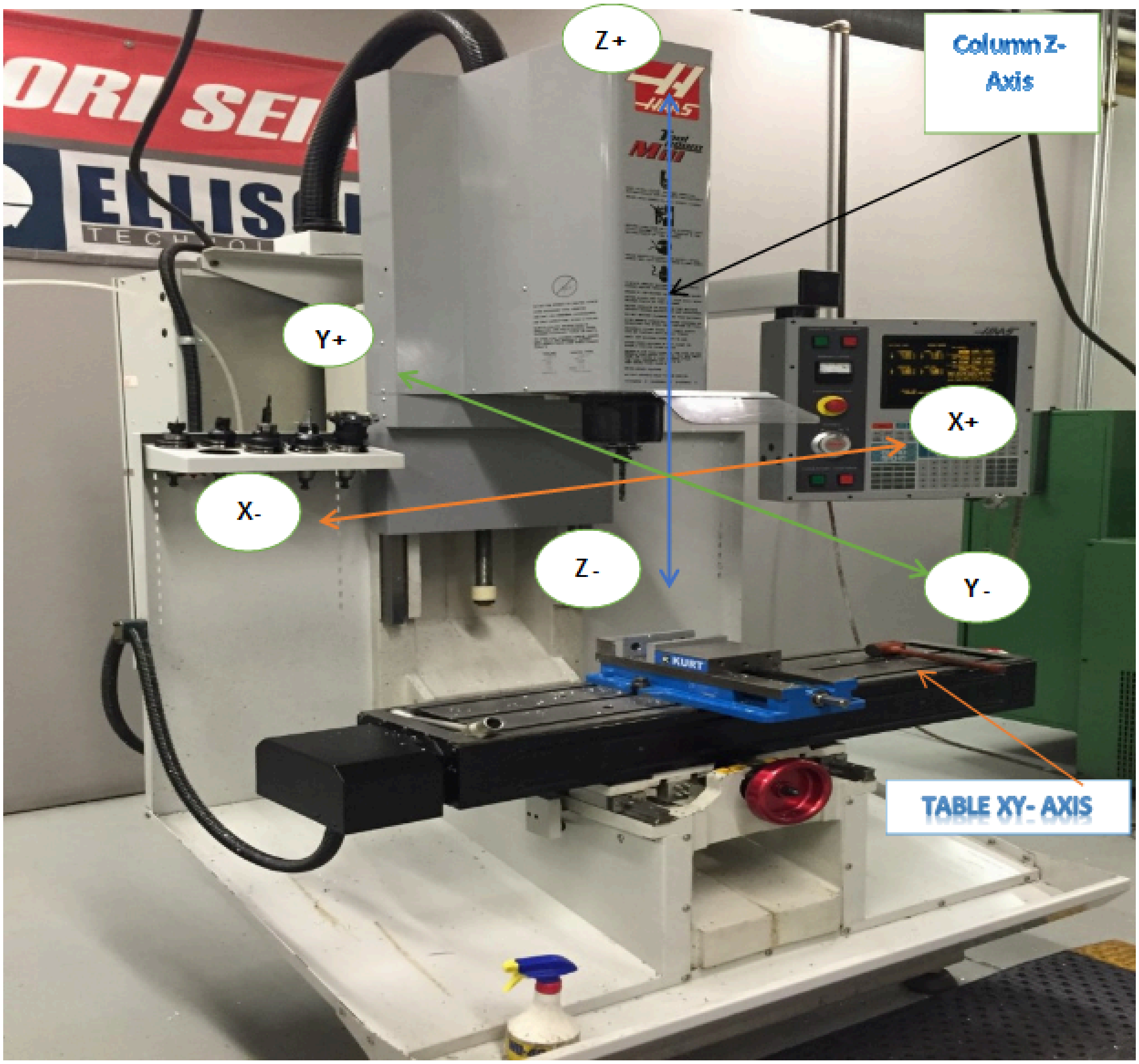


Figure 2. VMC Machine Coordinate System (At Home Position)

As shown in Figure 12, when working with a CNC, always think, work, and write CNC programs in terms of tool motion, not table motion. For example, increasing +X coordinate values move the tool right in relation to the table (though the table actually moves left). Likewise, increasing +Y coordinate values move the tool towards the back of the machine (the table moves towards the operator). Increasing +Z commands move the tool up (away from the table).

About Machine Home Position

When a CNC machine is first turned on, it does not know where the axes are positioned in the work space. Home position is found by the Power On Restart sequence initiated by the operator by pushing a button on the machine control after turning on the control power.

The Power On Restart sequence simply drives all three axes slowly towards their extreme limits (–X, +Y, +Z). As each axis reaches its mechanical limit, a microswitch is activated. This signals to the control that the home position for that axis is reached. Once all three axes have stopped moving, the machine is said to be "homed". Machine coordinates are thereafter in relation to this home position.

Work Coordinate System

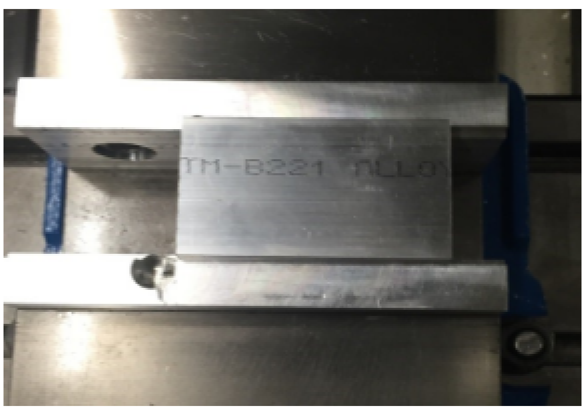
Obviously it would be difficult to write a CNC program in relation to Machine Coordinates. The home position is far away from the table, so values in the CNC program would be large and have no easily recognized relation to the part model. To make programming and setting up the CNC easier, a Work Coordinate System (WCS) is established for each CNC program.

The WCS is a point selected by the CNC programmer on the part, stock or fixture. While the WCS can be the same as the part origin in CAD, it does not have to be. While it can be located anywhere in the machine envelope, its selection requires careful consideration.

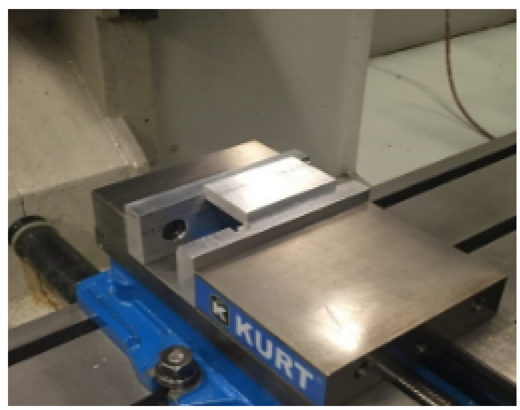
- The WCS location must be able to be found by mechanical means such as an edge finder, coaxial indicator or part probe.
- It must be located with high precision: typically plus or minus .001 inches or less.
- It must be repeatable: parts must be placed in exactly the same position every time.
- It should take into account how the part will be rotated and moved as different sides of the part are machined.

For example, Figure 13 shows a part gripped in a vise. The outside dimensions of the part have already been milled to size on a manual machine before being set on the CNC machine.

The CNC is used to make the holes, pockets, and slot in this part. The WCS is located in the upper-left corner of the block. This corner is easily found using an Edge Finder or Probe.



Top View



Side View

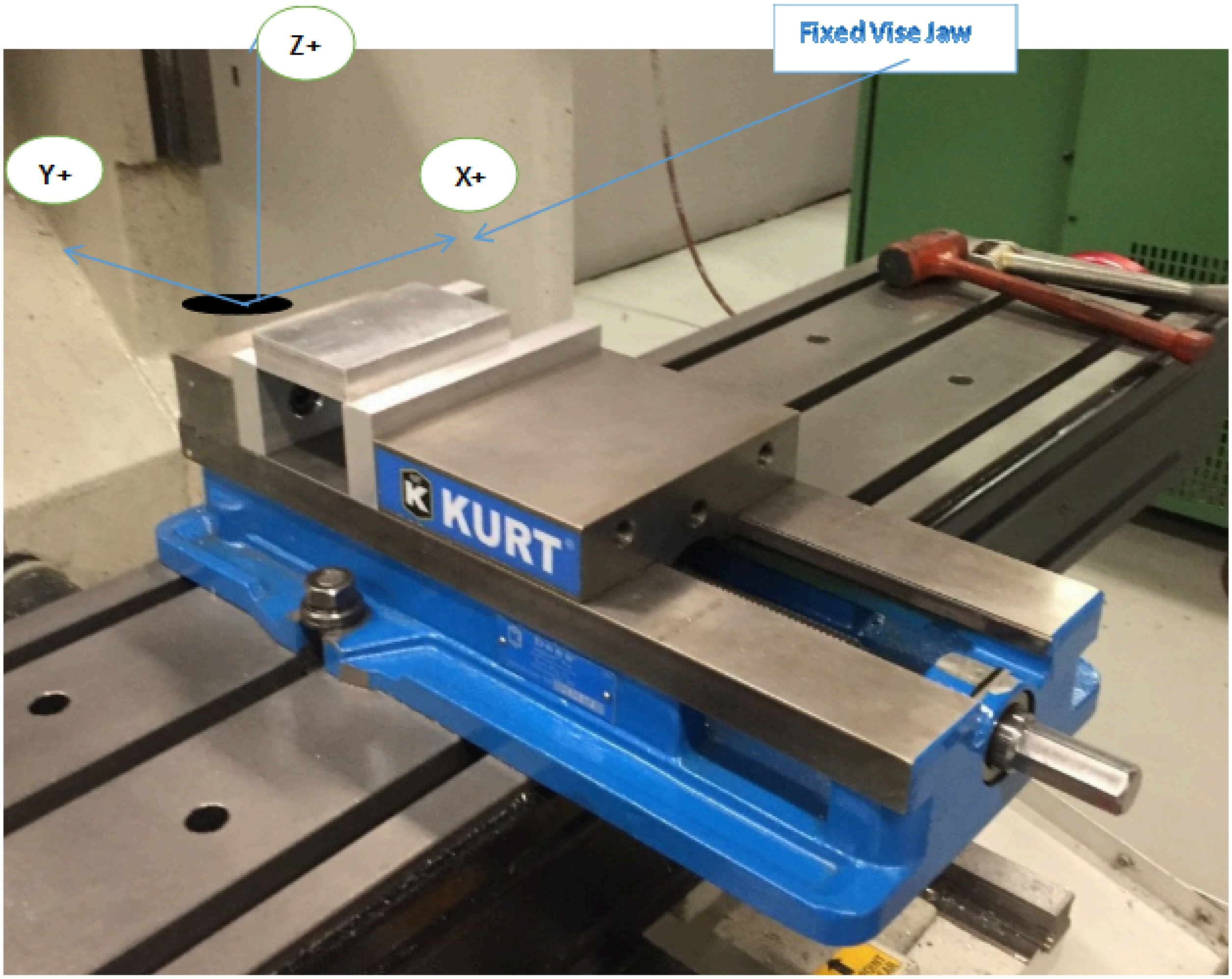


Figure 3. Work Coordinate System (WCS)

Machine and Tool Offsets

Machine Offsets:

Because it is difficult to place a vise in the exact same position on the machine each time, the distance from Home to the WCS is usually not known until the vise is set and aligned with the machine. Machine set up is best done after the program is completely written, because it is expensive to keep a CNC machine idle waiting for the CNC programming to be done. Besides, the programmer may change their mind during the CAM process, rendering any pre-planned setup obsolete.

To complicate matters further, different tools extend out from the machine spindle different lengths, also a value difficult to determine in advance. For example, a long end mill extends further from the spindle face than a stub length drill. If the tool wears or breaks and must be replaced, it is almost impossible to set it the exact length out of the tool holder each time.

Therefore, there must be some way to relate the Machine Coordinate system to the part WCS and take into account varying tool lengths. This is done using machine Tool and Fixture Offsets. There are many offsets available on CNC machines. Understanding how they work and to correctly use them together is essential for successful CNC machining.

Part Offset XY:

Fixture offsets provide a way for the CNC control to know the distance from the machine home position and the part WCS. In conjunction with Tool Offsets, Fixture Offsets allow programs to be written in relation to the WCS instead of the Machine Coordinates. They make setups easier because the exact location of the part in the machine envelop does not need to be known before the CNC program is written.

As long as the part is positioned where the tool can reach all machining operations it can be located anywhere in the machine envelope. Once the Fixture Offset values are found, entered into the control, and activated by the CNC program, the CNC control works behind the scene to translate program coordinates to WCS coordinates.

Notice in Figure 14 how Part Offsets (+X, -Y) are used to shift the centerline of the machine spindle directly over the WCS.

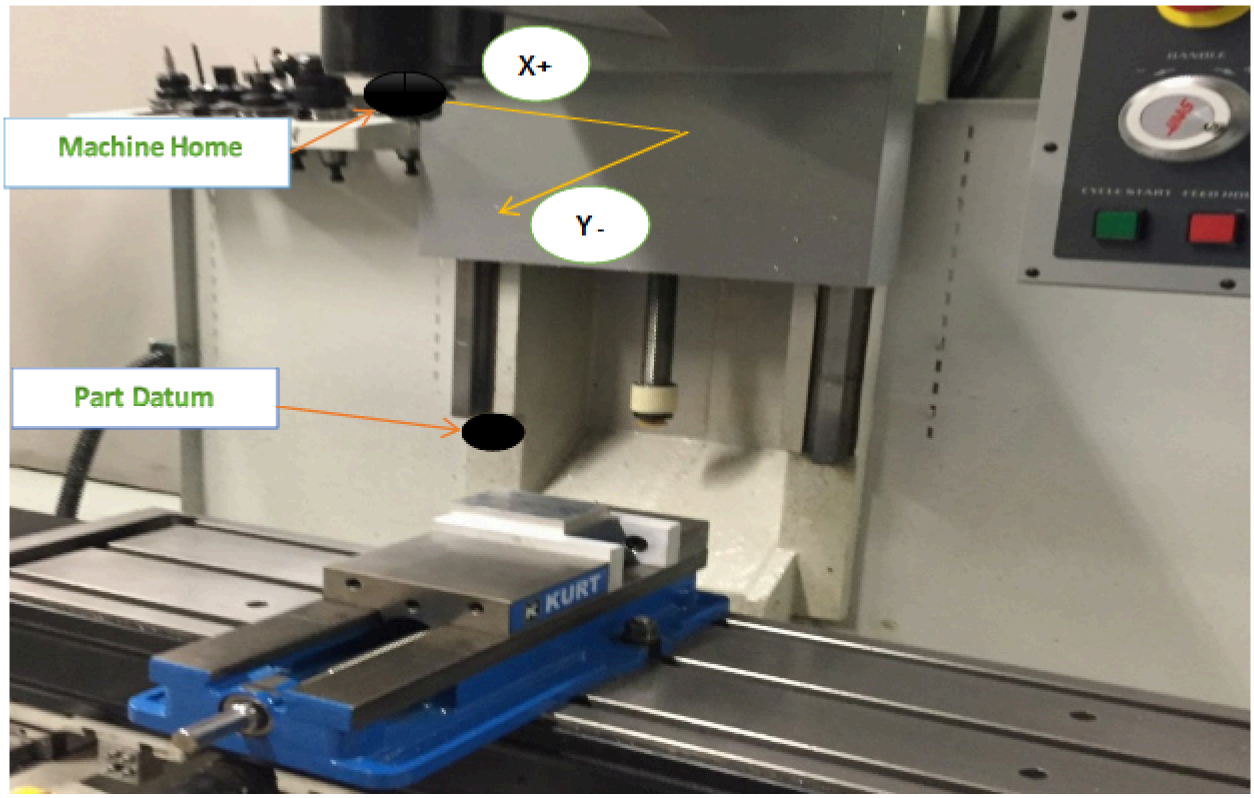


Figure 4: PartOffset Shifts Machine to WCS

Part Offset Z

The Part Offset Z value is combined with the Tool Length offset to indicate to the machine how to shift the Z-datum from part home to the part Z-zero, taking into account the length of the tool. Fixture Offset Z may or may not be used, depending on how the machine is set up and operated.

Tool Length Offset (TLO)

Every tool loaded into the machine is a different length. In fact, if a tool is replaced due to wear or breaking, the length of its replacement will likely change because it is almost impossible to set a new tool in the holder in exactly the same place as the old one. The CNC machine needs some way of knowing how far each tool extends from the spindle to the tip. This is accomplished using a Tool Length Offset (TLO).

In its simplest use, the TLO is found by jogging the spindle with tool from the machine home Z-position to the part Z-zero position, as shown on the far left in Figure 17 below. The tool is jogged to the part datum Z and the distance travelled is measured. This value is entered in the TLO register for that tool. Problems with this method include the need to face mill the part to the correct depth before setting tools. Also, if the Z-datum is cut away (typical of 3D surfaced parts) it is impossible to set the datum should a tool break or wear and need to be replaced. All tools must be reset whenever a new job is set up. When this method is used, the Fixture Offset Z is not used, but set to zero.

The method shown in the center is much better and used in this book. All tools are set to a known Z-position, such the top of a precision 1-2-3 block resting on the machine table. This makes it very easy to reset tools if worn or broken.

A tool probe is very similar to the 1-2-3 block method, except the machine uses a special cycle to automatically find the TLO. It does this slowly lowering the tool until the tip touches the probe and then updates the TLO register. This method is fast, safe and accurate but requires the machine be equipped with a tool probe. Also, tool probes are expensive so care must be taken to never crash the tool into the probe.

Both the 2nd and 3rd methods also require the distance from the tool setting position (the top of the 1-2-3 block or tool probe) to the part datum to be found and entered in the Fixture Offset Z. The machine adds the two values together to determine the total tool length offset. A method for doing this is included in.

3-Ways to Set Tool Length Offset

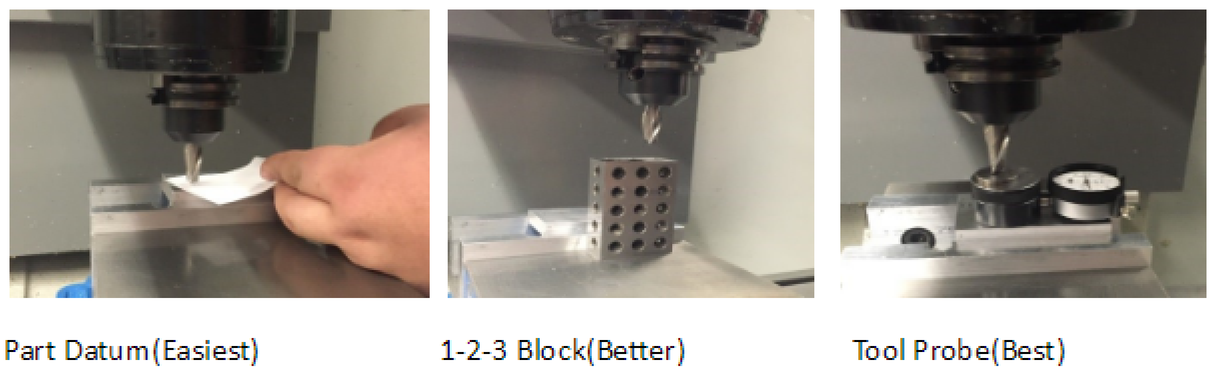


Figure 5. Ways to set TLO

UNIT TEST

1. Explain Machine home position.

2. On the Vertical Milling Center(VMC) the X axis move the table in what direction.

3. On the Vertical Milling Center(VMC) the Y axis move the table in what direction.

4. On the Vertical Milling Center(VMC) the Z axis move the table in what direction.

5. Please lists 3-ways to set Tool length offsets.

21

Unit 4: CNC Language and Structure

OBJECTIVE

After completing this unit, you should be able to:

- Identify the programs list instructions.
- Understand the Program Format
- Describe Letter Address Commands codes
- Describe Special Character Code Definitions.
- Understand the G & M Codes.

CNC programs list instructions to be performed in the order they are written. They read like a book, left to right and top-down. Each sentence in a CNC program is written on a separate line, called a Block. Blocks are arranged in a specific sequence that promotes safety, predictability and readability, so it is important to adhere to a standard program structure.

The blocks are arranged in the following order:

- Program Start
- Load Tool
- Spindle On
- Coolant On
- Rapid to position above part
- Machining operation

- Coolant Off
- Spindle Off
- Move to safe position
- End program

The steps listed above represent the simplest type of CNC program, where only one tool is used and one operation performed. Programs that use multiple tools repeat steps two through nine for each.

Table 3 and Table 4 in section **G & M Codes** show the most common G and M codes that should be memorized if possible.

Like any language, the G-code language has rules. For example, some codes are modal, meaning they do not have to be repeated if they do not change between blocks. Some codes have different meanings depending on how and where there are used.

While these rules are covered in this chapter, do not concern yourself with learning every nuance of the language. It is the job of the job of the CAD/CAM software Post Processor to properly format and write the CNC program.

Program Format

The program in Table 1: below machines a square contour and drills a hole.

Block	Description	Purpose
%	Start of program.	Start Program
O1234	Program number (Program Name).	
(T1 0.25 END MILL)	Tool description for operator.	
G17 G20 G40 G49 G80 G90	Safety block to ensure machine is in safe mode.	
T1 M6	Load Tool #1.	Change Tool
S9200 M3	Spindle Speed 9200 RPM, On CW.	
G54	Use Fixture Offset #1.	Move to Position
M8	Coolant On.	
G00 X-0.025 Y-0.275	Rapid above part.	
G43 Z1.H1	Rapid to safe plane, use Tool Length Offset #1.	
Z0.1	Rapid to feed plane.	
G01 Z-0.1 F18.	Line move to cutting depth at 18 IPM.	
G41 Y0.1 D1 F36.	CDC Left, Lead in line, Dia. Offset #1, 36 IPM.	Machine Contour
Y2.025	Line move.	
X2.025	Line move.	
Y-0.025	Line move.	
X-0.025	Line move.	
G40 X-0.4	Turn CDC off with lead-out move.	
G00 Z1.	Rapid to safe plane.	
M5	Spindle Off.	Change Tool
M9	Coolant Off.	
(T2 0.25 DRILL)	Tool description for operator.	
T2 M6	Load Tool #2.	
S3820 M3	Spindle Speed 3820 RPM, On CW.	

M8	Coolant On.	
X1. Y1.	Rapid above hole.	Move to Position
G43 Z1.H2	Rapid to safe plane, use Tool Length Offset 2.	
Z0.25	Rapid to feed plane.	
G98 G81 Z-0.325 R0.1 F12.	Drill hole (canned) cycle, Depth Z-.325, F12.	
G80	Cancel drill cycle.	Drill Hole
Z1.	Rapid to safe plane.	
M5	Spindle Off.	
M9	Coolant Off.	
G91 G28 Z0	Return to machine Home position in Z.	
G91 G28 X0 Y0	Return to machine Home position in XY.	End Program
G90	Reset to absolute positioning mode (for safety).	
M30	Reset program to beginning.	
%	End Program.	

Letter Address Commands codes

The command block controls the machine tool through the use of letter address commands. Some are used more than once, and their meaning changes based on which G-code appears in the same block.

Codes are either modal, which means they remain in effect until cancelled or changed, or non-modal, which means they are effective only in the current block. As you can see, many of the letter addresses are chosen in a logical manner (T for tool, S for spindle, F for feed rate, etc.).

The table below lists the most common Letter Address Commands codes.

Table 2: Letter Address Commands Codes

Variable	Description	Definitions
A	Absolute or incremental position ofAaxis (rotational axis around X axis)	A,B,C – 4th/5th Axis Rotary Motion Rotation about the X, Y or Z-axis respectively.The angle is in degrees and up to three decimal places precision. G01 A45.325B90.
B	Absolute or incremental position of B axis (rotational axis around Y axis)	Same as A
C	Absolute or incremental position of C axis (rotational axis around Z axis)	Same as B
D	Defines diameter or radial offset used for cutter compensation	Used to compensate for tool diameter wear and deflection.D is accompanied by an integer that isthe same as the tool number (T5 uses D5,etc). No decimal point is used. Itis always usedin conjunction with G41 or G42 and a XY move (never an arc). When called, the control reads the register and offsets the tool path left (G41) or right (G42) by the value in the register. G01 G41 X2.D1
E	Precision feed rate for threading on lathes	
F	Defines feed rate	Sets the feed rate when machining lines, arcs or drill cycles.Feed rate can be in Inches per Minute (G94 mode) or Inverse Time (G93 mode). Feed rates can be up to three decimal placesaccuracy (for tap cycles) and require a decimal point. G01 X2.Y0. F30.
G	Address for preparatory commands	G commands often tell the control what kind of motion is wanted (e.g., rapid positioning, linear feed, circular feed, fixed cycle) or what offset value to use. G02 X2.Y2.I.50J0.
H	Defines tool length offset; Incremental axis corresponding to C axis (e.g., on a turn-mill)	This code calls a tool length offset (TLO) register on the control. The control combines the TLO and Fixture Offset Z values to know where the tool is in relation to the part datum.It is always accompanied by an integer (H1, H2,etc), G43, and Z coordinate. G43 H1 Z2.

I	Defines arc size inX axisfor G02 or G03 arc commands. Also used as a parameter within some fixed cycles.	For arc moves (G2/G3), this is the incremental X-distance from the arc start point to the arc center. Certain drill cycles also use I as an optional parameter. G02 X.5 Y2.500I0.J0.250
J	Defines arc size inY axisfor G02 or G03 arc commands. Also used as a parameter within some fixed cycles.	For arc moves (G2/G3), this is the incremental Y-distance from the arc start point to the arc center. Certain drill cycles also use J as an optional parameter. G02 X.5 Y2.500 I0.J0.250
K	Defines arc size inZ axisfor G02 or G03 arc commands. Also used as a parameter within some fixed cycles, equal to L address.	For anarcmove (G2/G3) this is the incremental Z-distance from the arc start point to the arc center. In the G17 plane, this is the incremental Z-distance for helical moves. Certain drill cycles also use J as an optional parameter. G18 G03 X.3 Z2.500 I0.K0.250
L	Fixed cycle loop count; Specification of what register to edit using G10	Fixed cycle loop count: Defines number of repetitions ("loops") of a fixed cycle at each position. Assumed to be 1 unless programmed with another integer. Sometimes the K addressis usedinstead of L. With incremental positioning (G91), a series of equally spaced holes can be programmed as a loop rather than as individual positions. G10 use: Specificationof what register to edit (work offsets, tool radius offsets, tool length offsets, etc.).
M	Miscellaneous function	Always accompanied by an integer that determines its meaning. Only one M-codeis allowedin each block of code. Expanded definitions of M-codes appear later in this chapter. M08
N	Line (block) number in program; System parameter number to be changed using G10	Block numbers can make the CNC program easier to read. They are seldom required for CAD/CAM generated programs with no subprograms. Because they take up controlmemorymost 3D programs do not use block numbers. Block numbers are integers up to five characters long with no decimal point. They cannot appear before the tape start/end character (%) and usually do not appear before a comment only block. N100 T02 M06
O	Program name	Programs are stored on the control by their program number. Thisis an integer thatis preceded by the letter O and has no decimal places. O1234 (Exercise 1)

P	Serves as parameter address for various G and M codes	Dwell (delay) in seconds.Accompanied by G4 unless used within certain drill cycles. G4 P.1
Q	Peck increment in canned cycles	The incremental feed distance per pass in a peck drill cycle. G83 X2.000 Y2.000 Z-.625 F20.R.2 Q.2 P9.
R	Defines size of arc radius or defines retract height in canned cycles	Arcs can be defined using the arc radius R or I,J,Kvectors. IJK's are more reliable than R's so itis recommendedto use them instead. Ris also usedby drill cycles as the return plane Z value. G83 Z-.625 F20.R.2 Q.2 P9.
S	Defines speed, either spindle speed or surface speed depending on mode	Spindle speed in revolutions per minute (RPM). It is an integer value with no decimal, and always used in conjunction with M03 (Spindle on CW) or M04 (Spindle on CCW). S2500M03
T	Tool selection	Selects tool. It is an integer value always accompanied by M6 (tool change code). T01 M06
U	Incremental axis corresponding to X axis (typically only lathe group A controls) Also defines dwell time on some machines.	In these controls, X and U obviate G90 and G91, respectively. On these lathes, G90 is instead a fixed cycle address for roughing.
V	Incremental axis corresponding to Y axis	Until the 2000s, the V address was very rarely used, because most lathes that used U andWdidn'thave a Y-axis, so they didn't use V. (Green et al 1996 did not even list V in their table of addresses.) That is still often the case, although the proliferation of live lathe tooling and turn-mill machining has made V address usage less rare than it used to be (Smid2008 shows an example).
W	Incremental axis corresponding to Z axis (typically only lathe group A controls)	In these controls, Z and W obviate G90 and G91, respectively. On these lathes, G90 is instead a fixed cycle address for roughing.
X	Absolute or incremental position ofX axis.	Coordinate data for the X-axis. Up to four places after the decimalare allowedand trailing zeros are not used. Coordinates are modal, so there is no need to repeat them in subsequent blocks if they do not change. G01 X2.250F20.

Y	Absolute or incremental position of Y axis	Coordinate data for the Y-axis. G01 Y2.250 F20.
Z	Absolute or incremental position of Z axis	Coordinate data for the Z-axis.

Special Character Code Definitions

The following is a list of commonly used special characters, their meaning, use, and restrictions.

% – Program Start or End

All programs begin and end with % on a block by itself. This code is called tape rewind character (a holdover from the days when programs were loaded using paper tapes).

( ) – Comments

Comments to the operator must be all caps and enclosed within brackets. The maximum length of a comment is 40 characters and all characters are capitalized.
(T02: 5/8 END MILL)

/ – Block Delete

Codes after this character are ignored if the Block Delete switch on the control is on.
/ M00

; – End of Block

This character is not visible when the CNC program is read in a text editor (carriage return), but does appear at the end of every block of code when the program is displayed on the machine control.
N8 Z0.750 ;

G & M Codes

G&M Codes make up the most of the contents of the CNC program. The definition of each class of code and specific meanings of the most important codes are covered next.

G-Codes

Codes that begin with G are called preparatory words because they prepare the machine for a certain type of motion.
Table 3: G-Code

Code	Description
G00	Rapid motion.Used to position the machine for non-milling moves.
G01	Line motion at a specified feed rate.
G02	Clockwise arc.
G03	Counterclockwise arc.
G04	Dwell.
G28	Return to machine home position.
G40	Cutter Diameter Compensation (CDC) off.
G41	Cutter Diameter Compensation (CDC) left.
G42	Cutter Diameter Compensation (CDC) right.
G43	Tool length offset (TLO).
G54	Fixture Offset #1.
G55	Fixture Offset #2.
G56	Fixture Offset #3.
G57	Fixture Offset #4.
G58	Fixture Offset #5.
G59	Fixture Offset #6.
G80	Cancel drill cycle.
G81	Simple drill cycle.
G82	Simple drill cycle with dwell.
G83	Peck drill cycle.
G84	Tap cycle.

G90	Absolute coordinate programming mode.
G91	Incremental coordinate programming mode.
G98	Drill cycle return to Initial point (R).
G99	Drill cycle return to Reference plane (last Z Height)

M-Codes

Codes that begin with M are called miscellaneous words. They control machine auxiliary options like coolant and spindle direction. Only one M-code can appear in each block of code.

Table 4: M-Codes

Code	Description
M00	Program stop.Press Cycle Start button to continue.
M01	Optional stop.
M02	End of program.
M03	Spindle on Clockwise.
M04	Spindle on Counterclockwise.
M05	Spindle stop.
M06	Change tool.
M08	Coolant on.
M09	Coolant off.
M30	End program and press Cycle Start to run it again.

Select G-Code Definitions (Expanded)

G00 – Rapid Move

This code commands the machine to move as fast as it can to a specified point. It is always used with a coordinate position and is modal. Unlike G01, G00 does not coordinate the axes to move in a straight line. Rather, each axis moves at its maximum speed until it is satisfied. This results in motion as shown in Figure 18, below.

G00 X0. Y0.

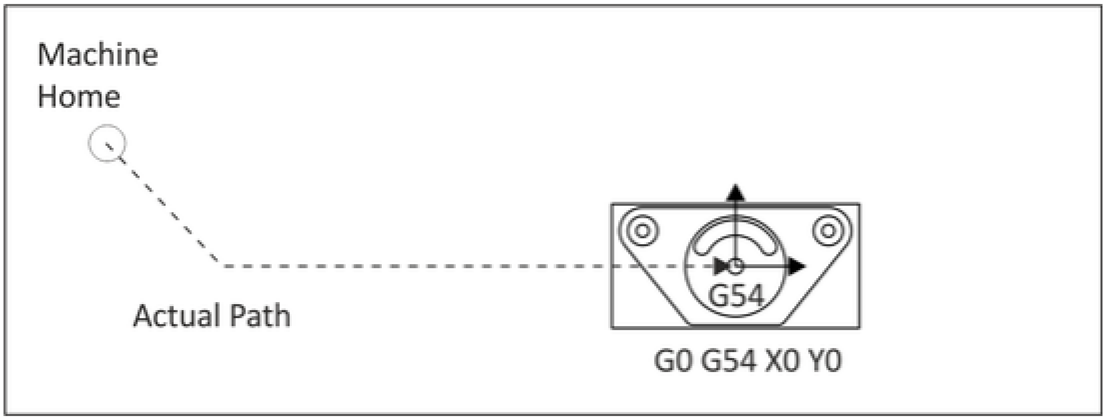


Figure 1. G00 Motion

Caution: The rapid speed of some machines can exceed 1. An incorrect offset or coordinate move can crash the machine faster than the operator can hit the emergency stop. Use the rapid feed override on the machine when running a program for the first time.

Linear Motion is Straight Line Motion:

G-Code is about motion, and the most common kind of motion found in part programs is straight line or linear motion. Motion is another one of those things in G-Code that is modal. You tell the controller what kind of motion you'd like with a G-Code and it remembers to always make that kind of motion until you tell it to change using another G-Code.

G00 for Fast Positioning ; Rapids Motion as fast as your machine will go. Used to move the cutter through air to the next position it will be cutting.

G01 for Slower Cutting Motion; Feed Motion slower, for cutting. Feedrate set by "F" G-Code.

F-word = "F" as in "Feedrate".

S-word = "S" as in "Spindle Speed", address is rpm.

Specifying Linear Motion With X, Y, and Z:

Specifying G00 or G01 does not cause any motion to happen–they merely tell the controller what type of motion is expected when you finally tell it where to move to. For actual motion you need to specify a destination using the X, Y, and Z words. To move to the part zero, we might issue a command like this:

G00 X0 Y0 Z0   Or use G01 if you want to go slower G01 X0 Y0 Z0 F40.

Interpolated motion or an interpolated move, When we specify multiple coordinates on a line, means more than one axis of the machine is moving at the same time. In fact, the controller will move them all at exactly the right speed relative to one another so that the cutter follows a straight line to the destination and moves at the feedrate.

If we specify the same destination, but spread the coordinates over multiple lines, each line is a separate move:

G00 X0 Y0 (Move to X0 Y0 in one move, keeping Z constant)

Z0 (Move to Z0 in one move, keeping X and Y constant)

G00 and G01 are modal, so we only have to specify them when we want to change modes.

Z Axis:

The concept of interpolated moves raises an interesting issue for the Z axis. It's often a good idea to move the depth-of-cut-axis on its own, rather than as coordinated motion with other axes(X and Y). Whether you're going to have a problem (collision) as the cutter gets close to the workpiece and fixturing. First moving in X and Y and then moving in Z , it's much easier to judge whether an accidental collision is about to take place. You're also much less likely to hit some random object sticking up, like a clamp, if you keep the cutter high until you're directly over where you want to start cutting.

G02 and G03 Circular Motion is a Mode Initiated:

G02 establishes a mode for clockwise circular arcs.
G03 establishes a mode for counter-clockwise circular arcs.

The G02 or G03 mode is established, arcs are defined in G-Code by identifying their 2 endpoints and the center which must be equi-distant from each endpoint. The endpoints are easy. The current control point, or location when the block is begun establishes one endpoint. The other may be established by XYZ coordinates. The center is most commonly identified by using I, J, or K to establish relative offsets from the starting point of the arc to the center.

EXAMPLE OF CLOCKWISE ARC:

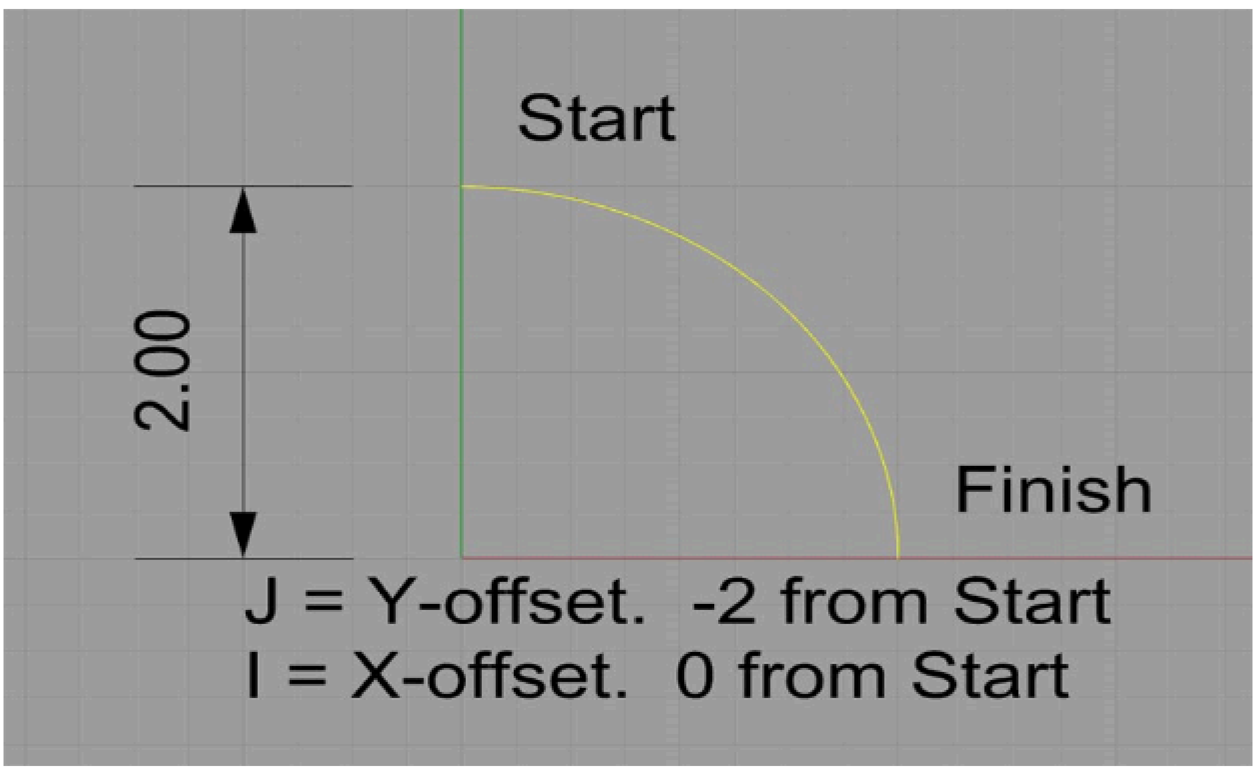


Figure 2. An Arc's center with IJK

This arc starts at X0Y2. and finishes at X2.Y0. It's center is at X0Y0. We could specify it in G-code like this:

G02 (Set up the clockwise arc mode)

X2Y0 I0J-2.0

The Center Using Radius "R".

The center just by specifying the radius of the circle. Circle has a radius of 2, so the G-Code might be simply:

G02 X2Y0 R2

G17/G18/G19 – Plane Designation

Arcs must exist on a plane designated by the command G17 (XY), G18 (XZ) or G19 (YZ). G17 is the machine default.

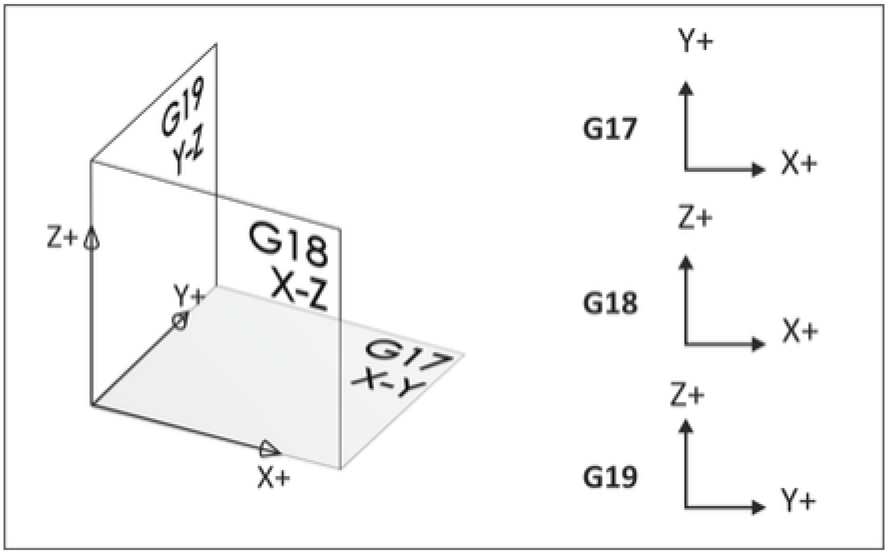


Figure 3. Plane Designation

G40/G41/G42 – Cutter Diameter Compensation (CDC)

CDC is a key to precision CNC machining, allowing the operator to compensate for tool wear and deflection by commanding the machine to veer left (G41) or right (G42) from the programmed path. G40 cancels cutter compensation. The amount of offset is entered in a CNC control D-register. The wear register can be thought of like a table that the control refers to with every move.

Table 5: Diameter Offset Register

Tool Diameter Offset	Value
D1	0.0125
D2	0.0000
D3	0.0000
D4	0.0000
D5	0.0000
D6	0.0000

The value in the D-register is calculated by the machine tool operator, who monitors the finished size of part features, compares them with the print, and enters the difference in the register as needed to keep the part within specifications. If there is no deviation, the register is set to zero.

G01 G41 D1 X1.0 Y.25 F40.

G43 – Tool Length Compensation

G43 activates tool length compensation. It is always accompanied by an H-code and Z-move, where H is the tool length offset (TLO) register to read, and Z is the height to go to in reference to the part datum.

The (TLO) can be thought of like a table on the control:
Table 6: Work Offsets

Tool Length Resister	Z
H1	10.236
H2	4.7510
H3	6.9652
H4	7.6841
H5	12.4483
H6	8.2250

The TLO is combined with the active fixture offset on the control so the machine knows where the tip of the tool is in relation to the part datum.

G43 H1 Z1.

G54 – Work Offset

Work offsets are data registers in the CNC control that hold the distance from the machine home X, Y, Z position to the part datum. These offsets can be thought of like a table on the control:

Table 7: Work Offset

Work Offset	X	Y	Z
G54	14.2567	6.6597	0.0000
G55	0.0000	0.0000	0.0000
G56	0.0000	0.0000	0.0000
G57	0.0000	0.0000	0.0000
G58	0.0000	0.0000	0.0000
G59	0.0000	0.0000	0.0000

Tip: G54 is usually usedfor the first machining setup. Additional offsets are used to machine other sides of the part.

The X and Y values represent the distance from the machine home to part datum XY. The Z value is the distance from the tool reference point (for example, the top of a 1-2-3 block) and the part Z-datum.

G54 X0. Y0.

UNIT TEST

1. Please describe the CNC program list instruction.

2. All CNC program start and end with what?

3. Describe letter address Commands codes.

4. Please lists three special character codes.

5. Describe G and M codes.

6. Please describe G00 G90 G54 X0 Y0.

7. Please describe G00 G90 G43 H1 Z1.

8. What is the different between G00 and G01?

9. Explain the different between G02 and G03.

10. Please Describe the F and S word.

Unit 5: CNC Operation

OBJECTIVE

After completing this unit, you should be able to:

- Understand the CNC Operation.
- List the steps to set up and operate a CNC mill.
- Identify the location and purpose of the operating controls on the Haas CNC Mill control.
- Start and home a CNC machine.
- Load tools into tool carousel.
- Set Tool Length Offsets.
- Set Part Offsets.
- Load a CNC program into the machine control.
- Dry run
- Safely run a new CNC program.
- Adjust offsets to account for tool wear and deflection.
- Shut down a CNC machine correctly.

Overview of CNC Setup and Operation

CNC machine setup and operation follows the process below:

1. Pre-Start
2. Start/Home
3. Load Tools
4. Mount Remove Part into the vise
5. Set Tool Length Offsets Z
6. Set Part Offset XY
7. Load CNC Program
8. Dry Run
9. Run Program
10. Adjust Offsets as Needed
11. Shut Down

1. Pre-Start

Before starting the machine, check to ensure oil and coolant levels are full. Check the machine maintenance manual if you are unsure about how to service it. Ensure the work area is clear of any loose tools or equipment. If the machine requires an air supply, ensure the compressor is on and pressure meets the machine requirements.

2. Start/Home

Turn power on the machine and control. The main breaker is located at the back of the machine. The machine power button is located in the upper-left corner on the control face.

3. Load Tools

Load all tools into the tool carousel in the order listed in the CNC program tool list.

4. Mount the Part in the Vise

Place the Part to be machine in the vise and tighten.

5. Set Tool Length Offsets

Set Tool Length Offsets For each tool used in the order listed in the CNC program, jog the Tools to the top of the part and then set the TLO.

6. Set Part Offset XY

Once the vise or other Part is properly installed and aligned on the machine, set the fixture offset to locate the part XY datum.

7. Load CNC Program

Load your CNC program into CNC machine control using USB flash memory, or floppy disk.

8. Dry Run

Run the program in the air about 2.00 in. above the part .

9. Run Program

Run the program, using extra caution until the program is proven to be error-free.

10. Adjust Offsets as Required

Check the part features and adjust the CDC or TLO registers as needed to ensure the part is within design specifications.

11. Shut Down

Remove part from the vise and tools from the spindle, clean the work area, and properly shut down the machine. Be sure to clean the work area and leave the machine and tools in the location and condition you found them.

UNIT TEST

1. Please list the CNC setup and operation process steps.

2. Describe each process.

23

Unit 6: Haas Control

OBJECTIVE

After completing this unit, you should be able to:

- Identify the Haas Control.
- Identify the Keyboard.
- Describe Start/Home Machine procedure.
- Describe Door Override procedure.
- Describe Load Tools procedure.
- Describe Tool Length Offset (TLO) for each tool.
- Verify part zero offset(XY) using MDI.
- Describe the setting tool offset.
- Verify Tool Length offset using MDI.
- Describe the procedure of load CNC program.
- Describe the procedure of save CNC program.
- Explain how to run CNC program.
- Describe the use of cutter diameter compensation.
- Describe the shut down program.

Haas Control

The Haas control is shown in Figures 18 and 19. Familiarize yourself with the location of buttons and controls. Detailed instructions on the following pages show how to operate the control.

Figure 1. Haas CNC Mill ControlHaas Keyboard

Keyboard

Keyboard keys are grouped into these functional areas:

1. Function Keys

2. Cursor Keys

3. Display Keys

4. Mode Keys

5. Numeric Keys

6. Alpha Keys

7. Jog Keys

8. Overrides Keys

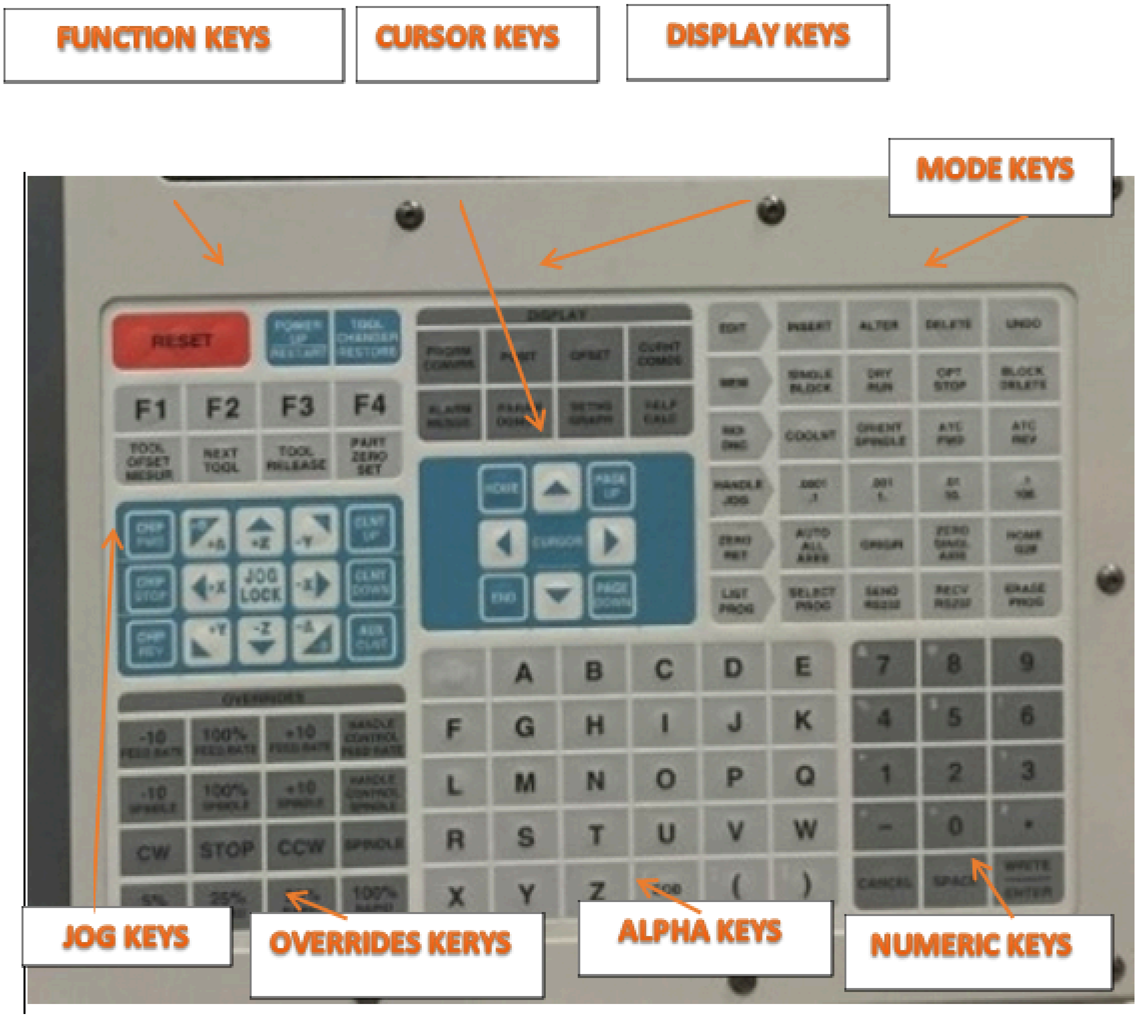


Figure 2. Haas CNC Control Buttons/ Keyboard keys

Start/Home Machine

Checklist:

1. Work Area: Made sure work area is Clear
2. Main Breaker: Turn On
3. Air Supply: Turn On Air to Correct pressure (at least 70PSI for tool changer to operate)
4. POWER ON: Press Green Button
5. Ensure Emergency Stop is not tripped. If it is, twist red knob right to release.
6. Wait until message 102 SERVOS OFF appears before proceeding.
7. RESET

8. Power On Restart

9. Ensure doors are closed and work area is clear.

10. Allow all machine axes to home before proceeding

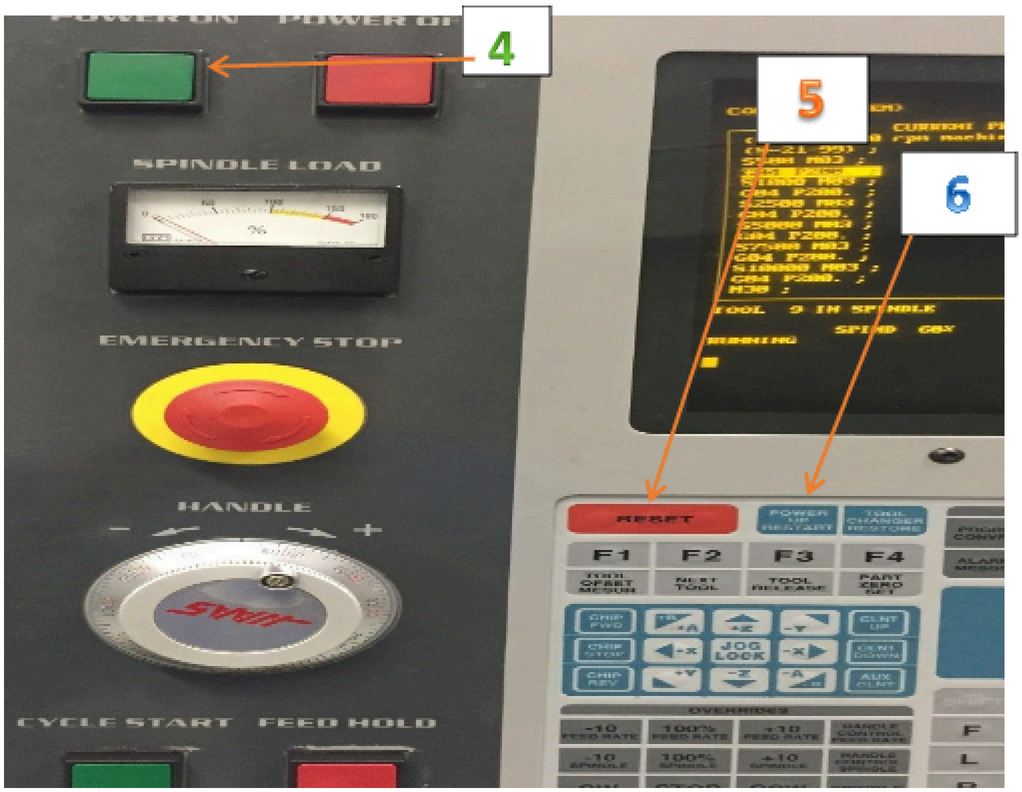


Figure 3. Start/Home Machine

Door Override

1. Mem: Select and Press Mem.

2. Setting Graph: Select and Press Setting Graph

3. Enter 51

4. Cursor: Press down arrow key and then the right arrow key to turn off

5. Write/Enter: Select and Press Write/Enter

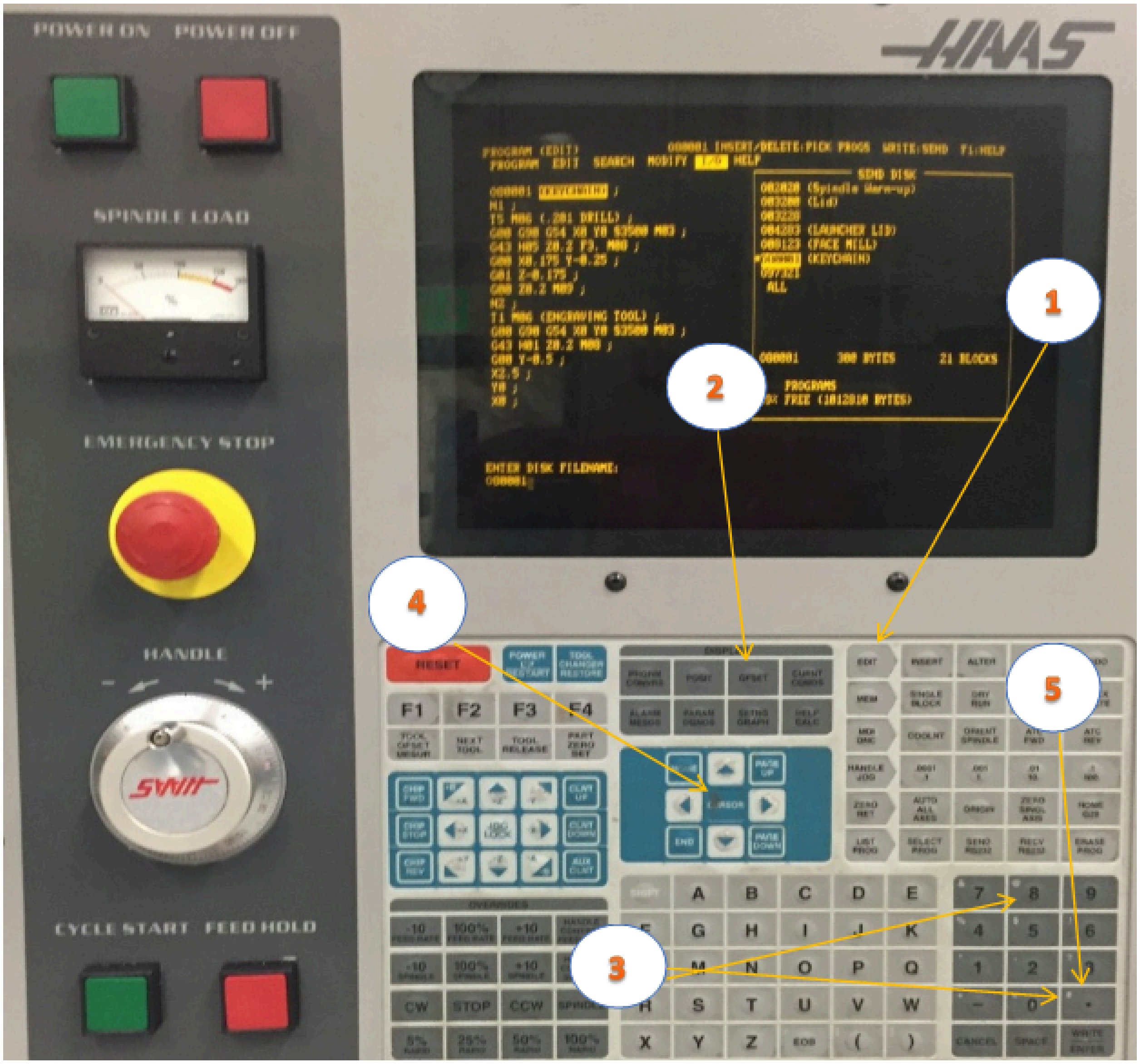


Figure 4. Door Override

Load Tools

Checklist:

1. MDI/DNC Key: Press the MDI/DNC button.

2. Tool Number:

- For example, to position the tool changer to T1,
- Press the T and then the 1 buttons.

3. ATC FWD: Press the ATC FWD button.

- Tool carousel will index to T1 position.

4. Position Tool in Spindle

- Do not grip by tool cutting flutes!
- Ensure tool taper is clean.
- Grip tool holder below V-flange to prevent pinching.
- Push tool into spindle.
- Ensure "dogs" on spindle line up with slots on tool holder.

5. Tool Release: Press the Tool Release button

- Machine will blow air thru spindle to clear debris.
- Gently push the tool upward and then release the Tool Release button.
- Ensure tool is securely gripped by spindle before releasing it.

6. Repeat steps 2-5 until all tools are loaded.

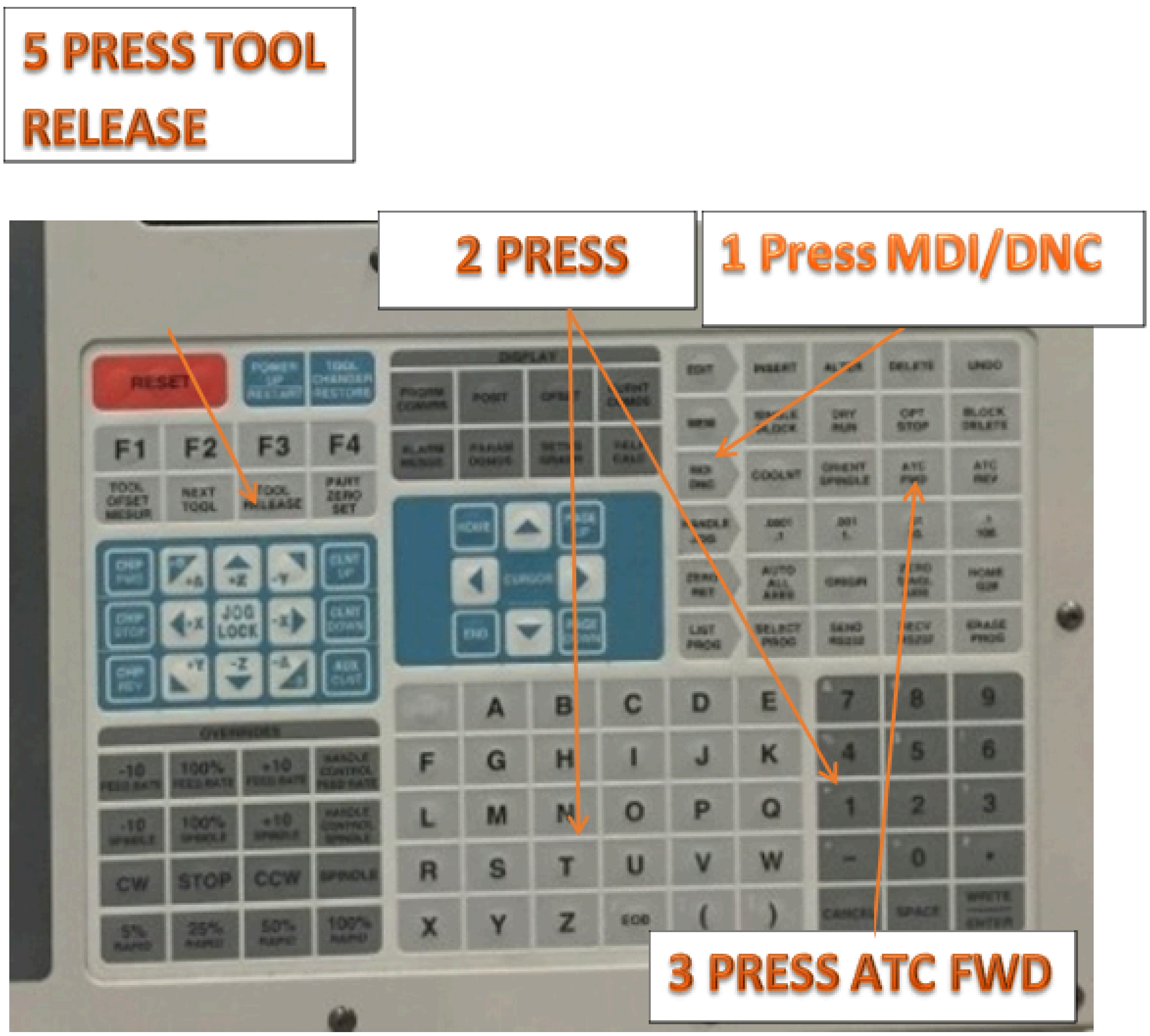


Figure 5. Load Tools

Setting Offsets

To machine a part accurately, the mill needs to know where the part is located on the table and the distance from the tip of the tools to the top of the part (tool offset from home position).

To manually enter offsets:

1. Choose one of the offsets pages.

2. Move the cursor to the desired column.

3. Type the offset value you want to use.

4. Press (ENTER) or (F1). The value is entered into the column.

5. Enter a positive or negative value and press (ENTER) to add the amount entered to the number in the selected column; press (F1) to replace the number in the column.

Jog Mode

Jog mode lets you jog the machine axes to a desired location. Before you can jog an axis, the machine must establish its home position. The control does this at machine power-up.

To enter jog mode:

1. Press (HANDLE JOG).

2. Press the desired axis (+X, -X, +Y,-Y, +Z, -Z).

3. There are different increment speeds that can be used while in jog mode; they are (.0001), (.001), (.01) and (.1). Each click of the jog handle moves the axis the distance defined by the current jog rate. You can also use an optional Remote Jog Handle (RJH) to jog the axes.

4. Press and hold the handle jog buttons or use the jog handle control to move the axis.

Set Tool Length Offset (TLO)

Checklist:

1. Handle Jog Mode: Select the Handle Jog button.

   - This sets machine to be controlled by the hand wheel.

2. Jog Increment: .01

   - This sets the job increment so each click of the hand wheel moves the tool .01 inches in the jog direction.

3.Jog Direction: Press the Z button

   - This sets the tool to move in Z when the jog handle is moved.

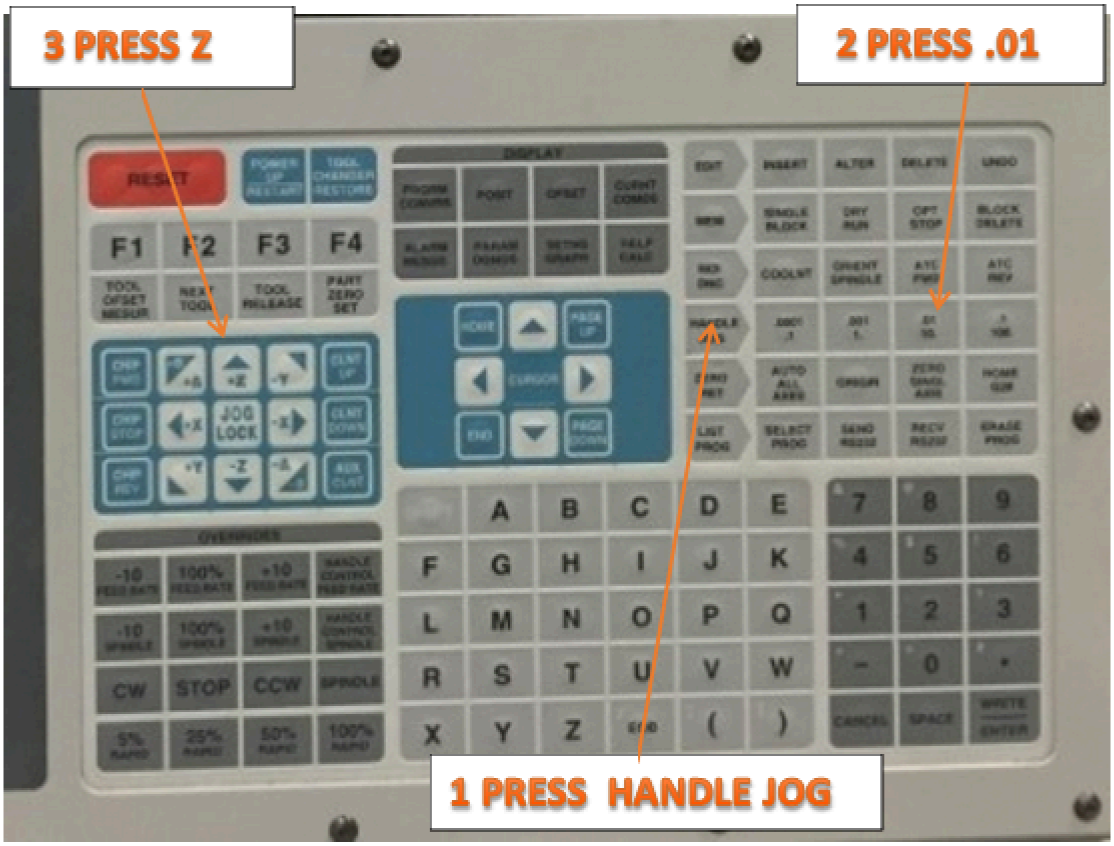


Figure 6. TLO

4. Offsets: Select and Press Offset

- Tool Offset page displays.

5. Cursor Arrows: Align for active tool

- Use the Up-Dn cursor keys (if needed) to move the highlighted bar on the graphics display over the offset values for the currently active tool.

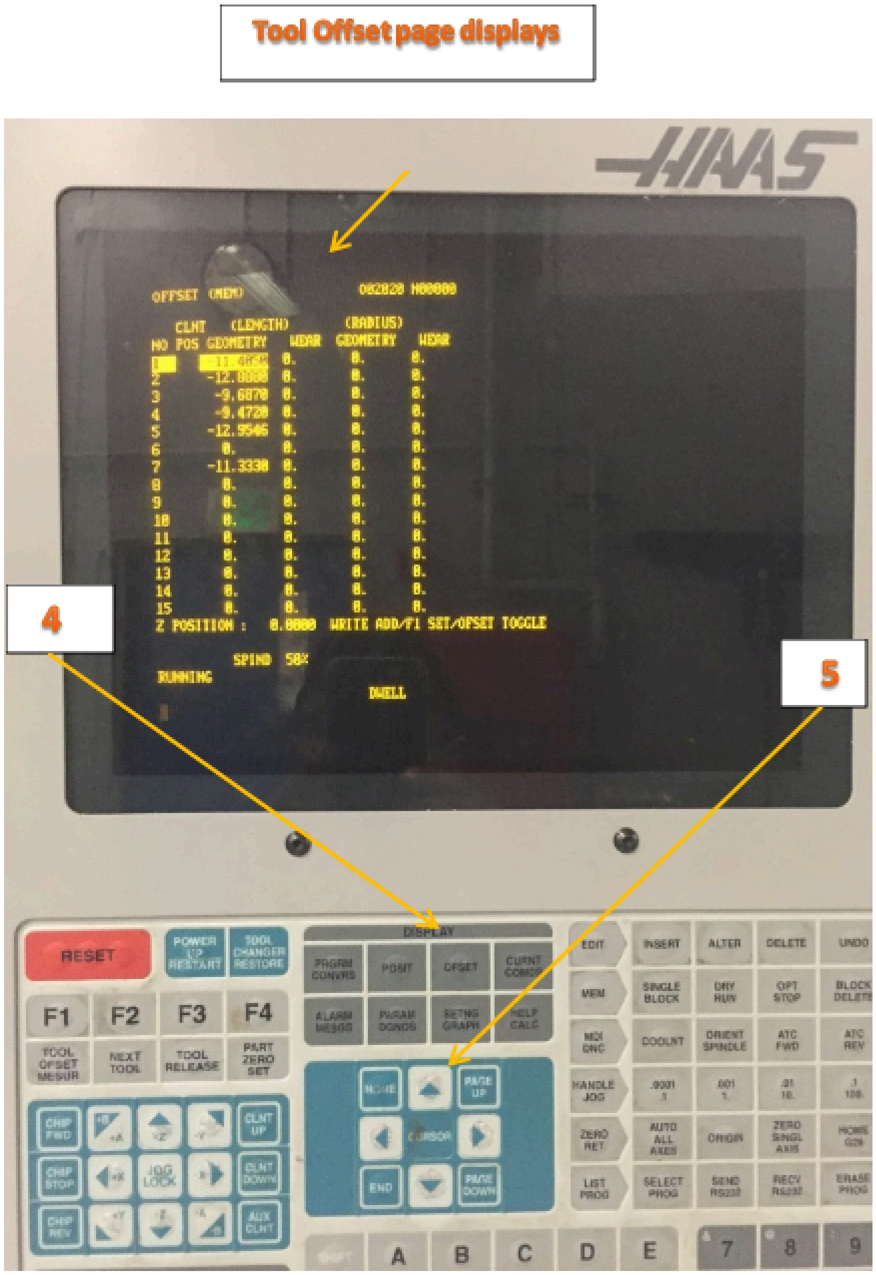


Figure 7. Offset

6.Use 1-2-3 Block to Set Tool Length

- Jog so tool is below top block.
- Apply slight pressure to block against tool. Use Jog Wheel to raise tool until the block just slides underneath it.
- Move block out of way and then move tool back down .01 inches below top of block.

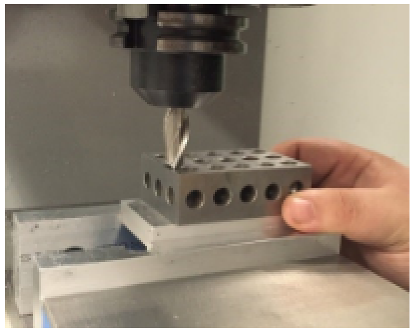


Figure 8

7. Jog Increment: .001

- Reduce jog increment and use jog handle to raise tool in .001 increments until it just slides under the block again.

8. Tool Offset Measure: Select and Press Tool Offset Measure

- This causes the control to enter the current position of the tool in the length offset register.
- Make sure the tool length number updates before proceeding.

9. Next Tool: Select and Press Next Tool

- This causes the current tool to be put away and the next tool to be loaded.
- Repeat steps 1 thru 9 until all tools are set.

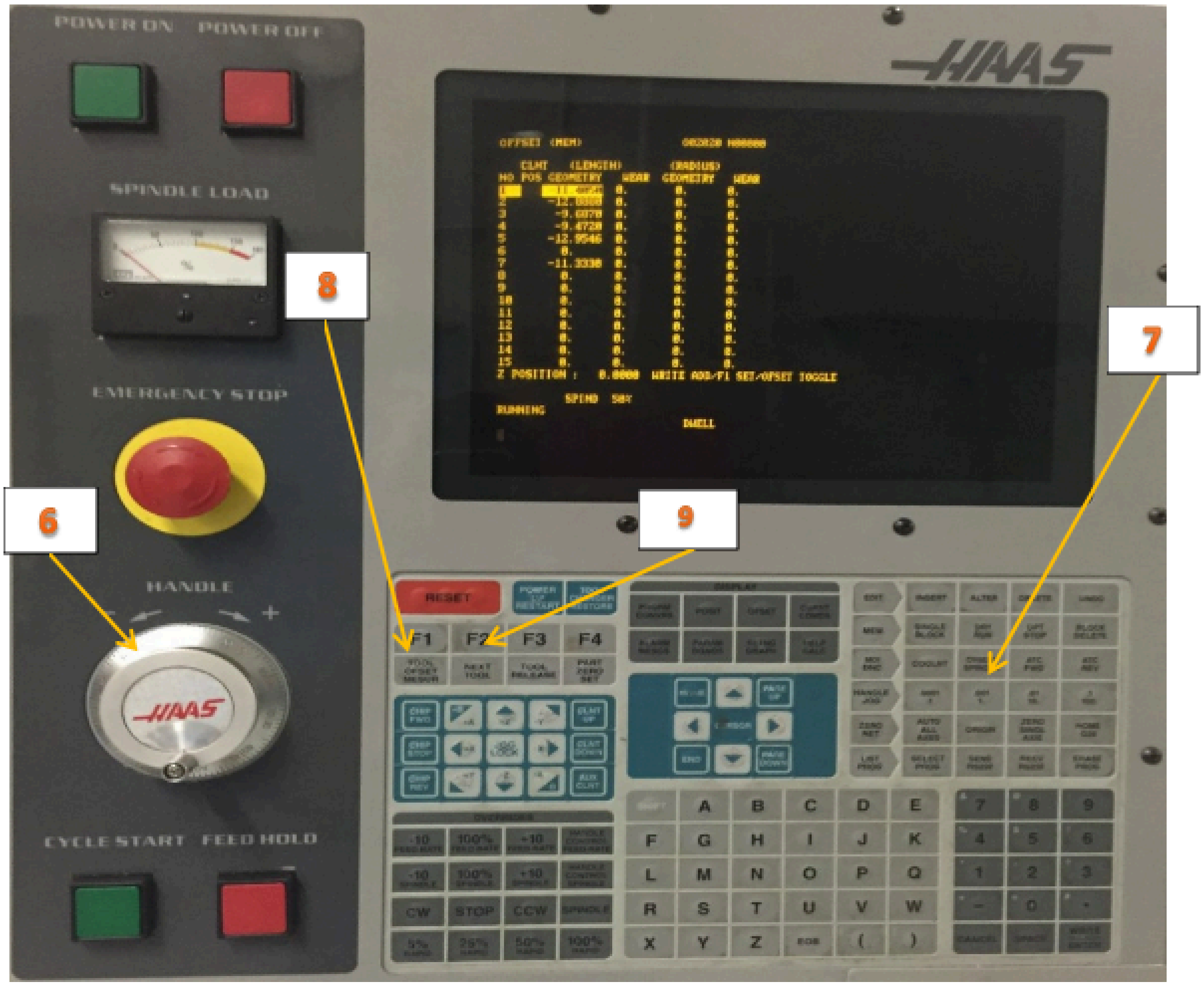


Figure 9. Offset Display Page

NOTE:

Setting tools requires manually jogging the machine with hands in the machine work envelope. Use extreme caution and observe the following rules:

• The spindle must be off.

• Never place your hand between the tool and the workpiece.

• Ensure the correct axis and jog increment are set before jogging.

• Move the handle slowly and deliberately. Keep your eyes on your hands and the tool position at all times.

• Never allow anyone else to operate the control when your hand is in the work area.

How to Access MDI

The Manual Data Input Mode (MDI) is one of the modes your CNC machine can operate in. The idea is to enter G-Codes or M-Codes on a line which are executed immediately by the machine–you don't have to write an entire g-code program when a line or two will suffice. MDI offers a lot of power while requiring very little learning. You can even use MDI commands to machine your part. With MDI, CNC can be quick and dirty just like manual machining.

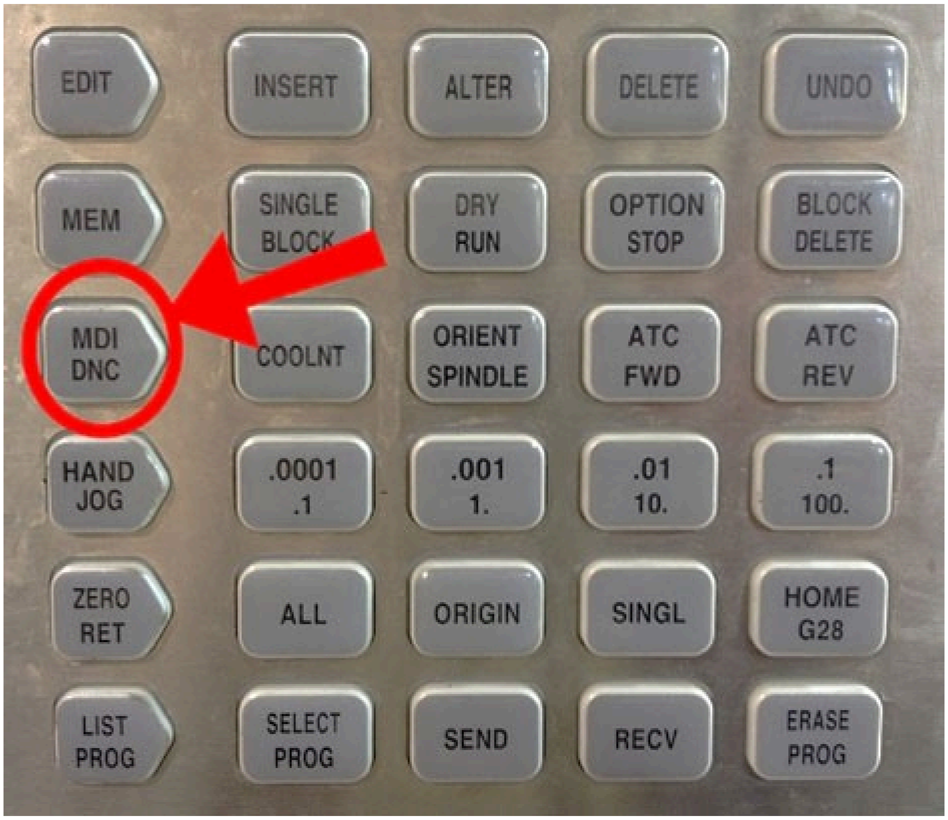


Figure 10. MDI

Press the MDI key on your CNC control panel to go to MDI mode.

For Example:

Press MDI/DNC

Erase Prog: Select and Press (to clear any commands)

Enter S1200 M03 (Spindle Speed 1200 RPM, On CW)

Setting Part Zero Offset OffsetXY(Using Edge Finder)

Checklist:

1. MDI/DNC Key: Select and Press MDI/DNC

2. Erase Prog: Select and Press Erase Prog to clear any commands

3. Turn on Spindle Speed: S1200

   - Press S1200 M03(Input): Select: Write/Enter

4. Cycle Start: Select

- Spindle will start CW at 1200 RPM

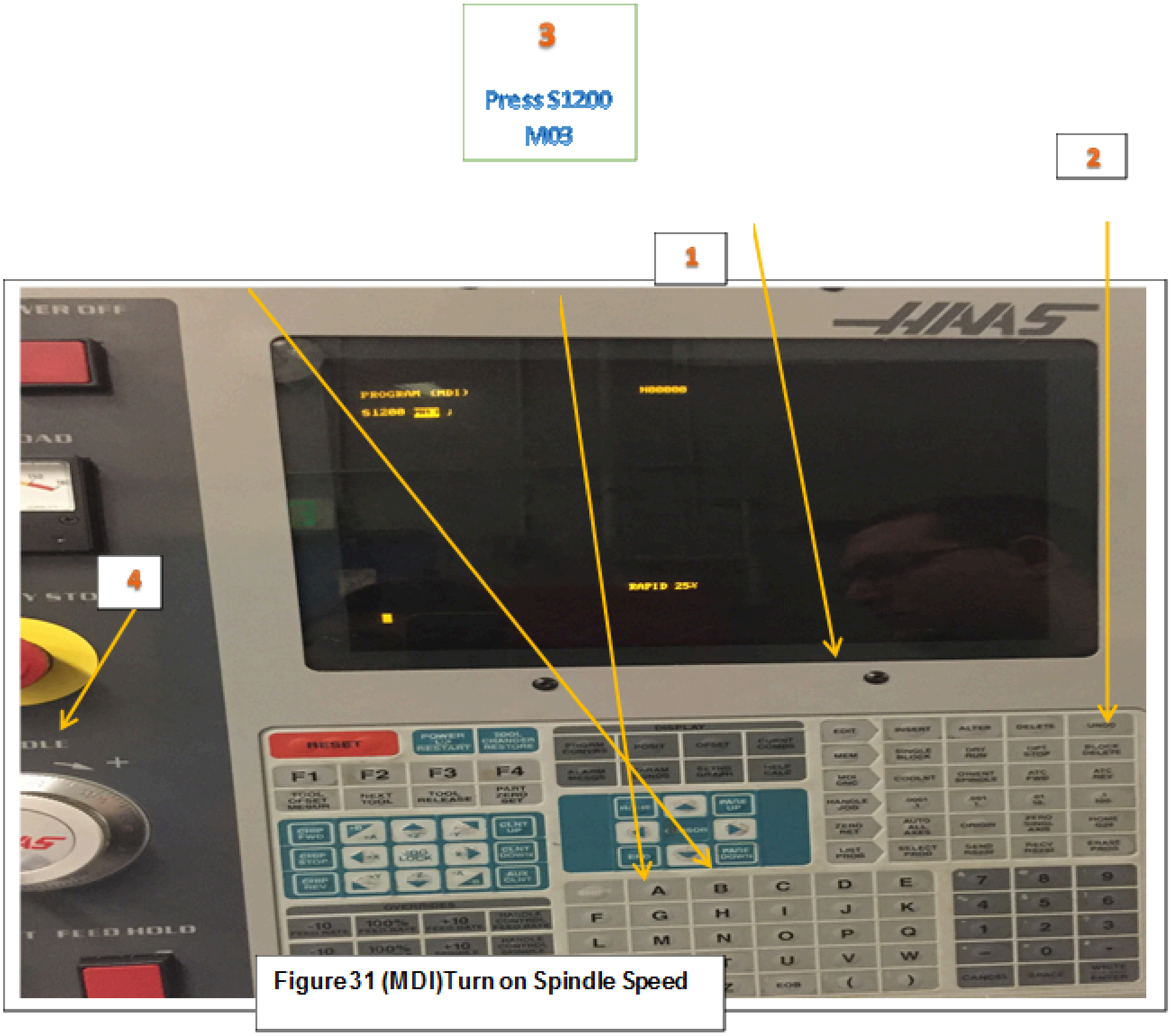
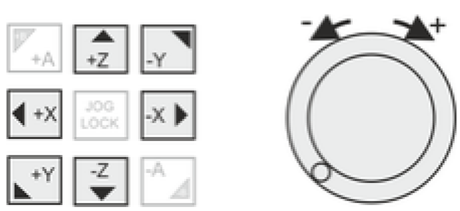


Figure 11. Turn on Spindle Speed (MDI)

5. Handle Jog: Select Handle Jog and Jog Increment: .01

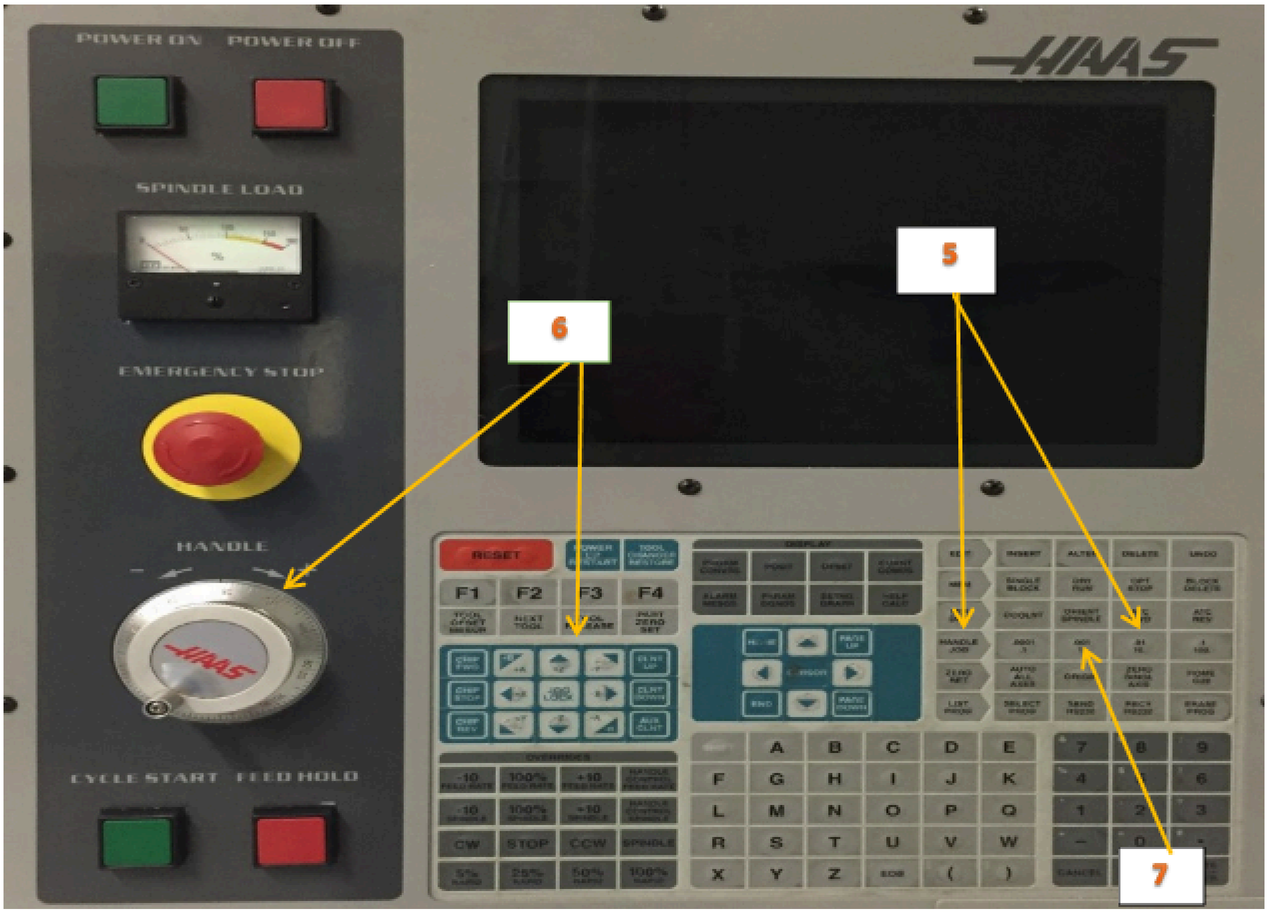
6. Jog Handle: As Needed

    - Select jog direction and use handle as required to place edge finder stylus alongside the left part edge.



7. Jog Increment: .001

- Move edge finder slowly until it just trips off center as shown below.
- This places the center of the spindle exactly .100 from the part edge



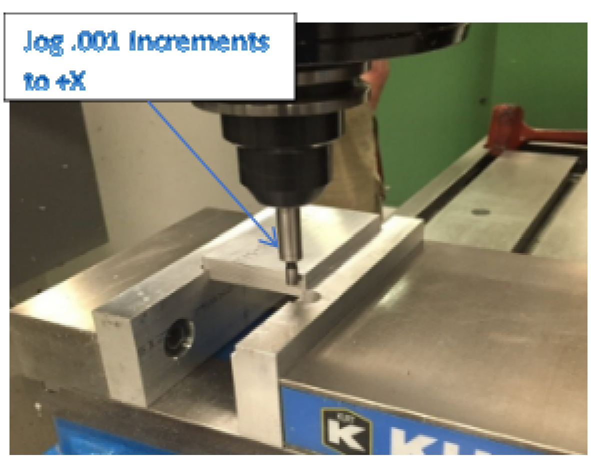


Figure 12. Just before tripping

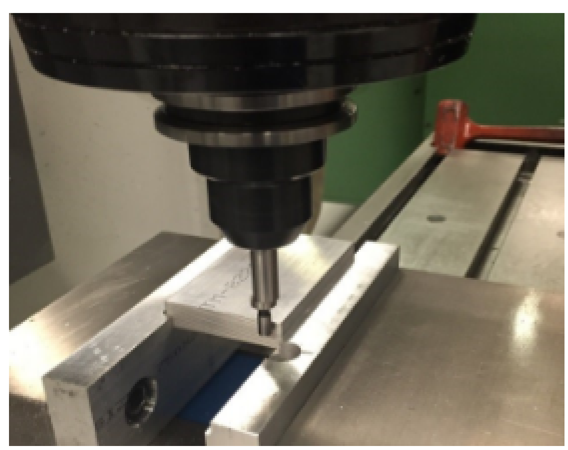
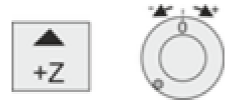


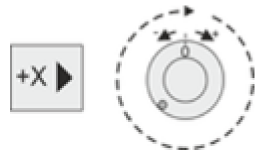
Figure 13. Just after tripping

8 .Jog Handle: Retract in Z



• Jog straight upward in Z until edge finder is above part and jog handle reads zero on the dial.

9. Jog Handle: Set jog direction to +X and rotate handle one full turn clockwise.



Since the control is in .001 increment mode, rotating the dial exactly one full turn places the center of the spindle directly over the left part edge.

10. Offset Page: Select and Press

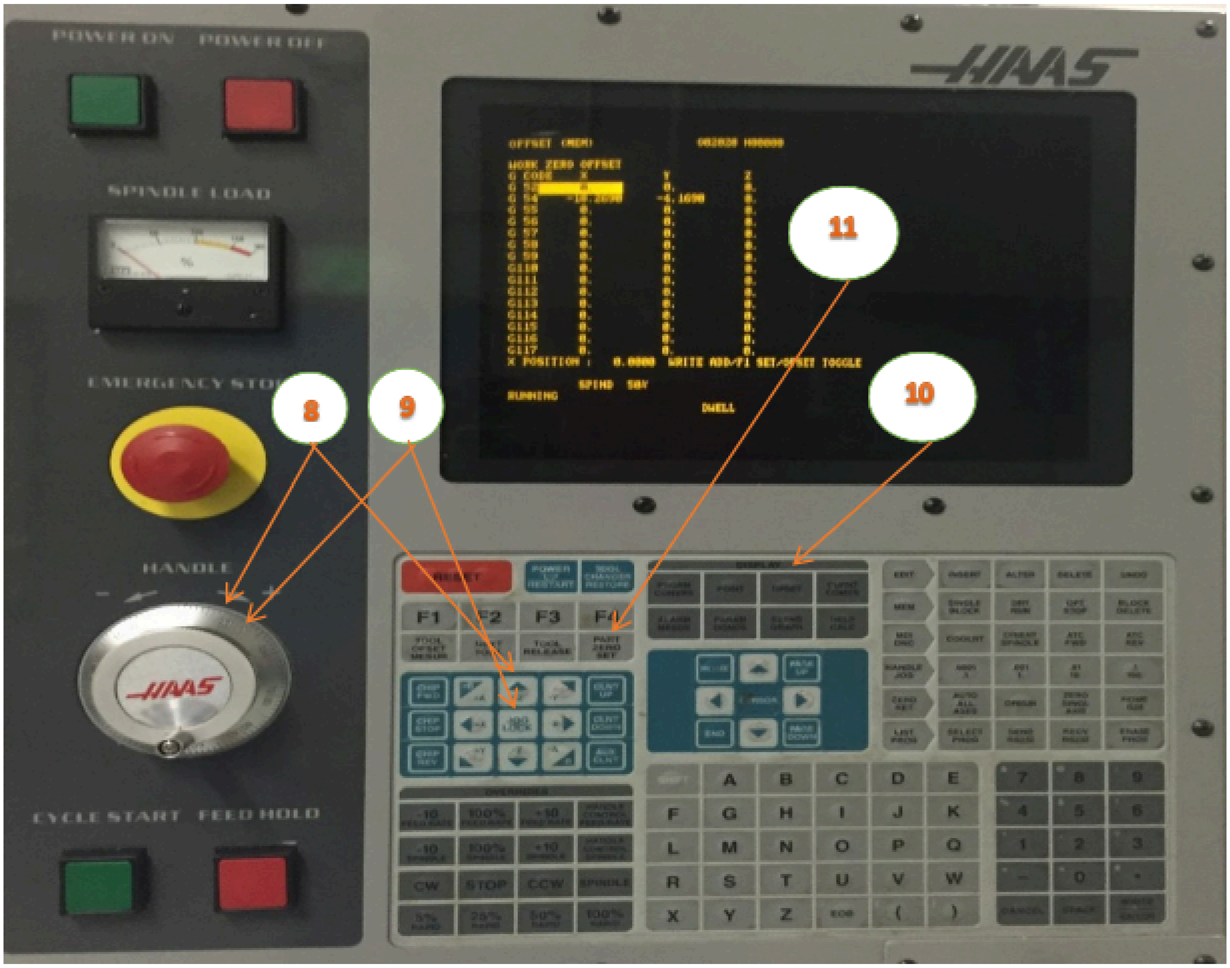
• Select Offset button and PgUp/PgDn buttons until Work Zero Offset page appears. Use Arrow keys to highlight G54 (or whatever fixture offset is to be set).

11. Part Zero Set: Press Part Zero Set

• This sets the G54 X value to the current spindle position.

12. Spindle Stop: Press spindle stop

13. For Setting the Y-axis repeat steps 6–11



Using MDI to verify Part zero offset

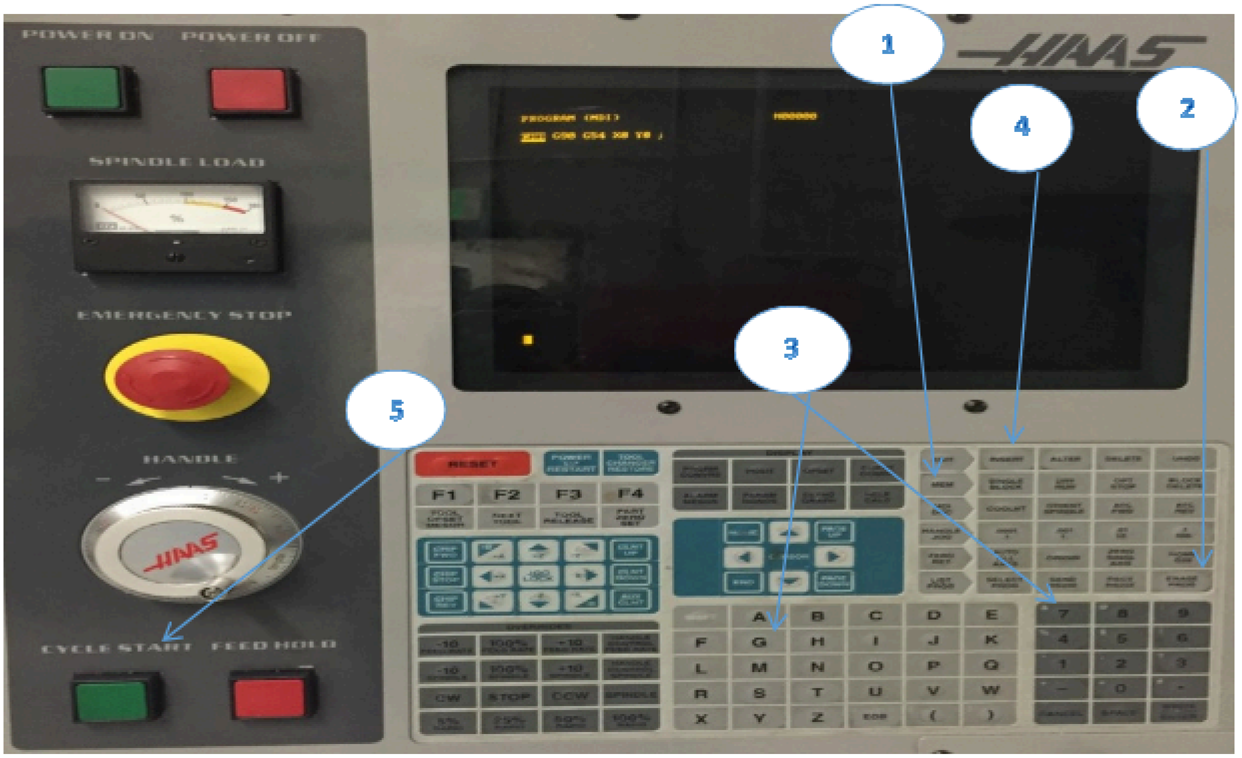
1. MDI/DNC Key: Select and Press MDI/DNC

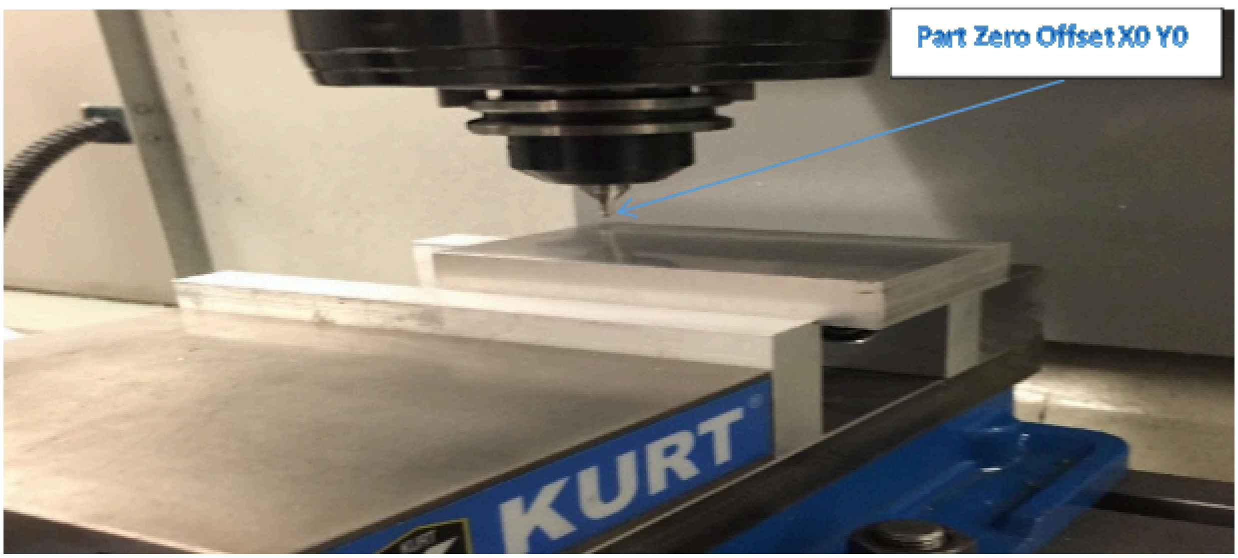
2. Erase Prog: Select and Press (to clear any commands)

3. Enter G00 G90 G54 X0 Y0

4. Insert: Select and Press Insert

5. Cycle Start: Select and Press Cycle Start





Part Zero Offset X0 Y0 (MDI)

- To shift the datum RIGHT in relation to the machine operator, ADD a shift amount to the offset X-value. For example, to shift X+.1, input .1 WRITE/ENTER.
- To shift the datum CLOSER to the machine operator, SUBTRACT a shift amount from the offset Y-value. For example, to shift Y-.1, input -.1 WRITE/ENTER

Setting Part Zero Offset XY(Using Mechanical pointer)

In order to machine a part, the mill needs to know where the part is located on

the table. You can use an edge finder, an electronic probe, or many other tools and methods to establish part zero. To set the part zero offset with a mechanical pointer:

1. Place the material (1) in the vise and tighten.

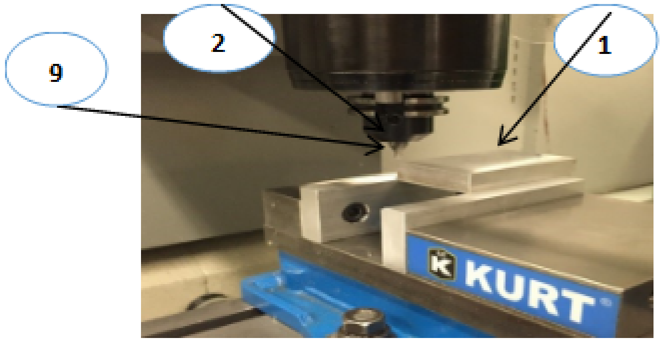


Figure 14. Setting Part Zero XY (Using Mechanical pointer)

2. Load a pointer tool (2) in the spindle.

3. Press (HANDLE JOG) (E).

4. Press (.1/100.) (F) (The mill will move at a fast

speed when the handle is turned).

5. Press (+Z) (A).

6. Handle jog (J) the Z-Axis approximately 1″ above the part.

7. Press (.001/1.) (G) (The mill will move at a slow speed when the handle is turned).

8. Handle jog (J)the Z-Axis approximately. 0.2″ above the part.

9. Select between the X and Y axes (I) and handle jog (J) the tool to the upper left corner

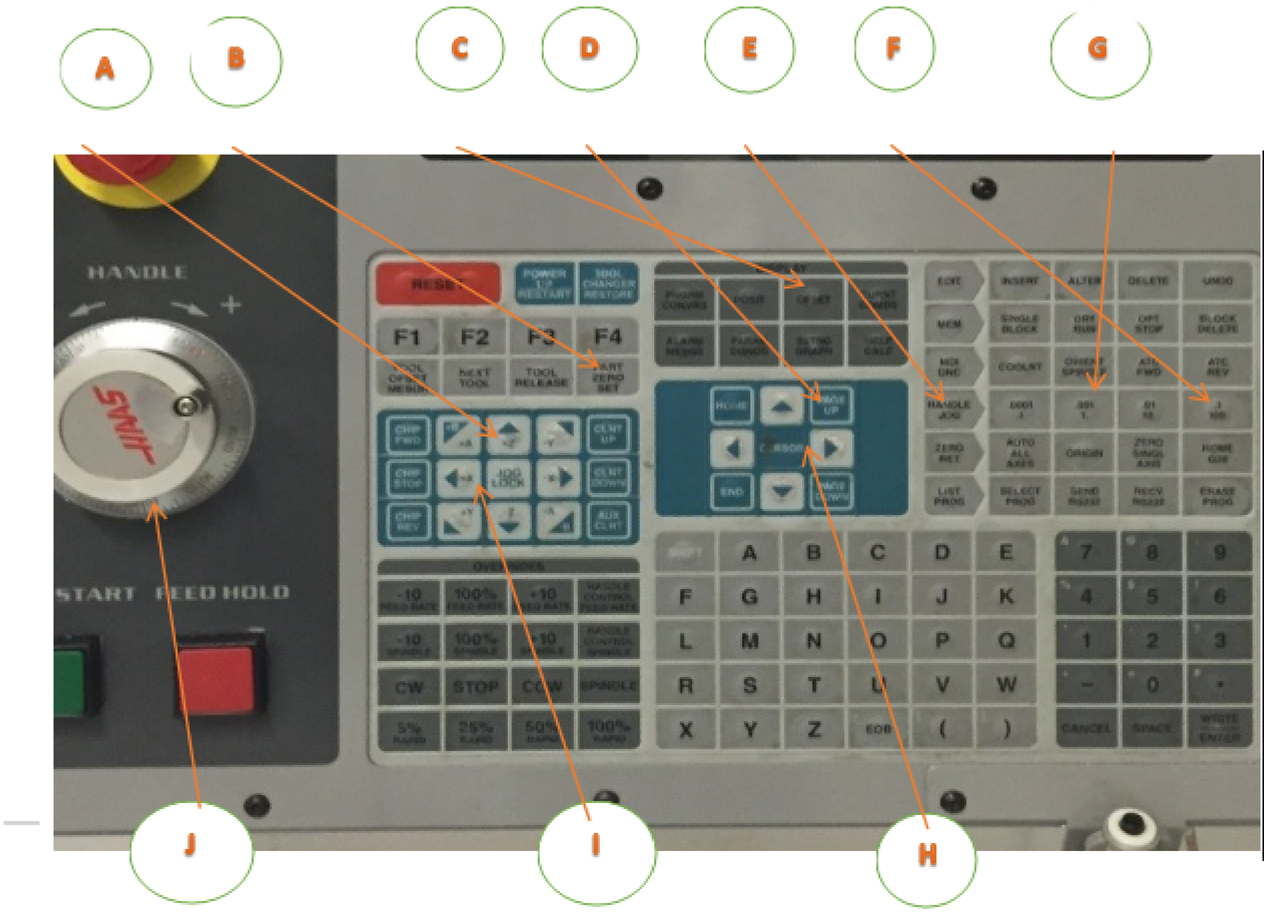
of the part (See Figure 36 above (9).

10. Press (OFFSET) (C) until the Active Work Offset pane is active.

11. Cursor (H) to G54 X-Axis column.

12. Press [PART ZERO SET] (B) to load the value into the X-Axis column. The second

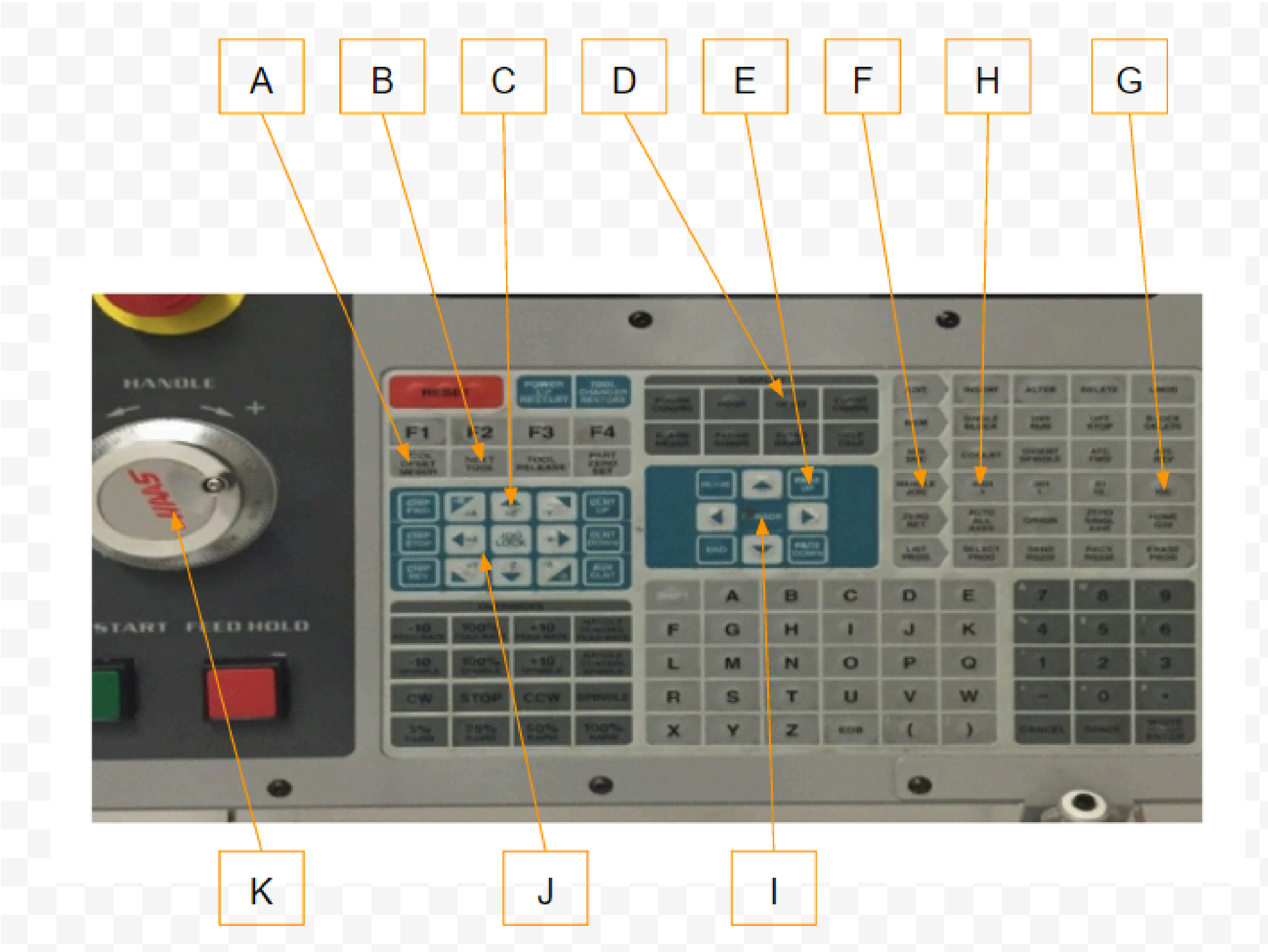
press of [PART ZERO SET] (B) loads the value into the Y-Axis column.



Setting Tool Offset

The next step is to touch off the tools. This defines the distance from the tip of the tool to the top of the part. Another name for this is Tool Length Offset, which is designated as H in a line of machine code. The distance for each tool is entered into the Tool Offset Table.

Setting Tool Offset. With the Z Axis at its home position, Tool Length Offset is measured from the tip of the tool (1) to the top of the part (2). See Figure 15.



1. Load the tool in the spindle (1).

2. Press (HANDLE JOG) (F).

3. Press (.1/100.)(G) (The mill moves at a fast rate when the handle is turned).

4. Select between the X and Y axes (J) and handle jog (K) the tool near the center of the part.

5. Press (+Z) (C).

6. Handle jog (K) the Z Axis approximately 1″ above the part.

7. Press (.0001/.1) (H) (The mill moves at a slow rate when the handle is turned).

8. Place a sheet of paper between the tool and the work piece. Carefully move the tool down to the top of the part, as close as possible, and still be able to move the paper.

9. Press (OFFSET) (D).

10. Press (PAGE UP) (E) until you display the Program Tool Offsets window. Scroll to tool #1.

11. Cursor (I])to Geometry for position #1.

12. Press [TOOL OFFSET MEASURE] (A).

The next step causes the spindle to move rapidly in the Z Axis.

13. Press (NEXT TOOL) (B).

14. Repeat the offset process for each tool.

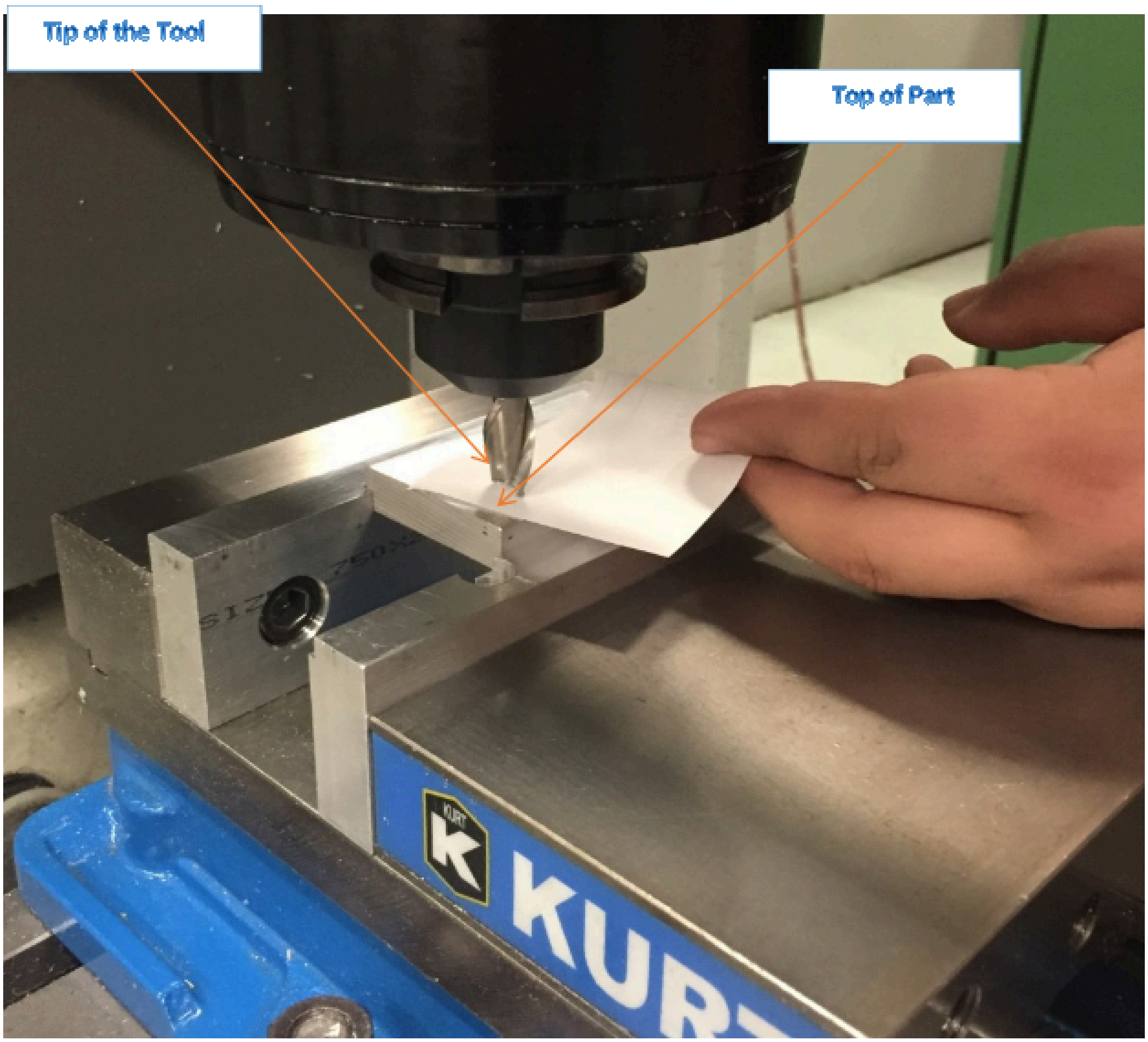


Figure 15. Setting Tool Offset (sheet of paper)

Using MDI to verify Tool Length offset

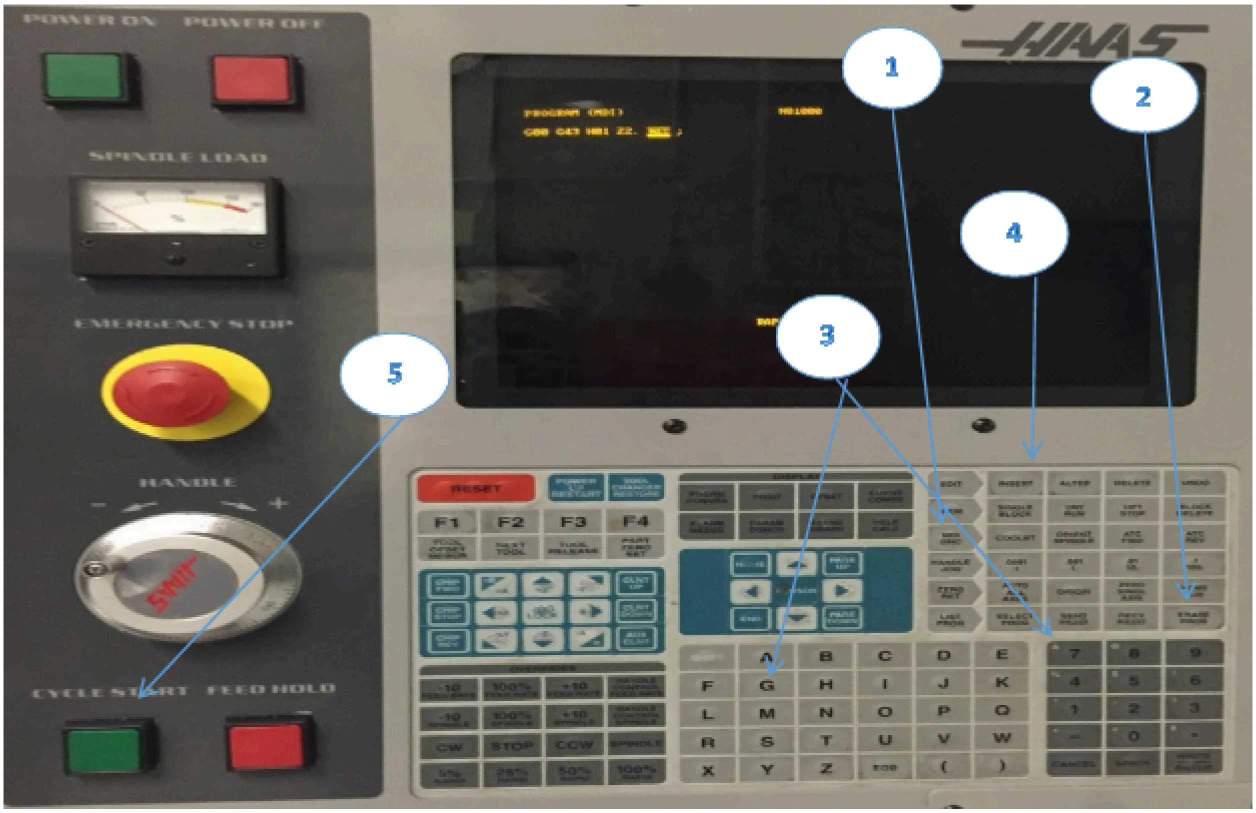
1. MDI/DNC Key: Select and Press MDI/DNC

2. Erase Prog: Select and Press (to clear any commands)

3. Enter G00 G90 G43 H01 Z2.00

4. Insert: Select and Press Insert

5. Cycle Start: Select and Press Cycle Start



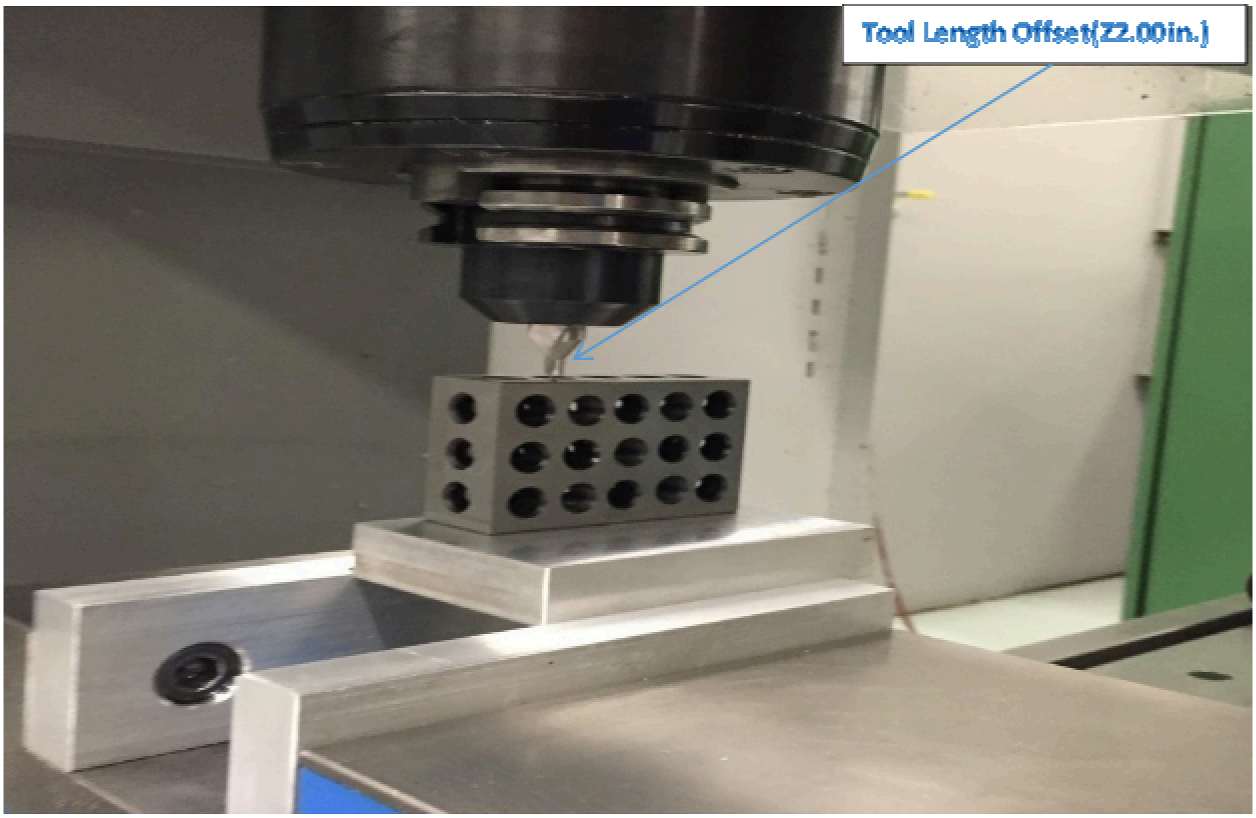


Figure 16. Tool length Offset (2.00 inch above part using 1 2 3 block to verify)

Load CNC Program

1. Edit: Select and Press Edit

2. F1: Select and Press F1

3. Cursor: Press the left arrow key to I/O and then the DN arrow key to move the highlight bar to Disk Directory.

4. Write/Enter: Select and Press Write/Enter

5. Cursor (Disk Directory): Press the DN arrow key to program to load.

6. Write/Enter: Select and Press Write/Enter

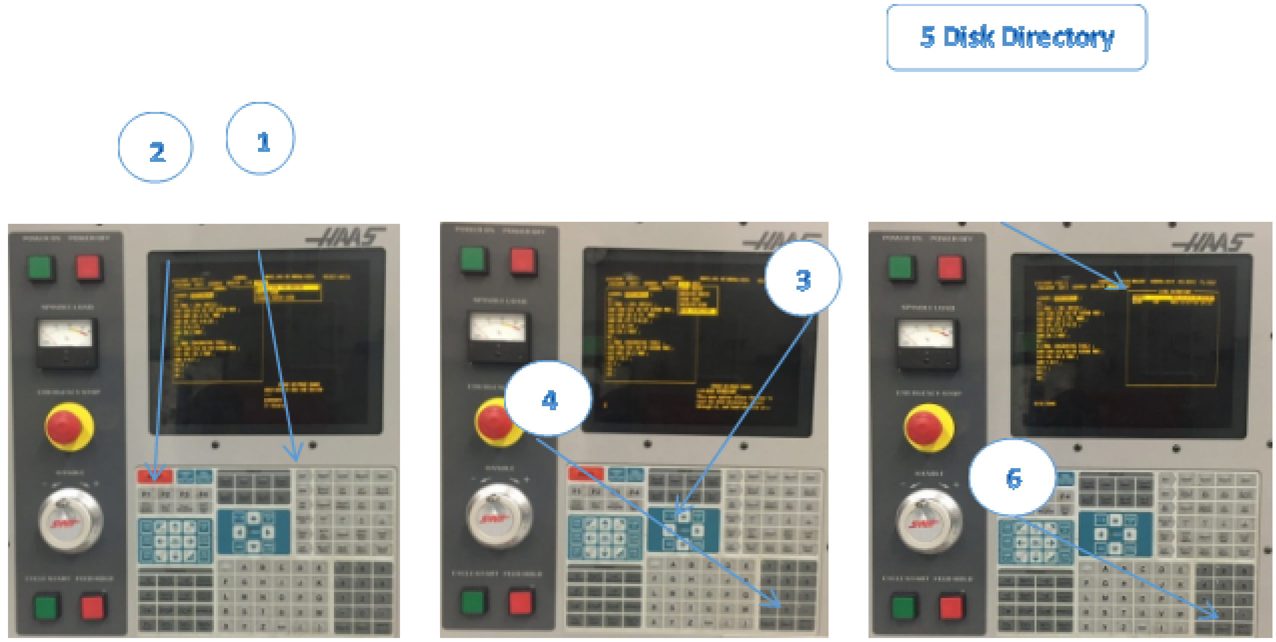


Figure 17. Load CNC Program

Save CNC Program

1. Edit: Select and Press Edit

2. F1: Select and Press F1

3. Cursor: Press the left arrow key to I/O and then the DN arrow key to move the highlight bar to Send Disk.

4. Write/Enter: Select and Press Write/Enter

5. Enter Disk File Name: O80001

6. Write/Enter

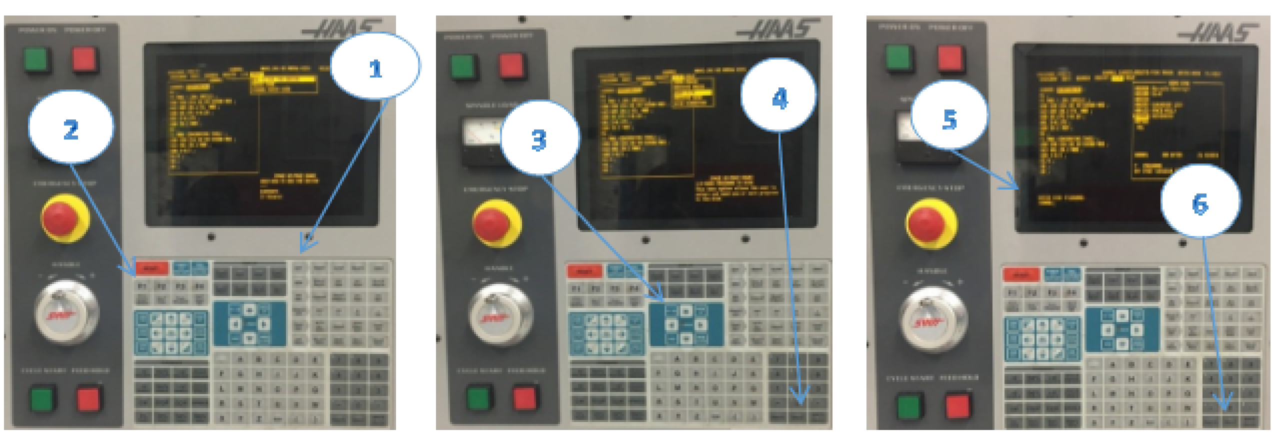


Figure 18. Save CNC Program

Run CNC Program

This is the preferred process for running a new program. Once a program is proven, all feed rates can be set to 100% and single block mode can be set to off.

Dry Run Operation

The machine executes all motions exactly as programmed. Do not use a work piece in the machine while dry run is operating. The Dry Run function is used to check a program quickly without actually cutting parts.

To select Dry Run:

1. While in MEM or MDI mode, press (DRY RUN).

When in Dry Run, all rapids and feeds are run at the speed selected with the jog speed buttons.

2. Dry Run can only be turned on or off when a program has finished or [RESET] is

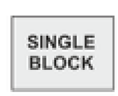
pressed. Dry Run makes all of the commanded X Y Z moves and requested tool

changes. The override keys can be used to adjust the Spindle speeds.

Checklist:

1. Pre-Start

- Ensure vise or fixture is secure and that you have a safe setup.
- There should be no possibility that the work holding will fail to perform as required.
- Remove vise handles.
- Clear the work area of any tools or other objects.
- Close the machine doors.



- Turn Single Block Mode On.
- Press Rapid Feedrate -10 button eight times to set rapid Feed Rate Override to 20% of maximum.

2. Start

- Place one hand on Feed Hold button and be ready to press it in case there are any problems.
- Press Cycle Start Button.

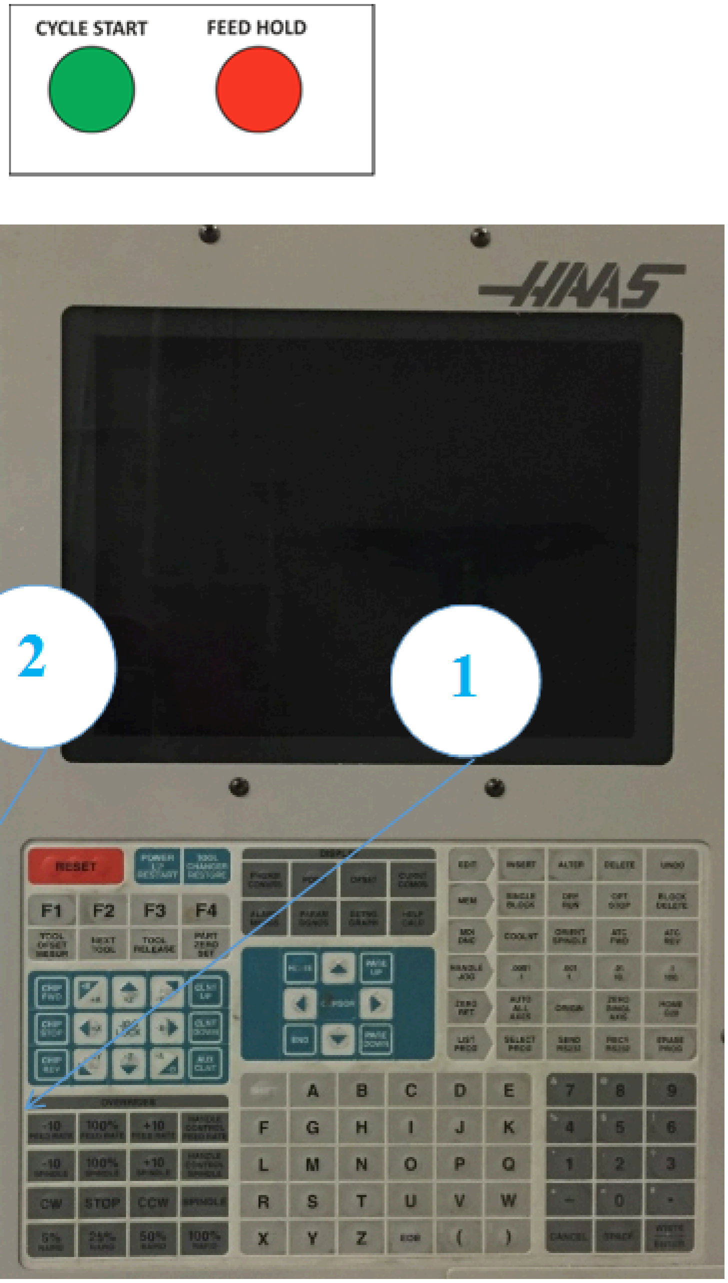


Figure 19. Run CNC Program

A common error is setting the Fixture or Tool Length offset incorrectly. When running a program for the first time, set the machine to Single block mode. Reduce rapid feed rate to 25%, and proceed with caution. Once the tool is cutting, turn off single block mode and let the program run. Do not leave the machine unattended, and keep one hand on the feed hold button. Listen, watch chip formation, and be ready to adjust cutting feed rates to suite cutting conditions.

Adjusting CDC Offsets

Machining operations that use Cutter Diameter Compensation (CDC: G41/G42) can be adjusted to account for tool wear and deflection. Measure across a finished feature on part and compare it with the desired value. Subtract the Actual from the Target sizes and enter the difference into the CDC register on the control for that tool. For example:

(Target Feature Size: 2.5000) – (Actual Feature Size: 2.5150) = Wear Value: -0.0150

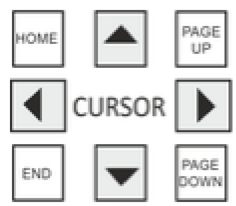
The tool path will now be compensatedfor the size difference. Running the same operation again should result in the feature being exactly the target size

Wear compensation is used only on contour passes. It is not used for face milling, 3D milling, or drill cycles. Select the Wear Compensation option in your CAD/CAM software and, if needed, set a Tool Diameter Wear value as shown above. When used, the wear value is always a negative number. Always set Tool Diameter Geometry to zero for all tools since CAD/CAM software already accounts for the tool diameter by programming the tool center line path.

Checklist:

1. Offset Page: Select and Press Offset

2. Adjust Diameter Offset: Select and Enter Value



- Pg Up/Dn to highlight the tool to be adjusted.
- Enter a value using the numeric keypad.
- Example Tool Diameter Wear value as shown above: -0.0150
- Select and Press Writ/Enter

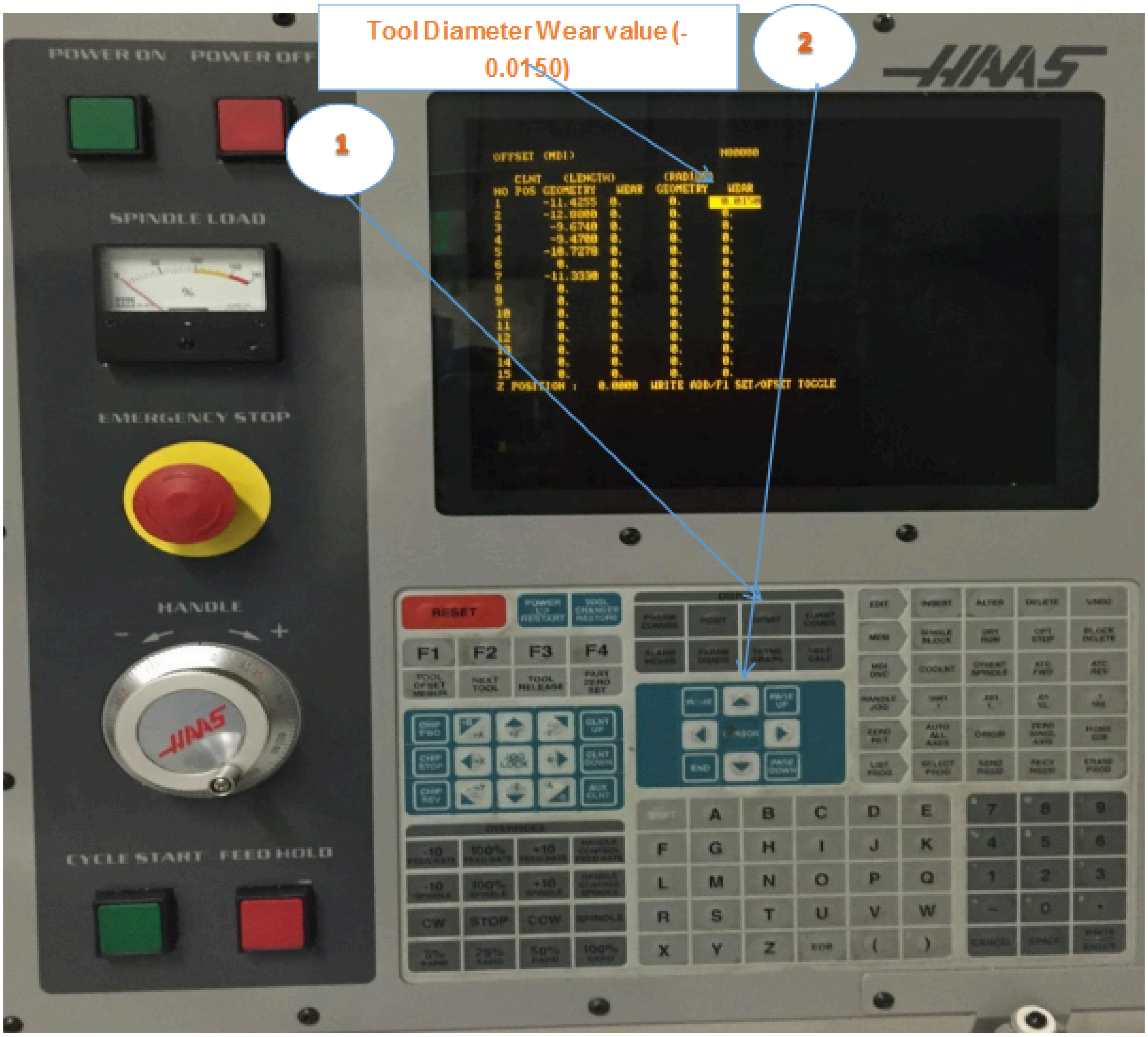


Figure 20. Adjusting CDC Offsets

Shut Down CNCMachine

Checklist:

1.Remove tool from spindle:

- Enter the number of an empty tool carousel.
- Select ATC FWD

2. Jog Machine to Safe Area:

- Select Jog

3.Shut Down Button: Press POWER OFF

Post Power-Down Checklist:

• Wipe spindle with a soft clean rag to remove coolant and prevent rusting.

• Put away tools.

• Clean up work area.

• Always leave the machine, tools, and equipment in the same or better condition than when you found them.

It is important to clean the machine after each use to prevent corrosion, promote a safe work environment, and as a professional courtesy to others. Allow at least 15-30 minutes at the end of each class for cleaning. At the very least, put away all unused tools and tooling, wash down the machine with coolant, remove standing coolant from the table, and run the chip conveyor.

UNIT TEST

1. Please lists eight functional areas of HAAS Keyboard.

2. Describe how to Door override.

3. Explain how to load Tools.

4. Describe MDI and give one example.

5. Explain how to set part zero offset.

6. Please describe using MDI to verify part zero offset.

7. Explain how to set Tool Length offset

8. Please describe using MDI to verify Tool Length offset.

9. Describe how to save CNC program.

10. Please Explain the shut down procedure.

24

Unit 7: Mastercam

OBJECTIVE

After completing this unit, you should be able to:

- Describe save image.
- Describe the load image to Mastercam.
- Describe scaling and dragging or translate image.
- Describe Mastercam setup, stock setup, and Tool setting.
- Describe creating Toolpaths.

Mastercam

Raster To Vector Images Conversion:

This Worksheet walks you through how to import Raster images into Mastercam. Mastercam is a vector software, where most images for the internet and other sources are Raster Images. Raster images are made up of thousand of pixels of differing color, where vector images are image of line that use mathematical formulas to determine their shape. In this activity you will learn how to convert Raster images to Vector images that Mastercam can use.

1. Using the internet search for your desired Image. Logos or images with sharp color changer work best, try to avoid pictures taken with a camera. .jpg, .gif, .bmp are the file extension currently supported by Mastercam.

2. Try to get the larges image possible. When Using Google examine the images sizing information located under the image. The Larger the number the better.

3. Then navigate to the raw picture by clicking on the image.

4. Click on See full size image.

5. RIGHT CLICK on the image and select Save Image As…

6. Save your raster image to a location that you know.

7. Launch Mastercam 2017

8. Once it loads Press ALT-C or Run User Application (under Setting) a window will open called Chooks. This is a collection of add on file to mastercam.

9. Navigate and click on Rast2Vec.dli

10. It many ask you of you want to keep current Geometry. Then, Click Yes

11.Navigate to your saved image and click on it.

12. The Black White conversion window will open. Slide the Threshold slider in the Black White conversion dialog box until the image on right shows the desired amount of detail.

13. When you like what you see. Then, click OK.

14. Rast2Vec window will popup no modification should be needed. Then, click OK.

15. When the Adjust Geometry opens. Then, click OK.

16. Your image should now be on the screen.

17. Click Yes to exit Rast2Vec.

Scaling and Dragging or Translate your image

After you have imported a image it often not in the right location or the proper size that you need. You need to Scale it up or down and Drag it to the position that you would like.

Scaling:

1. From The ToolBar >Xform>Scale

2. Then Select all the line that part of your image. The Line you select should turn yellow. You can use a window or select each line individually. If you pick a line you do not what just pick it again and it should change back form yellow.

3. Once you have selected your entire image then click on the Green Ball.

4. The Scale window will open. Change To MOVE and to PERCENTAGE, and then adjust the percent up or down until your image is the size you would like. After typing the new percentage number hit ENTER on the keyboard. This will preview that size change. Once the size is what you are looking for. Then, click OK.

Dragging or Translate

1. From the Tool Bar>xfrom>Drag or Translate

2. Once again select all the line that part of your image.

3. Once you have selected your entire image then click on the Green Ball.

4. Change from Copy To Move.

5. Click near your Image in the graphics screen. As you move your mouse the image will move.

6. Once the image is in the proper place LEFT Click again and it will place the image.

MASTERCAM SETUP

1. Turn on the Operation Manager window by pressing Alt-O

2. Click on Machine Type – Mill, then select the HASS 3X MINI MILL – TOOLROM.MMD-5. You should see the HASS 3X MINI pop up in the Operation Manager window.

STOCK SETUP

1. Click on the plus sign next to Properties to see the drop down list and click on stock setup. The Machine Group Properties dialog box should appear. Check the box next to Display to activate it, then click on the Bounding Box button. The Bounding Box dialog box should appear. Confirm that X, Y AND Z are all be set to zero then click OK.

2. Change the value of Z to stock you are using, then click OK. On an isometric view, you should see your part go from 2D to 3D with the image on the top surface.

TOOL SETTING

1. Click on Tool Setting. Enter a program number….Feed Calculation should be "From Tool." Under Toolpath Configuration, check "Assign tool number sequentially" and "Warn of duplicate tool numbers." Under Advanced Operation, check the "Override defaults with modal values" box and then check all three selections below it. Under Sequence #, change the start number to 10. Select the material by clicking on the Select button. Click on the drop down arrow associated with Source and select "Mill – library." From the list, select "ALUMINUM inch – 6061" then click OK. Click OK to exit the Machine Group Properties dialog box.

CREATING TOOLPATHS

1. Choose Toolpath – Contour. The chaining dialog box should appear. Select Window to choose your engraving elements, then click anywhere to establish an approximate start point (selection will change to yellow). Click OK to bring up the 2D Toolpaths – Contour dialog box.

2. Toolpath Type should automatically be set to Contour.

3. Click on Tool (below Toolpath Type) then click on Select Library Tool button. A Tool Selection dialog box will appear. To limit the list to a particular type of tool (ball end mills which will use for engraving), Click on Filter then select/de-select tool types so that only Endmill2 Sphere is highlighted. Click OK. Choose the 1/32 Ball Endmill followed by click OK. Change federate to 5.0 and Spindle Speed to 4000 then click OK.

4. Under Cut Parameters, Compensation Type should be off.

5. Under Lead In/Lead Out, Uncheck the Lead In/Out box as will not be using this feature.

6. Under Linking Parameters, Clearance should be set to 0.5, Retract should be 0.1, feed Plane should be 0.1, Top of stock should be O.0 and Depth should be -0.015. Then click OK.

7. Run Verify Selected Operations in the Operations Manager to see the toolpath.

UNIT TEST

1. What software is Mastercam is?

2. Please explain the Threshold Slider.

3. Lists file supported by Mastercam.

4. Describe scaling and dragging or translate image.

5. Describe stock setup.

6. Describe the tool setting.

7. Explain how to create Toolpaths.