Manufacturing Processes 4-5

. . .

LamNgeun Virasak

OpenOregon
Educational Resources

MANUFACTURING PROCESSES 4-5

MANUFACTURING PROCESSES 4-5

LAMNGEUN VIRASAK

Open Oregon Educational Resources

Contents

Introduction 1
Preface iii
Table of Contents v

Part I. Chapter 1: Milling Machines

1. Milling Machines 9
2. Unit 1: Tramming the Head 11
3. Unit 2: Speeds, Feeds, and Tapping 23
4. Unit 3: Sine Bar 31
5. Unit 4: Offset Boring Head 37

Part II. Chapter 2: Lathe Machines

6. Chapter 2: Lathe Machine 49
7. Unit 2: Speed and Feed 59
8. Unit 3: Chucks 65
9. Unit 4: Turning 69
10. Unit 5: Tapping 83
11. Unit 6: Lathe Threading 91

Part III. Chapter 3: Drill Presses

12. Chapter 3: Drill Press 105

Part IV. Chapter 4: Bandsaws

13. Chapter 4: Bandsaw 113

Part V. Chapter 5: Surface Grinders

14. Chapter 5: Surface Grinder 127

Part VI. Chapter 6: Heat Treating

15. Chapter 6: Heat Treating 141
16. Unit 2: Hardness Testing 145

Part VII. Chapter 7: Lean Manufacturing

17. Chapter 7: Lean Manufacturing 151

Part VIII. Chapter 8: CNC

18. Chapter 8: CNC 161
19. Unit 2: CNC Machine Tool Programmable Axes and Position Dimensioning Systems 163
20. Unit 3: Vertical Milling Center Machine Motion 173
21. Unit 4: CNC Language and Structure 181
22. Unit 5: CNC Operation 195
23. Unit 6: Haas Control 197
24. Unit 7: Mastercam 227

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

Chapter 1. Milling Machine
Unit One: Tramming the head
Unit Two: Cutting speed
Unit Three: Sine bar
Unit Four: Sine bar and Rotary Table

Chapter 2. Lathe Machine
Unit One: The Engine Lathe
Unit Two: Speed and Feed
Unit Three: Chucks
Unit Four: Turning
Unit Five: Tapping
Unit Six: Threading

Chapter 3. Drill Press
Unit One: Introduction to Drill Press and Safety

Chapter 4. Bandsaw
Unit One: Introduction to Bandsaw and Safety

Chapter 5. Surface Grinder
Unit One: Introduction to Surface Grinder and Safety

Chapter 6. Heat Treating
Unit One: Introduction to Heat Treating and Safety
Unit Two: Hardness Testing

Chapter 7. Lean Manufacturing
Unit One: Introduction to Lean Manufacturing

Chapter 8. CNC
Unit One: Introduction to CNC
Unit Two: CNC machine tool programmable axes and position dimensioning system.
Unit Three: Vertical Milling Center Machine Motion.
Unit Four: CNC Language and Structure
Unit Five: CNC Operation
Unit Six: Haas Control
Unit Seven: Mastercam

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.

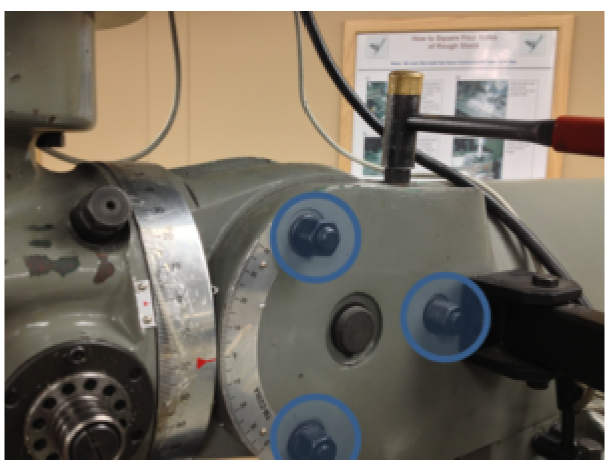
A dial indicator attached to the spindle for precise mill head alignment.

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.

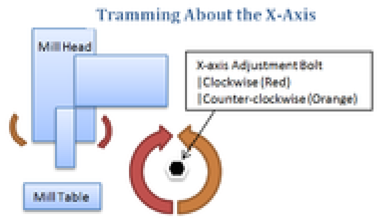
1. After loosening the bolts, re-tighten them by hand plus a ¼ of a turn using the mill wrench.
2. The adjustment bolt that moves the mill head up and down around the x-axis is located at the back of the mill.

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.

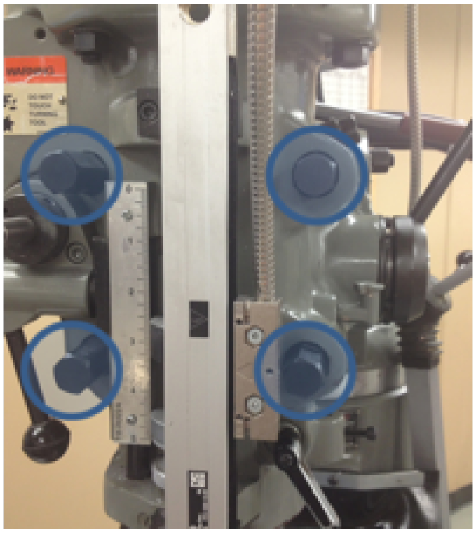


Mill head adjustment about the X-axis.

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.

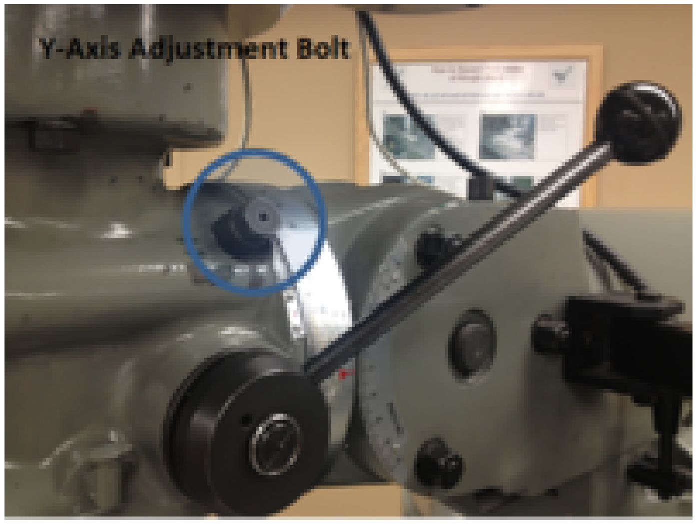
Tramming Process for the Y-Axis

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).



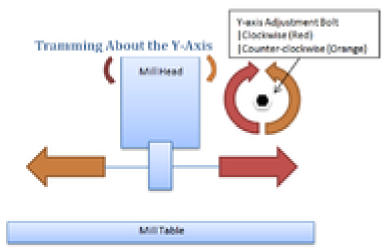
Location of the bolts to be loosened to allow the head to rotate about the Y-axis.

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.

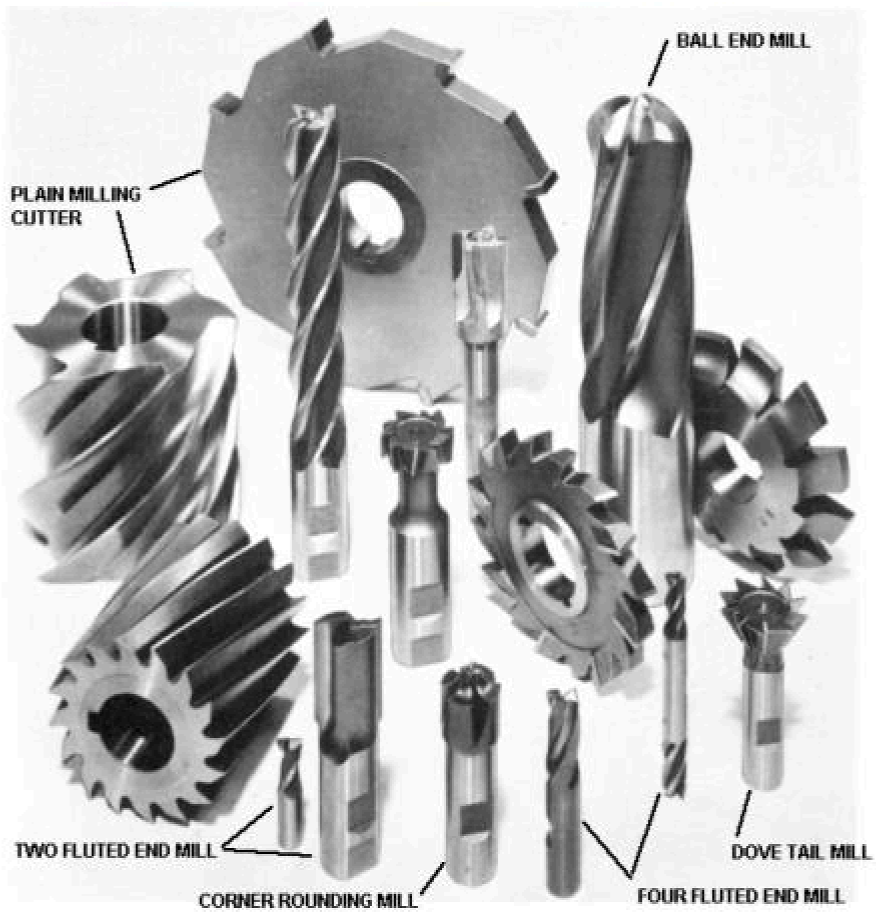
1. The figure above shows how the adjustment bolt for tramming about the y-axis affects the mill head. Use the same process as described for tramming about the x-axis, however, use locations left and right of the mill head as your reference points in contrast to the front and the rear as done previously.
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.

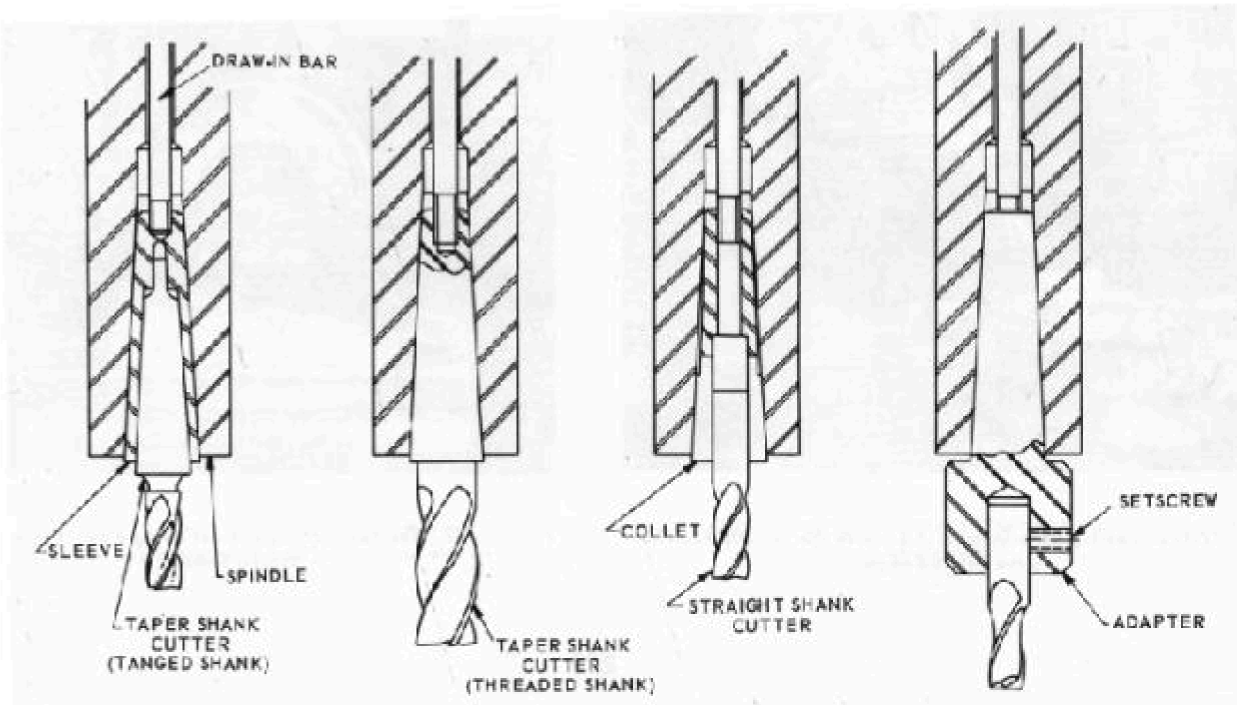
9. If the indicator does not stay at zero, realign the vice by lightly tapping with a soft hammer until the indicator reads half of its previous value.
10. Repeat the process until the dial indicator shows zero through a complete travel from one side of the vice to the other.
11. Fasten the T-bolts securely,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

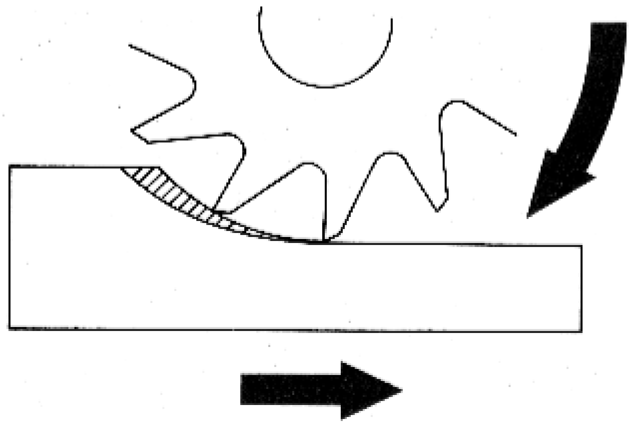
1. If a tool needs to be removed, lock the quill at the highest position.
2. Next, loosen the drawbar with a wrench while using the brake.
3. Make sure that the threads of the draw bar remain engaged in the collet. If they are not engaged, the cutter will fall and potentially be damaged when the collet 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

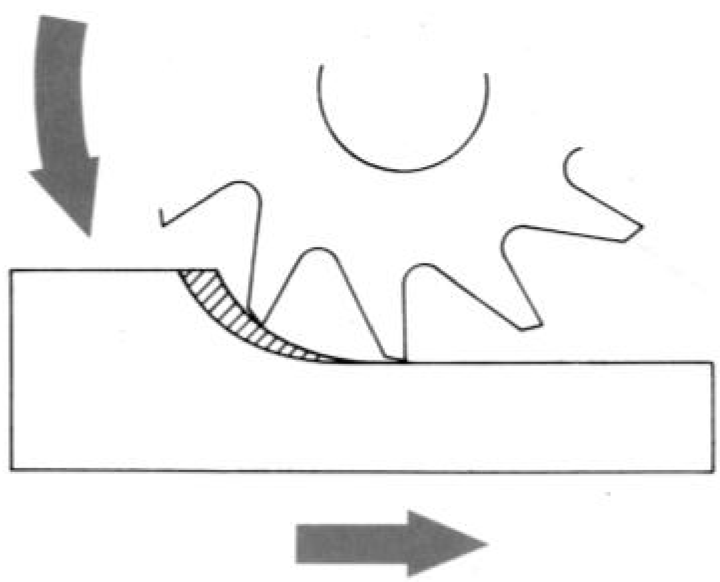
1. The workpiece is fed against the rotation of the cutter.
2. Conventional milling is usually preferred for roughing cuts.
3. Conventional milling requires less force than climb milling.
4. Does not require a backlash eliminator and tight table gibs.
5. Recommended when machining castings and hot-rolled 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).

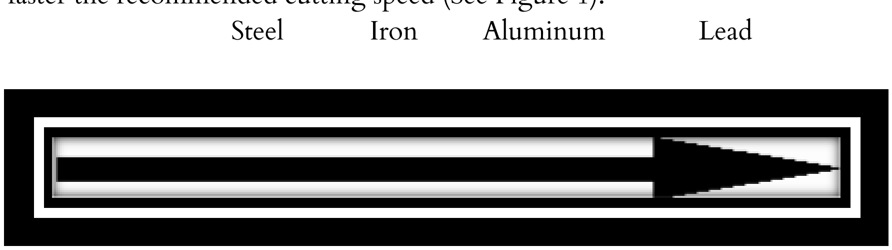


Figure 1: Increasing Cutting Speed Based on work material hardness

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).

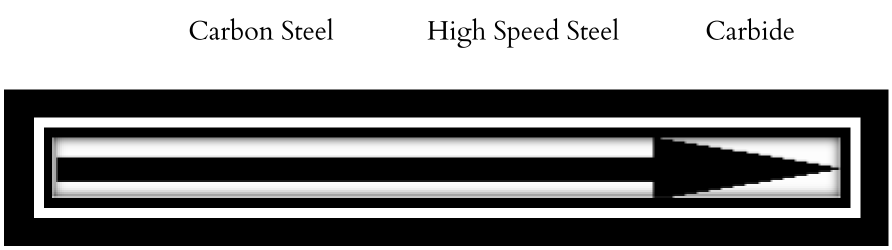


Figure 2: Increasing Cutting Speed Based on Cutting tool hardness

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				

		12	.4603	31/64	.4844	33/64	.5156				
9/16	.5625	18	.4943	33/64	.5156	17/32	.5312	37/64	.5781	19/32	.5938
		24	.5114	33/64	.5156	17/32	.5312				
		11	.5135	17/32	.5312	9/16	.5625				
5/8	.6250	18	.5568	37/64	.5781	19/32	.5938	41/64	.6406	21/32	.6562
		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
		10	.6273	21/32	.6562	11/16	.6875				
3/4	.7500	16	.6733	11/16	.6875	45/64	.7031	49/64	.7656	25/32	.7812
		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
		9	.7387	49/64	.7656	51/64	.7969				
7/8	.8750	14	.7874	13/16	.8125	53/64	.8281	57/64	.8906	29/32	.9062
		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.

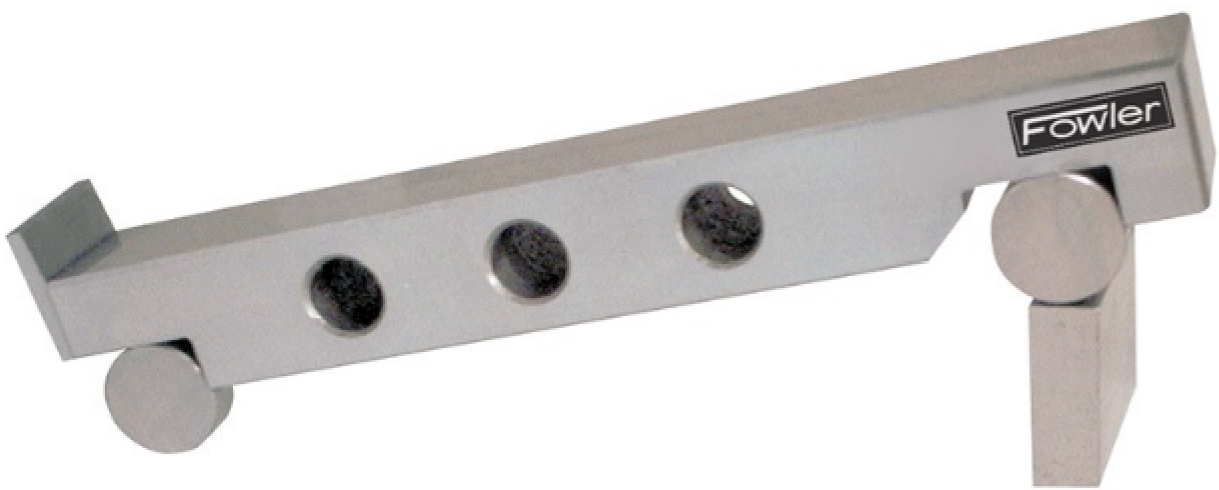


Figure 1. The Sine Bar

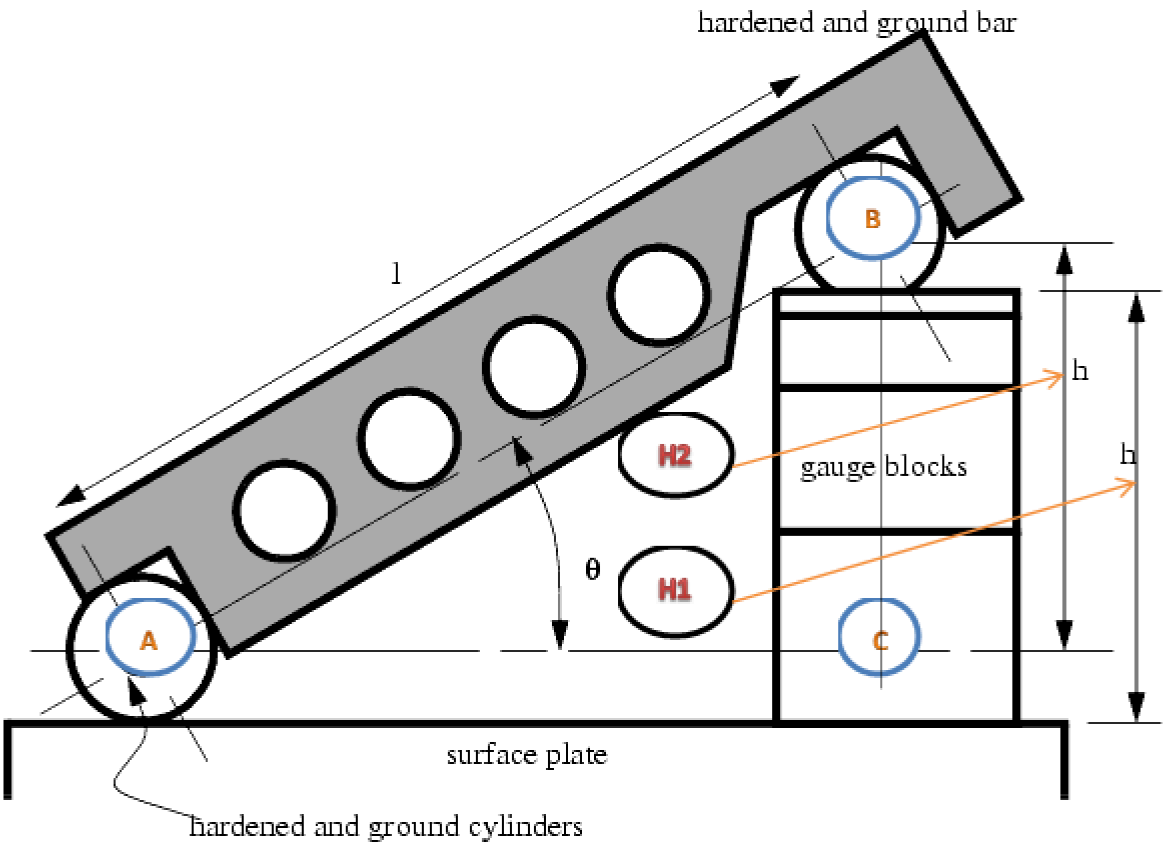
Two cylinders of equal diameter are placed at the ends of the bar. The axes of these two cylinders are mutually parallel to each other, and are also parallel to, and at equal distance from, the upper surface of the sine bar. Accuracy up to 0.01mm/m of length of the sine bar can be obtained.

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″**: **$H = 5.000'' * \sin(13°)$**. 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

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

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.

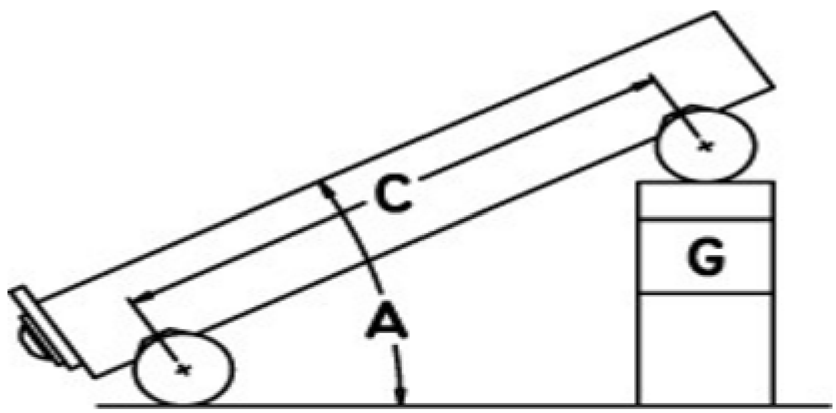


Figure 3: Uses the formula: G = C * Sin(A)

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.

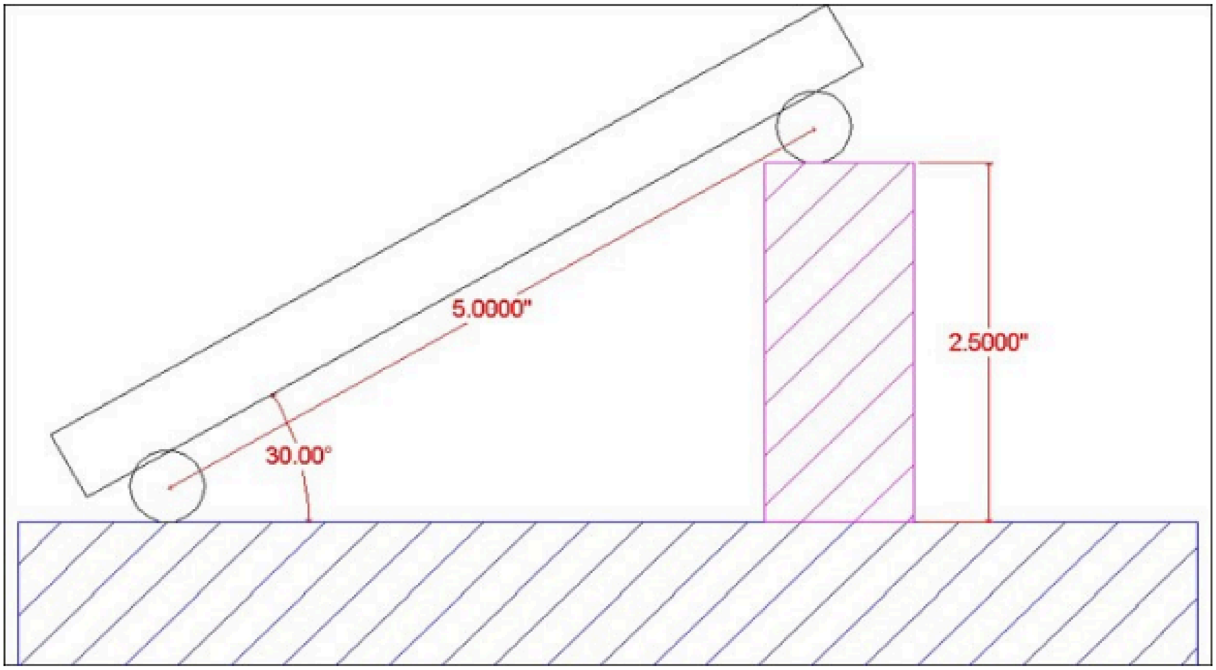


Figure 4. Sine Bar

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?

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.

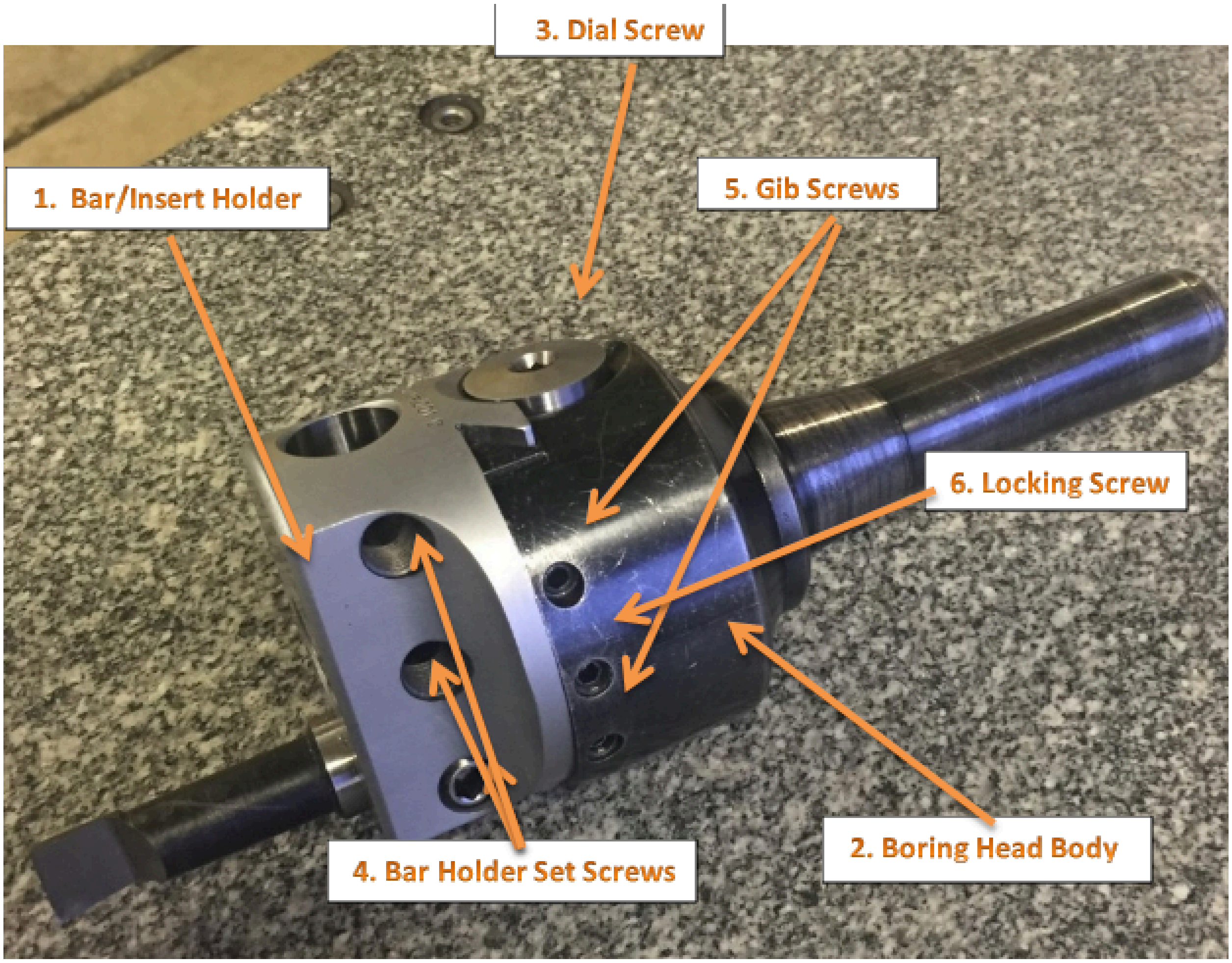


Figure 1. Offset Boring Head

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.

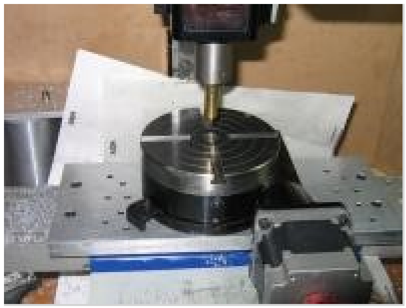
6. Bring the dial indicator into contact with the diameter of the plug, and rotate the spindle by hand.

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

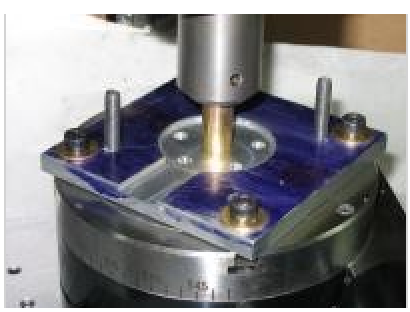
Visual Position

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 watc
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.

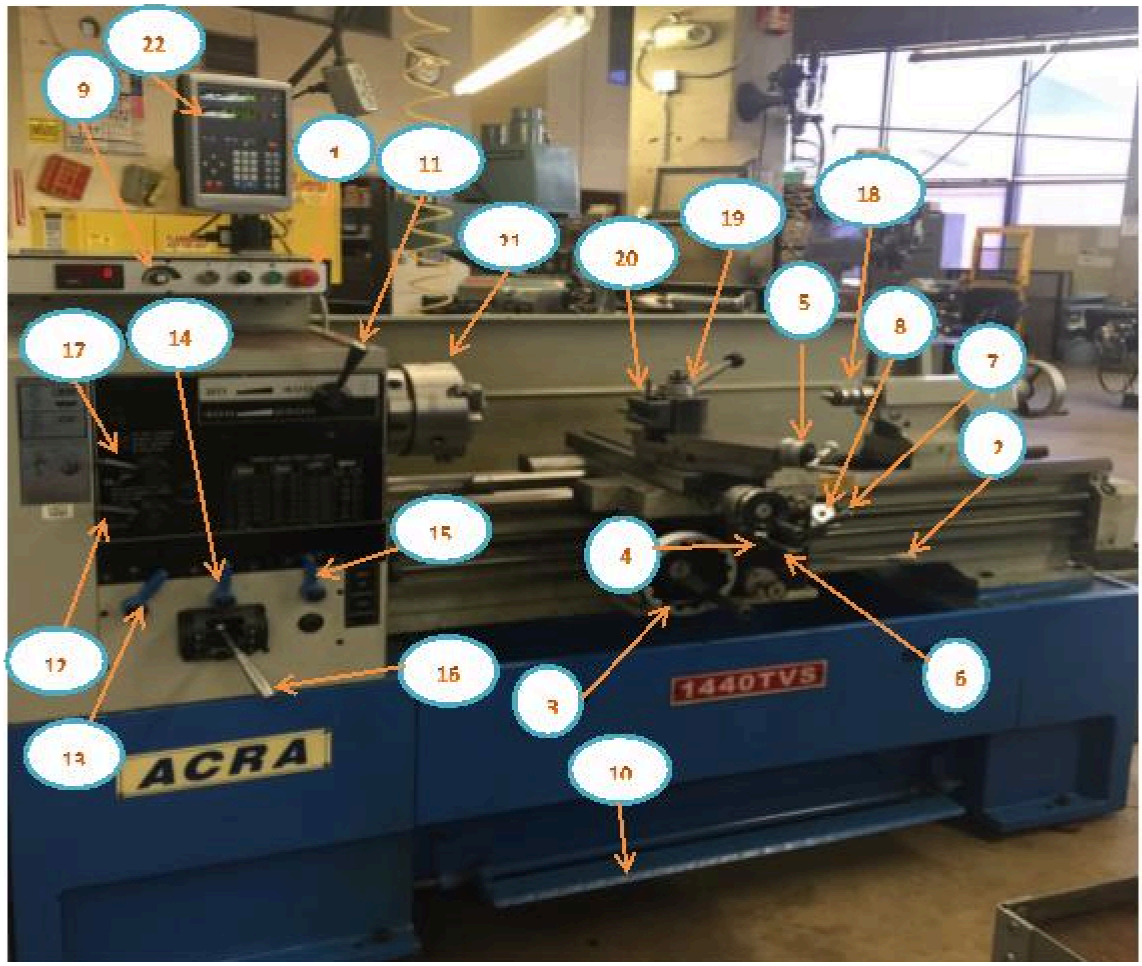


Figure 1. Parts of a lathe

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 usec 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.

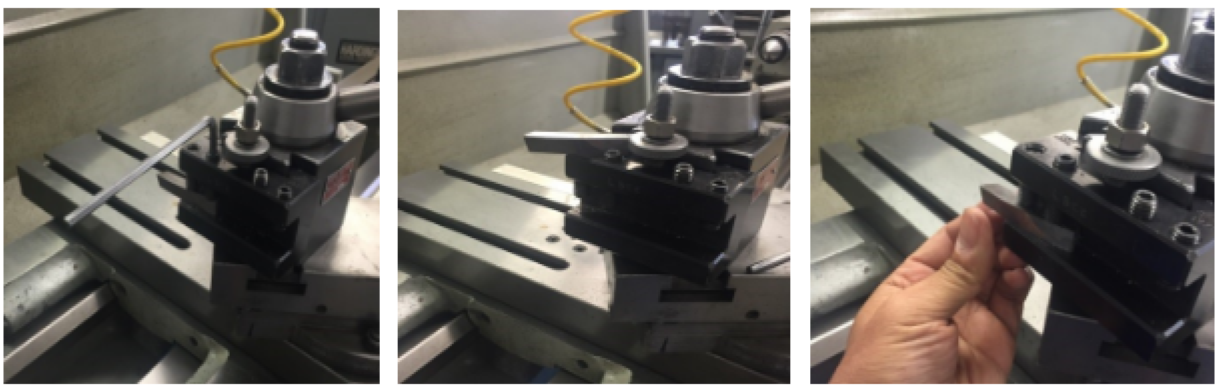


Figure 4: Installing a Cutting Tool

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.

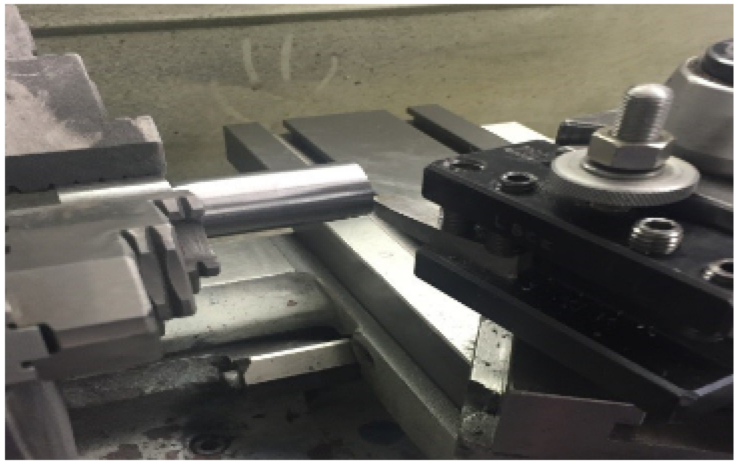
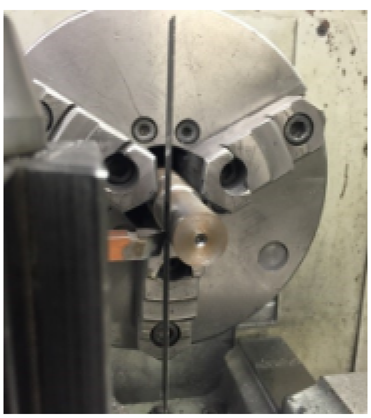


Figure 5: Positioning the Tool

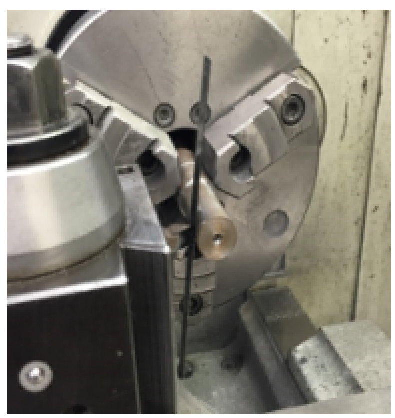
Centering the Workpiece

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

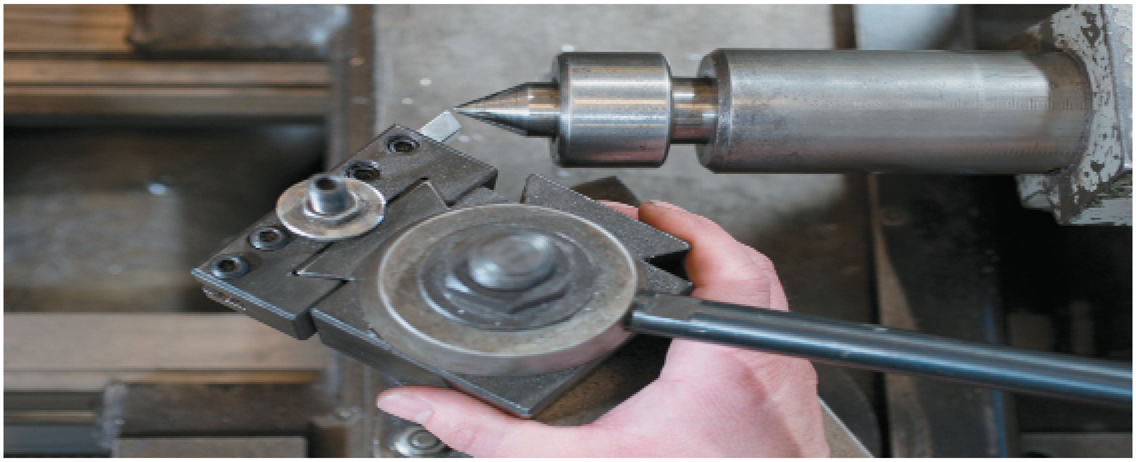


Tool is High

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.

Steel Iron Aluminum Lead

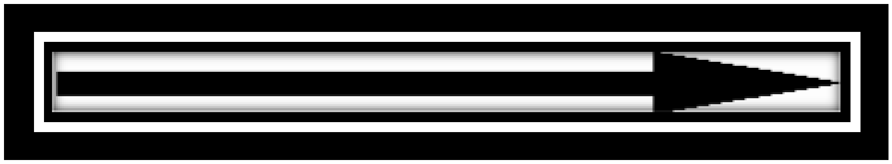


Figure 1: Increasing Cutting Speed Based on work material hardness

Carbon Steel High Speed Steel Carbide

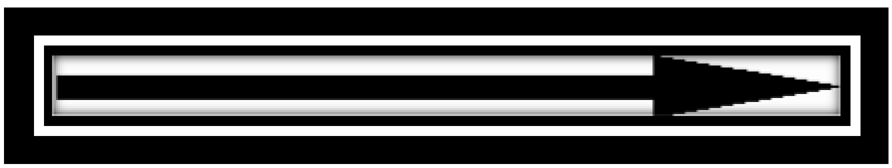


Figure 2: Increasing Cutting Speed Based on Cutting tool hardness

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

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.

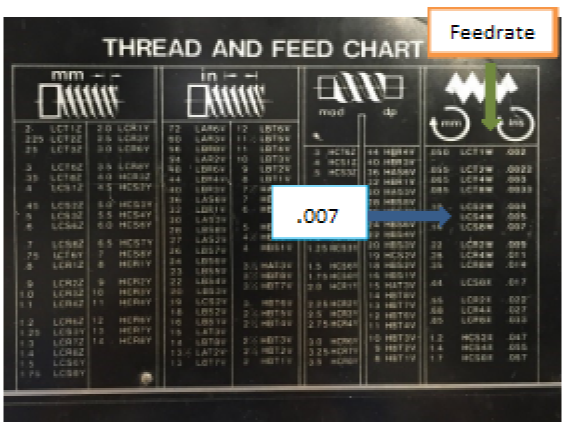


Figure 5: Thread and Feedrate Chart

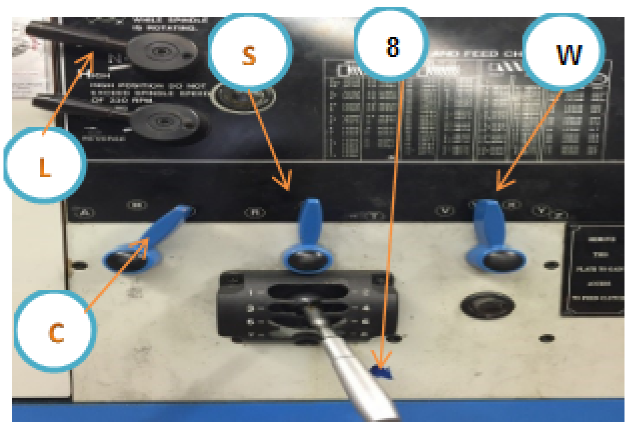


Figure 6: Quick Change Gearbox

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.