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.

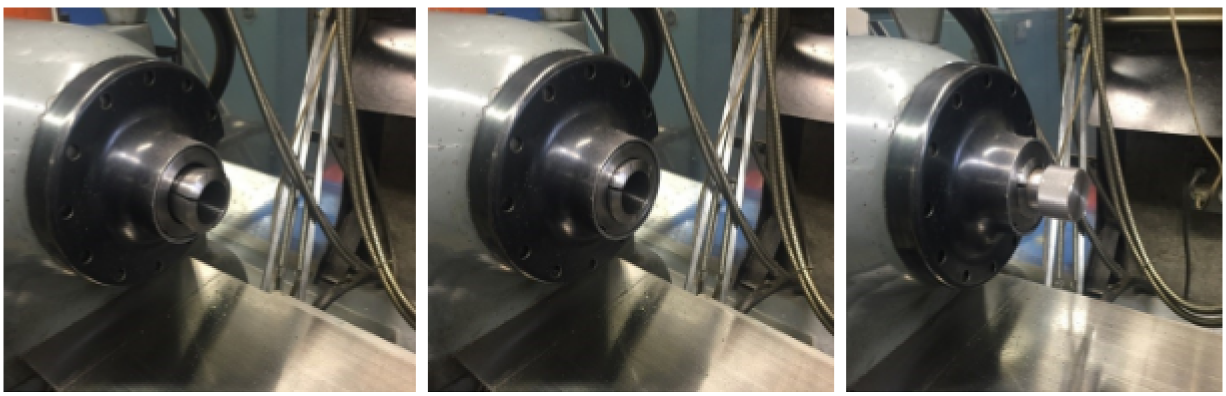


Figure 3: Collect chuck

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.

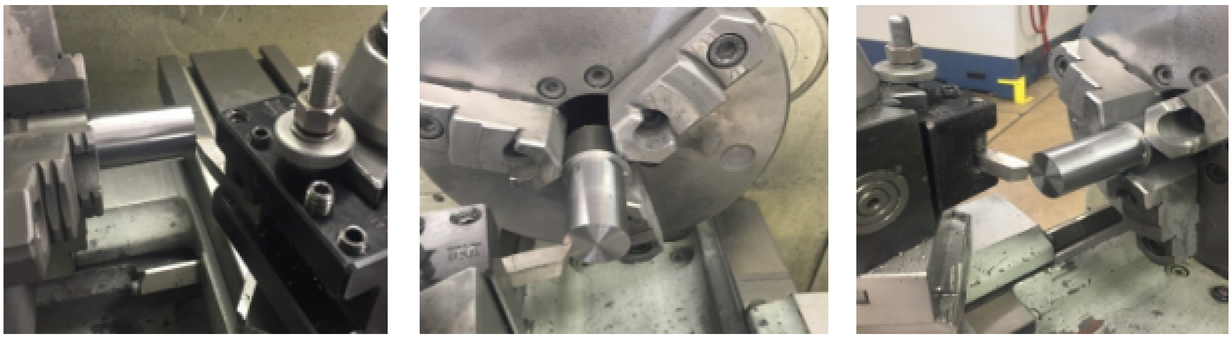


Figure 1. Facing Operation

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.

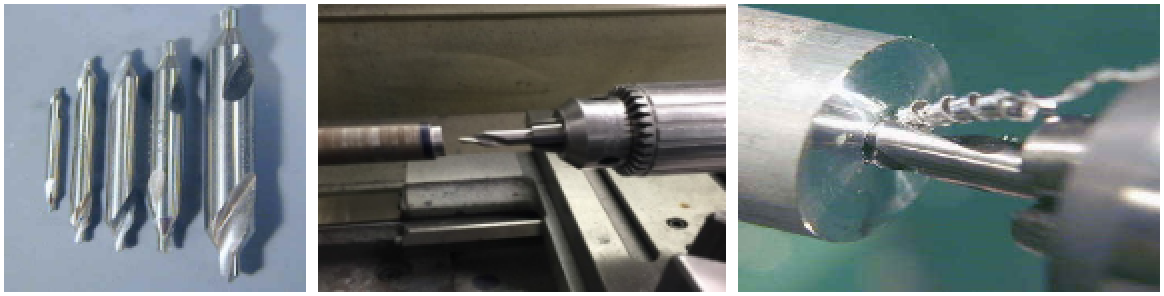


Figure 2: Center/Spot Tool

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

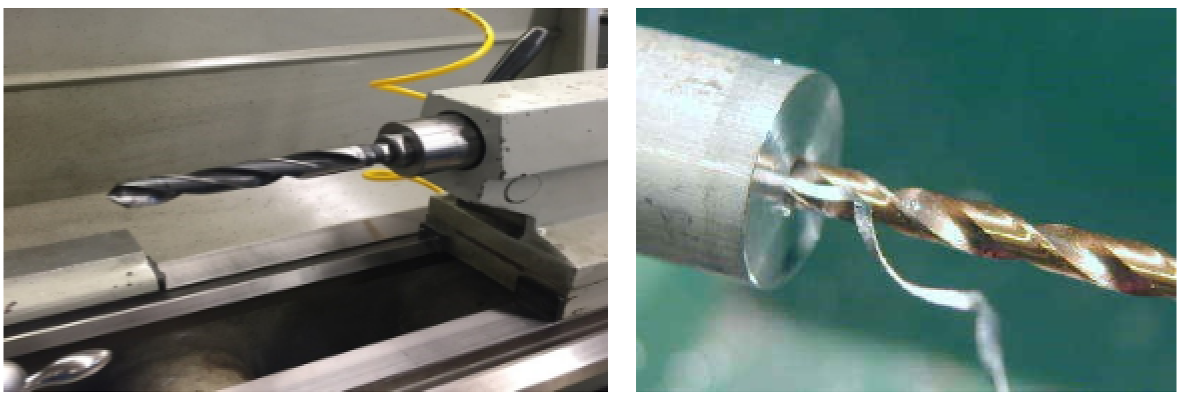


Figure 3. Drill

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.

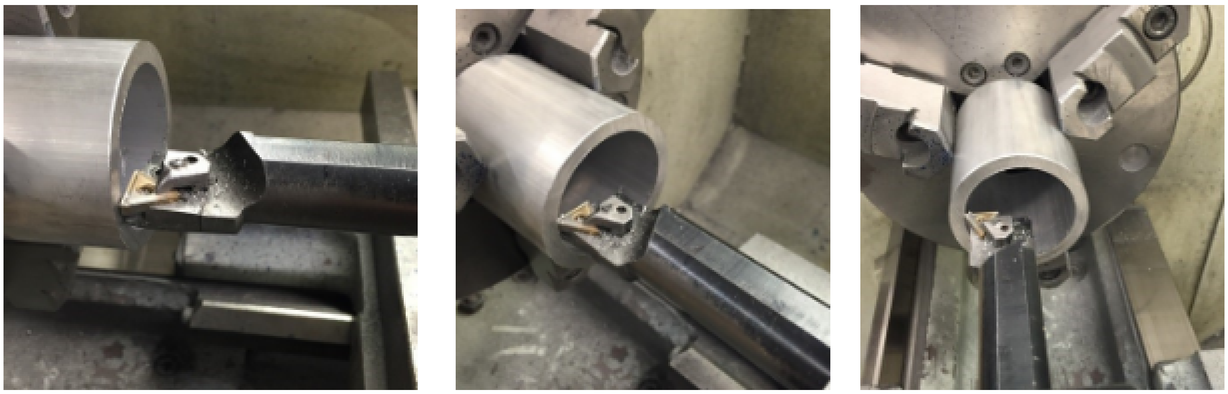


Figure 4. Boring on a lathe

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.

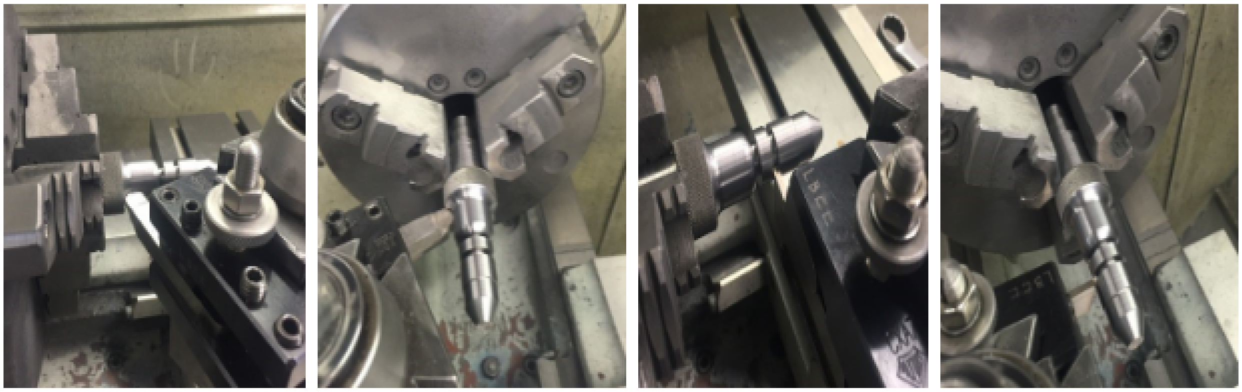


Figure 6. Cutting a Groove

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.

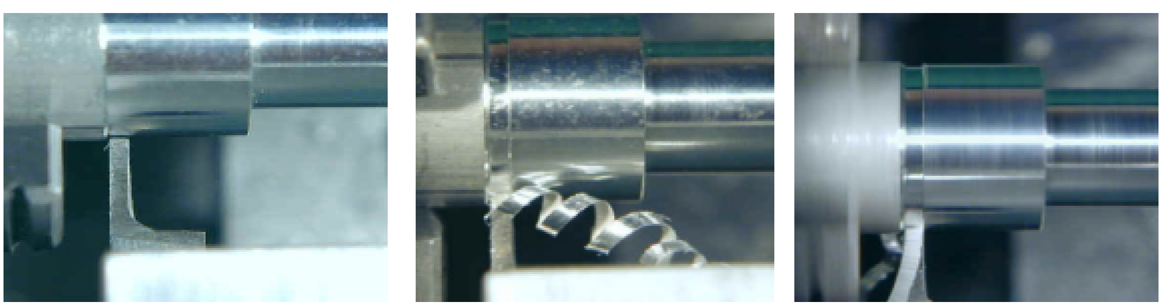


Figure 7. Parting

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.

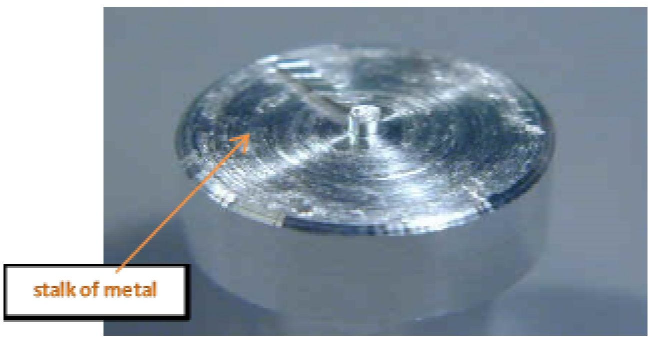


Figure 8. Workpiece Cutoff

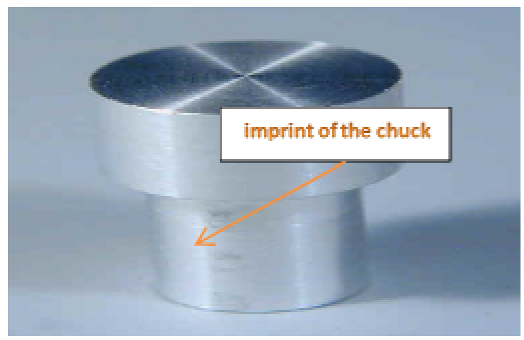


Figure 9. Finished Workpiece

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.

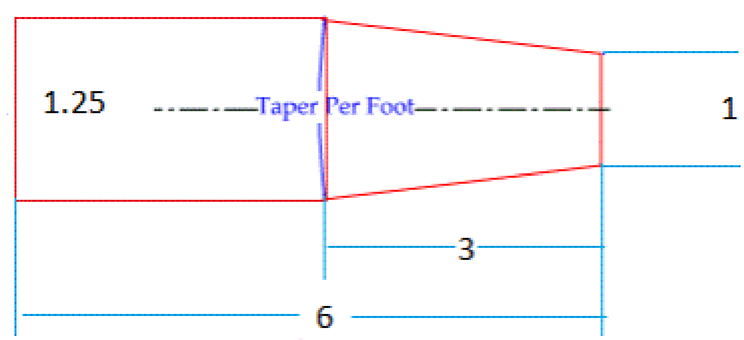


Figure 10. The main part of an inch taper

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.

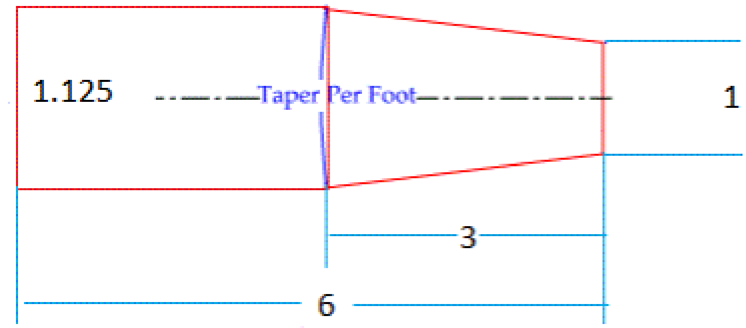


Figure 11. Dimension of a workpiece having a taper

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.

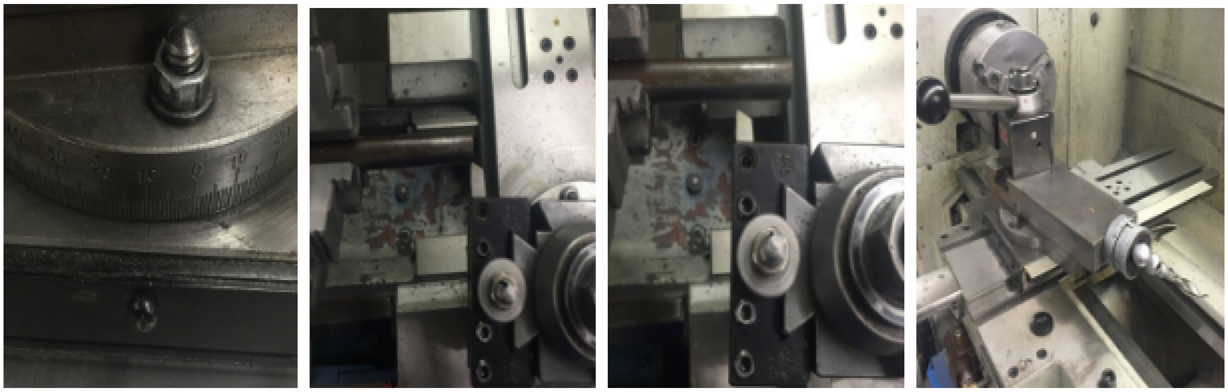


Figure 12. Taper Turning Operation

True workpiece in a 4-jaw chuck

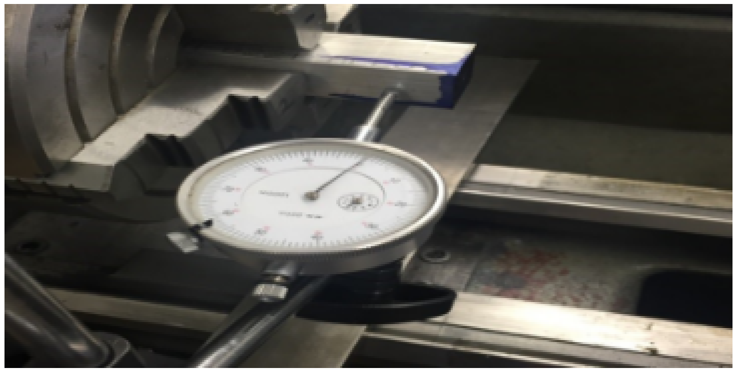
1. A dial or test indicator should be used whenever a machined diameter must be aligned to within a thousandths of an inch.

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.

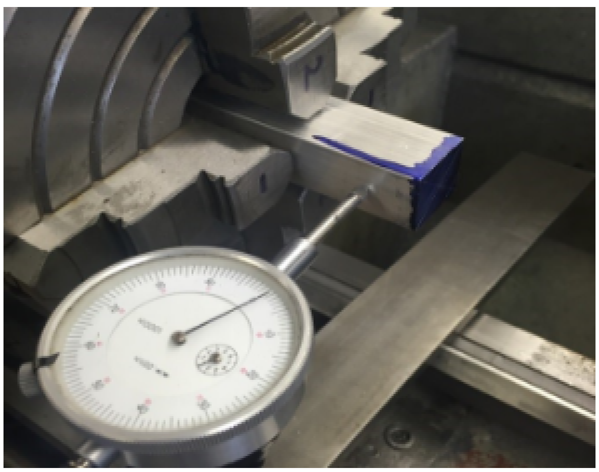
5. Set the indicator spindle in a horizontal position with the contact point set to the center height.

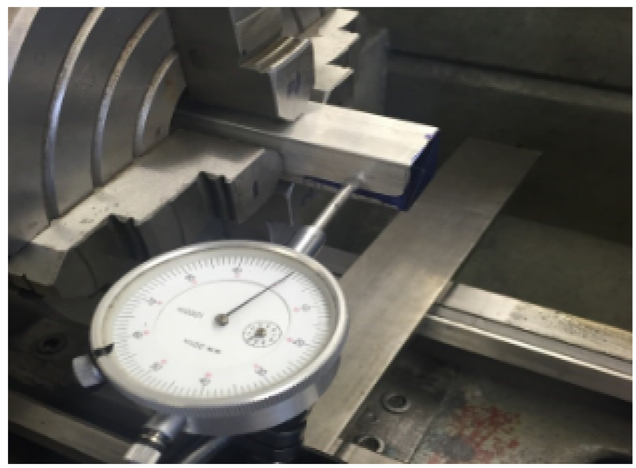


6. Bring the indicator point against the workpiece diameter so that it registers about .020 and rotate the lathe spindle by hand.

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.

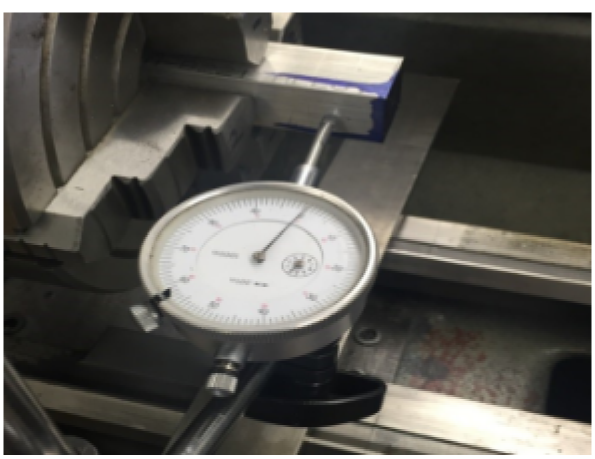


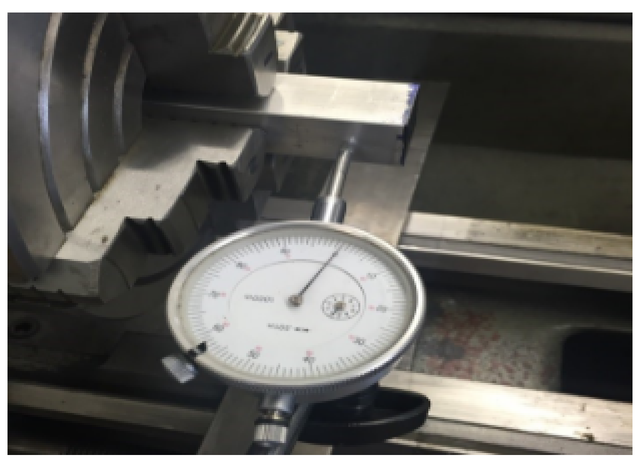


Side 1. Left and Right Side

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.

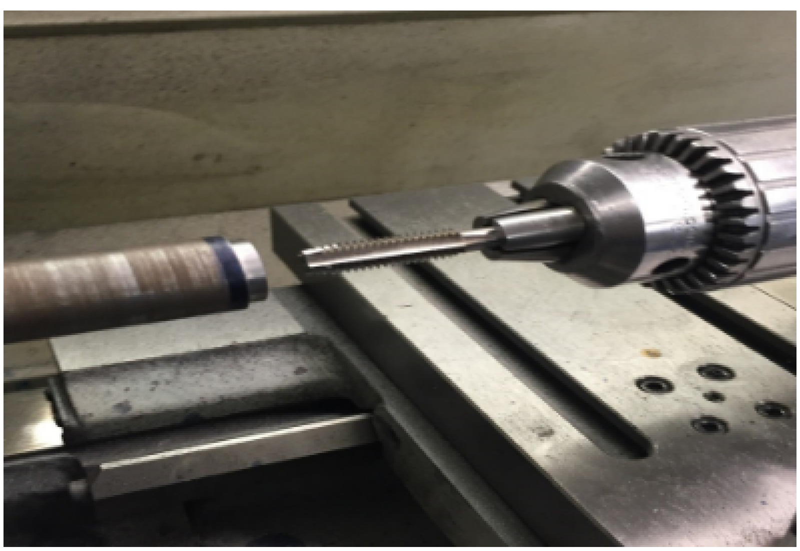


Figure 1. Tap

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.

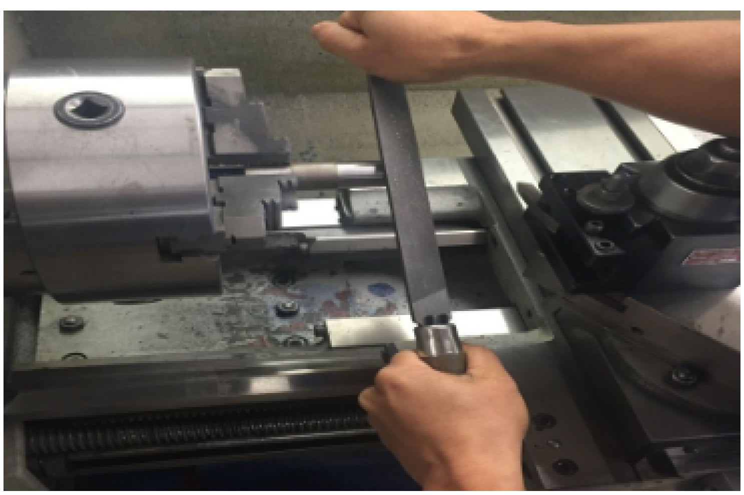


Figure 2. Filing

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.

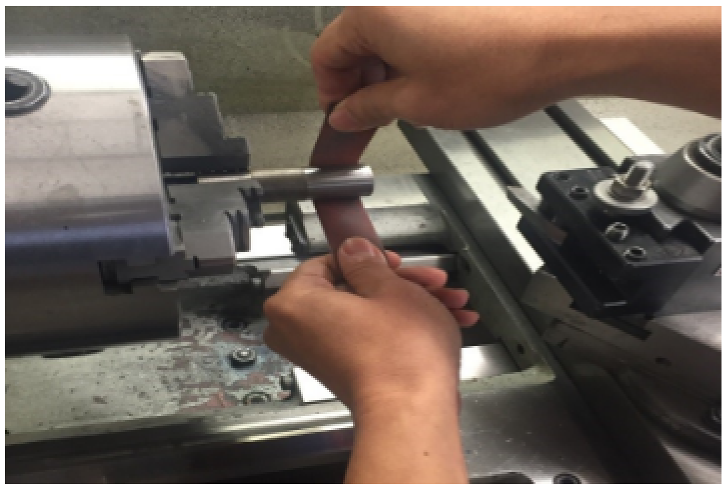


Figure 3. Polishing

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)

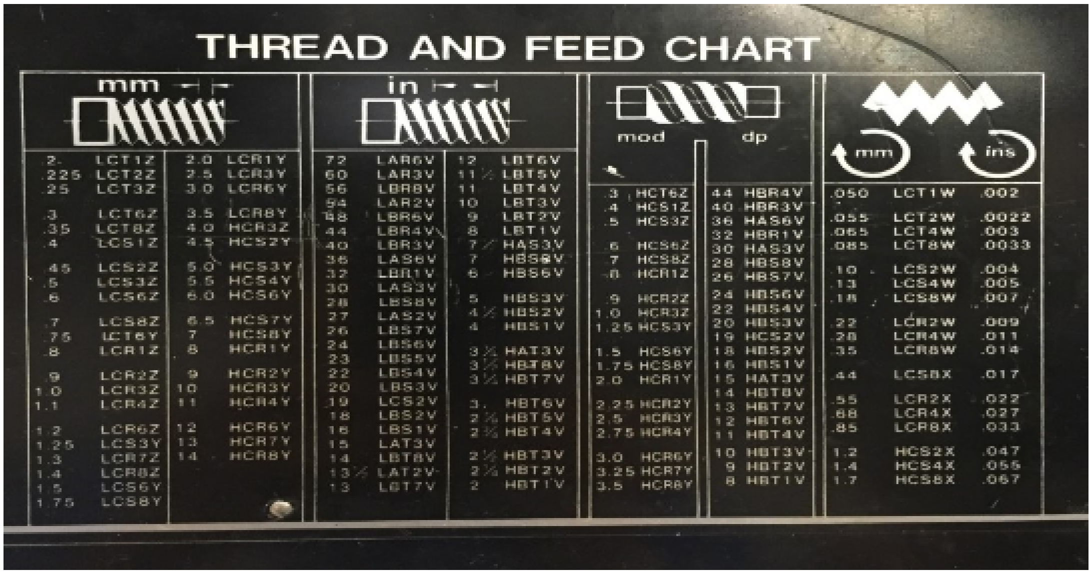


Figure 1. Thread and Feed Chart

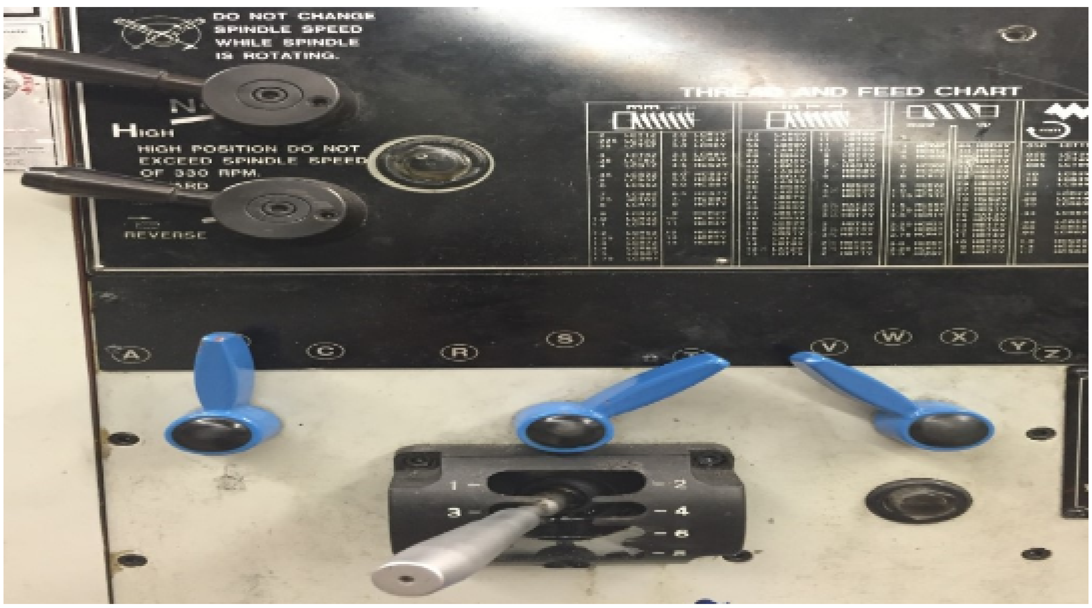


Figure 2. Setting Gearbox

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

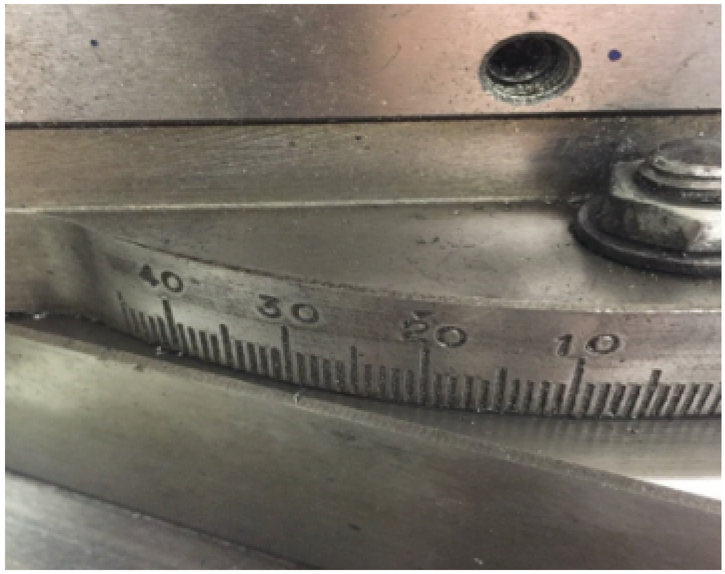


Figure 3. 29 Degrees

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

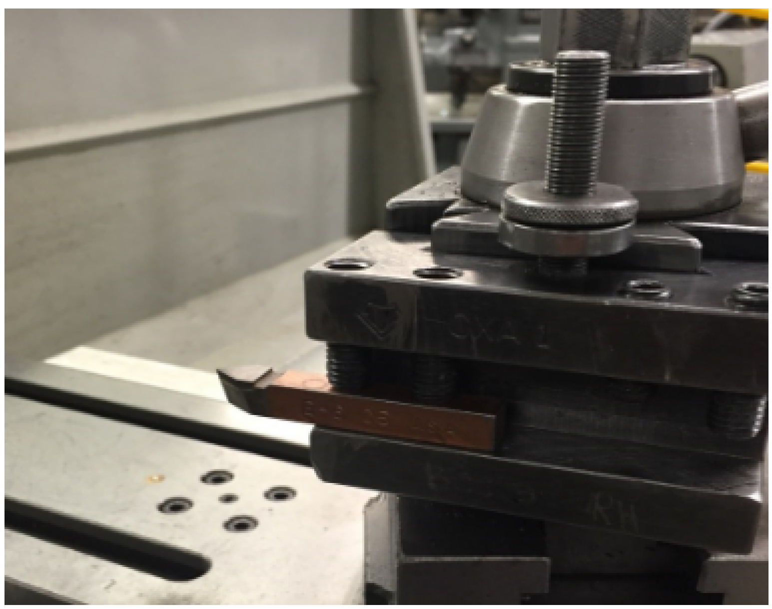


Figure 4. 60 Degree Threading Tool

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

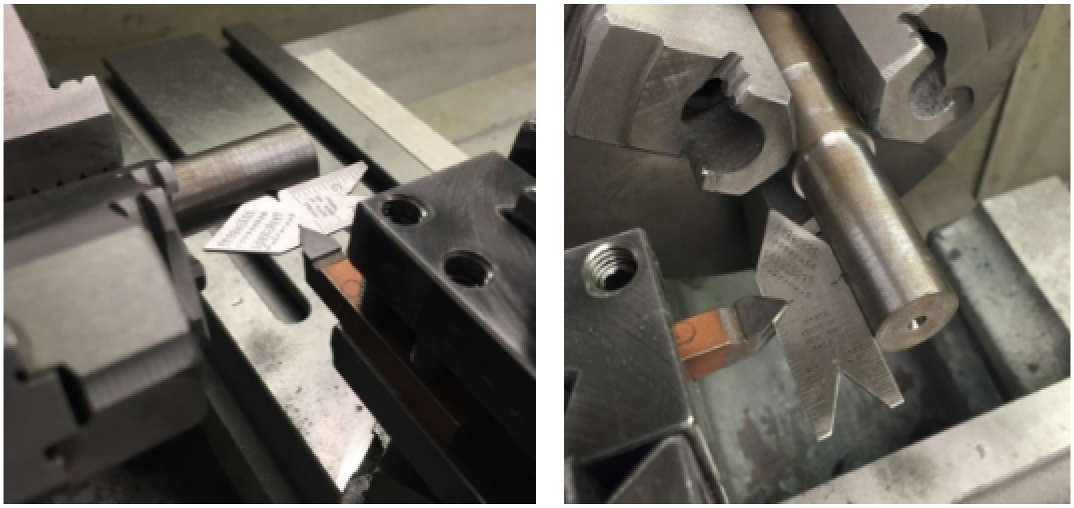


Figure 5. Using the Center gage to positionthe tool for machining Threads

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

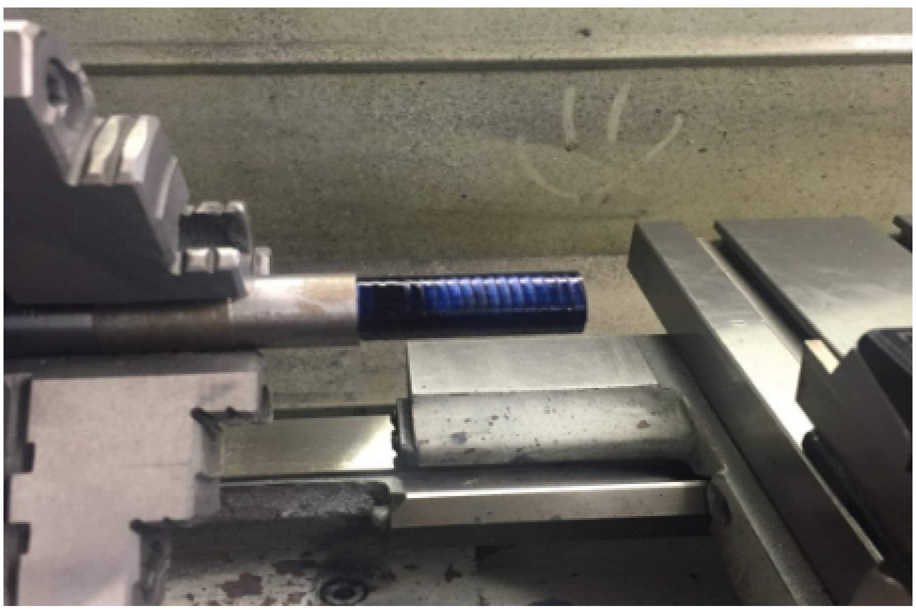


Figure 6. Layout

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.

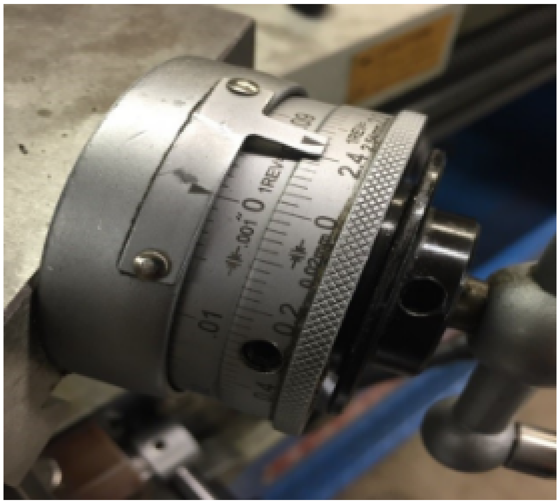


Figure 7. Compound

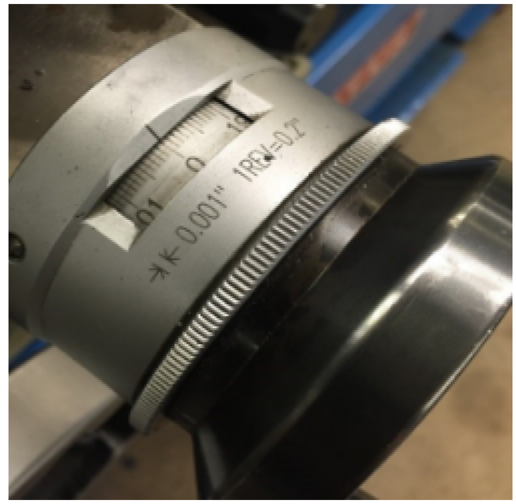


Figure 8. Cross Feed

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.

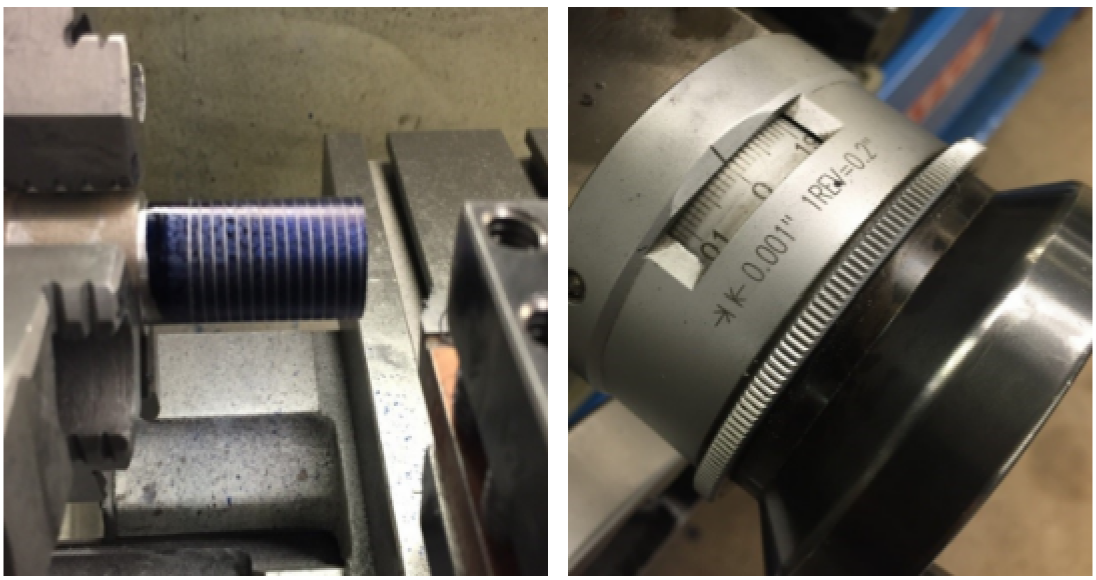


Figure 9. End of the part and Cross feed to Zero

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

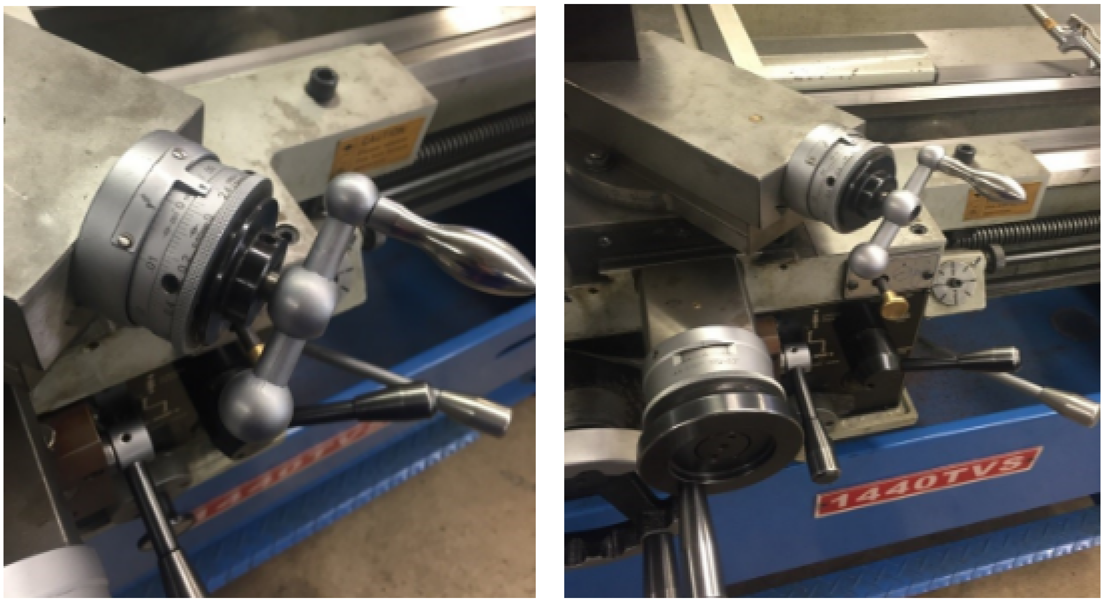


Figure 10: Compound feed in .002 inch

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.

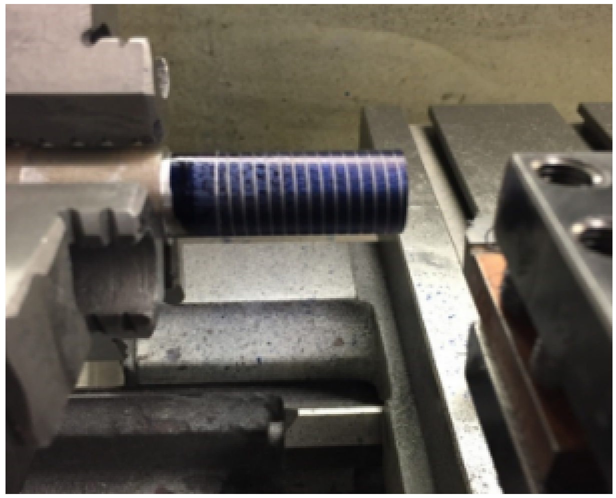


Figure 12. Starting Position

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

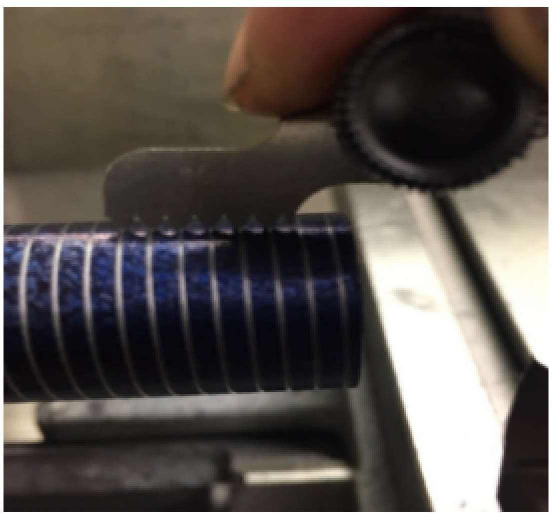


Figure 13. Screw Pitch Gage Figure 14. Screw Pitch Gage(10)

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.

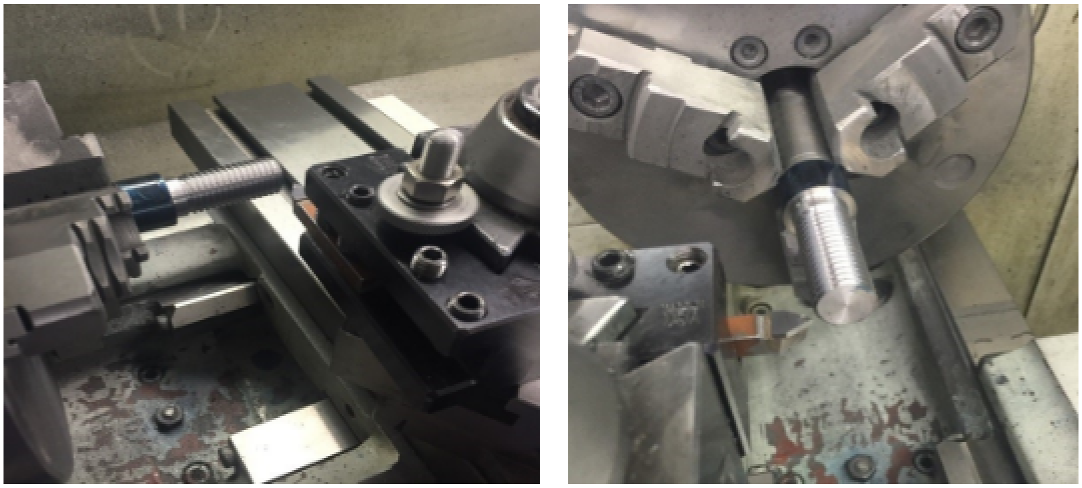


Figure 15. Threading operation

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.

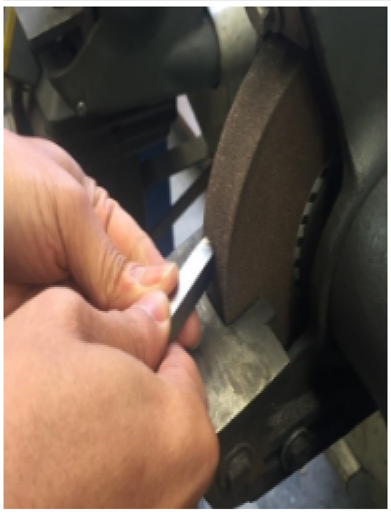
Reaming with a Machine Reamer

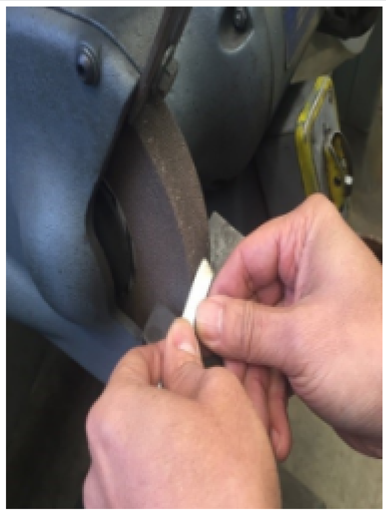
The hole to be reamed with a machine reamer must be drilled or bored to within 0.010 inch of the finished size so that the machine reamer will only have to remove the cutter bit marks. Use plenty cutting fluid for reaming.

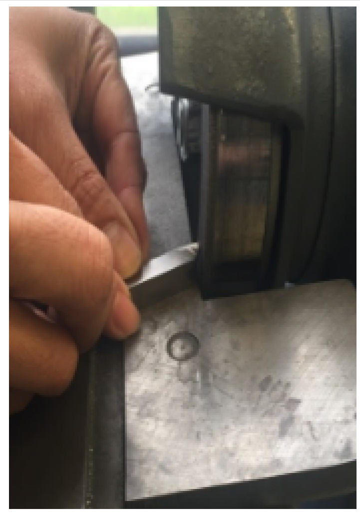
Grind a Lathe Tool bit

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

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

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

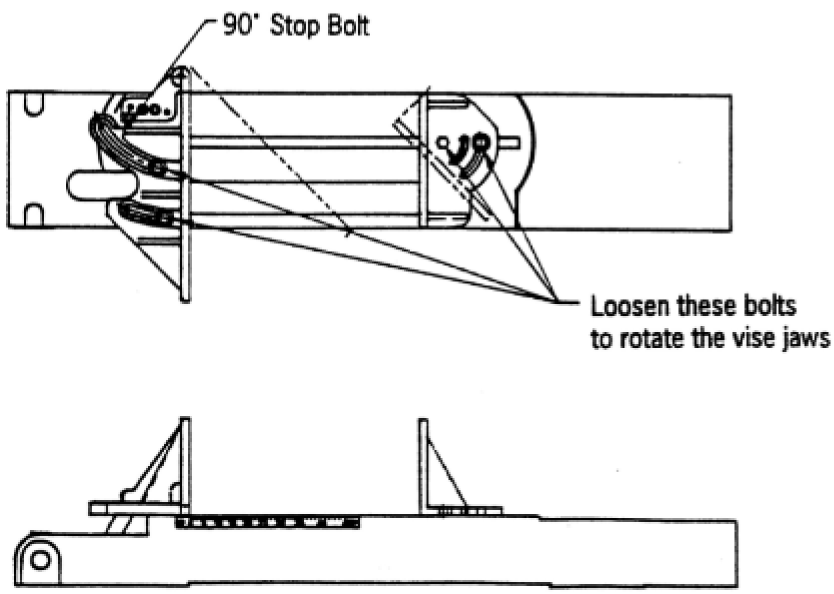
1. Turning the handle to the left will loosen the vice. Turning it to the right will tighten it.

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.

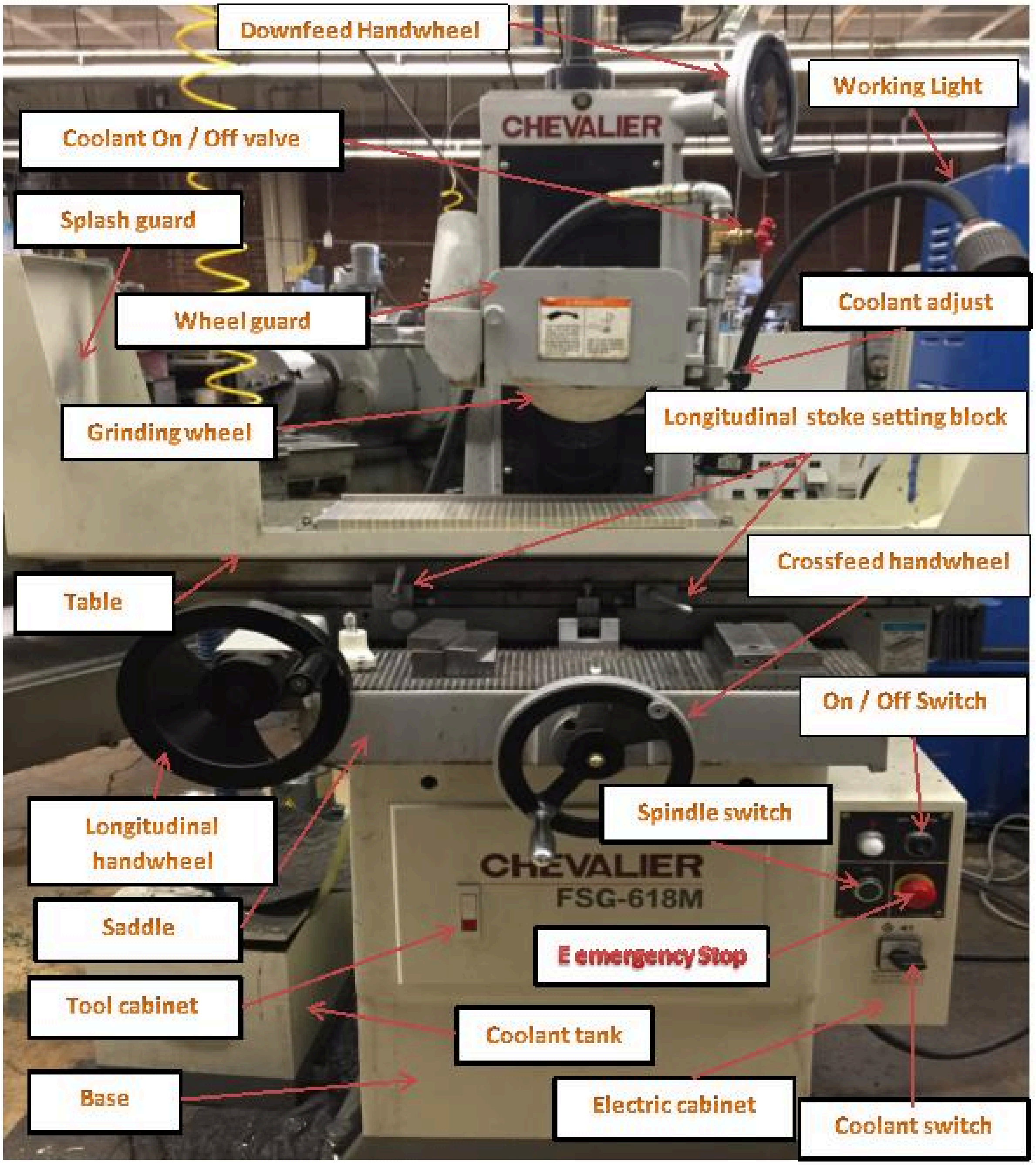


Figure 1. Chevalier Surface Grinder

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.

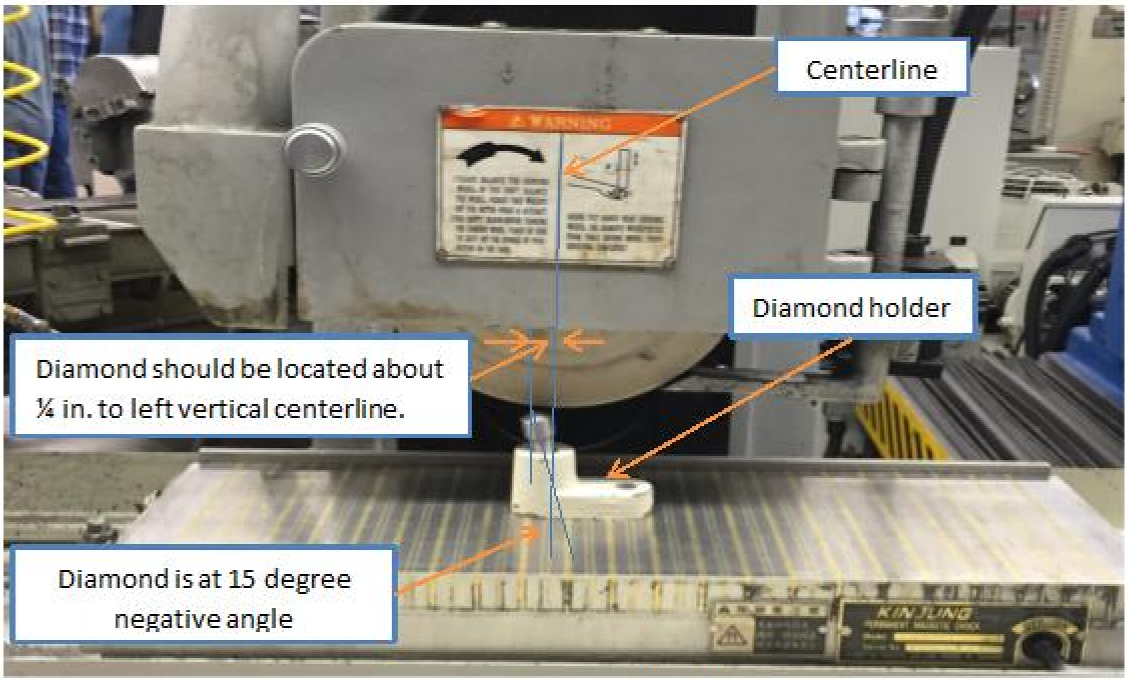


Figure 2. Dressing the wheel

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.

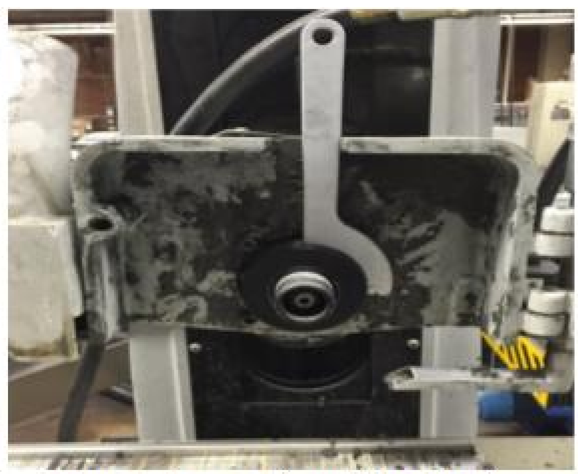


Figure 4. Brace wrench into hole Figure 5. Remove the grinding wheel

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