Basic Lathe Manual
# Table of Contents

Preface .................................................................................................................................................. iii

1. Machine Shop Safety ......................................................................................................................... 1

2. Lathe Introduction ................................................................................................................................. 3
   2.1 The Metal Lathe ................................................................................................................................. 3
   2.2 Standard Components ......................................................................................................................... 4
   2.3 Common Operations ............................................................................................................................. 7
   2.4 Tools ..................................................................................................................................................... 9

3. Cutting Speeds and Feeds ...................................................................................................................... 11
   3.1 Cutting Speeds .................................................................................................................................. 11
   3.2 Feed Rates/ Depth of Cut .................................................................................................................... 12

4. Lathe Basics ........................................................................................................................................ 15
   4.1 Operating the Lathe ............................................................................................................................. 15
      Setting Spindle Speed and Feed Rate ..................................................................................................... 15
      Turning on the Lathe ............................................................................................................................... 15
      Feeding the Tool .................................................................................................................................. 15
      Mounting the Workpiece ....................................................................................................................... 16
      Supporting the Workpiece with the Tailstock ....................................................................................... 17
      Mounting Tools ..................................................................................................................................... 18
   4.3 Measurement ..................................................................................................................................... 20
      Caliper ................................................................................................................................................. 20
4.4 Automatic Feed ................................................................. 24
4.5 Lubricants/Coolant ............................................................... 25

5. Lathe Operations ................................................................. 27
5.1 Facing .................................................................................. 27
5.2 Turning ............................................................................... 28
5.3 Drilling/Reaming ................................................................. 29
5.4 Parting/Grooving ................................................................. 31
5.5 Boring ............................................................................... 32
5.6 Taper Turning ..................................................................... 33

Compound Rest ......................................................................... 33

Taper Attachment ...................................................................... 34

5.7 Knurling ............................................................................ 36

6. Sources .................................................................................. 38
Preface

The lathe is directly related to manufacturing and a firm understanding of its operations is critical to your mechanical engineering future. This manual is designed to be a supplement as you learn the different lathe processes. It is not designed to replace lab instruction, but by reviewing its contents before, during, and after lab you will be prepared to learn and accomplish much more than otherwise possible; you will be able to devote more lab time to hands on lathe experience rather than listening to a lecture by referring to this manual before seeking assistance from the TA’s.

The sections of this manual are arranged in the order you must understand information to successfully complete a lathe operation. Later sections will refer to skills discussed in previous sections. Safety is the most important thing you must learn in the machine shop. Review this section often as hazards in the machine shop have the potential to cause serious injury. If you do not understand how to use a machine or perform an operation do not hesitate to ask a TA for help.
1. **Machine Shop Safety**

Safety is the most important thing students must learn when they work in the machine shop. The lathe is a very useful tool, but it can cause serious injury if used improperly. Read the following list of safety points carefully before using the lathe. Periodically reread this safety section; this section will gain more meaning as you become more familiar with the lathe.

- If you do not know how to use a tool or machine, **DO NOT USE IT WITHOUT HELP!** Ask a TA or the machine shop staff for help.
- If the lathe makes an unusual noise, stop the machine and ask for help.
- Do not wear jewelry such as necklaces and rings when using the lathe.
- Do not wear long sleeved shirts or loose clothing that could become caught in the machine.
- Always wear long pants and close toed shoes.
- Always wear safety glasses.
- Only one student should operate a single machine at a time.
- Use only sharp tools that are in good condition.
- If a tool or machine is damaged notify a TA or lab supervisor.
- Always use the correct cutting speeds for a job.
- Clean the machine after use.
- If cutting oil or fluid is spilled clean it up immediately.
• To not attempt to stop a rotating chuck with your hands.

• Do not remove metal chips from the workpiece with your hands; these chips are very hot and sharp.

• Do not test the surface finish of the workpiece with your finger while it is spinning.

• Keep the carriage clear of the rotating chuck.

• Always use the correct tool for a job.

• Workpiece that extends more than 2.5 times its diameter out of the chuck must be supported by the tail workpiece.

• Never leave the chuck wrench in the chuck.

• Do not leave a machine running unattended.

• The machine should be stopped to take measurements.

• Absolutely no horseplay is allowed in the machine shop!
2. Lathe Introduction

2.1 The Metal Lathe

A metal lathe (Figure 1) is a machine that rotates and cuts a workpiece by pressing a fixed tool against it. The lathe is an important part of a mechanical engineer’s education because every company that manufactures products (everyone you will work for) works with a lathe.

Figure 1: A metal lathe containing labeled components 1 to 15 (unless noted otherwise) [1].
2.2 Standard Components

The **bed**, made of cast iron, is used to provide a secure frame for the main components to be mounted on.

The **brake** is a foot pedal that is used to stop the spindle after the machine has been turned off.

The **carriage** provides a rigid structure for tools to mount to. The carriage is moving along the ways to feed the tool into the workpiece.

A **chuck** is mounted on the spindle and secures the being machined. (Different attachments can be mounted on the spindle, but we will usually use a chuck.)

The **digital readout screen** (not present in Figure 1) displays the coordinate location of the tool.

The **feed rod** transmits power from the headstock assembly to the carriage for feeding operations.

The **feed selector levers** set the automatic feed rate of the tool.

The **gearbox**, located inside the headstock assembly, provides different speeds for the spindle and automatic feed.

The **headstock assembly** consists of a motor, gears, and the spindle.

The **lead screw** transmits power from the headstock assembly to the carriage for threading operations.

The **spindle** is located on the right side of the headstock assembly and is used to hold and drive the workpiece.

The **spindle speed selector levers** are used to set the speed of the spindle.

The **steady rest** (not present in Figure 1) is clamped to the ways of the lathe and is used to support the workpiece.

The **tailstock assembly** is used to support the workpiece and mount drilling tools. The tailstock also slides on the ways to fit different workpiece lengths.

The **ways**, inner and outer, are guide rails used to ensure accurate movement of the carriage and tailstock.
Figure 2: A close-up of the tailstock assembly containing labeled components 16 to 19 [1].

The tailstock handwheel is used to manually feed the tools in the tailstock assembly toward the workpiece.

The tailstock locking lever locks the tailstock in place on the ways.

The tailstock quill is either used to mount cutting tools or support tools (live center). The quill is extended by rotating the tailstock handwheel.

The tailstock quill lock is used to lock the position of the quill.
The **carriage handwheel** is used to move the carriage and the tool in the y-direction.

The **compound rest**, located on the cross slide, is used for cutting threads and tapers.

The **compound rest feed handle** is used to manually position and feed the compound rest.

The **cross slide** is mounted on the carriage and feeds the tool in the x-direction.

The **cross slide feed handle** is used to manually feed the tool in the x-direction.

The **feed lever** engages the feed rod to the carriage for automatic feed.

The **follow rest** (not present in Figure 3) is mounted to the carriage and used to support the workpiece.
The **spindle clutch** is located on the right side of the carriage and controls the rotation and direction of the spindle.

The **split-nut control** connects the carriage directly to the lead screw and is used when threading.

The **tool post** (not present in Figure 3), found on the cross slide, is where tools are mounted.

## 2.3 Common Operations

Numerous operations can be completed on the lathe, but only those commonly used in Me En 282 are thoroughly discussed in this manual. To identify the tool used for each operation, see Section 2.4.

<table>
<thead>
<tr>
<th>Figure 4: <strong>Facing</strong> is a process in which a turning/facing tool is used to remove the face of the work-piece, creating a flat surface [3].</th>
</tr>
</thead>
<tbody>
<tr>
<td>Facing</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Figure 5: <strong>Turning</strong> is a process in which a turning/facing tool is used to reduce the diameter of a work-piece to a desired dimension [3].</th>
</tr>
</thead>
<tbody>
<tr>
<td>Straight turning</td>
</tr>
<tr>
<td>Figure 6: <strong>Drilling</strong> is a process where a drill-bit is used to create cylindrical holes in the end of workpiece [3].</td>
</tr>
<tr>
<td>---</td>
</tr>
<tr>
<td><img src="image" alt="Drilling" /></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Figure 7: <strong>Reaming</strong> is when a reamer is used to make a drilled hole more accurate and give it a better finish by removing a small amount of material [3].</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image" alt="Reaming" /></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Figure 8: <strong>Parting</strong> is when a parting tool is used to cut off the end of the workpiece. <strong>Grooving</strong> is where a parting tool is used to create grooves in the workpiece [3].</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image" alt="Parting or cutoff" /></td>
</tr>
</tbody>
</table>
Figure 9: **Boring** is a process in which a boring bar is used to enlarge a hole in the end of a workpiece [3].

![Boring](image)

Figure 10: **Taper Turning** is a process where a turning/facing tool is used to cut an angle on a workpiece [3].

![Taper turning](image)

Figure 11: **Knurling** is a process where a knurling tool is used to create pleasing, easy-to-grip pattern on the surface of a workpiece [3].

![Knurling](image)

### 2.4 Tools

The tools used in the operations above are pictured below. For details on tools, see the course text book.
Figure 12: A facing/turning tool

Figure 13: A drill-bit

Figure 14: A reamer

Figure 15: A parting tool

Figure 16: A small and large boring bar

Figure 17: A knurling tool
3. Cutting Speeds and Feeds

3.1 Cutting Speeds

Correct cutting speeds are critical to any lathe operation and have been tabulated for many different machining conditions (see the Machinery Handbook). Values found in tables will not work for every operation because many variables influence the cutting speeds that can be used; tabulated values should be used as a starting point to find optimal speeds. Factors that affect cutting speed are listed below.

Material being machined – Harder materials such as steel take more force to cut and require a slower cutting speed. Softer materials such as brass and aluminum can be cut at much higher speeds.

Type of operation/tool – Cutting speeds will vary for different operations. For example, parting and grooving speeds are about half the turning speeds. Drilling speeds are different from turning speeds, and reaming speeds are half of the drilling speeds.

Tool material – Carbide tools are harder and more rigid than high speed steel tools, and thus can cut the material at a faster speed. If you are unsure about the tool material, ask a TA.

Other factors – Other factors such as tool condition, rigidity of the set-up, machine horsepower, and coolant affect the cutting speed. Tabulated cutting speeds are generally speeds for optimum conditions (rigid set-up, sharp tool, flood coolant, etc.). We usually will not be machining at optimum conditions, so slower cutting speeds should be used.

Cutting speeds are measured in surface-feet-per-minute (sfpm). This is the distance the tool moves over the surface of the workpiece in one minute. The following equation is used to calculate the rpm (revolutions per minuet) of the lathe spindle.
\[ \text{RPM} = \frac{4 \times \text{SFPM}}{D} \]

D is the diameter of the workpiece for operations such as turning and parting and is the diameter of the drill bit or reamer for drilling operations. In this class, use the cutting speed values listed in Table 1 to calculate the rpm of the spindle.

Note: If you are not using a computer numerated control (CNC) machine, divide the cutting speeds in the table by two.

**3.2 Feed Rates/ Depth of Cut**

Feed is the amount of material removed from the workpiece per revolution of the spindle. The feed on the lathe is generally measured in inches per revolution. The depth of cut is the distance a tool is plunged into the workpiece. For turning, the depth of cut is one half of the difference between the initial and final diameters of the workpiece. The depth of cut and feed rate are related to the cutting speed. A deeper cut or faster feed will require a slower cutting speed as seen in Table 2 from the Machinery Handbook. If a deep cut is taken, a slower feed rate may also be necessary.

Starting points for the depth of cut and feeds can also be found in tables, but experience is required to find the best combinations because all lathes and set-ups differ in rigidity and horsepower; Using values in tables may prove disastrous in some situations and too light in others. A good way to learn acceptable values is by watching more experienced machinists. If you have a question about speeds or the depth of cut simply ask a TA. With a little experience, you will soon gain a good intuition of acceptable values. If cutting speeds or feeds are too fast for a given set-up you will see and hear chatter (vibration of the workpiece and tool), which will produce a poor surface finish. If chatter occurs, reduce cutting speed, feed, or depth of cut until good cutting results are obtained. You may also need to increase the rigidity of your set-up. Do not hesitate to ask a TA for help [4].
Table 1: Cutting Speed Adjustment Factors for High Speed Steel Tools. Note that a slower cutting speed is used for deeper cuts and faster feeds. This table does not necessarily need to be used, but it demonstrates how depth of cut and feed rate are related to cutting speed [5].

<table>
<thead>
<tr>
<th>Feed</th>
<th>Feed Factor $F_f$</th>
<th>Depth of Cut</th>
<th>Depth of Cut Factor $F_d$</th>
</tr>
</thead>
<tbody>
<tr>
<td>in.</td>
<td>mm</td>
<td>in.</td>
<td>mm</td>
</tr>
<tr>
<td>0.002</td>
<td>0.05</td>
<td>1.50</td>
<td>0.005</td>
</tr>
<tr>
<td>0.003</td>
<td>0.08</td>
<td>1.50</td>
<td>0.010</td>
</tr>
<tr>
<td>0.004</td>
<td>0.10</td>
<td>1.50</td>
<td>0.036</td>
</tr>
<tr>
<td>0.005</td>
<td>0.13</td>
<td>1.44</td>
<td>0.031</td>
</tr>
<tr>
<td>0.006</td>
<td>0.15</td>
<td>1.34</td>
<td>0.047</td>
</tr>
<tr>
<td>0.007</td>
<td>0.18</td>
<td>1.25</td>
<td>0.062</td>
</tr>
<tr>
<td>0.008</td>
<td>0.20</td>
<td>1.18</td>
<td>0.078</td>
</tr>
<tr>
<td>0.009</td>
<td>0.22</td>
<td>1.12</td>
<td>0.094</td>
</tr>
<tr>
<td>0.010</td>
<td>0.25</td>
<td>1.08</td>
<td>0.100</td>
</tr>
<tr>
<td>0.011</td>
<td>0.28</td>
<td>1.04</td>
<td>0.125</td>
</tr>
<tr>
<td>0.012</td>
<td>0.30</td>
<td>1.00</td>
<td>0.150</td>
</tr>
<tr>
<td>0.013</td>
<td>0.33</td>
<td>0.97</td>
<td>0.188</td>
</tr>
<tr>
<td>0.014</td>
<td>0.36</td>
<td>0.94</td>
<td>0.200</td>
</tr>
<tr>
<td>0.015</td>
<td>0.38</td>
<td>0.91</td>
<td>0.250</td>
</tr>
<tr>
<td>0.016</td>
<td>0.41</td>
<td>0.88</td>
<td>0.312</td>
</tr>
<tr>
<td>0.018</td>
<td>0.46</td>
<td>0.84</td>
<td>0.378</td>
</tr>
<tr>
<td>0.020</td>
<td>0.51</td>
<td>0.80</td>
<td>0.438</td>
</tr>
<tr>
<td>0.022</td>
<td>0.56</td>
<td>0.77</td>
<td>0.500</td>
</tr>
<tr>
<td>0.025</td>
<td>0.64</td>
<td>0.73</td>
<td>0.558</td>
</tr>
<tr>
<td>0.028</td>
<td>0.71</td>
<td>0.70</td>
<td>0.618</td>
</tr>
<tr>
<td>0.030</td>
<td>0.76</td>
<td>0.68</td>
<td>0.680</td>
</tr>
<tr>
<td>0.032</td>
<td>0.81</td>
<td>0.66</td>
<td>0.738</td>
</tr>
<tr>
<td>0.035</td>
<td>0.89</td>
<td>0.64</td>
<td>0.833</td>
</tr>
<tr>
<td>0.040</td>
<td>1.02</td>
<td>0.60</td>
<td>1.000</td>
</tr>
<tr>
<td>0.045</td>
<td>1.14</td>
<td>0.57</td>
<td>1.250</td>
</tr>
<tr>
<td>0.050</td>
<td>1.27</td>
<td>0.55</td>
<td>1.250</td>
</tr>
<tr>
<td>0.060</td>
<td>1.52</td>
<td>0.50</td>
<td>1.773</td>
</tr>
</tbody>
</table>

For use with HSS tool data only from Tables 1 through 9. Adjusted cutting speed $V = V_{HSS} \times F_f \times F_d$, where $V_{HSS}$ is the tabulated speed for turning with high-speed tools.
Table 2: Suggested feed rates for various materials. Use half of these speeds if you are not using a CNC machine. This table also includes milling speeds which are not applicable to lathe operations. Parting speeds are half of turning speeds [2].

<table>
<thead>
<tr>
<th>Work Material</th>
<th>Suggested Feed Rate (Feet/Minute)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>H.S. Drill</td>
</tr>
<tr>
<td>Aluminium</td>
<td>60-200</td>
</tr>
<tr>
<td>Magnesium</td>
<td>100-400</td>
</tr>
<tr>
<td>Brass &amp; Bronze Hard</td>
<td>125-300</td>
</tr>
<tr>
<td>Malleable Iron</td>
<td>125-200</td>
</tr>
<tr>
<td>Steel Malleable</td>
<td>100-150</td>
</tr>
<tr>
<td>Inconel</td>
<td>50-100</td>
</tr>
<tr>
<td>316L Stainless Steel</td>
<td>50-100</td>
</tr>
<tr>
<td>Tungsten</td>
<td>50-100</td>
</tr>
<tr>
<td>Tool Steel</td>
<td>50-100</td>
</tr>
<tr>
<td>Plastic</td>
<td>50-100</td>
</tr>
<tr>
<td>Rubber</td>
<td>50-100</td>
</tr>
<tr>
<td>Wood</td>
<td>50-100</td>
</tr>
</tbody>
</table>
4. Lathe Basics

4.1 Operating the Lathe

**Setting Spindle Speed and Feed Rate**

RPM and feed values are listed on the headstock of the lathe. To set the speed and feed, simply find the correct value (or something close to it) on the headstock and shift the levers as indicated by the list. Make sure the spindle is not turning when shifting speeds and feeds.

**Turning on the Lathe**

Make sure the red emergency stop switch is not pushed in and the correct spindle speed is selected. The guard over the chuck must be in the down position to operate the lathe. Pull the clutch lever on the carriage toward you and then up to run the lathe spindle in forward. Pulling the clutch lever toward you and then down will put the lathe spindle in reverse. Be sure that the spindle is turning the correct direction before making a cut. You will not likely have a reason to ever run the machine in reverse. To turn the lathe off push the clutch handle back to the middle position (neutral). The foot brake can be used to stop the spindle after the lathe has been turned off.

**Feeding the Tool**

Tools mounted on the carriage are moved in the y-direction with the carriage handwheel, x-direction with the cross slide feed handle, and at an angle with the compound rest feed handle. Tools mounted in the tailstock are fed into the workpiece with the tailstock handwheel.
4.2 Mounting Tools/The Workpiece

Mounting the Workpiece

Workpiece (the metal being machined) is mounted to the spindle of the machine with a chuck, faceplate, or collet. Workpiece can also be mounted between centers. The use of chucks will be covered in this manual. (For details on other mounting methods see the course textbook.) If the workpiece extends more than 2.5 times its diameter out of the chuck, the end must be supported by the tailstock to prevent deflection.

1. Clean the jaws of the chuck with a rag or paper towel to ensure accuracy.

2. Insert the workpiece into the chuck as far as possible, but leave plenty of clearance between the area of the workpiece to be cut and the chuck (Figure 18).

3. Tighten the jaws onto the workpiece with the chuck key and make sure the workpiece is centered in the spindle (Figure 19). Never leave the chuck key in the chuck!

Figure 18: Insert workpiece into chuck.

Figure 19: Tighten jaws with chuck
Supporting the Workpiece with the Tailstock

1. If the workpiece must be supported, mount the workpiece with a small amount protruding from the chuck. Face and center-drill the end of the workpiece (Figure 20; for more information on facing and center drilling see Sections 5.1 and 5.3).

![Figure 20: The workpiece after facing and center-drilled.]

2. Mount the live center in the tailstock; slide the tailstock into position and lock it into place (Figure 21).

![Figure 21: The live center being mounted in the tailstock.]

3. Loosen the jaws of the chuck and slide the workpiece out to the desired location. Using the live center to align the workpiece, retighten the chuck (Figure 22).

![Figure 22: The workpiece being aligned to the live center and retightened in the chuck.]
4. Apply a small amount of pressure to the live center with the tailstock handwheel and lock the quill in place with the quill lock lever (Figure 23).

When cutting very long pieces of material, deflection may still create a problem. This problem can be solved by using a steady rest of a follow rest. A steady rest mounts directly to the bed of the lathe and supports the center of the workpiece. A follow rest is mounted to the carriage and follows the tool as the workpiece is cut. If the use of one of these attachments is needed, obtain help from a TA or a lab supervisor.

**Mounting Tools**

Note: For details pertaining to mounting specific tools refer to the Section 5.

**Tools mounted on the tailstock; drilling and reaming**

Drills and reamers are mounted in a Jacob’s chuck that is inserted into quill of the tailstock. The tools are moved by extending the quill with the tailstock handwheel. The tailstock also can be moved by sliding it across the lathe bed. When drilling, the tailstock must be locked into position with the tailstock locking lever.

1. Mount the drill bit or reamer in the Jacob’s Chuck (Figure 24).
2. Slide the Jacob’s Chuck into the quill of the tailstock. The end of the Jacob’s Chuck has a tab that must fit into a groove in the end of the tailstock. This tab prevents the chuck from rotating. To ensure that the Jacob’s Chuck is seated properly rotate it with your hands until you feel it slip into the groove (Figure 25). (The quill must be extended slightly for the tab to fit down into the groove.)

Figure 25: Ensure proper seating of the Jacob’s chuck in quill of tailstock

Tool mounted on the carriage; turning, parting, grooving, knurling, boring, etc.

Tools are mounted on the tool post which is attached to the cross-slide; the tool is then moved by precision movements of the cross-slide and carriage. All of the tools used in the ME 282 lab are mounted to tool holders that can be quickly attached to the tool post.

1. Mount the tool in the tool holder by tightening the Allen-head set screws (Figure 26)

Figure 26: Tighten Allen-head set screws of tool holder

2. Slide the tool holder onto the tool post. The tool post handle may need to be moved to allow the tool holder to slide down onto the tool post (Figure 27).

Figure 27: Slide the tool holder onto tool post.
3. Lock the tool holder in place by pulling the tool post handle towards you.

4. The tool must be centered vertically in relation to the workpiece mounted in spindle. To center the tool, place a ruler between the tool and the workpiece (Figure 28). Remove the ruler and adjust the height of the nut on the tool holder until the ruler is vertical when placed back in between the workpiece and the tool. The tool holder must be loose when adjusting the height of the tool and should be locked in place when using the ruler.

5. If the tool must be rotated, loosen the nut on top of the tool holder. Rotate the tool holder to the desired position and retighten the nut (Figure 29).

**Figure 28: Use ruler to vertically center tool to workpiece mounted in spindle.**

**Figure 29: Tighten tool holder in desired position.**

### 4.3 Measurement

Measuring the size of the workpiece being cut and measuring the movements of tools are both critical in precision machining. Lengths, outside diameters, and inside diameters of workpiece can be measured with a variety of measuring instruments including calipers and micrometers. Movements of the tool are measured with a digital readout and micrometer collars on the handwheels. In this manual we will introduce the caliper, digital readout, and micrometer collars.

**Caliper**

The caliper can measure outside diameters, inside diameters, and lengths with accuracy up to one-thousandth of an inch. Increments of one-tenth of
an inch are marked on the caliper body, and a dial gives a reading be-
tween zero and one-hundred thousandths of an inch. To read the caliper,
add the reading on the dial to the length of the last mark not covered on
the caliper body.

Before taking any measurements clean the jaws of the caliper by closing
them on a piece of paper and pulling the paper out. The caliper must also
be calibrated. After cleaning the jaws, close the caliper all the way. The
dial should read zero. If the dial does not read zero loosen the screw at
the bottom of the dial, rotate the dial until it reads zero, and retighten the
screw.

The jaws are opened and closed by rotating a wheel by running the thumb
over it (Figures 30 and 31). Pressure is also to be applied to the jaws
while taking a measurement.

Figure 30: A caliper reads the outside dimension of a workpiece by closing the jaws on it.
Figure 31: A caliper reads the inside dimension of a workpiece by opening the jaws on it.

The jaws on the front of the caliper are used to measure outside dimensions (Figure 30). The back of the caliper is used to measure internal features such as holes (Figure 31), and the end of the caliper is used to measure depths.

**Micrometer Collars**

The handwheels, which move tools, all have micrometer collars attached. These collars show the distance the tool is moved when the handwheels are rotated. Each dial is divided up into increments that represent a certain distance. For example, each mark on the compound rest feed handle represents one thousandth of an inch. The distance represented by each mark varies for the different handwheels and can be found on the micrometer collar.

Figure 32: A close-up of the compound rest micrometer collar.
Each increment on the micrometer collar for the cross slide feed handle represents two thousands of an inch (Figures 32 and 33.) The tool actually only moves one thousandth of an inch, but because this amount is removed from the radius of the workpiece a total of two thousands of an inch of material is removed from the diameter.

![Figure 33: The cross slide feed handle being set to zero.](image)

To set the micrometer collar to zero, loosen the lock nut while holding the collar. Twist the micrometer collar until it reads zero and retighten the lock nut while holding the collar in position.

**Digital Readout**

The digital readout indicates x and y coordinates of the tool. This is very convenient and easy to use, but must be used with caution. Before use, ensure that the digital readout is working properly by comparing its reading to the reading of the micrometer collars. This could save you from ruining a part.

When the tool is in a desired position, the x and y coordinates on the digital readout can be set to zero. To set a zero select which coordinate direction you would like to zero by pressing x or y. Enter zero or a desired coordinate and press enter. The digital readout should not show this new coordinate.
The x direction on the digital readout has two setting, radius and diameter. When the diameter light is lit in the upper left hand corner, the digital readout will indicate the total amount of material being removed from the diameter of the part. When this light is not lit, the digital readout will indicate the actual distance the tool is being moved. To change this setting press shift, press the diameter button, and then press the x. The diameter light should either light up or turn off (Figure 34).

4.4 Automatic Feed

The lathe can automatically feed the tool in both the x and y directions. Using automatic feed can give better results and surface finish than feeding the tool manually, but can also result in less control of the tool. A fast feed will remove material quickly and leave a rough surface finish; a slow feed will take more time, leaving a better surface finish. The usable feed rate will vary with the type of tool, depth of cut, material of the workpiece, rigidity of the setup, and the horsepower of the machine. For information on calculating feed rates see Section 3.

Caution must be used when using automatic feed. Always make sure that the feed is set up correctly by engaging the feed before cutting the part. Keep the carriage clear of the rotating chuck; if the feed runs the carriage into the chuck serious damage will be done to you and the machine! Never turn off the machine while the automatic feed is engaged as this will
break the tool. If you have not used automatic feed before, obtain help from a TA or shop supervisor.

Set the feed rate while the spindle is not turning by shifting the levers on the headstock to the location indicated on the lathe’s feed list (Figure 35).

The feed rate is measured in inches per revolution of the spindle. A table of different feed rate settings can be found on a plate on the headstock. To set the feed rate, find the feed rate desired on the table and set the selector levers to the correct locations. Only change the feed rate when the machine is turned off and the spindle is not turning.

![Figure 35: The levers located on the headstock.](image)

To use the automatic feed, engage the automatic feed lever. The automatic feed lever has three positions: up engages the y direction feed, middle is neutral, and down engages the x direction feed. If the tool feeds in the wrong direction, a lever on the headstock will change the feed direction.

### 4.5 Lubricants/Coolant

Lubricants and coolants (Figure 36) are used to cool and lubricate the tool. Coolants are made up of mostly water and do not provide much lubrication but are very useful when tools become too hot. Cutting oils are used to lubricate the cutting edge of the tool. These oils reduce friction and are therefore also help reduce heat. Use cutting oil when using high speed
steel tools (drills, reamers, taps, some parting and turning tools, etc.) Cutting oil is not generally required when using carbide tools [4].

Figure 36: A bottle of cutting oil (left) and a bottle of coolant (right).
5. Lathe Operations

5.1 Facing

Facing is used to make the end of a workpiece flat and trim the length of a part (Figure 38).

Mount the workpiece and facing/turning tool (Section 4.1). The end of the workpiece is being machined, so it can not be supported and must not extend more than 2.5 diameters from the chuck. Calculate the rpm required for facing the workpiece material and set the machine speed (Section 2). Facing speeds are identical to turning speeds.

Figure 37: The tool cutting slowly towards the middle of the workpiece.

Figure 38: A faced piece of workpiece
To face the part, begin cutting at the outside diameter and slowly feed the tool towards the middle with the cross slide feed handle (Figure 37). When the tool reaches the middle of the part, move the tool away with the carriage handwheel.

Note: If the tool is not centered properly, material will be left in the middle of the part when the operation is complete.

5.2 Turning

Turning is one of the most often used and important lathe operations. The turning operation allows the workpiece diameter to be cut down to any desired diameter. Different lengths of the workpiece can be turned to different diameters.

1. Mount the workpiece and facing/turning tool. Support the end of the workpiece if necessary (Section 4.1).

2. Calculate the rpm required for turning the workpiece material and set the machine speed (Section 2).

3. Begin the operation at the right hand side of the part (Figure 39). Set the depth of cut with the cross slide feed handle. Feed the tool slowly from right to left with the carriage handwheel until the desired length is cut down to the smaller diameter.

4. Repeat this process until the desired diameter is reached. Measure the diameter of the part between passes of the tool to ensure that the correct amount of material is removed (Figure 40).
5.3 Drilling/Reaming

Drill bits and reamers can be mounted in a Jacob’s Chuck fitted to the tailstock and fed into the end of the rotating workpiece to create a hole. When drilling a hole, a center drill must first be used to start the hole. A center drill is a rigid tool that will start the hole in the correct location; a regular drill bit will ‘walk’ on the surface of the workpiece before starting and will not be centered. After starting a hole with a center drill, a regular drill bit is used to drill the hole to the desired depth. If a large diameter hole is required, multiple drill bits should be used to progressively make the hole larger. Drill bits will not create a perfectly round hole and may not be very accurate, so reamers are used to make an accurate round hole with a good surface finish.

Mount the workpiece and Jacob’s chuck and drill bit on the lathe (Section 4.1)

Calculate and set the correct rpm for the tool and material being drilled (Section 2).
1. Slide the tailstock close to the workpiece and lock it in place (Figure 41).

2. Using cutting oil, advance the drill into the rotating workpiece with the quill feed handle (Figure 42).

3. After advancing the drill bit a short distance, back it out to clear the chips out of the flutes. Repeat this process until the desired depth is reached (Figure 43)

Note: Be sure to use the proper speeds and coolant when using a reamer! Reaming speeds are generally half that of drilling speeds. Reamers are not designed to remove large amounts of material. A drill bit slightly smaller than the reamer should first be used to drill a pilot hole.
5.4 Parting/Grooving

A parting tool is used to cut material off the piece of workpiece mounted in the chuck. The parting tool can also be used to create a groove. A more rigid grooving tool is also available for creating grooves. The use of this tool is recommended when the depth of the groove does not exceed the tools length.

Mount the workpiece and tool (Section 4.1). Be sure that the tool is centered with the workpiece.

Calculate and set the proper rpm for the tool and material being cut (Section 2). This speed will vary with the rigidity of the setup. Parting and grooving speeds are generally half of turning speeds.

Cut the groove by feeding the tool into the part with the cross slide feed handle (Figure 44). If a wide groove is required, back the tool out, move the carriage, and feed the tool back into the part with the cross slide feed handle. **Do not attempt to cut with the side of the tool by moving the carriage!**

If parting off the workpiece, continue to feed the tool with the cross slide feed handle until the end of the workpiece falls off (Figure 45).

---

Figure 44: Feeding the parting tool into the rotating workpiece to create a groove

Figure 45: The workpiece immediately after being parted off.
5.5 Boring

Boring is used to create a hole in the end of a part similar to drilling, but the boring operation creates a much more precise hole. A drill bit may not drill a perfectly round hole and will not drill perfectly straight. Boring produces a very round straight hole, and can leave a very good surface finish. Boring can also be used to produce an odd sized hole that could not be produced with a standard sized drill bit.

Mount the workpiece and boring bar as (Section 4.1). Pre-drill a hole large enough to accept the boring bar.

Calculate and set the proper rpm for the tool and material being cut (Section 2). Boring speeds are identical to turning speeds. Use the finished inside diameter for the rpm calculation.

Set the depth of cut with the cross-slide and feed the boring bar into the workpiece with the carriage handwheel (Figure 46).

Repeat this process until the desired hole size is obtained (Figure 47). Measure often between passes to ensure that the correct amount of material is removed.

Figure 46: Feeding the boring bar into the workpiece

Figure 47: A workpiece after boring has been completed to the desired hole size
5.6 Taper Turning

In the CTB 101 shop, tapers can be cut on a part by using the compound rest and the taper attachment. The compound rest is a quick and easy way to cut a taper, but is not suitable for cutting long tapers because the compound rest only feeds a few inches. The taper attachment requires more work to set up, but allows long, shallow tapers to be cut.

![Taper Turning](image)

**Compound Rest**

1. Loosen the three set screws on the compound rest, set it to the desired taper angle, and retighten the set screws (Figure 48)

![Figure 48: Loosening a screw on the compound rest.](image)

2. Mount a turning tool and workpiece (Section 4.1). Tool alignment with the centerline of the workpiece is critical when turning tapers. You will likely need to rotate the tool post to position the tip of the tool correctly (Figure 49). (Only the tip of the tool should cut into the workpiece.)

![Figure 49: The aligned tool with a turned](image)
3. Calculate and set the proper rpm for the tool and material being cut.

4. Position the tool by moving the carriage and the cross-slide.

5. Cut the taper by feeding the tool into the workpiece with the compound rest feed handle.

6. After making a pass, reposition the tool and make another pass (Figure 50). Repeat this process until the desired taper depth is reached (Figure 51).

The compound rest can also be used to cut a tapered bore with a boring tool.

**Taper Attachment**

If you have not used the taper attachment (Figure 52) or automatic feed before ask a TA for help.
1. Set the angle on the taper attachment by loosening the set screw and turning the knob on the attachment until the taper bar is in the correct location (Figure 53). Retighten the set screw.

2. Attach the taper attachment to the lathe bed with the bed clamp (Figure 54).

3. Mount a turning tool and the workpiece (Section 4.1).

4. Calculate and set the proper rpm for the tool and material being cut. Set up the automatic feed (Section 4.4).

5. Make a test run with the automatic feed to ensure the taper attachment is working properly before cutting the workpiece.

6. Position the tool and begin cutting the taper by engaging the
automatic feed (in the y-direction).

7. When the tool reaches the end of the taper, disengage the automatic feed and reposition the tool to make another pass (Figure 55).

8. Repeat this process until the desired taper depth is reached (Figure 56).

5.7 Knurling

A knurl is a diamond shaped pattern that is pressed into a part to give it better grip. Examples of knurls can be found on handles of many machines in the machine shop.
1. Mount the workpiece and the knurling tool (Section 4.1). Make sure that the workpiece is rigidly supported as large forces are used when creating a knurl. The distance between the rollers on the knurling tool can be adjusted to the diameter of the workpiece. If the knurling tool cuts chips off the workpiece, the rollers should be moved farther apart. Make sure that the knurling tool is square to the workpiece. Note: the knurling tools in CTB 101 are self centering and the nut on the tool holder does not need to be adjusted (Figure 57).

2. Set the rpm to 60 for knurling.

3. Using cutting oil, feed the knurling tool into the part with the cross-slide until the desired knurl depth is reached (Figures 58, 59 and 60).
6. **Sources**


