ESPL Machine Shop Training Course

Machine Operations:
• Turning
• Drilling & tapping
• Milling
• Bandsaw cutting

Topics Covered:
1. Theory  “Safety isn't an accident”
2. Operation and Best Practices  “Machines and tools can be replaced… student’s can’t”
3. Safety  Not wearing safety glasses, or working with a buddy will get you expelled from the machine shop!

Proper attire

• No long sleeves in the machine shop
• No gloves while operating machines
• Long hair must be tied up and back
• Close-toed shoes required
• Long pants recommended
• Safety glasses required
• No jewelry
Buddy system required!

- To use any tools in the machine shop you need a “buddy” to be with you at all times.
- This “buddy” must have passed the online Machine Shop Safety Test
- The “buddy” does not have to have passed the hands on training yet.

Calipers

1. Outside measurements
2. Inside measurements
3. Depth measurements
4. Main scale (mm)
5. Main scale (inch)
6. Vernier scale (mm)
7. Vernier scale (inch)
Digital readout (simple instructions)

Lathe:
• To zero, press ‘X’ or ‘Y’ on the right
• For offset, press ‘X’ or ‘Y’ on the pad, then enter a number, then ‘load’
• Check if readout is in diameter or radius mode. This is denoted by a “D” on the readout.

Mill
• Press ‘X’, then ‘ABS’ to zero or enter a number then ‘ENT’ to set an offset.
• Be sure to account for mill bit or edgefinder diameter when determining offset for milling.

Fair use & cleanliness

• Be considerate of others wanting to use machines
  – Everyone has deadlines and critical parts to make
  – Be considerate with how long parts take to make; tell others when machine will be free.

• Vacuum, clean machines and floor after EVERY USE
  – Do not leave machines dirty or with parts or tools still loaded.

• Not being considerate will result in temporary loss of access.
Turning

1. Theory

What is a turning operation?

• Turning – a single-point tool removes material from the surface of a rotating work piece on a Lathe.
### Parts of a lathe

- **Headstock**
- **Spindle**
- **Feed**
- **Tool post**
- **Tailstock**
- **Cross slide**
- **Carriage**
- **Ways**
- **Lead screw (threaded)**
- **Bed**

### Holding a workpiece

**3-Jaw**: jaws move simultaneously achieving automatic centering on round or hexagonal shaped parts. It applies even holding pressure quickest, but it is the least accurate method.

**4-Jaw**: each jaw moves independently. Permits odd shape work pieces and non-concentric turning, but takes time to accurately load.

**Spring Collet**: accurate and reasonably quick, but only for round parts < ~1.5” dia.
Holding workpiece in a lathe

Primary method is using a 3-Jaw:

4-Jaw:

Collet:

Tool post and carriage manipulation
Part identification - carriage

- TOOL POST
- CARRIAGE
- LEAD SCREW
- BED

Part identification - carriage

- CROSS SLIDE
- WAYS
- HAND CONTROLS
Part identification – carriage movement

- HAND CONTROLS
  - ADJUST X (cross feed)
  - ADJUST Y (feed)
  - CLUTCH LEVER

Tool handling

- Release tool using horizontal lever
- Ensure tools are placed in the proper location next to the lathe they are made for
  - Metric – Clausing Lathe
  - English – South Bend Lathe
Part identification – speed & feed

- SPEED CONTROLS
- FEED DIRECTION
- POWER SWITCHES

Part identification - tailstock

- DRILL CHUCK
- TAILSTOCK
- LIVE CENTER
Lathe operations

All cutting is made from **right-to-left** or from **out-to-in**

- **Turning** – cutting tool is fed parallel to the rotating axis reducing the outside diameter of the part.
- **Facing** – cutting tool is fed radially inward to create a flat face on the part end.

Lathe operations

- **Profile turning** – cutting tool is fed along a part or “profile” to obtained the desired geometry. CNC control is required.

- **Chamfering** – cutting tool is fed into an edge to break the sharp edge. If large amount of material is removed it becomes profile turning.
Lathe operations

• **Threading** – feed of cutting tool matches the rotational speed.

• **Knurling** – does not remove material but does create a rough, diamond, or square patterned surface for better grip.

Lathe operations

• **Drilling** – drill bit is *stationary*, mounted to the tailstock not toolpost. Hole is concentric.

• **Boring** – cutting tool removes material from an existing hole.

• **Parting or Cutoff** – a thin cutting tool is fed radially inward to remove a finished part.
Tool types

Ceramic Insert

Boring Bar

Cutoff or Parting Tool

Turning/Facing Tools

Cutting conditions in machining

• Material removal rate:
  \[ M_{RR} = v \cdot f \cdot d \]

  where
  \( v \) = cutting speed;
  \( f \) = feed;
  \( d \) = depth of cut

  cutting speed = \( \omega \cdot \pi \cdot D \)

  \( D \) - use outer most diameter

• Cutting Time:
  \[ T_c = \frac{L}{f \cdot \omega} \]
Roughing vs. finishing

• Roughing - removes large amounts of material Close to desired geometry (to about 0.010-0.030"
  – Feeds: large (~0.005 – 0.010” / revolution)
  – Cutting speeds: slow (50-80% of finishing speeds)

• Finishing - completes part geometry
  – Final dimensions, tolerances, and finish
  – Feeds: small (~0.001-0.005” / revolution)
  – Cutting speeds: fast (see chart)

• Biggest difference between them is: **Depth of Cut**

---

Recommended lathe maximum RPM

### Graph:

- **Aluminum**
- **Steel**
- **Stainless**

<table>
<thead>
<tr>
<th>Piece Diameter (inches)</th>
<th>Aluminum</th>
<th>Material: Steel</th>
<th>Stainless</th>
</tr>
</thead>
<tbody>
<tr>
<td>1/2</td>
<td>1400</td>
<td>1000</td>
<td>600</td>
</tr>
<tr>
<td>1</td>
<td>700</td>
<td>500</td>
<td>300</td>
</tr>
<tr>
<td>1 1/2</td>
<td>500</td>
<td>333</td>
<td>200</td>
</tr>
<tr>
<td>2</td>
<td>325</td>
<td>275</td>
<td>150</td>
</tr>
<tr>
<td>2 1/2</td>
<td>280</td>
<td>225</td>
<td>120</td>
</tr>
<tr>
<td>3</td>
<td>225</td>
<td>166</td>
<td>100</td>
</tr>
</tbody>
</table>

---

Bruce Flachsbart © 2019 University of Illinois at Urbana-Champaign, All Rights Reserved
Power requirement

The specific energy:

Cutting power per unit volume removal rate

\[ U = \frac{P_c}{M_{RR}} = \frac{F_c v}{v f d} = \frac{F_c}{f d} \]

Approximation: \( U \sim \text{hardness } H \)

- Cutting Power:
  \[ P_c = F_c \cdot v \]
  where \( F_c = \text{cutting force down on tool} \)
  \( v = \text{cutting speed}; \)

Specific energy (U) for cutting

Approximate specific-energy requirements

<table>
<thead>
<tr>
<th>MATERIAL</th>
<th>SPECIFIC ENERGY*</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>J/mm³</td>
</tr>
<tr>
<td>Al alloys</td>
<td>0.4-1.1</td>
</tr>
<tr>
<td>Cast irons</td>
<td>1.6-5.5</td>
</tr>
<tr>
<td>Copper alloys</td>
<td>1.4-3.3</td>
</tr>
<tr>
<td>High-Temp alloys</td>
<td>3.3-8.5</td>
</tr>
<tr>
<td>Mg alloys</td>
<td>0.4-0.6</td>
</tr>
<tr>
<td>Ni alloys</td>
<td>4.9-6.8</td>
</tr>
<tr>
<td>Refractory alloys</td>
<td>3.8-9.6</td>
</tr>
<tr>
<td>Stainless steels</td>
<td>3.0-5.2</td>
</tr>
<tr>
<td>Steels</td>
<td>2.7-9.3</td>
</tr>
<tr>
<td>Titanium alloys</td>
<td>3.0-4.1</td>
</tr>
</tbody>
</table>

* At drive motor, corrected for 80% efficiency;

1 hp = 550 ft-lbs/s = 396,000 in-lbs/min
Example – assuming a rough cut

Aluminum rod 1.5”, $U = 0.3$ hp.min/in³
Find:
RPM = 250 rpm   Feed = 0.005”   Depth of cut = 0.015”

$M_{RR}$ (in³/min) = $250(\pi1.5)0.005(0.015) = 0.0884$ in³/min.

Force down on tool post (lbs) = $U \cdot f \cdot d$
= $0.3$ hp.min/in³(0.005”)(0.015”)396,000 in∙lbs/min
= 8.91 lbs

Cutting Time (min) = $\frac{L}{f \cdot \omega} = \frac{3”}{(0.005”) \frac{500\text{rev}}{\text{min}}} = 0.24$ min

Example – assuming a finishing cut

Aluminum rod 1.5”, $U = 0.3$ hp.min/in³
Find:
RPM = 500 rpm   Feed = 0.001”   Depth of cut = 0.005”

$M_{RR}$ (in³/min) = $500(\pi1.47)0.001(0.005) = 0.0115$ in³/min.

Force down on tool post (lbs) = $U \cdot f \cdot d$
= $0.3$ hp.min/in³(0.001”)(0.005”)396,000 in∙lbs/min
= 0.594 lbs  (less than 1/10th the force of roughing)

Cutting Time (min) = $\frac{L}{f \cdot \omega} = \frac{3”}{(0.001”) \frac{500\text{rev}}{\text{min}}} = 0.6$ min
**Example – larger diameter part comparison**

Using the previous material, but assume a 3" diameter being reduced to 2.920" (everything doubled). Determine:

- \( \text{RPM} = 100 \text{ rpm} \)
- \( \text{Feed} = 0.005" \)
- \( \text{Depth of cut} = 0.035" \)

\[
M_{RR} \text{ (in}^3/\text{min)} = 100(\pi 3)(0.005)(0.035) = 0.165 \text{ in}^3/\text{min \ (double)}
\]

**Force down on tool post (lbs)**

\[
F = 0.3 \text{ hp.min/in}^3(0.005")(0.035")396,000 \text{ in} \cdot \text{lbf/min}
\]

\[
= 20.8 \text{ lbs \ (more than double)}
\]

**Cutting Time (min)**

\[
T = \frac{L}{f \cdot \omega} = \frac{3"}{(0.005")(100 \text{ rev/min})} = 6 \text{ min \ (25 times!)}
\]
Preparation & cleaning

- The first (and last) procedure is cleaning.
- Without clean equipment, the tolerance of the finished product diminishes.
- Make sure turning, facing, and parting tools are where they should and cutting edges are sharp
- Locate spindle chuck key (not in chuck!)
- Set cutting speed, turn on disconnect on wall
- Before starting the lathe double check that nothing will obstruct the Spindle movement.
- Plan out your work before you start.

Reduction of torque and vibrations

- Have the workpart extend a little as possible from the Spindle
- Have the cutting tool extend as little as possible from the Tool Post
- Make sure both are secure
- Make sure cutting edge is lined up with exact rotation center of workpart
- If a part is longer than \(~4x \text{ its diameter}\) it should be drilled with a centerdrill and held with a live center (rotates with spindle) to eliminate whirling and reduce vibration
Setting the speed and engaging the spindle

1. Set spindle speed
2. Power on lathe
3. Move clutch lever *upwards* to start spindle

Part facing

- Align cutting tip to center of rotating part
- Bring tool in close to part in Y direction
- SLOWLY move towards part until a small bit of material is removed
- Press the Y button on the DRO (Digital Readout)
- Move cutting tool inward (feed zeroing)
Autofeed and clutch usage

(1) Autofeed: move lever to the right and carriage will move
(1) Stop autofeed by replacing lever
(2) Stop spindle by moving clutch lever downward.
(2) Or, press brake on floor which stops spindle faster
(3) Switch off power before removing part

Lathe drilling

• Use a countersink bit (or center bit) to start the hole so the drill bit doesn’t “walk” along surface
• The countersink bit and then drill bit are mounted in the tailstock.
• The tailstock is slide forward and locked into place before advancing with handwheel.
“Pecking” when drilling

- When drilling holes deeper than 2x the diameter of the drill bit… use “Pecking”
  - Push drill bit into material ~half its diameter, then back the drill out to clear chips
  - Continue pecking as the hole becomes deeper as needed
  - Limits heat inside the part and wear of the drill bit

Material - best practices

- Steel
  - Hard material, requires special carbide bits, small bit of lubrication
  - Slow speeds help improve tool life
  - “Blue Chips” indicate a good feed/speed
- Aluminum
  - Softer material, can use higher speeds
  - Prone to noise, vibration
  - Using WD-40 can improve surface finish on final pass (remember to clean machine)
- Plastic
  - Very soft material
  - Prone to melting if speed/feed too high
  - Delrin is easiest to machine, burn resistant
Non-round workpieces

• Ensure non-round part is securely chucked

• Small feed passes until part is perfectly round

• Measure with calipers, then begin turning

Part handling

• Metal parts will be hot after some turning

• Metal parts will be VERY hot after parting
  – Allow time for the part to cool
  – Handle with a thick rag (after turning power off to the Lockout)
  – Use compressed air to assist with cooling.
3. Safety

Be aware of “snagging” potential

- Spinning headstock and workpart can snag (catch) loose material very quickly.

- Student at Yale University died when she got pulled into the machine. It was believed she was leaning over when her hair became entangled.

  [YouTube N9grSq-TWMQ](https://www.youtube.com/watch?v=N9grSq-TWMQ)
  Human versus Lathe?

  [YouTube KkaRMsF184](https://www.youtube.com/watch?v=KkaRMsF184)
Safety concerns

- Always know where the *emergency off switch* is.
- Never let go of the *chuck key* until it has been returned to its storage location (on the table).
- Before turning on the lathe, *rotate the chuck by hand* to ensure there is no obstruction of workpart.
- Never place *tools on top* of the ways of the lathe.
- Always stop the lathe before making measurements, adjustments, or cleaning.
- If *odd noise or vibration* occurs, then stop and investigate. Could be warning of something catastrophic (e.g. dull tool, etc.)

Things that are sharp...

- Remove sharp edges and burrs from the workpart before removing it from the lathe.
- Chips are sharp. Do not attempt to remove them with your hand, especially when built up around the toolpost. Stop the machine and use short sticks to collect them.
- When loading or unloading workparts, move the tailstock and the carriage away to avoid accidental contact with cutting tools.
Drilling and Tapping

1. Theory

Drill press

- Used to perform drilling, reaming, countersinking, counterboring, and tapping.
Drill bit basics

- Tool usually has 2 flutes and cuts only along the *rotational* axis.
- Can be performed on a mill or drill press (*toolbit rotates*) or a lathe (*workpart rotates*).
- *Typically 118° point angle, 90° angle for plastics,* and *150° angle for steels,* but requires a pilot hole.
- Chisel edge does not cut but cold works the material toward the cutting edges.

Through hole vs. blind hole

(a) Through hole – drill exits opposite side of work. *Do not drill into table or vice!*
(b) blind hole – drill does not exit opposite side.
Recommended RPL for drilling

<table>
<thead>
<tr>
<th>Drill Diameter (inches):</th>
<th>Aluminum</th>
<th>Steel</th>
<th>Stainless</th>
</tr>
</thead>
<tbody>
<tr>
<td>1/4</td>
<td>1200</td>
<td>750</td>
<td>500</td>
</tr>
<tr>
<td>3/8</td>
<td>900</td>
<td>500</td>
<td>375</td>
</tr>
<tr>
<td>1/2</td>
<td>600</td>
<td>250</td>
<td>250</td>
</tr>
<tr>
<td>5/8</td>
<td>360</td>
<td>200</td>
<td>150</td>
</tr>
<tr>
<td>3/4</td>
<td>300</td>
<td>150</td>
<td>120</td>
</tr>
<tr>
<td>1</td>
<td>180</td>
<td>100</td>
<td>85</td>
</tr>
</tbody>
</table>

More operations related to drilling

(a) **Reaming** – enlarges an existing hole to a precise dia.
(b) **Center drilling** – for starting holes to avoid drill bits from deflecting or “walking” along the surface.
(c) **Counterboring** – adds a blind flat to hide screw heads
(d) **Countersinking** – chamfer along leading edge for safety, to guide parts, or to hide flat head screws
Center punch

- A manual way to create a divot in a material to make it easier to start a hole, and keep drill bit from wandering during drilling.

Tapping - cutting internal threads

- A tap is a spiral cutting tool that is *tapered* over 3-5 threads.
- Bottom taps are tapered over 1-2 threads so can tap the bottom of blind holes.
- Tool is rotated *forward and backward* to break the cutting chips so they don’t jam the tap.
- Through holes are preferred to prevent clogging with chips.
- *Large torque* in needed since up to 20 chips are being created simultaneously.
- Use cutting or *tapping fluid*
## Maximum torque for thread size (inch-lbs)

<table>
<thead>
<tr>
<th>Bolt Size</th>
<th>18-8 Stainless Steel</th>
<th>Brass</th>
<th>Aluminum 2024-T4</th>
<th>316 Stainless Steel</th>
<th>Nylon</th>
</tr>
</thead>
<tbody>
<tr>
<td>2 - 56</td>
<td>2.5</td>
<td>2.0</td>
<td>1.4</td>
<td>0.4</td>
<td>2.6</td>
</tr>
<tr>
<td>4 - 40</td>
<td>5.2</td>
<td>4.3</td>
<td>2.9</td>
<td>5.5</td>
<td>1.19</td>
</tr>
<tr>
<td>4 - 48</td>
<td>6.6</td>
<td>5.4</td>
<td>3.6</td>
<td>6.9</td>
<td></td>
</tr>
<tr>
<td>6 - 32</td>
<td>9.6</td>
<td>7.9</td>
<td>5.3</td>
<td>10.1</td>
<td>2.14</td>
</tr>
<tr>
<td>6 - 40</td>
<td>12.1</td>
<td>9.9</td>
<td>6.6</td>
<td>12.7</td>
<td></td>
</tr>
<tr>
<td>8 - 32</td>
<td>19.8</td>
<td>16.2</td>
<td>10.8</td>
<td>20.7</td>
<td>4.30</td>
</tr>
<tr>
<td>8 - 36</td>
<td>22.0</td>
<td>18.0</td>
<td>12.0</td>
<td>23.0</td>
<td></td>
</tr>
<tr>
<td>10 - 24</td>
<td>22.8</td>
<td>18.6</td>
<td>13.8</td>
<td>23.8</td>
<td>6.61</td>
</tr>
<tr>
<td>10 - 32</td>
<td>31.7</td>
<td>25.9</td>
<td>19.2</td>
<td>33.1</td>
<td>8.20</td>
</tr>
<tr>
<td>1/4&quot; - 20</td>
<td>75.2</td>
<td>61.5</td>
<td>45.6</td>
<td>78.8</td>
<td>16.00</td>
</tr>
<tr>
<td>1/4&quot; - 28</td>
<td>94.0</td>
<td>77.0</td>
<td>57.0</td>
<td>99.0</td>
<td>20.80</td>
</tr>
<tr>
<td>5/16&quot; - 18</td>
<td>132.0</td>
<td>107.0</td>
<td>80.0</td>
<td>138.0</td>
<td>34.90</td>
</tr>
<tr>
<td>5/16&quot; - 24</td>
<td>142.0</td>
<td>116.0</td>
<td>86.0</td>
<td>147.0</td>
<td></td>
</tr>
<tr>
<td>3/8&quot; - 16</td>
<td>236.0</td>
<td>192.0</td>
<td>143.0</td>
<td>247.0</td>
<td></td>
</tr>
<tr>
<td>3/8&quot; - 24</td>
<td>259.0</td>
<td>212.0</td>
<td>157.0</td>
<td>271.0</td>
<td></td>
</tr>
<tr>
<td>7/16&quot; - 14</td>
<td>376.0</td>
<td>317.0</td>
<td>228.0</td>
<td>393.0</td>
<td></td>
</tr>
<tr>
<td>7/16&quot; - 20</td>
<td>400.0</td>
<td>357.0</td>
<td>242.0</td>
<td>418.0</td>
<td></td>
</tr>
<tr>
<td>1/2&quot; - 13</td>
<td>517.0</td>
<td>422.0</td>
<td>313.0</td>
<td>542.0</td>
<td></td>
</tr>
<tr>
<td>1/2&quot; - 20</td>
<td>541.0</td>
<td>443.0</td>
<td>328.0</td>
<td>565.0</td>
<td></td>
</tr>
</tbody>
</table>

https://www.engineersedge.com/torque_table_sae.htm

---

## Cutting external screw threads

- (left) Single-point thread cutting and (right) threading die

---

Bruce Flachsbart © 2019 University of Illinois at Urbana-Champaign, All Rights Reserved

slide 55
Drilling and Tapping

2. Operation and Best Practices

Drilling & Reaming

- Because the chisel area does not cut, it is recommended that for holes larger than 3/8” dia a pilot hole is drilled first.
- Use an interrupted feed, called peck drilling, to break up chips if they get too big.
- Precision holes should be started with a center punch, a center drill, or a mill bit.
- The cutting speed for reaming should be ~1/3 the speed used for drilling.
- Cutting fluid should always be used for reaming.
- Never stop the machine with the reamer in the hole.
Standard Operating Procedure

• Assemble Tap and Handle
• Clamp with hole pointed straight in desired direction
• Insert tap into hole and start to turn clockwise while applying a small bit of inward pressure

Standard Operating Procedure

• When tap starts to cut threads, ensure that it is aligned axially with the hole
• Continue to turn in for about 2 full turns
• Turn the handle backwards ¼-turn until resistance stops (this breaks the chips formed while cutting)
Standard Operating Procedure

• Turn tap handle ½-turn in, then ¼-turn back to break chips
• Continue as far as desired, then back tap completely out, same process for Die

Broken taps

• Try to back tap out with a strong pliers.
• If that doesn’t work, hardened material can not usually be drilled out, either hammered it out or try shattering a broken tap inside the hole.
• Broken taps are often the result of misalignment to a hole or overloaded chips
Drilling and Tapping

3. Safety

Precautions

• Clamping workpiece is critical
• Do not measure or removed part while drill is still rotating.
• Always remove the chuck key immediately after using it. A key left in the chuck can be thrown at a high velocity and cause injury.
• Remove burrs from drilled workpieces and sharp edges.
• Taps are much sharper than drill bits, so use a rag to handle and remove chips from the tap
Milling

1. Theory

- Fixed spindle with workpart clamped to precision 3-axis gantry
Parts of a Mill

Computer Numerical Control (CNC)

- Computer control of machining axis enable rounded and complex parts
Part identification

- HEAD
- SPEED ADJUSTMENT CRANK
- DIGITAL READOUT (DRO)
- HIGH-LOW GEAR
- QUILL FEED LEVER
- SPINDLE BRAKE

Part identification

- MOTOR
- DRAWBAR
- SPINDLE SWITCH
Part identification

- QUILL LOCK (Z-axis)
- SPINDLE
- DRILL CHUCK (interchangeable with collets)

Part identification

- TOE CLAMPS
- COLUMN
- VICE
- WORKTABLE
- HAND CRANKS
- X-AXIS LOCK
- Y-AXIS LOCK
Part identification

- ADJUST Y
- ADJUST X
- ADJUST Z
- TABLE HEIGHT
- LOCK IS BEHIND CASE

End Mill – standard mill cutting tool

- 2-flutes (Aluminum & plastics) and 4-flutes (steel, etc.)
  - More flutes are *stiffer* and provide *better surface finish*,
  - Less flutes *increase material removal* rate of soft materials and are *less expensive*.
- All 2-flute mill bits can plunge straight into the material and cut laterally. But not all.
  4-flute mill bits can plunge.
Other types of mill cutting bits

- **Ball Mill** – creates inside corner radius, but not flat surfaces. Used with CNCs.
- **“Bull Nose” Mill** – makes smooth surfaces like standard mill and creates fillets in corners like ball mill.
- **Tapered Mill** – provides uniform draft angles to sides (e.g. for mold casting)
- **Chamfer Tool** – removes sharp edges
- **Outside Corner Rounding Tool**
- **Face Mill** – large diameter mill to create very flat surfaces. They do not remove much material.

Climb vs conventional milling

- In **Climb** milling the tool is “pulled” into the work material, deflecting into the part, so gantry table must be very stiff. Can produce a smoother finish but use only for finishing cuts.
- **Conventional** milling is safer especially for older tools. Use for ALL roughing cuts, and finishing cuts if you want.
- All plastics should be conventionally milled (i.e. anything sticky)
Design Advisor – limit your setup changes

- Design for fewest number of part setup changes
  - Features created in the *same setup* will have *better dimensional accuracy* than using multiple setups

![Diagram](image1)

Design Advisor – corners & edges

- Sharp 3-point corners *cannot be machined*
- Use *fillet* on inside corner *parallel to cutting axis*
- Use fillet with the same radius as the tool (e.g. for 0.25” end mill, make fillet radius of *0.125”*)
- To mate to a rectangular part, *overcut corners*

![Diagram](image2)
Design Advisor – limit any thin sections

- Parts with narrow features, thin walls, or long protrusions should be avoided. Why?
  - *Part must be rigid enough to withstand cutting forces*

![Diagram illustrating the concept of thin sections and their impact on part rigidity.]

Design Advisor – chamfer outside corners

- Unlike inside corners it is easier and less expensive to chamfer outside corners.

![Diagram illustrating the benefits of chamfering outside corners.]

Bruce Flachsbart © 2019 University of Illinois at Urbana-Champaign, All Rights Reserved
Design Advisor – tool size and reach

- Design part so that largest tool can be used.
- **Larger** tools are *more expensive*, but *wear longer* and are stiff enough to create *deep features fast*.

Design Advisor – undercut features

- Undercut features require special tools or multiple setups. Avoid if at all possible.
Mill

2. Operation and Best Practices

Preparation

- Clean machine and immediate floor with vacuum after each use
- Sparingly use cutting oil during large milling operations, vacuum away oil after each pass to ensure no buildup occurs
Part clamping & using parallels

- Make sure *chips* are cleared from vice.
- Use *parallels* for small parts, raising parts from vice bottom.
- Tighten clamp partially, then use a *rubber mallet* to hammer part against parallels.
- Clamp workpart firmly in vice.

Part and tool loading

1. Clamp part firmly into vice.
2. Adjust Z-height to fit the tool, then lock.
3. Insert collet and end mill into spindle – Quill feed must be raised all the way to access drawbar.
4. While holding collet with one hand, tighten the spindle drawbar with the other hand until hand tight.
5. Hold brake and use wrench to firmly tighten drawbar. *Then remove wrench from Drawbar.*
Running spindle

1. Press black button to power on the machine
2. Turn switch to “High Speed” to start spindle
3. **AFTER** spindle reaches full speed, adjust spindle speed by turning the speed adjustment crank
   - Verify mill bit is turning clockwise!

The Blue Bridgeport has a DIFFERENT way to adjust speed. Adjust belts when breaker is off.

Controlling direction and speed of spindle

- **Forward/Reverse**
- **High/Low gear**
- **Speed Dial Indicator**

<table>
<thead>
<tr>
<th>Operation</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Forward + High gear</td>
<td>Forward Fast – read <em>inside</em> of dial indicator</td>
</tr>
<tr>
<td>Forward + Low gear</td>
<td>Reverse Slow – read outside of dial</td>
</tr>
<tr>
<td>Reverse + High gear</td>
<td>Reverse Fast – read inside of dial</td>
</tr>
<tr>
<td>Reverse + Low gear</td>
<td>Forward Slow – read <em>outside</em> of dial</td>
</tr>
</tbody>
</table>
Recommended RPM for milling

- Limit plunging to **1 ½** times the mill tool diameter

Depth of cut and feed limits
Milling workpart

- Use hand cranks to move part relative to end mill in X and Y directions
- Use lever to move end mill up/down in Z
- Use lock levers to stop motion in one direction or to create resistance to finely adjust a position

Tool unloading

- To remove loosen the drawbar on top of mill HEAD
- Keep your hand below the end mill to catch it
- Replace collets when finished
- If collet gets stuck, tap drawbar top with wrench to free it.
How to precisely locate a surface using an edgefinder

- Edgefinder: “finds” the edge of a part
- Move spindle away from the workpart, and insert edgefinder into 3/8” collet.
- AFTER starting the spindle, turn speed down to ~800 rpm.
- Choose one axis to move in from, and lock all others.
- Move edgefinder and very slowly touch it against the workpart until both parts of the edgefinder become concentric.
- When the edgefinder again pushes off to one side, the edge has been “found.”
- Zero that axis on the DRO (Digital Readout)

Ways to use the Edgefinder

- The diameter of the edgefinder is 0.2000”
- To set zero ON the edge, move the axis in by 0.1000” and then re-zero the DRO
- To set the zero to reference a hole or feature, add/subtract the 0.1000” to account for the center to edge offset of the edgefinder.
When finished…

- When finished milling, turn the large switch to the “off” position
- Press the red power button to cut power to the machine
- Remove part from vice
- Vacuum chips, fluid from table, and immediate floor area.
- If you move the Mill Head you must see Dave to re-indicate the Mill Head.
- Turn the disconnect at the wall off.

Milling

3. Safety
Potential milling accidents

- Part comes loose from vice
  - Securely fastened using vice or onto the table
  - Never clamp multiple parts in vice
- Hands or clothes getting wrapped in end mill
  - Keep a safe distance
- Breaking end mills (possible parts flying)
  - Do not climb mill when roughing
  - Do not dive quickly (plunge) into material.
  - Do not use high feed rates
  - Do not run mill into vice or table.

Other safety precautions

- Make sure the red power button is OFF before changing mill bit.
- Stop the machine before making any measurements.
- Use shields or guards to deflect chips
- Stop the machine before removing chips
- Use a rag to protect your hands from mill tool cutting edges when loading and unloading.
Bandsaw

1. Theory

Band saw

Workpiece is fed into linear cutting blade
Part identification

BLADE START

STOP

There is another stop button below the work table!

Twist button to restart machine

Part identification

• BLADE TENSIONER

Set tension for ½” wide blade
Part identification

- BLADE GUIDES
- GUARD
- BLADE
- WORK TABLE

Fine vs coarse blade

- Fine blade is for:
  - steel
  - carbon fiber (may require separate blade)
  - very dense plastics
- Coarse blade is for:
  - aluminum
  - wood
  - fiberglass
  - most plastics
Bandsaw

2. Operation and Best Practices

Standard operating procedure

- Ensure correct blade is installed and properly tensioned (set to 1/2”)
- Close doors and set guard height to 1/4” higher than part thickness
- Position part as desired for cut
- Press BLADE START
How to change the blade

- Cut power to machine
- Open top and bottom doors
- Loosen blade tension until blade sags
- Carefully pull blade from guards and around doors
- Wind blade and hang on shelf
- This step will be demonstrated during laboratory session

Changing the blade

- Carefully replace new blade ensuring that the sharp end points towards you
- Set blade on top of top pulley and fit through guards
- Set blade around bottom pulley, tighten or loosen tension as necessary
- Once aligned on wheels and guides, tighten the blade tension to \( \frac{1}{2} \)"
- Close doors and guards
Part handling

- Hold part securely with both hands
- Place hands equally apart and never inline with the blade
- Do not let fingers come within 2” of moving blade
- Press firmly, but not with entire body weight
- If blade deviates from straight cut, turn part as necessary, but only <5°. Cut in segments if required
- If parts are long, have a helper support one end.

Blade handling

- Ensure blade is facing right direction
- Be careful with sharp end of blade while loose
- To wind blade, step on bottom and turn twice with hands while folding over
- Replace blade if worn, broken or missing teeth
Horizontal bandsaw

- On switch that automatically shuts off when blade finished cutting.
- Part Clamp
- Lever to keep blade from lowering

- *No hollow tubes* are to be cut on this tool!
- Used only for *large square stock* >1” in all directions

Bandsaw

3. Safety
Potential bandsaw accidents

- **Clothing snagged by the cutting blade**
  - Do not lean close to blade or touch blade when moving

- **Part breaks or slips and hands end up in the blade**
  - Do not lean on part while cutting
  - When close to an edge, reduce pressure and cut the last bit slowly

- **Debris inhalation**
  - Wear proper PPE (respirator & vacuum) when cutting composites

- **Blade breaks**
  - Turn motor off immediately by pressing a STOP button
  - Guards are in place to ensure the blade will not fly off

How much is TOO much force?

- Do not use entire body weight to push part into blade

- Apply firm, but controlled pressure with hands against part, this will reduce chip load and make cutting faster and cleaner
Accidents - bad things can happen

- These last slides are to show what can happen when people are careless - machine shop tools are very powerful!
- It only takes a split second for a bad decision to cause a loss of a finger, an arm, an eye, or your life.
- Almost every shop accident starts with the belief “it can never happen to me”
- Don’t be the person who is scared for life because you were careless or rushing for a deadline.

Machine shop accidents:

What happens when flesh meets a moving cutting edge.

What happens when a moving cutting edge meets a gloved hand.
When tools fly…

Don’t drill or mill carelessly

Didn’t clamping down a workpiece  Why you don’t Use Gloves
Air hose accident

- Careful using air hose to blow off dust or debris, because eyes are very sensitive to compressed air flows!
- Quick disconnect fittings can retain enough pressure to remove teeth.

Always *respect the equipment* and stay SAFE!