ESPL Machine Shop Training Course

Machine Operations:
- *Turning*
- *Drilling & tapping*
- *Milling*
- *Bandsaw cutting*

Topics Covered:
1. Theory
2. Operation and Best Practices
3. Safety

"Safety isn't an accident"

"Machines and tools can be replaced… student's can't"

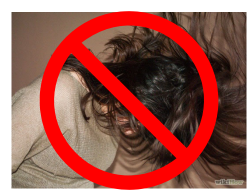
Not wearing safety glasses, or working with a buddy will get you expelled from the machine shop!

 slide 1

ILLINOIS

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

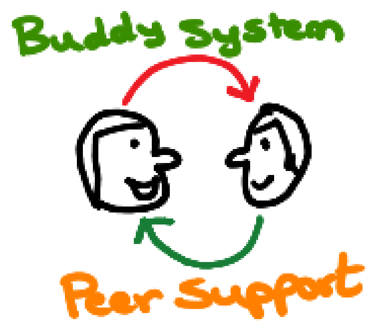


 slide 2

ILLINOIS

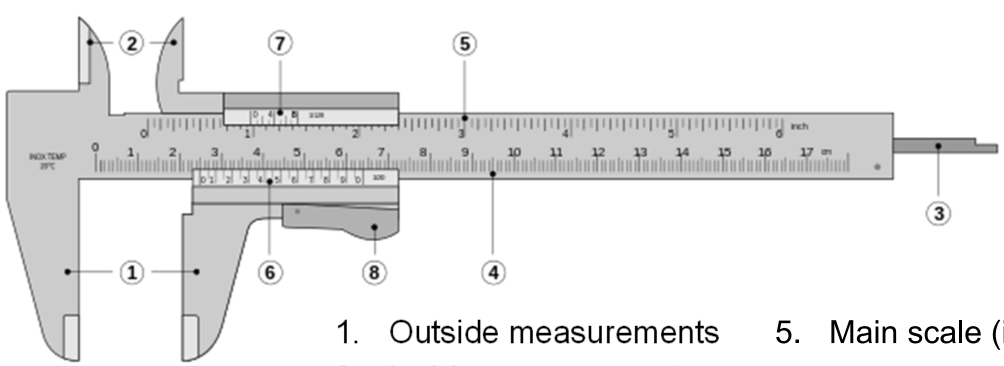
Buddy system required!

- 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.



ILLINOIS

Calipers

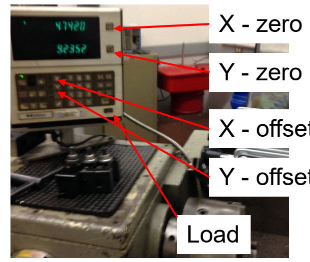


1. Outside measurements
2. Inside measurements
3. Depth measurements
4. Main scale (mm)
5. Main scale (inch)
6. Vernier scale (mm)
7. Vernier scale (inch)

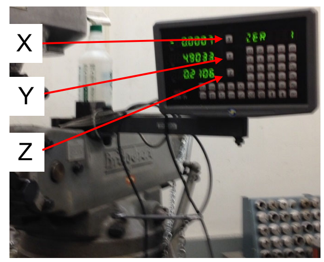
ILLINOIS

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.



ILLINOIS

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.**

ILLINOIS

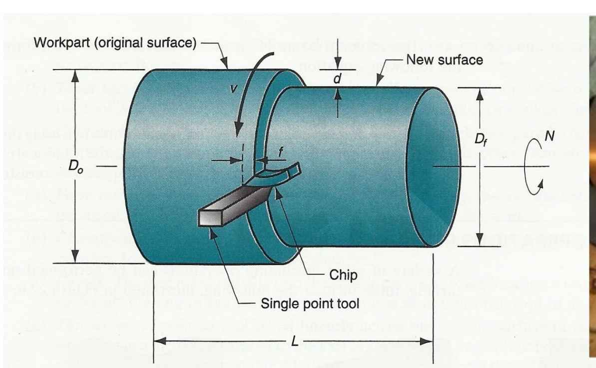
Turning

1. Theory

ILLINOIS

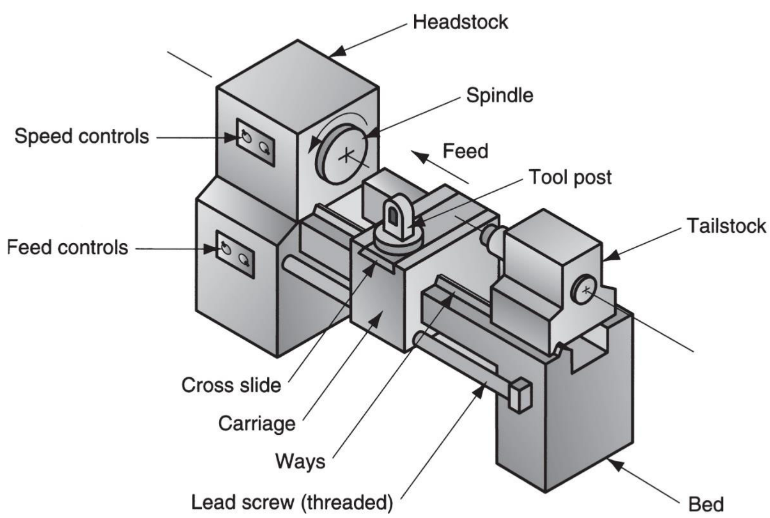
What is a turning operation?

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



ILLINOIS

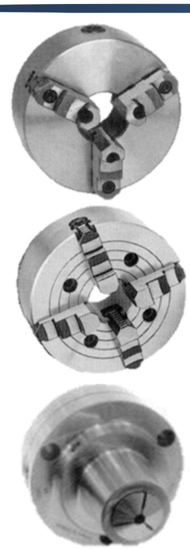
Parts of a lathe



 slide 9

ILLINOIS

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.

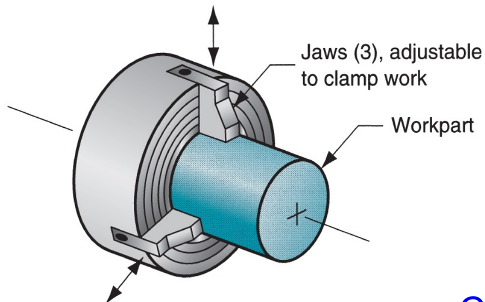
Spring Collet: accurate and reasonably quick, but only for round parts < ~1.5" dia.

 slide 10

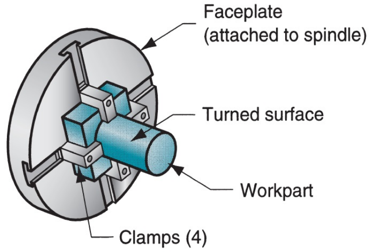
ILLINOIS

Holding workpiece in a lathe

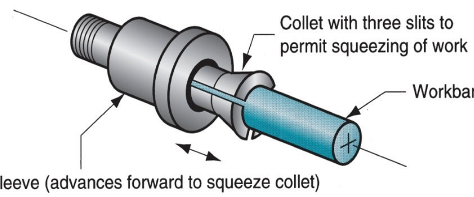
Primary method is using a 3-Jaw:



4-Jaw:

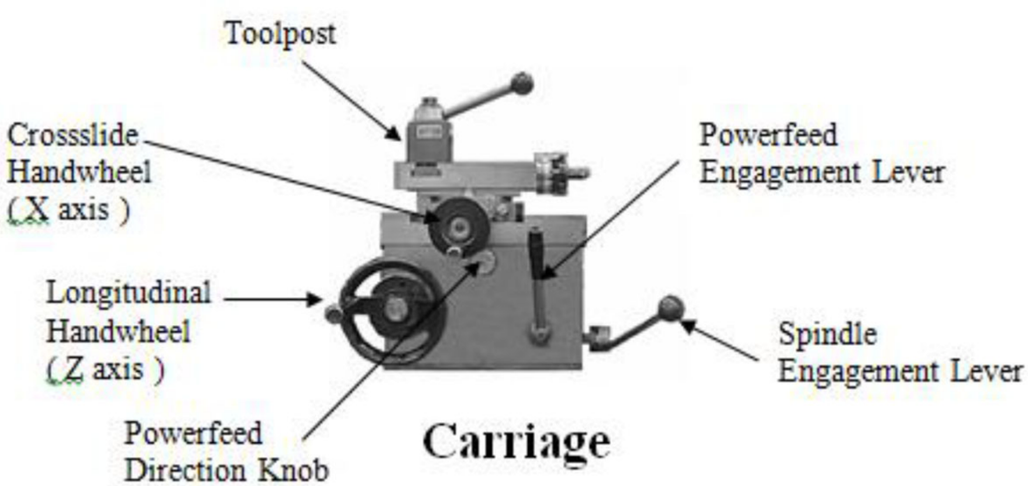


Collet:



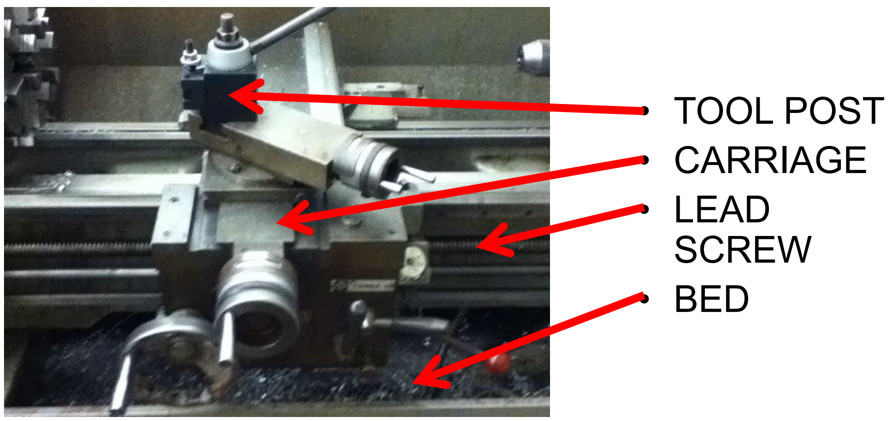
 slide 11 ILLINOIS

Tool post and carriage manipulation



 slide 12 ILLINOIS

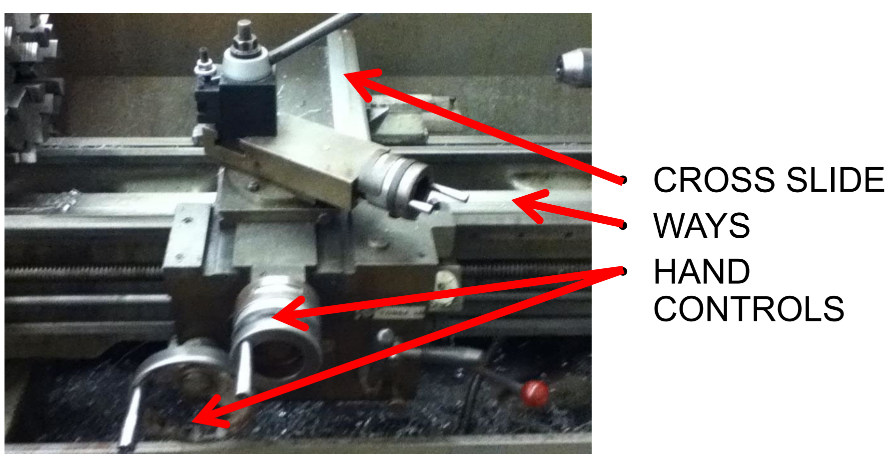
Part identification - carriage



 slide 13

ILLINOIS

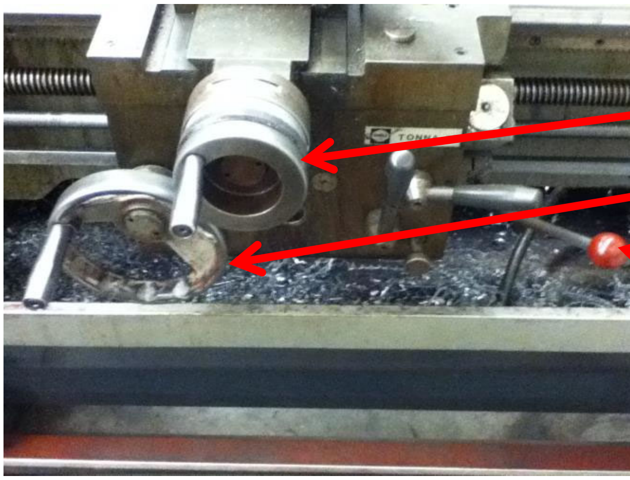
Part identification - carriage



 slide 14

ILLINOIS

Part identification – carriage movement



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

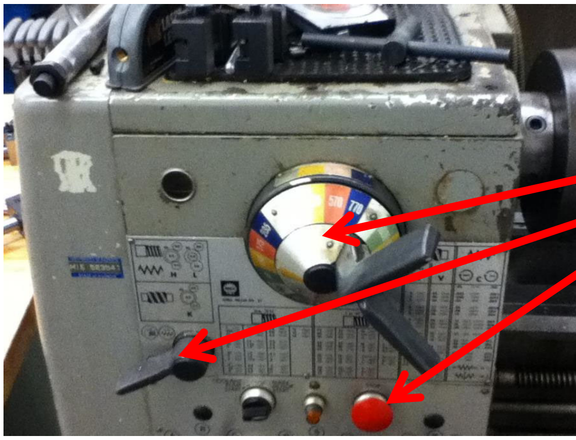
ILLINOIS

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

ILLINOIS

Part identification – speed & feed



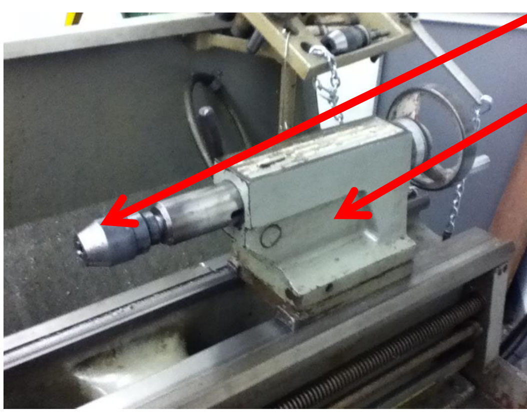
- SPEED CONRTOLS
- FEED DIRECTION
- POWER SWITCHES

 slide 17

ILLINOIS

Part identification - tailstock

- DRILL CHUCK
- TAILSTOCK
- LIVE CENTER



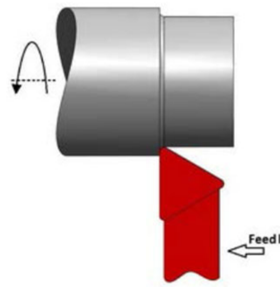
 slide 18

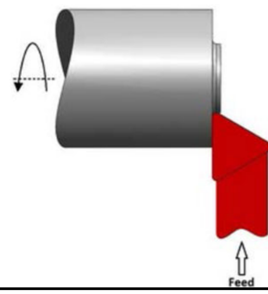
ILLINOIS

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.



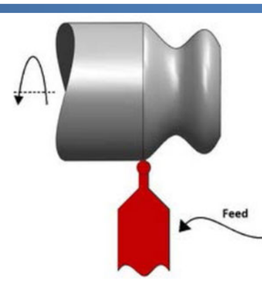


 slide 19

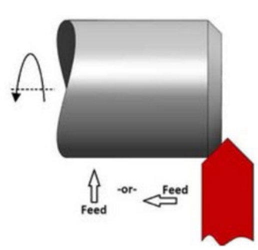
ILLINOIS

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.

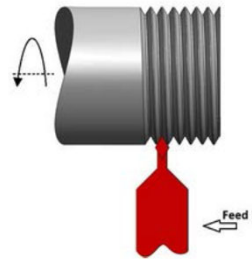


 slide 20

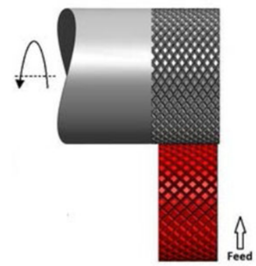
ILLINOIS

Lathe operations

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



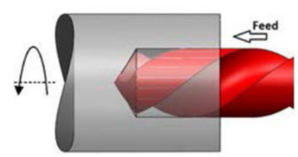
- *Knurling* – does not removed material but does create a rough, diamond, or square patterned surface for better grip.



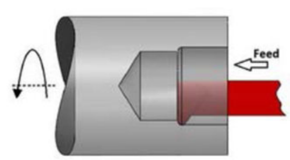
ILLINOIS

Lathe operations

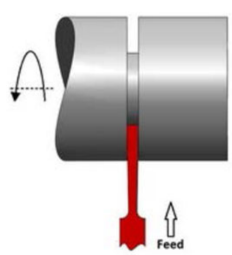
- *Drilling* – drill bit is *stationary*, mounted to the tailstock not toolpost. Hole is concentric.



- *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.



ILLINOIS

Tool types

Ceramic Insert

Boring Bar

Turning/Facing Tools

Cutoff or Parting Tool

Cutting conditions in machining

- Material removal rate:

$$M_{RR} = v \cdot f \cdot d$$

where v = *cutting speed*;

f = *feed*;

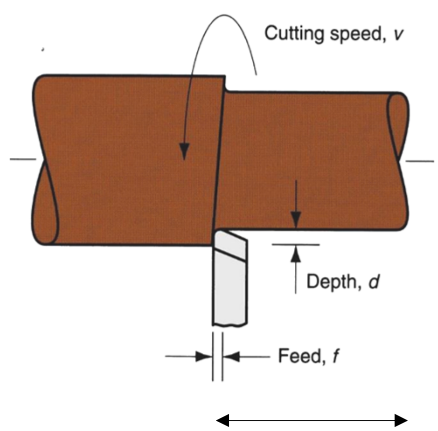
d = *depth of cut*

cutting speed = $\omega \cdot \pi \cdot D$

(D - use outer most diameter)

- Cutting Time:

$$T_c = \frac{L}{f \cdot \omega}$$



Roughing vs. finishing

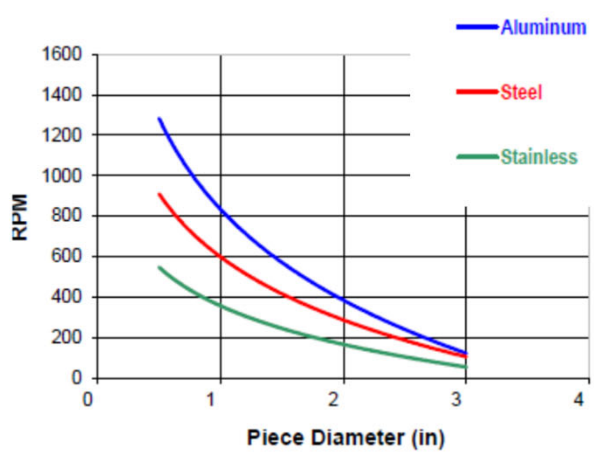
- Roughing - removes large amounts of material Close to desired geometry (to *about* ***0.010****-0.030"*)
  - Feeds: *large (~****0.005*** *– 0.010" / revolution)*
  - Cutting speeds: *slow (****50****-80% of finishing speeds)*
- Finishing - completes part geometry
  - Final dimensions, tolerances, and finish
  - Feeds: *small (~****0.001****-0.005" / revolution)*
  - Cutting speeds: *fast (see chart)*
- Biggest difference between them is: ***Depth of Cut***

 slide 25 ILLINOIS

Recommended lathe maximum RPM

Piece Diameter (inches): / Material:

Piece Diameter (inches)	Aluminum	Steel	Stainless
1/2	1400	1000	600
1	700	500	300
1 1/2	500	333	200
2	325	275	150
2 1/2	280	225	120
3	225	166	100



 slide 26 ILLINOIS

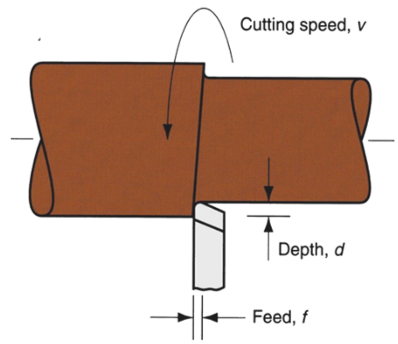
Power requirement

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 ~hardness H



- Cutting Power:

$$P_c = F_c \cdot v$$

where F_c = *cutting force down on tool*

v = *cutting speed*;

ILLINOIS

Specific energy (U) for cutting

Approximate specific-energy requirements

MATERIAL	SPECIFIC ENERGY*	
	J/mm^3	$hp\text{-}min/in^3$
Al alloys	0.4-1.1	0.15-0.4
Cast irons	1.6-5.5	0.6-2.0
Copper alloys	1.4-3.3	0.5-1.2
High-Temp alloys	3.3-8.5	1.2-3.1
Mg alloys	0.4-0.6	0.15-0.2
Ni alloys	4.9-6.8	1.8-2.5
Refractory alloys	3.8-9.6	1.1-3.5
Stainless steels	3.0-5.2	1.1-1.9
Steels	2.7-9.3	1.0-3.4
Titanium alloys	3.0-4.1	1.1-1.5

* At drive motor, corrected for 80% efficiency;

1 hp = 550 ft·lbs/s = 396,000 in·lbs/min

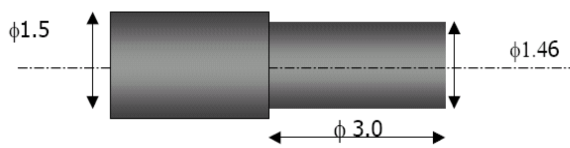
ILLINOIS

Example – assuming a rough cut

Aluminum rod 1.5”,

U = 0.3 hp.min/in^3

Find:



RPM = 250 rpm Feed = 0.005” Depth of cut = 0.015”

M_{RR} (in^3/min) = *250(π1.5)0.005(0.015)* = 0.0884 in^3/min.

Force down on tool post (lbs) = *U • f • d*

= 0.3 hp.min/in^3(0.005”)(0.015”)396,000 in·lbs/min

= 8.91 lbs

Cutting Time (min) = $\frac{L}{f \cdot \omega} = \frac{3"}{(\frac{0.005"}{rev})(\frac{250rev}{min})}$ = 0.24 min

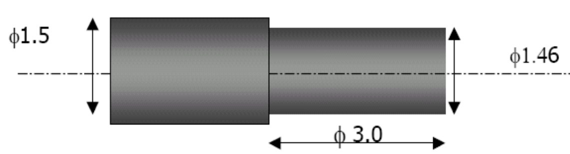
ILLINOIS

Example – assuming a finishing cut

Aluminum rod 1.5”,

U = 0.3 hp.min/in^3

Find:



RPM = 500 rpm Feed = 0.001” Depth of cut = 0.005”

M_{RR} (in^3/min) = *500(π1.47)0.001(0.005)* = 0.0115 in^3/min.

Force down on tool post (lbs) = *U • f • d*

= 0.3 hp.min/in^3(0.001”)(0.005”)396,000 in·lbs/min

= 0.594 lbs *(less than 1/10th the force of roughing)*

Cutting Time (min) = $\frac{L}{f \cdot \omega} = \frac{3"}{(\frac{0.001"}{rev})(\frac{500rev}{min})}$ = 0.6 min

ILLINOIS

Example – larger diameter part comparison

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

RPM = 100 rpm   Feed = 0.005”   Depth of cut = 0.035”

M_{RR} (in^3/min) = *100(π3)0.005(0.035)* = 0.165 in^3/min (double)

Force down on tool post (lbs) = *U • f • d*

= 0.3 hp.min/in^3(0.005”)(0.035”)396,000 in·lbs/min

= 20.8 lbs (more than double)

Cutting Time (min) = $\frac{L}{f \cdot \omega} = \frac{3"}{(\frac{0.005"}{rev})(\frac{100rev}{min})}$ = 6 min (*25 times!*)

ILLINOIS

Lathe

2. Operation & Best Practices

ILLINOIS

Preparation & cleaning

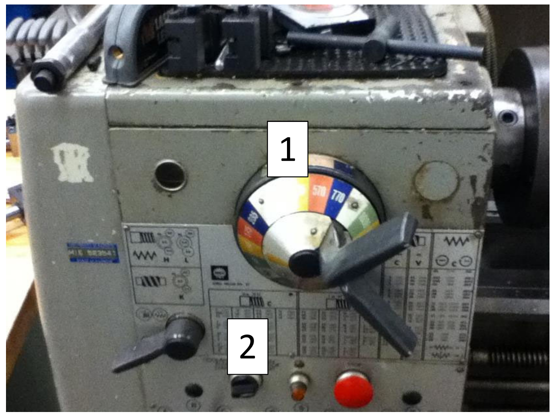
- The first (and last) procedure is cleaning.
- Without clean equipment, the tolerance of the finished product diminishes.
- Make sure turning, facing, and parting tools are where they should and cutting edges are sharp
- Locate spindle chuck key (not in chuck!)
- Set cutting speed, turn on disconnect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

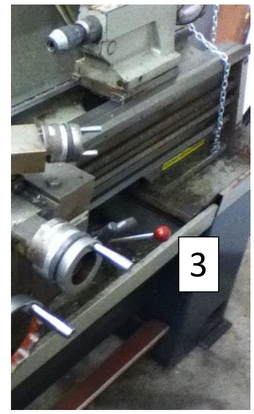
Reduction of torque and vibrations

- 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 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

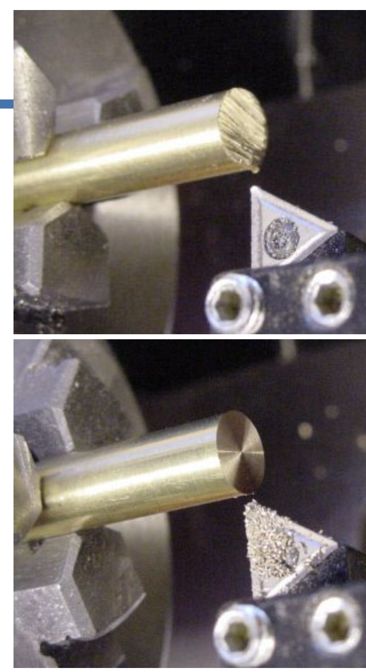




ILLINOIS

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)



 slide 36

ILLINOIS

Autofeed and clutch usage

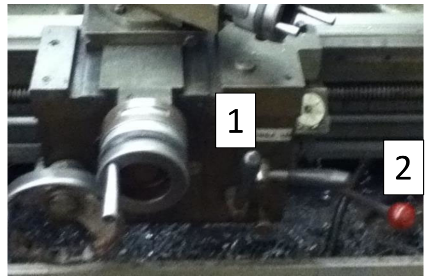
(1) Autofeed: move lever to the right and carriage will move

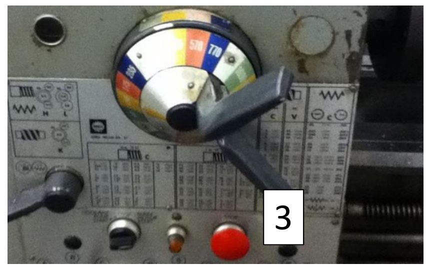
(1) Stop autofeed by replacing lever

(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





I ILLINOIS

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.

I ILLINOIS

“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

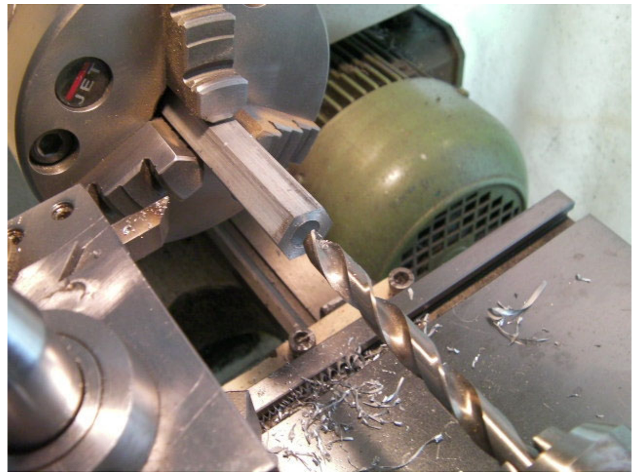
Material - best practices

- 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

ILLINOIS

Non-round workpieces

- Ensure non-round part is securely chucked
- Small feed passes until part is perfectly round
- Measure with calipers, then begin turning



ILLINOIS

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.

ILLINOIS

Lathe

3. Safety

 slide 43

ILLINOIS

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

Human versus Lathe?

YouTube KkaRMsF1l84

 slide 44

ILLINOIS

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.

ILLINOIS

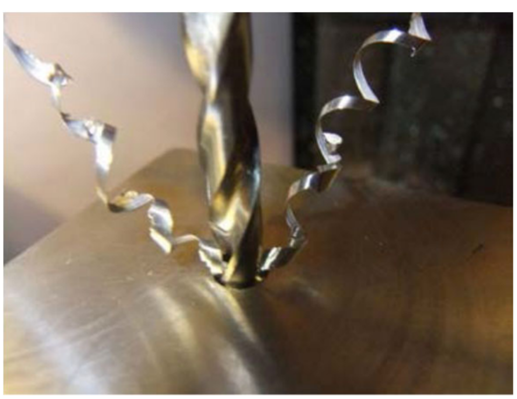
Drilling and Tapping

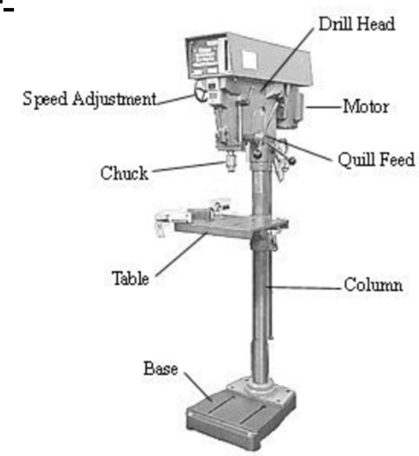
1. Theory

ILLINOIS

Drill press

- Used to perform drilling, reaming, counter-sinking, counterboring, and tapping.



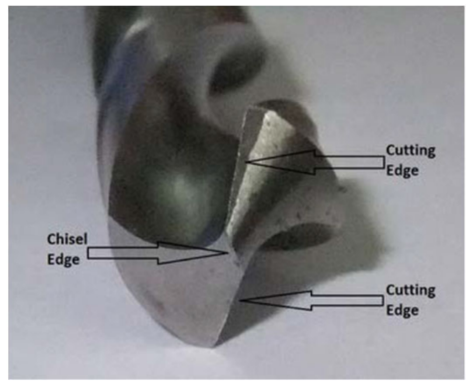


Drill Press

ILLINOIS

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.

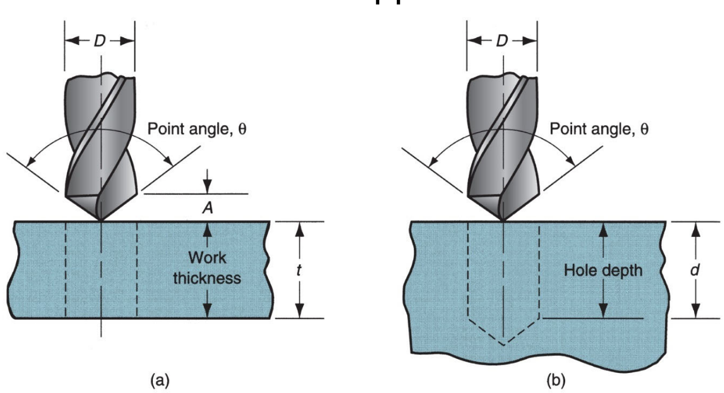


 slide 49 ILLINOIS

Through hole vs. blind hole

(a) Through hole – drill exits opposite side of work.
Do not drill into table or vice!

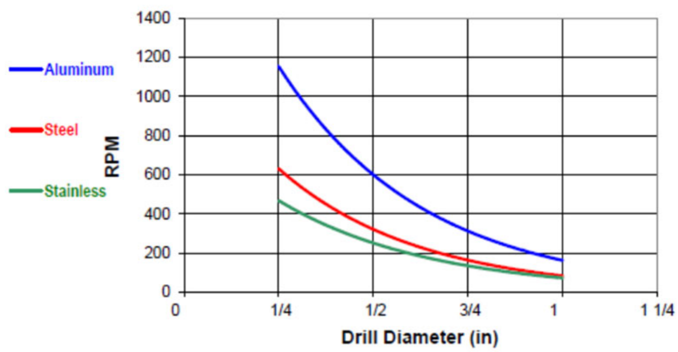
(b) blind hole – drill does not exit opposite side.



 slide 50 ILLINOIS

Recommended RPL for drilling

Drill Diameter (inches):	Material: Aluminum	Steel	Stainless
1/4	1200	750	500
3/8	900	500	375
1/2	600	250	250
5/8	360	200	150
3/4	300	150	120
1	180	100	85



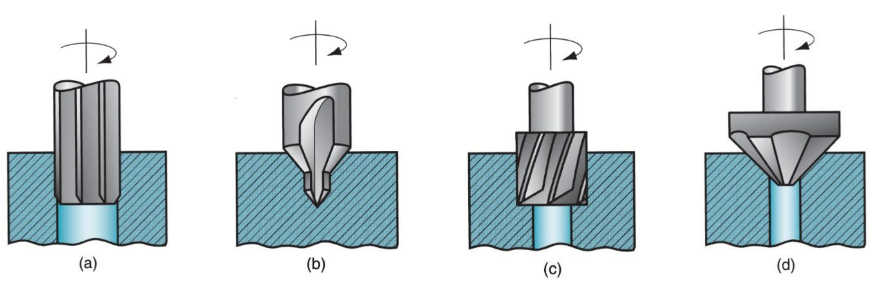
More operations related to drilling

(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.

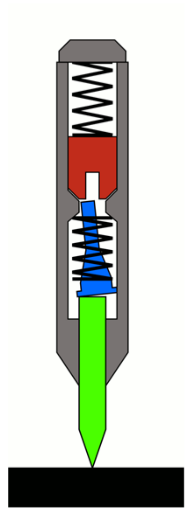
(c) Counterboring – adds a blind flat to hide screw heads

(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.

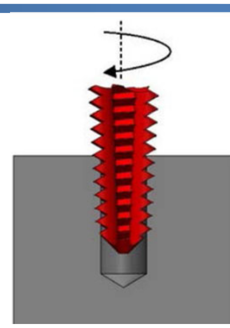


 slide 53

ILLINOIS

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*



 slide 54

ILLINOIS

Maximum torque for thread size (inch-lbs)

Bolt Size	18-8 Stainless Steel	Brass	Aluminum 2024-T4	316 Stainless Steel	Nylon
2 - 56	2.5	2.0	1.4	2.6	0.44
4 - 40	5.2	4.3	2.9	5.5	1.19
4 - 48	6.6	5.4	3.6	6.9	
6 - 32	9.6	7.9	5.3	10.1	2.14
6 - 40	12.1	9.9	6.6	12.7	
8 - 32	19.8	16.2	10.8	20.7	4.30
8 - 36	22.0	18.0	12.0	23.0	
10 - 24	22.8	18.6	13.8	23.8	6.61
10 - 32	31.7	25.9	19.2	33.1	8.20
1/4" - 20	75.2	61.5	45.6	78.8	16.00
1/4" - 28	94.0	77.0	57.0	99.0	20.80
5/16" - 18	132.0	107.0	80.0	138.0	34.90
5/16" - 24	142.0	116.0	86.0	147.0	
3/8" - 16	236.0	192.0	143.0	247.0	
3-8" - 24	259.0	212.0	157.0	271.0	
7/16" - 14	376.0	317.0	228.0	393.0	
7/16" - 20	400.0	357.0	242.0	418.0	
1/2" - 13	517.0	422.0	313.0	542.0	
1/2" - 20	541.0	443.0	328.0	565.0	

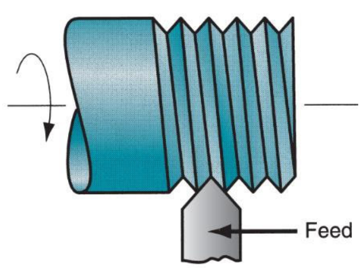
https://www.engineersedge.com/torque_table_sae.htm

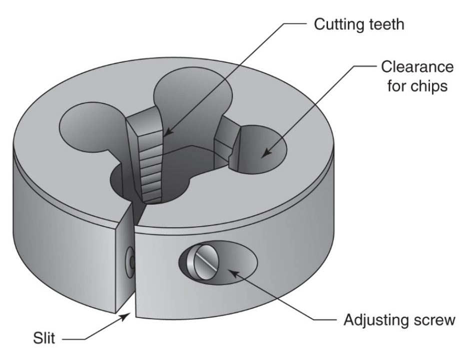
 slide 55

ILLINOIS

Cutting external screw threads

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





 slide 56

ILLINOIS

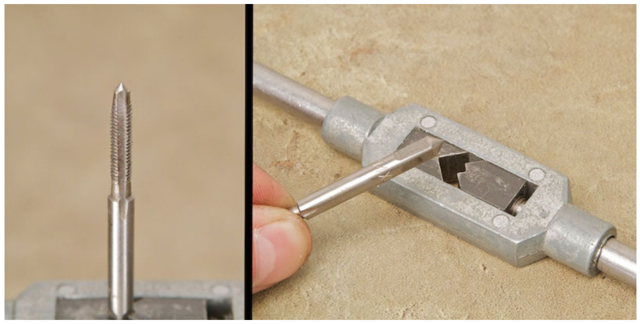
Drilling and Tapping

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

ILLINOIS

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)

ILLINOIS

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

ILLINOIS

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

ILLINOIS

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

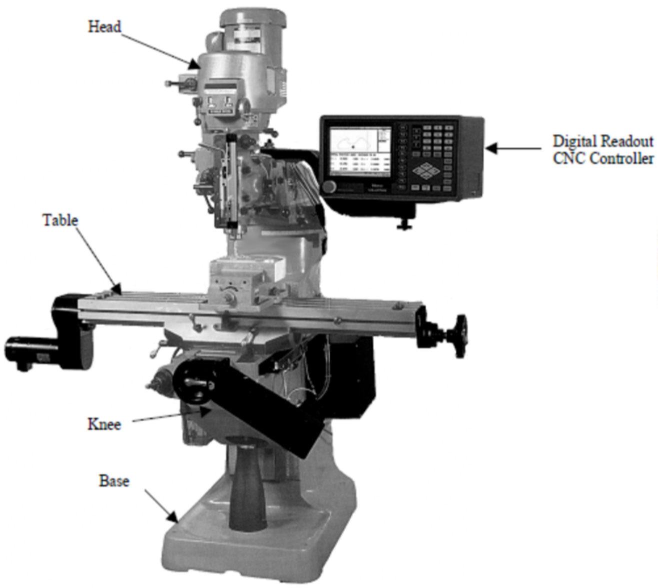
Milling

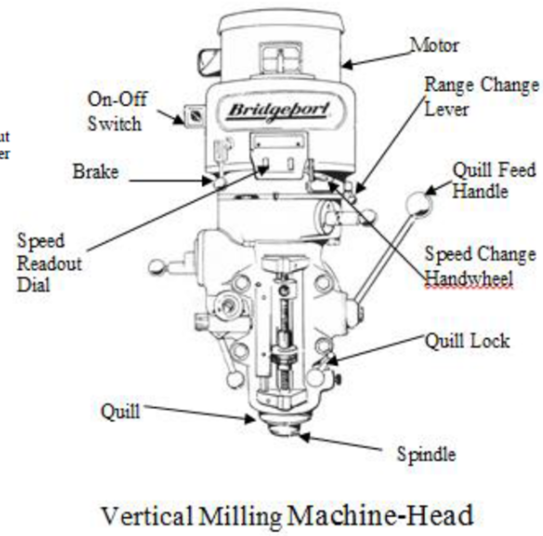
1. Theory

Milling

- Fixed spindle with workpart clamped to precision 3-axis gantry

Parts of a Mill

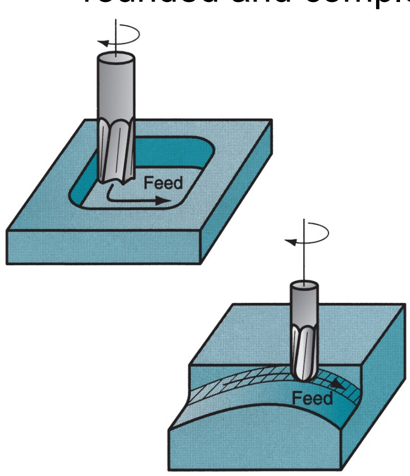


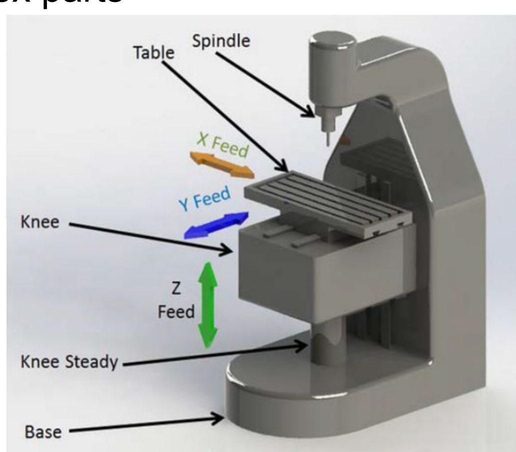


ILLINOIS

Computer Numerical Control (CNC)

- Computer control of machining axis enable rounded and complex parts

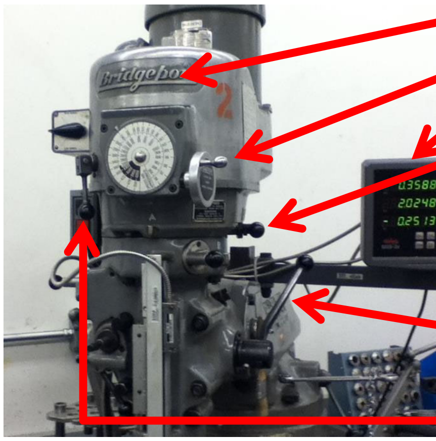




 slide 68

ILLINOIS

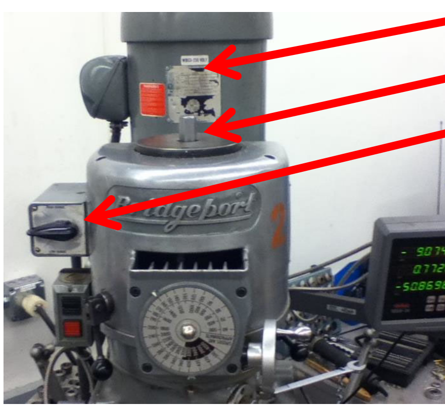
Part identification



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

ILLINOIS

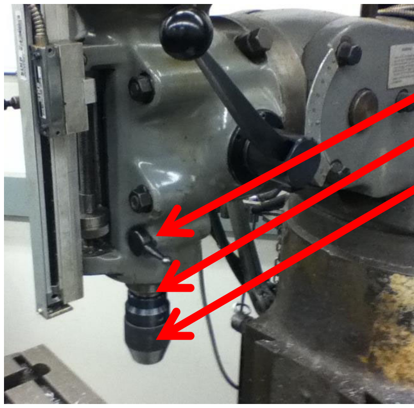
Part identification



- MOTOR
- DRAWBAR
- SPINDLE SWITCH

ILLINOIS

Part identification

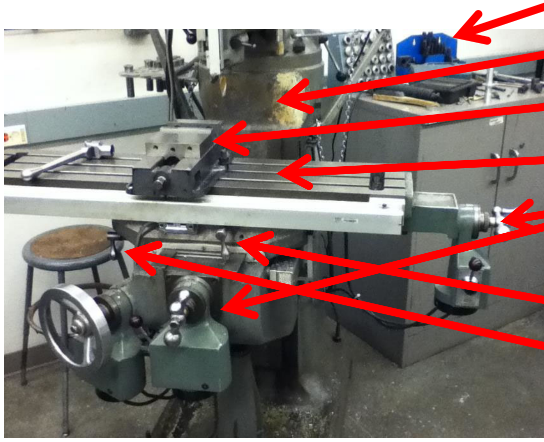


- QUILL LOCK (Z-axis)
- SPINDLE
- DRILL CHUCK (interchangeable with collets)

 slide 71

ILLINOIS

Part identification

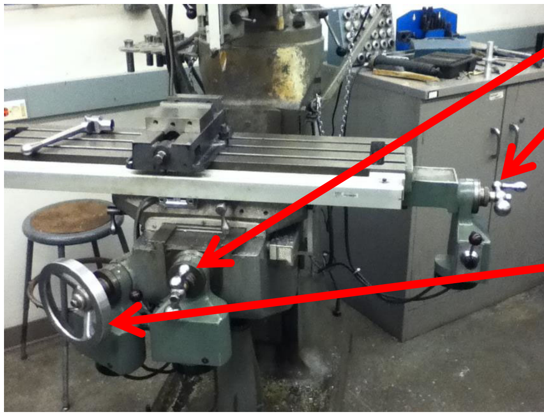


- TOE CLAMPS
- COLUMN
- VICE
- WORKTABLE
- HAND CRANKS
- X-AXIS LOCK
- Y-AXIS LOCK

 slide 72

ILLINOIS

Part identification



- ADJUST Y
- ADJUST X

ADJUST Z
TABLE HEIGHT
LOCK IS
BEHIND CASE

ILLINOIS

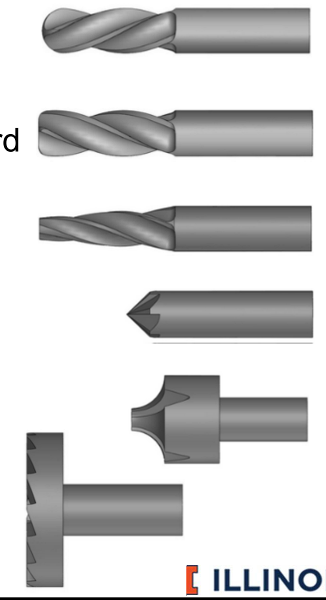
End Mill – standard mill cutting tool

- 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.

ILLINOIS

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.

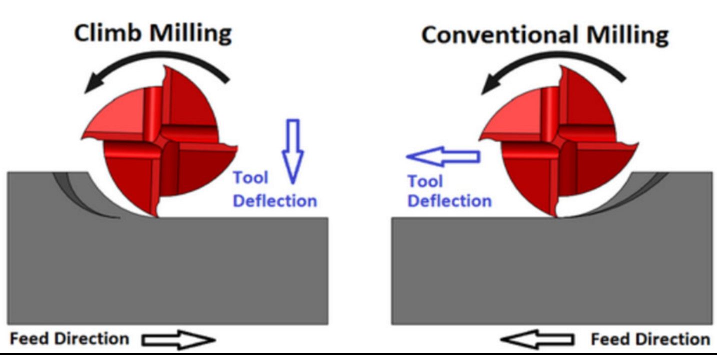


 slide 75

ILLINOIS

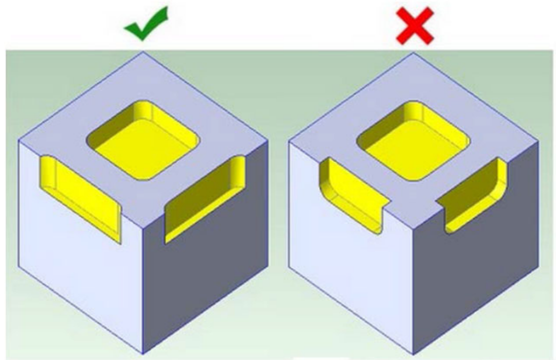
Climb vs conventional milling

- In <u>Climb</u> 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 <u>*only*</u> for finishing cuts.
- <u>*Conventional*</u> 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

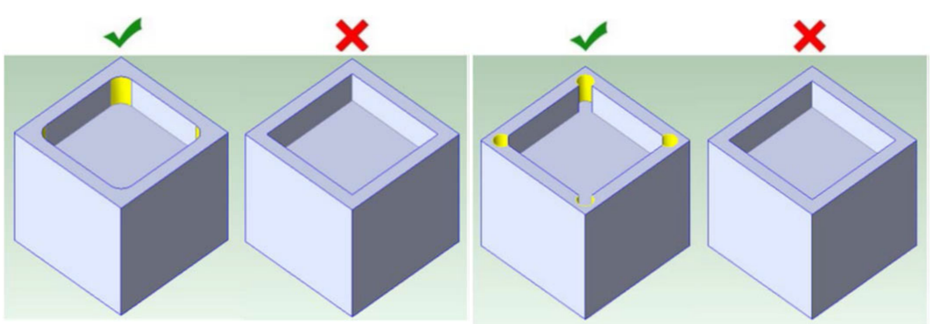
- Design for fewest number of part setup changes
  - Features created in the *same setup* will have *better dimensional accuracy* than using multiple setups



 slide 77 ILLINOIS

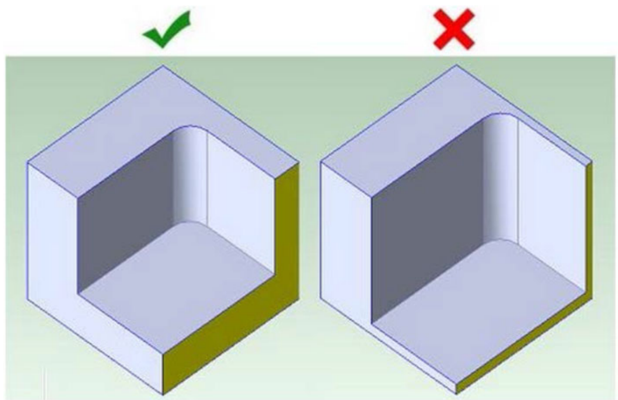
Design Advisor – corners & edges

- Sharp 3-point corners <u>*cannot*</u> <u>*be machined*</u>
- 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*



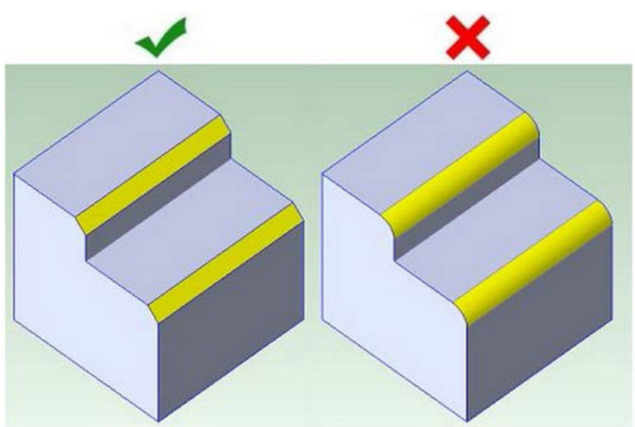
Design Advisor – limit any thin sections

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



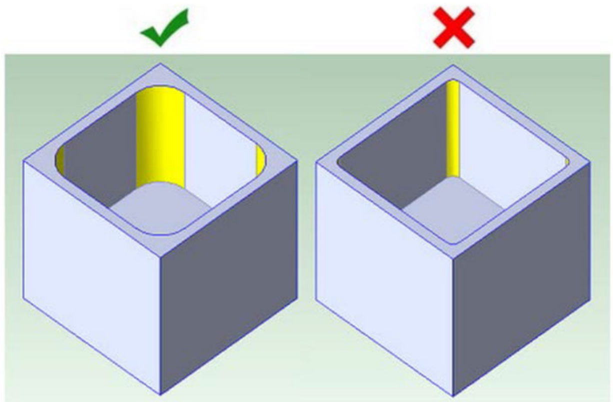
Design Advisor – chamfer outside corners

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



Design Advisor – tool size and reach

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

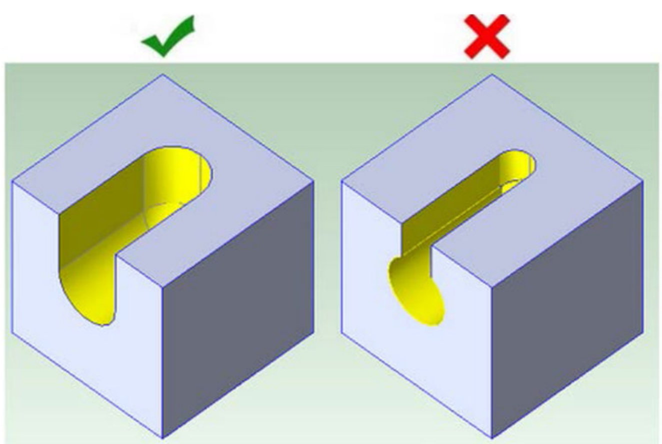


Bruce Flachsbar

ILLINOIS

Design Advisor – undercut features

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



Mill

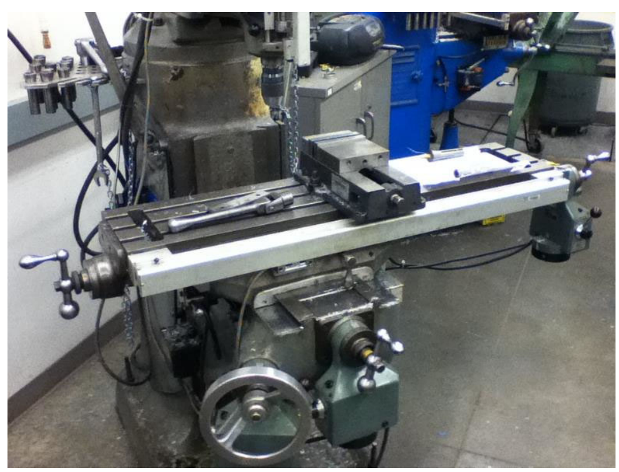
2. Operation and Best Practices

 slide 83

ILLINOIS

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



 slide 84

ILLINOIS

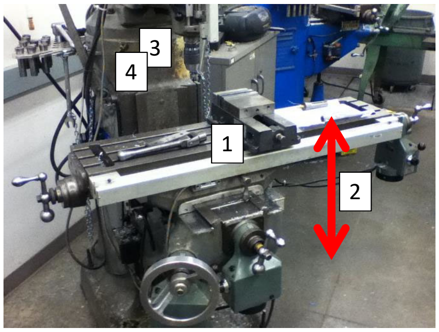
Part clamping & using parallels

- 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.

 slide 85 ILLINOIS

Part and tool loading

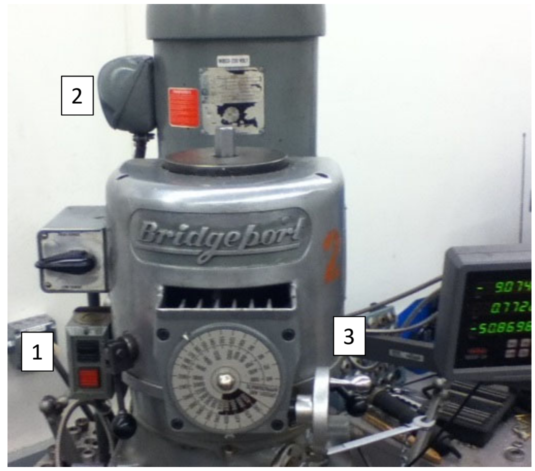
1. Clamp part firmly into vice.
2. Adjust Z-height to fit the tool, then lock.
3. Insert collet and end mill into spindle – Quill feed must be raised all the way to access drawbar
4. While holding collet with one hand, tighten the spindle drawbar with the other hand until hand tight
5. Hold brake and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



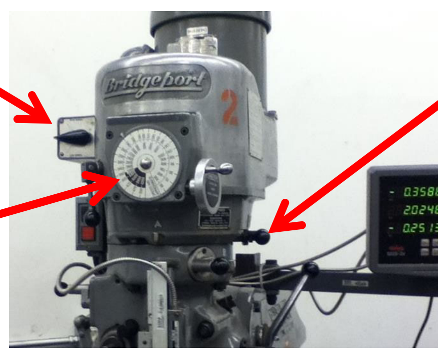
The Blue Bridgeport has a DIFFERENT way to adjust speed. Adjust belts when breaker is off.

 slide 87 ILLINOIS

Controlling direction and speed of spindle

Forward/ Reverse

Speed Dial Indicator



High/ Low gear

Forward + High gear = Forward Fast – read *inside* of dial indicator

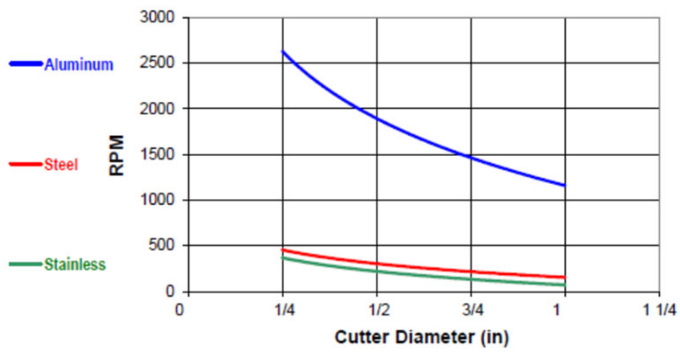
Forward + Low gear = Reverse Slow – read outside of dial

Reverse + High gear = Reverse Fast – read inside of dial

Reverse + Low gear = Forward Slow – read *outside* of dial

Recommended RPM for milling

Cutter Diameter (inches):	Material: Aluminum	Steel	Stainless
1/4	2500	450	375
3/8	2250	375	300
1/2	2000	300	200
5/8	1750	250	150
3/4	1500	225	120
1	1000	150	100



Depth of cut and feed limits

Depth of Cut for the Mill (in)

	Aluminum	Steel	Stainless
Rough	0.1	0.08	0.06
Finish	0.005	0.005	0.005

Feed Rate for the Mill (ipm)

	Aluminum	Steel	Stainless
Rough	5	3	1.5
Finish	3	2	0.75

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

 slide 90 ILLINOIS

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

ILLINOIS

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.

ILLINOIS

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)

ILLINOIS

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.

ILLINOIS

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.

 slide 97

ILLINOIS

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.

 slide 98

ILLINOIS

Bandsaw

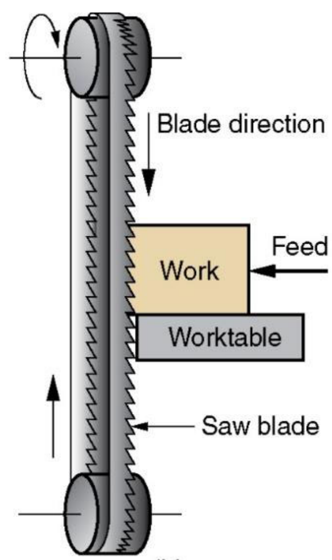
1. Theory

 slide 99

ILLINOIS

Band saw

Workpiece is fed into linear cutting blade



 slide 100

ILLINOIS

Part identification

- BLADE START
- STOP

There is another stop button below the work table!

Twist button to restart machine

ILLINOIS

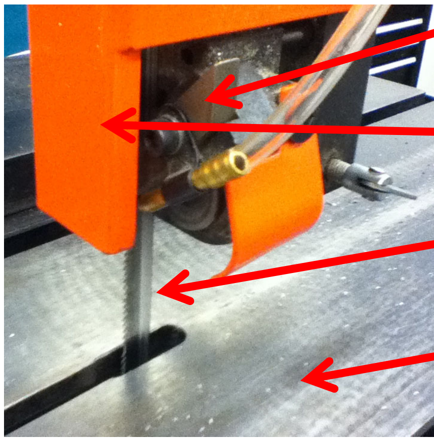
Part identification

- BLADE TENSIONER

Set tension for ½” wide blade

ILLINOIS

Part identification



- BLADE GUIDES
- GUARD
- BLADE
- WORK TABLE

 slide 103 ILLINOIS

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

 slide 104 ILLINOIS

Bandsaw

2. Operation and Best Practices

 slide 105

ILLINOIS

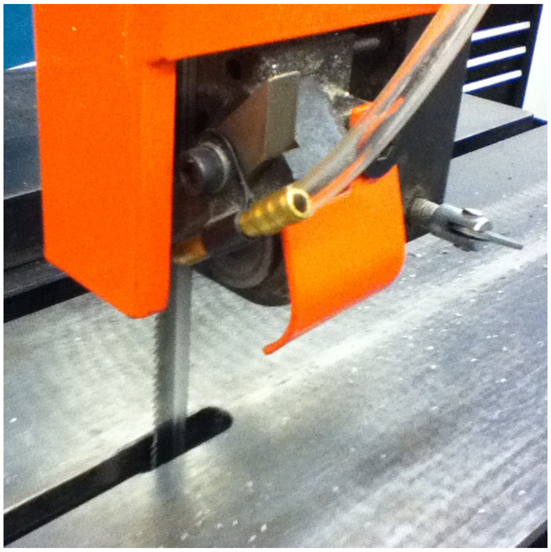
Standard operating procedure

- 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*

 slide 106

ILLINOIS

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

ILLINOIS

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 ½”
- Close doors and guards

ILLINOIS

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

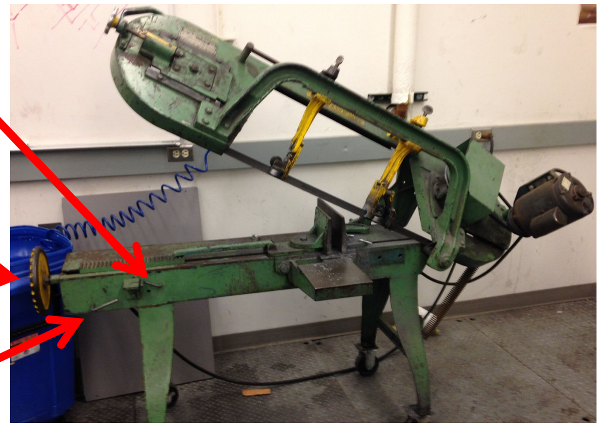
- Ensure blade is facing right direction
- Be careful with sharp end of blade while loose
- To wind blade, step on bottom and turn twice with hands while folding over
- Replace blade if worn, broken or missing teeth

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

ILLINOIS

Bandsaw

3. Safety

ILLINOIS

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

 slide 113 ILLINOIS

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

 slide 114 ILLINOIS

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.

ILLINOIS

Machine shop accidents:

What happens when flesh meets a moving cutting edge.

What happens when a moving cutting edge meets a gloved hand.

ILLINOIS

When tools fly…

 slide 117

ILLINOIS

Don't drill or mill carelessly

Didn't clamping down a workpiece

Why you don't Use Gloves

 slide 118

ILLINOIS

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.

 slide 119

ILLINOIS

Always *respect the equipment* and stay SAFE!

 slide 120

ILLINOIS