

# Machine Shop

## Part 4 –Mill

Prof. Steven S. Saliterman  
Introductory Medical Device Prototyping

Department of Biomedical Engineering, University of Minnesota

<http://saliterman.umn.edu/>

# Safety Notice

- ▶ You must complete safety instruction before using tools and equipment in the Medical Device Center, ME Student Shop and CSE Workshops.
- ▶ All machinery can be dangerous. You must have a trained individual instruct you first when using unfamiliar equipment.
- ▶ Only authorized and trained individuals may operate CNC equipment.
- ▶ Code examples shown are for illustration purposes only, and are not meant for operation or programming actual equipment. They may be incomplete or contain errors.
- ▶ Always abide by shop safety instructions and never engage in horseplay.
- ▶ Remember to wear OSHA approved eye and ear protection in the shop, short sleeves, leather or steel toed shoes, and secure long hair, avoid loose clothing, and take off rings, watches and bracelets when using power equipment.
- ▶ These slides are part of the “Introductory Medical Device Prototyping” course at the University of Minnesota, and are not meant for any other purpose.

# Eye Protection & First Aid



- Always wear OSHA approved eye and ear protection.
- Familiarize yourself with the shop first aid kit, location of telephone, and emergency phone numbers.

# Machine Shop Topics

- ▶ Facilities
- ▶ Hand Tools
- ▶ Machine Tools
- ▶ Mill
- ▶ Lathe

# Tools of the Trade

- ▶ Safety
- ▶ Mill
  - Mill features.
  - Collets
  - Aligning the vise.
  - Securing the work piece.
  - Facing a surface.
  - Using edge-finder for work zero.
  - Milling
  - Edge-rounding and countersinking.
  - Circular milling.
  - Drilling
  - Counterboring and countersinking.
  - Spindle and cutting speeds.
  - Lubricants/coolants
- ▶ Cleanup
- ▶ Appendix
  - Formulas & Tables

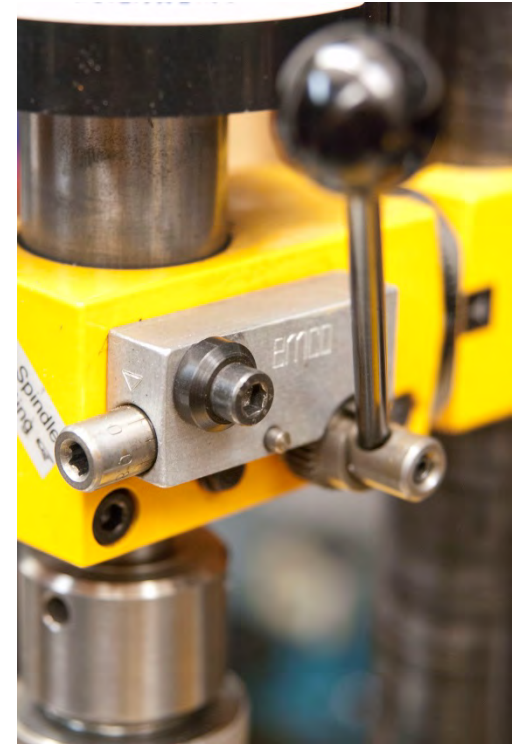
# Mill Features



EMCO mill.



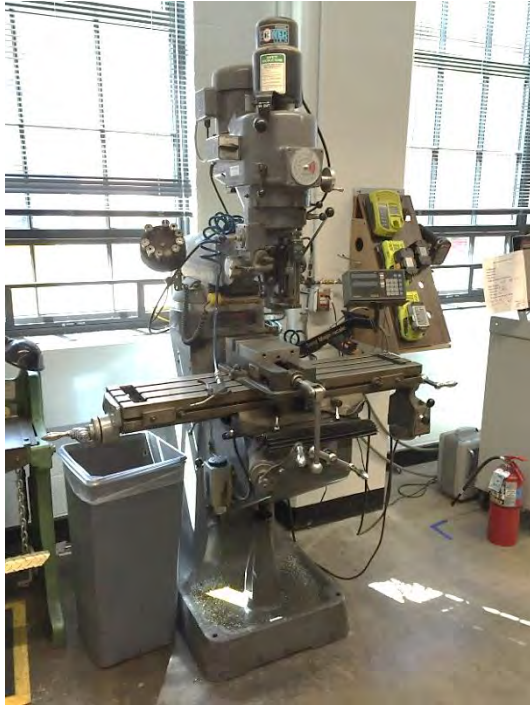
Spindle, collet holder & collet.



Manual Quill for Z-Axis.



# *Similar Features on Different Mills...*



ME Bridgeport mill.



MDC Jet mill.



HAAS VF2 CNC mill.

# *Tool Set*



Each mill is going to have its own set of specific adjustment tools.



# *Fractional Compression Collet Set...*



Each collet accepts a small range of stock, shaft or end-mill diameters.

# End Mills...



Fractional End Mills – 2 Flute Center Cutting and 4 Flute



Ball-Nose (Round) End Mills



45° Chamfer



Roughing



Finishing



Corner Rounding  
(Radius) & Taper

# *Collet Adaptor & End Mill...*



Placing collet in adaptor and tightening to collet holder on spindle.

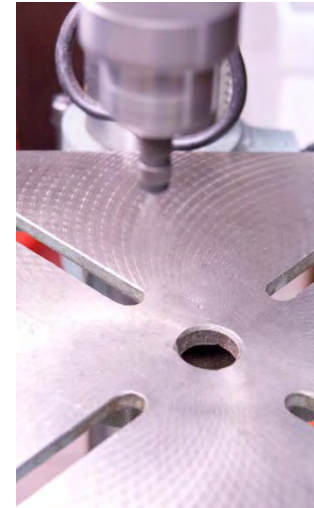
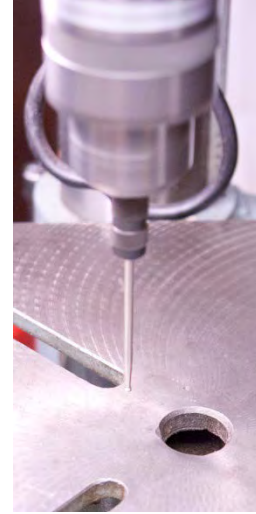
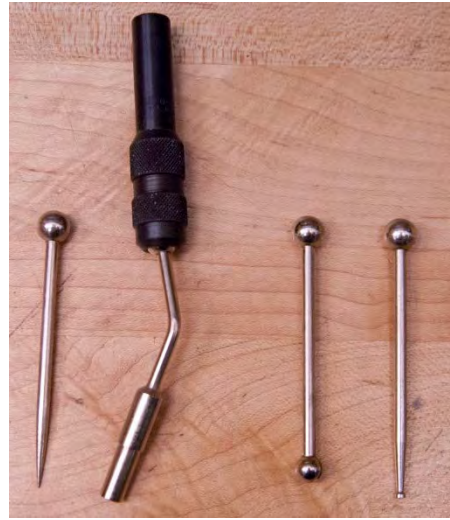


# Edge Finding/Zeroing



Top – Laser center and edge finder.  
Right – Different spring-loaded edge-finders.

# *Center Finder – Wiggler...*

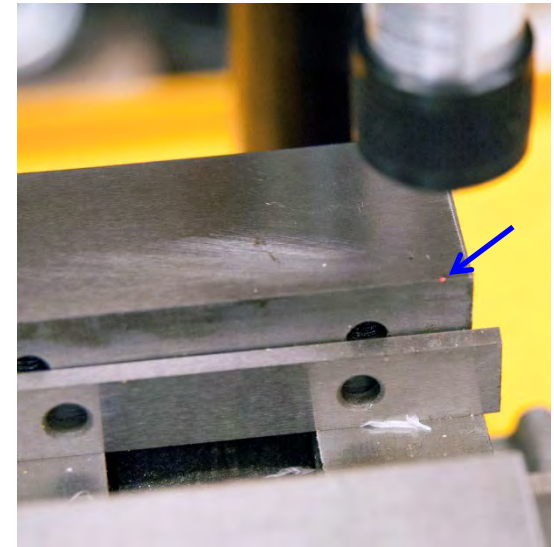
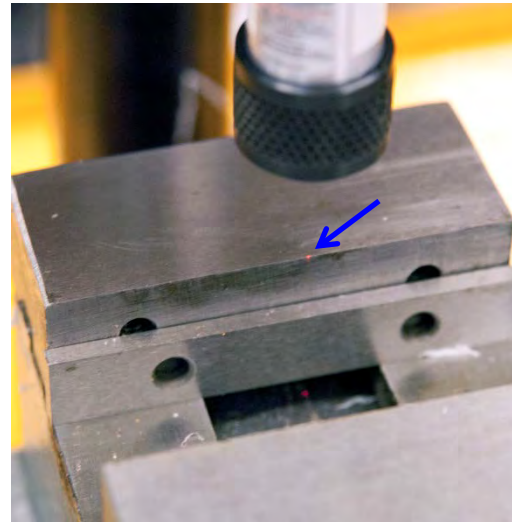
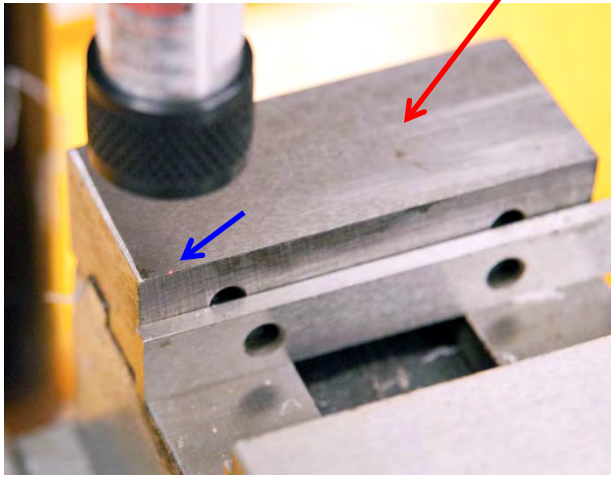


Used for mill and drill part alignment.



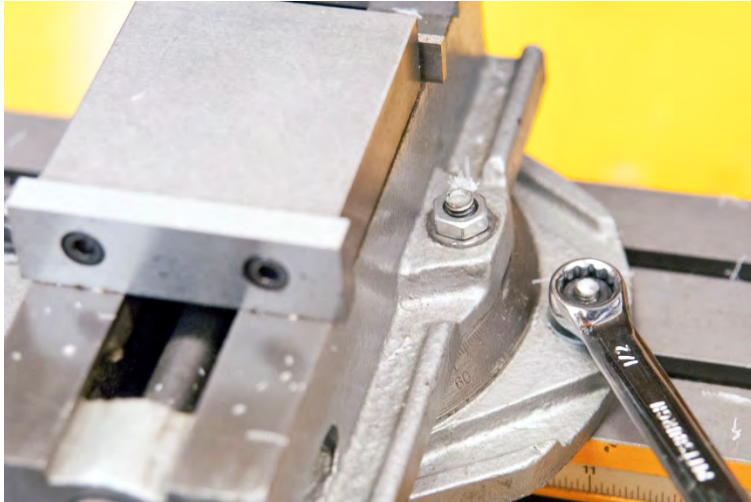
# *Align Back of Vice – the Fixed Jaw – First ...*

Fixed jaw.



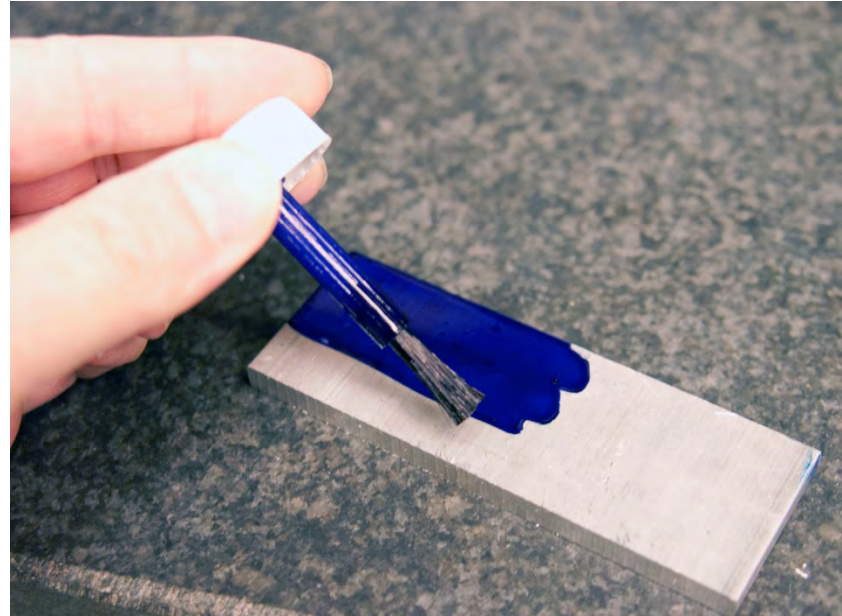
Laser is shown above, but  
spring-loaded edge-finder  
works fine too.

# *Move Vise with Gentle Tap...*



If vise needs alignment, loosen only one of two nuts holding the vise to the table, not the rotation nut!

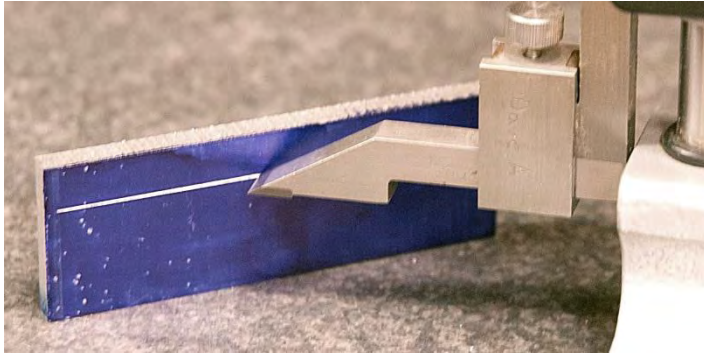
# Work Layout



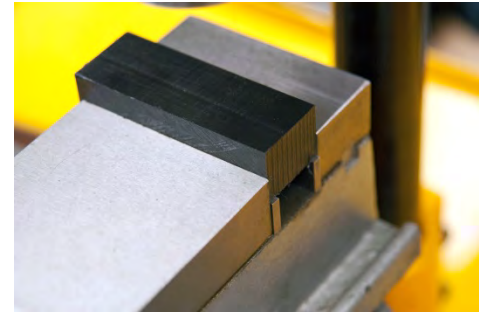
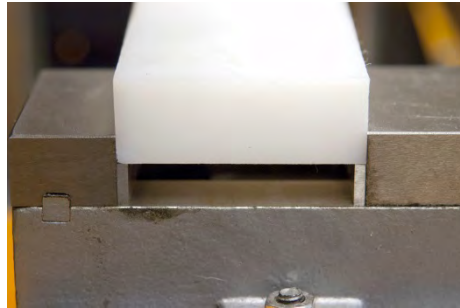
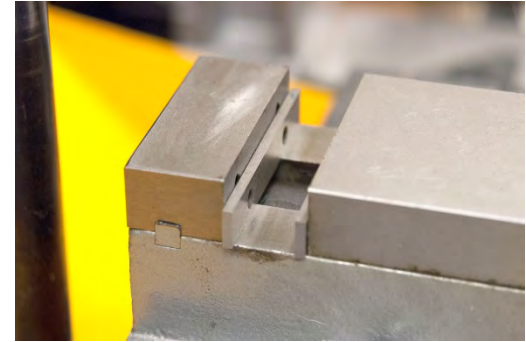
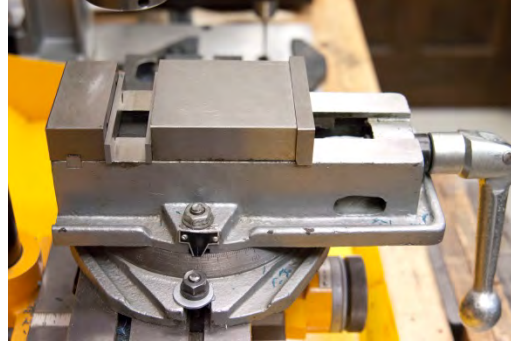
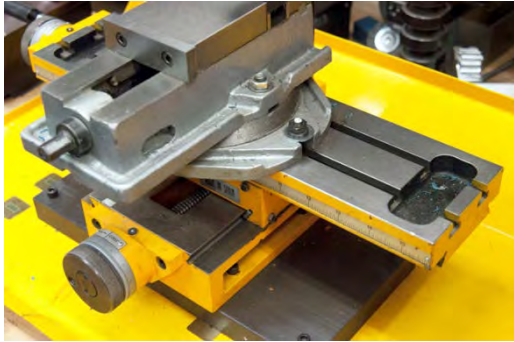
When machining without a DRO, it is often helpful to precisely layout your work – including holes. Polymers are tricky, as the Dykem is hard to remove. Pre-test.



# Height Gage & Granite Surface...



# Securing Work Piece



T-Slot table, rotatable vise and parallel plates



# *Securing Work...*

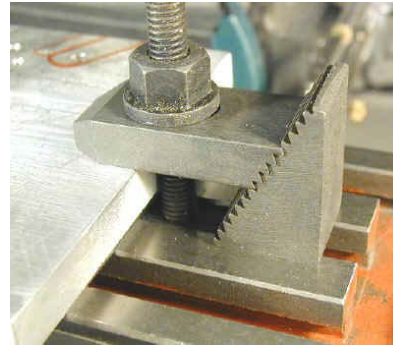


Tap lightly while tightening vise to ensure work is flat against parallel plates.

# *Clamp Sets...*



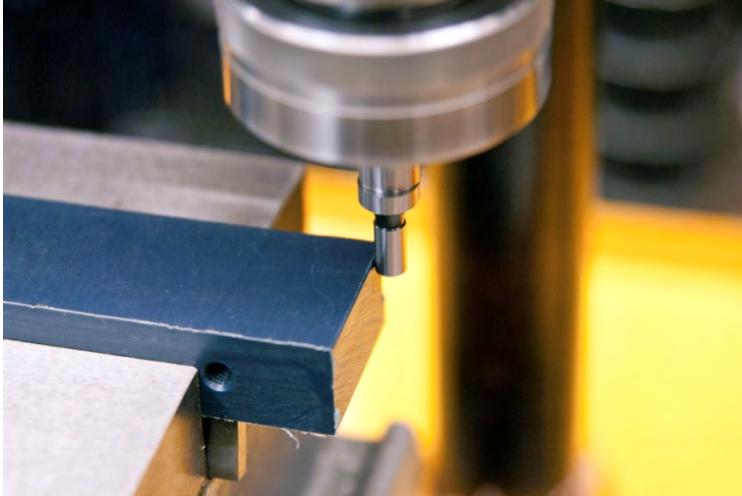
Image courtesy of Jet.



Images courtesy of Frank J. Hoose, Jr.

T-nuts, washer-nuts, threaded stems, step blocks and flanges can hold down vises, work piece and other objects to the table.

# Using an Edge Finder...



- Using an edge-finder to zero each work axis.
- Induce wobble, then move slowly against work piece until wobble stops.
- Remember to subtract radius of edge finder when zeroing DRO.

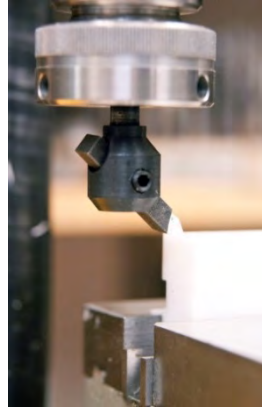
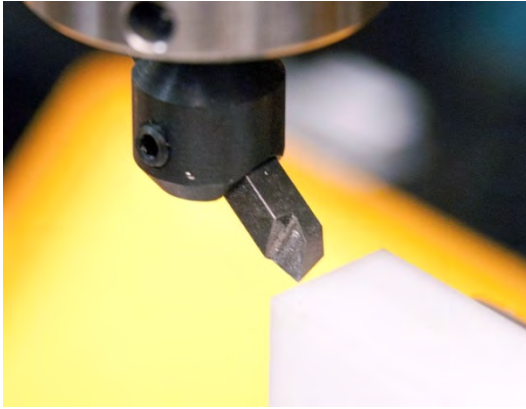


# *Handle Micrometers for X and Y Axis...*



- The handle micrometers are used when no Digital Readout (DRO) is present.
- The inner dial rotates separately from the outer handle table adjust.
- While holding the outer handle firmly, rotate inner dial to zero.

# Face the Work Piece...



- The first surface you face becomes your reference for further facing or other operations. HSS fly-cutter tool shown above.
- Take off .03" or less at a time to prevent jamming the work. Visually double check your clearances from all angles before starting.
- If necessary, continue to face your stock by rotating faced surface to the fixed jaw.
- Generally use a lubricant/coolant for metal machining.



# Other Face Tools



Image courtesy of Micro-Mark



Fly Cutter – Tormach, Inc.



Indexable – Kennametal, Inc.

# *Adjusting Z- Axis & Milling...*



Quill is engaged by a gear system and hex-driver adjusts Z- Axis.  
(Hex driver is removed when spindle is on.)

# *Clearing Chips...*



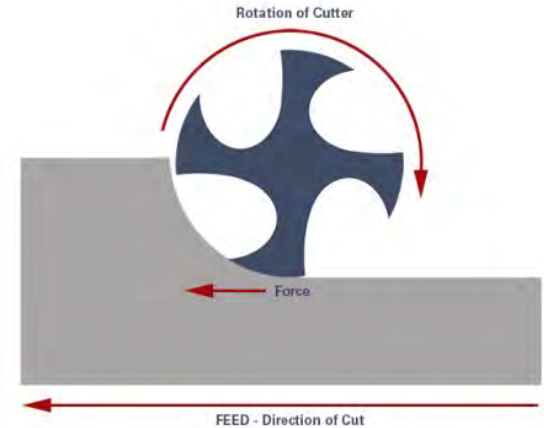
Never use your hands to clear chips.



Adjust Z axis no more than about .03" to .05" with each pass (material and machine dependent).

# Use “Conventional Milling”

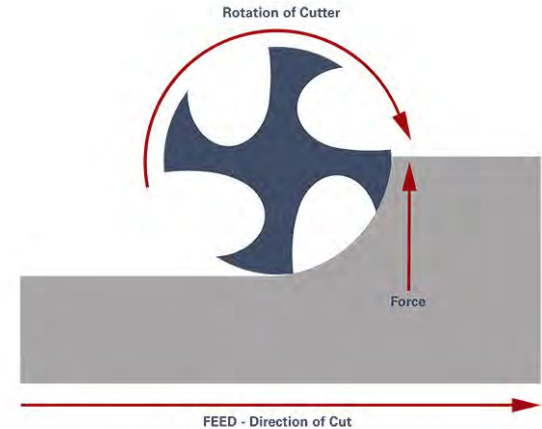
- Used with manual Bridgeport lathe.
- With the tool rotating clockwise conventional milling goes AGAINST the rotation.
- The flutes of your cutter are hitting the material and pushing against the rotation, depositing chips IN FRONT of the cut. As expected, that will result in re-cutting of the chips which will both increase tool wear and decrease surface quality.
- Since the tool hits at the bottom of the part and the flute cuts upward with the chip getting heavier as it cuts, you are creating upward force on the part which can cause work holding issues.
- Preferred for hot rolled steel and cast iron.
- Tool deflection with a conventional mill tends to be parallel to the tool, it engages the rough surface at a more forgiving rate.



**Counterclockwise around the perimeter, clockwise in a pocket.**

# *In Contrast to “Climb Milling”...*

- Preferred method for CNC.
- Think of the flutes, or teeth of the cutter as pulling the material, or CLIMBING through the material.
- When climb milling the flute hits the material at the top of the cut, and the thickness of the chip decreases as the flute cuts.
- This results in the chips being deposited BEHIND the cut, which is important. The chips clear the cutter, which means you are not re-cutting chips. Since you are not re-cutting chips, the result is a better surface finish and longer tool life.
- Less power is required from the spindle to climb mill, and the result of the cut is down-force on the material, which can simplify work holding considerations.
- Also when finishing the floor of a feature or face milling thin material the down force can assist in stabilizing the part.
- Problematic with old manual lathes because of backlash in gears.



**Clockwise around the perimeter, counter-clockwise in a pocket.**



# Milling a Round Edge

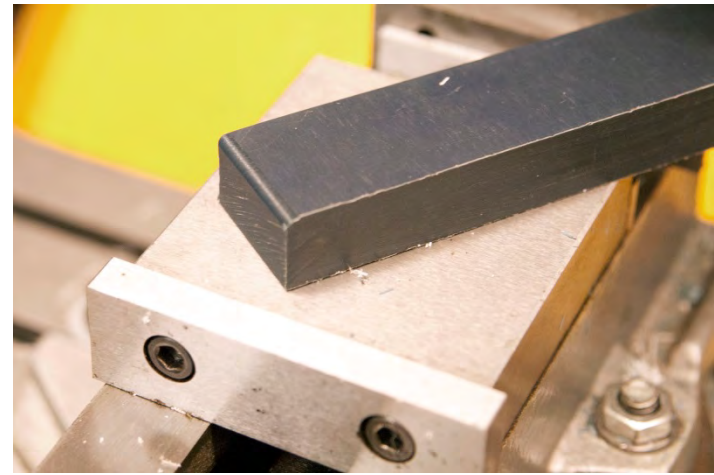
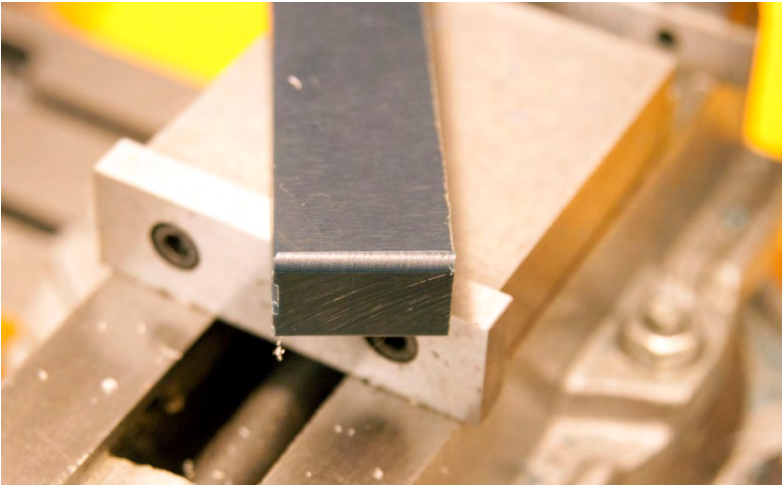


First align with lower inside edge of cutter – being sure the end mill rotates freely.



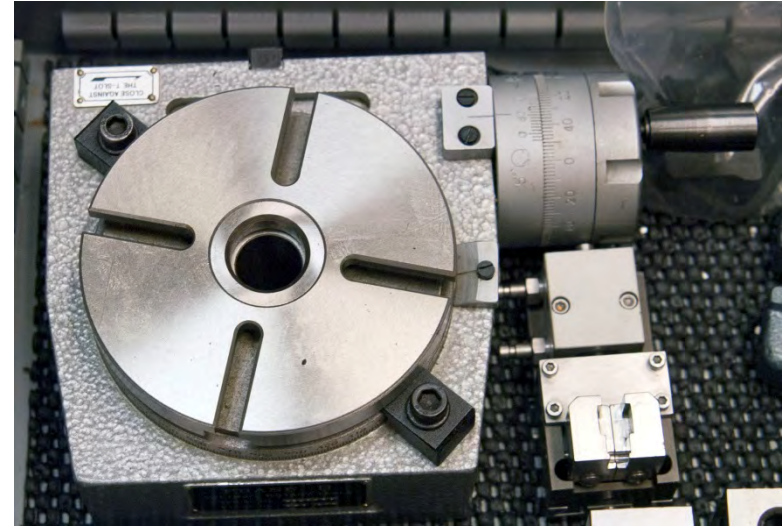
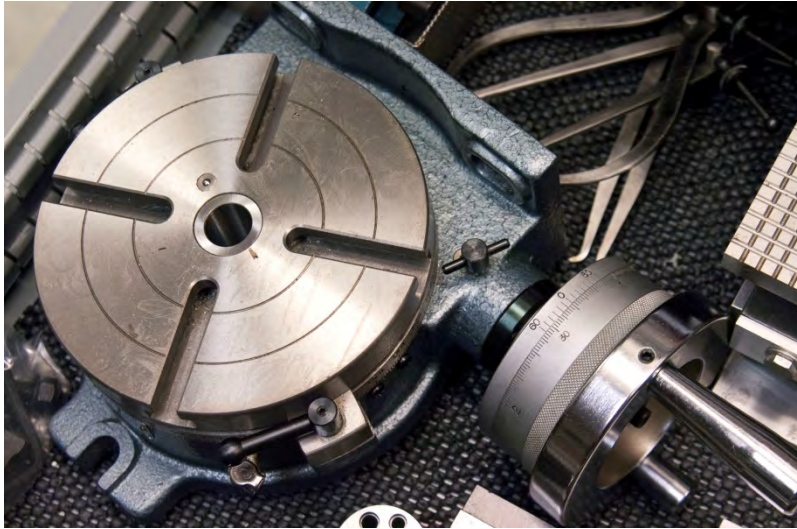
Cut by adjusting the Z axis slightly with each Y axis pass. The X axis stays fixed. (Reverse if cutting along the X axis.)

# *Finished Edge Radius...*



Example of a finished edge (outside) radius.

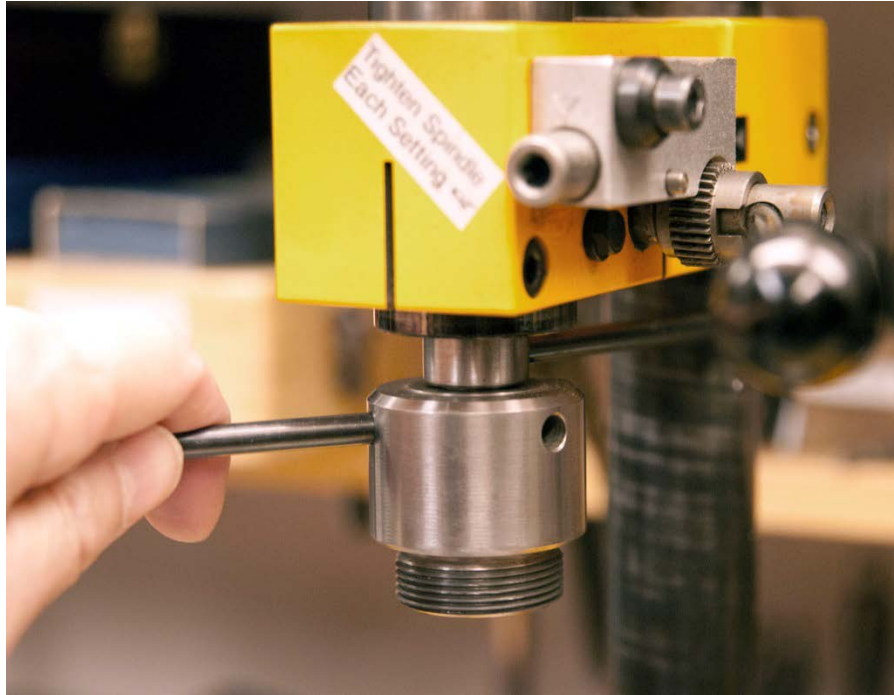
# Circular Milling



Large or small circular table are attached to mill table.



# Drilling with a Mill



Remove collet holder from spindle.



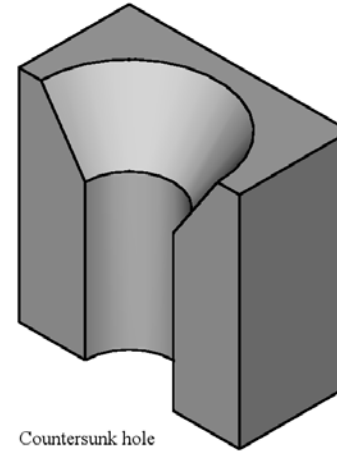
Attach drill chuck, drill bit  
and disengage quill.

# *Drill Bits...*

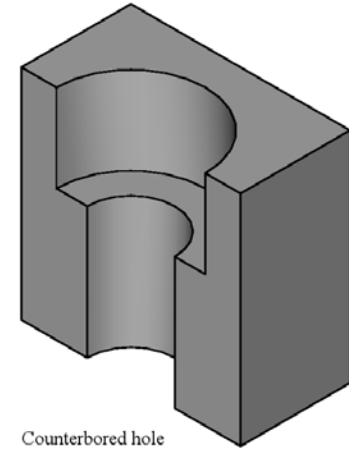


- Above: English drill bits – fractional, letter and numbered.
- Center-cutting end mills can also be used for making holes.

# *Locate and drill holes...*



Countersunk hole

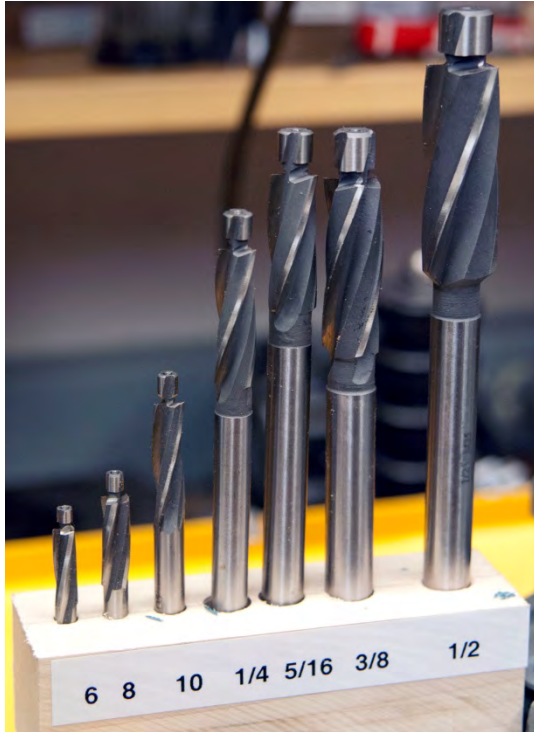


Counterbored hole

- Locate holes with the DRO, micrometer handles or attach a dial indicator.
- Spot-drilling or center-drilling may be necessary first for large drill bits.
- Countersink or counterbore holes if required.
- Hand tap or use attachable tapping tool.



# *Counterboring & Countersinking Tools...*



Counterbore set.



Countersinks



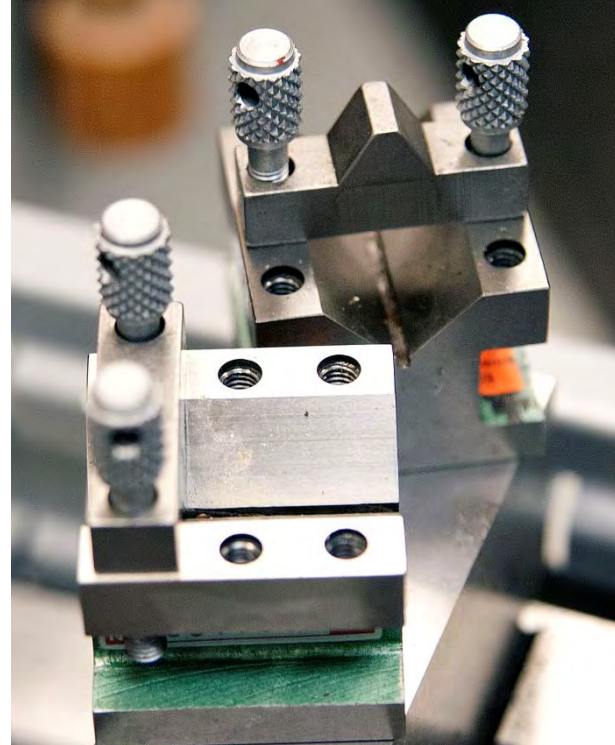
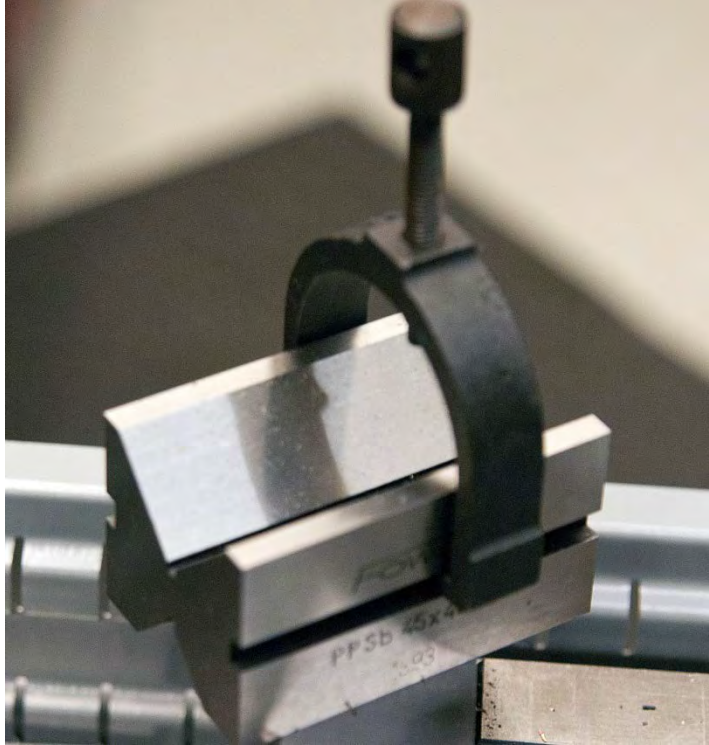
# Circular Saw



# T-Slot & Keyway Cutters



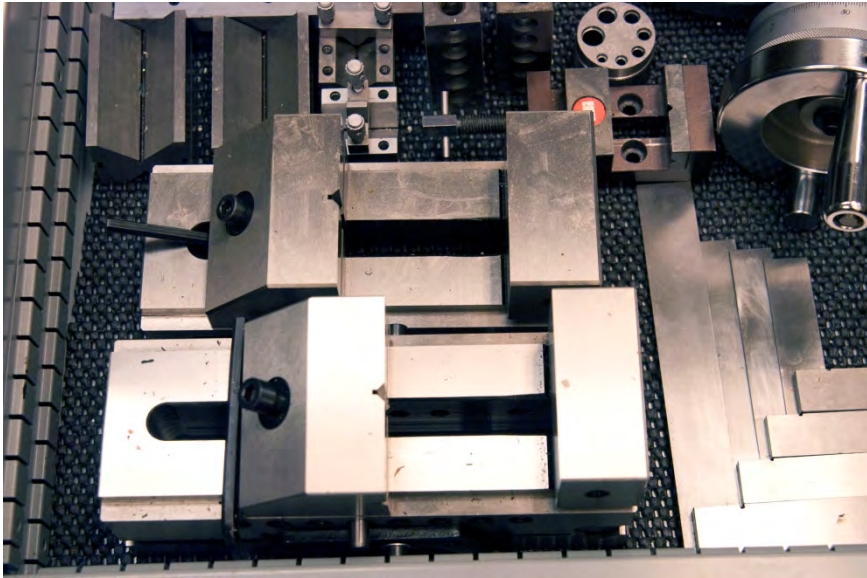
# Working With Cylindrical Stock



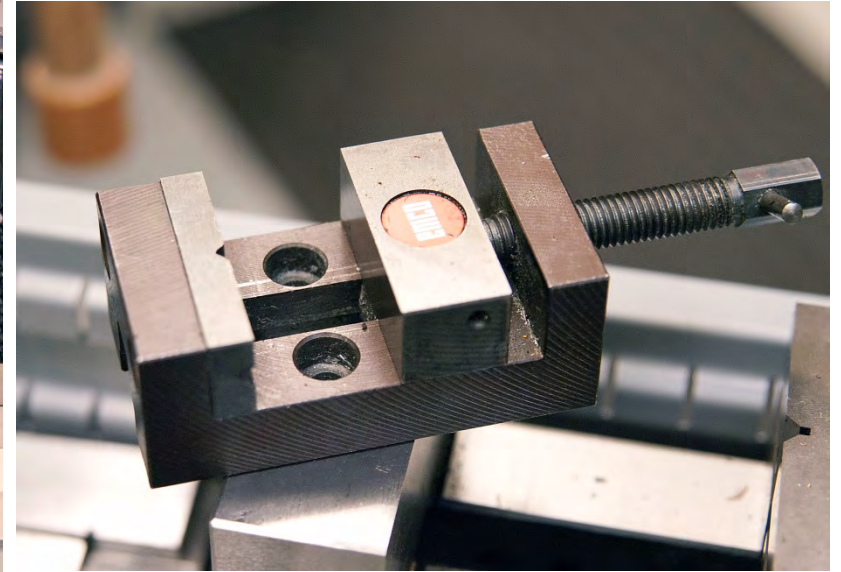
Medium and small “V” blocks and clamps.



# Optional Vises



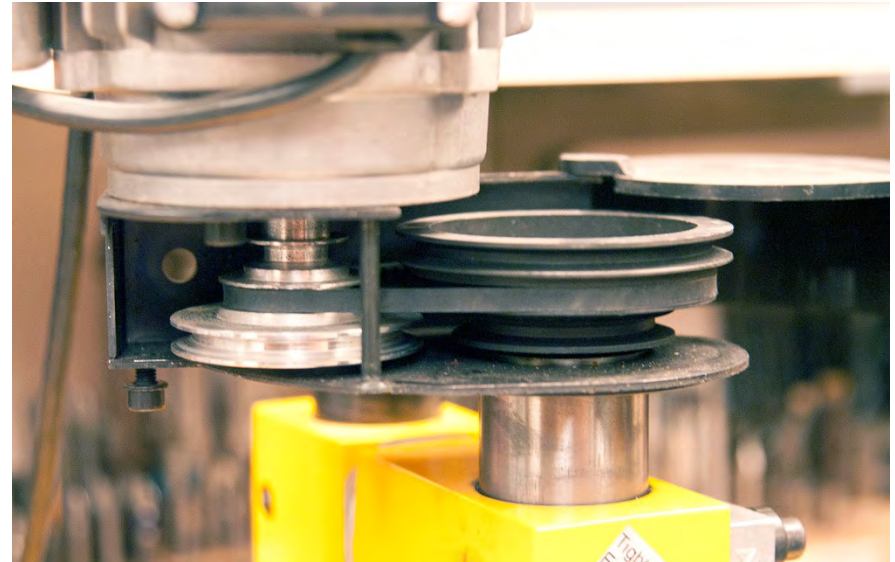
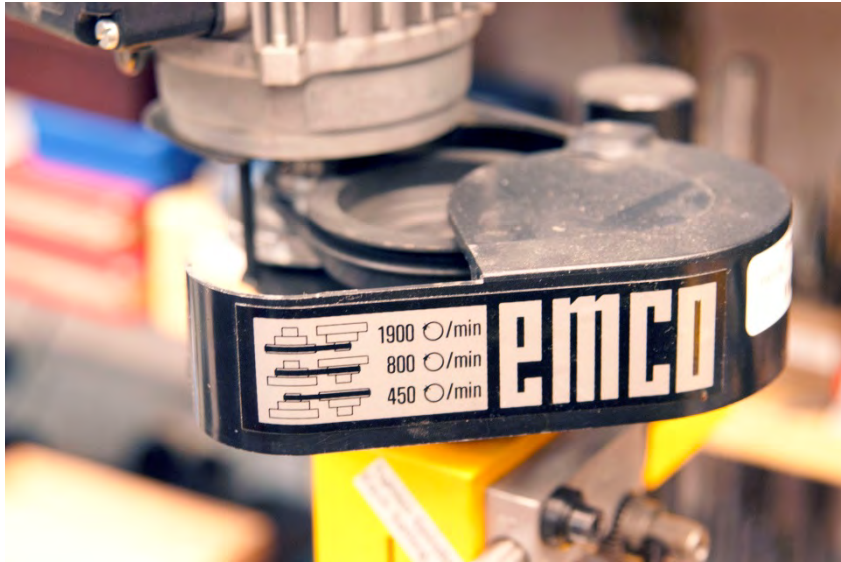
Adjustable angle vise  
(foreground).



Miniature vise.



# *Adjusting Spindle Speed...*



Change spindle speed by changing the belt.

# Speed, Feed and Tapping Formulas

- ▶ IPR (inches per revolution)
- ▶  $S = \text{Spindle Speed in RPM}$
- ▶  $RPM = 3.82 \times \frac{SFM}{\text{Cutter Diameter}}$  (revolutions per minutes)
- ▶  $SFM = 0.262 \times \text{Cutter Diameter} \times RPM$  (surface feet per minute)
- ▶  $Feed = IPM = IPR \times RPM$  (inches per minute)
- ▶  $IPR = \text{specified, or if chip load per flute} \times \text{number of flutes}$
- ▶ For tap,  $F(\text{inch per min}) = \frac{RPM}{TPI}$
- ▶ For twist drill,  $F(\text{inch per min}) = F(\text{inches per revolution}) \times RPM$
- ▶ F mills,  $F(\text{inches per min}) = \left( \frac{\text{Feed}}{\text{tooth}} \times n \right) \times RPM$

# *Mill Cutting Speeds...*

<b>Mill Cutting Speeds (SFM) Surface Feet Per Minute</b>		
<b>Material</b>	<b>HSS</b>	<b>Carbide</b>
Aluminum	600	800
Brass	175	175
Delrin	400	800
Polycarbonate	300	500
Stainless Steel (303)	80	300
Steel (4140)	70	350

Check the reference guide for the particular mill you are using for the correct speeds.

# Lubricants / Coolants



Most polymer materials will not need lubricant or coolant. Heavy-duty aluminum, brass and all steel machining should be lubricated/cooled. Wipe clean tools and oil as indicated.



# Cleanup



- ▶ “Brush and sweep” is preferable when you are done.
- ▶ Careful “puffs” of air can be useful, but do not make chips fly carelessly into equipment and onto others.
- ▶ Solvent/oily rags should be disposed in an air tight receptacle to prevent spontaneous combustion.

# Flammables



Store all flammable liquids in a designated cabinet.

# Summary

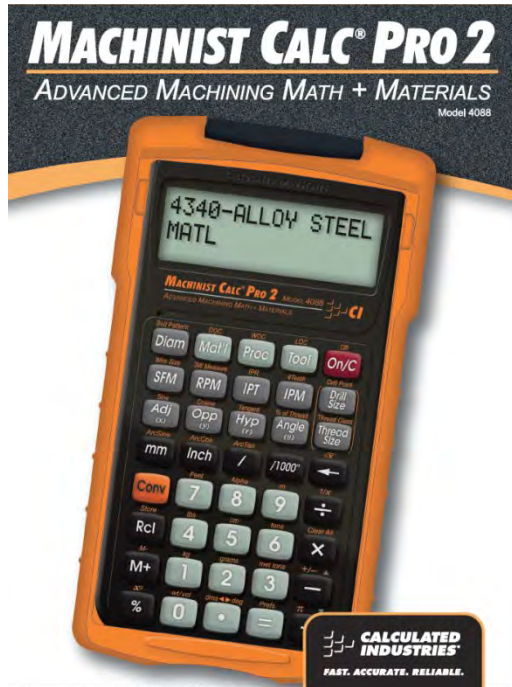
- Safety
- Machine features.
- Collets
- Aligning the vise.
- Securing the work piece.
- Facing a surface.
- Using edge-finder for work zero.
- Milling
- Edge-rounding and countersinking.
- Circular milling.
- Drilling
- Counterboring and countersinking.
- Spindle and cutting speeds.
- Lubricants/coolants
- Cleanup

# Abbreviations and Units

- ▶ **°C** = Degrees Celsius
- ▶ **DIA** = Diameter
- ▶ **d** = Depth of Cut
- ▶ **F** = Feed in Inches or mm Per Minute (F)
- ▶ **°F** = Degrees Fahrenheit
- ▶ **FPR** = Feed Per Revolution (F)
- ▶ **FPT** = Feed Per Tooth
- ▶ **IPM** = Inches Per Minute
- ▶ **IPR** = Inches Per Revolution
- ▶ **L** = Length of Cut
- ▶ **MRR** = Metal Removal Rate (cubic in./min.)
- ▶ **RPM** = Revolutions Per Minute
- ▶ **SFM** = Surface Feed Per Minute
- ▶ **SMPM** = Surface Meters Per Minute
- ▶ **MMPR** = Millimeters Per Revolution
- ▶ **T** = Number of Teeth in a Cutter
- ▶ **TCm** = Time Cutting in Minutes
- ▶ **TCs** = Time Cutting in Seconds
- ▶ **TPI** = Threads Per Inch
- ▶ **W** = Width of Cut



# Mill & Lathe Formulas



Cutting Speed (surface feet/min.)

$$\text{SFM} = 0.262 \times \text{DIA} \times \text{RPM}$$

Revolutions Per Minute

$$\text{RPM} = 3.82 \times \text{SFM} \div \text{DIA}$$

Feed Rate (in/min.)

$$\text{IPM} = \text{FPT} \times \text{T} \times \text{RPM}$$

Feed Per Revolution

$$\text{FPR} = \text{IPM} \div \text{RPM}$$

Feed Per Tooth (in)

$$\text{FPT} = \text{IPM} \div (\text{RPM} \times \text{T})$$

Metal Removal Rate

$$\text{MRR} = \text{W} \times \text{d} \times \text{F}$$

Converting IPR to IPM

$$\text{IPM} = \text{IPR} \times \text{RPM}$$

Converting IPM to IPR

$$\text{IPR} = \text{IPM} \div \text{RPM}$$

Converting SFM to SMPM

$$\text{SMPM} = \text{SFM} \times .3048$$

Converting IPR to MMPR

$$\text{MMPR} = \text{IPR} \times 25.40$$

Distance over Time (in minutes)

$$\text{L} = \text{IPM} \times \text{TCm}$$

Time Cutting over Distance (Mill) (minutes)

$$\text{TCm} = \text{L} \div \text{IPM}$$

Time Cutting over Distance (Mill) (seconds)

$$\text{TCs} = \text{L} \div \text{IPM} \times 60$$

Time Cutting over Distance (Lathe) (seconds)

$$\text{TCs} = \text{L} \div (\text{IPR} \times \text{RPM}) \times 60$$

# Inch Metric Conversion

## INCH METRIC CONVERSION

$$\text{mm} \times 0.03937 = \text{in.}$$

$$\text{in.} \times 25.4 = \text{mm}$$

$$\text{m} \times 39.37 = \text{in.}$$

$$\text{in.} \times 0.0254 = \text{m}$$

$$\text{m} \times 3.2808 = \text{ft}$$

$$\text{ft} \times 0.3048 = \text{m}$$

$$\text{m} \times 1.0936 = \text{yd}$$

$$\text{yd} \times 0.9144 = \text{m}$$

$$\text{km} \times 0.621 = \text{mi}$$

$$\text{mi} \times 1.6093 = \text{km}$$

Celsius to Fahrenheit  
 $(^{\circ}\text{C} \times 1.8) + 32 = ^{\circ}\text{F}$

Fahrenheit to Celsius  
 $(^{\circ}\text{F} - 32) \div 1.8 = ^{\circ}\text{C}$

# Tapping & Threading Formula

---

## INCH TAPS

---

$$\text{Tap Drill Size (inch)} = \text{Thread Diameter} - \frac{0.01299 \times \% \text{ of Full Thread}}{\text{Number of TPI}}$$

$$\% \text{ of Full Thread (inch)} = \text{Number of TPI} \times \frac{\text{Major DIA of Thread} - \text{Drilled DIA}}{0.01299}$$

$$\text{IPM (Mill Tapping Feed Rate)} = \text{RPM} \div \text{TPI}$$

$$\text{IPR (Lathe Threading)} = 1 \div \text{TPI}$$

$$\text{Form Tap Drill Size} = \text{Basic Tap DIA} - \frac{0.0068 \times \% \text{ of Full Thread}}{\text{Number of TPI}}$$

Recommended 65% form thread:

$$\text{Form Tap Drill Size} = \text{Basic Tap DIA} - \frac{0.442}{\text{Number of TPI}}$$

---