Introduction to Machining:
Milling Machine
Milling Machines

- Knee-and-column Milling Machine
  - Horizontal and Vertical types
  - Universal and Ram types
- Bed-type Mill
- Planer-type Mills – the largest category
- Tracer (profile) Mill – reproduce an irregular part geometry
- CNC Milling machine
Vertical Milling Machine is Most Common

- Milling machines are very versatile. They are usually used to machine flat surfaces, but can also produce irregular surfaces. They can also be used to drill, bore, cut gears, and produce slots.
- The type of milling machine most commonly found in student shops is a vertical spindle machine with a swiveling head. Although there are several other types of milling machines, shown is the vertical milling machine.
- A milling machine removes metal by rotating a multi-toothed cutter that is fed into the moving workpiece. The spindle can be fed up and down with a quill feed lever on the head.
- The bed can also be fed in the x, y, and z axes manually. Once an axis is located at a desired position and will no longer be fed, it should be locked into position with the gibb locks.
- Most milling machines are equipped with power feed for one or more axes. Power feed is smoother than manual feed and, therefore, can produce a better surface finish. Power feed also reduces operator fatigue on long cuts. On some machines, the power feed is controlled by a forward reverse lever and a speed control knob.
Typical, Basic Milling Machine
Tramming the Head

- The head of a vertical milling machine can be tilted from side to side and from front to back. This allows for versatility of the machine, but these adjustments can drift. Occasionally, one should check and adjust the head so that the spindle will be normal to the plane of the table.
Milling Vises
Vises

- Work on a milling machine is most often held in a vise clamped onto the bed. To make features aligned with the edges of the stock, it's necessary to align the vise with the feed axes of the mill. To do this, mount the vise on the bed and secure it with T-bolts, but only lightly so as to permit adjustment of the orientation of the vise.
Cutters

• In vertical mills, milling cutters with solid shafts are usually used. Milling cutters with keyed holes are predominantly for use in horizontal mills. End mills are designed for cutting slots, keyways and pockets. Two fluted end mills can be used to plunge into work like a drill. End mills with more than two flutes should not be plunged into the work. Ball end mills can produce a fillet. Formed milling cutters can be used to produce a variety of features including round edges.

• Use More flutes for finer finish at lower MRR.
Cutters (Mills)
End Mills

• Have cutting teeth at one end, as well as on the sides
Cutting conditions

- Milling cutters
  - Plain milling cutters
  - Form milling cutters
  - Face milling cutters
  - End milling cutters

- Cutting conditions
  Spindle rotation speed: \( N = \frac{v}{\pi D} \)
  Feed rate: \( f_r = N n_t f \)
  Material Removal Rate: \( MRR = w d f_r \)
Installing & Removing Cutters

- End mill cutters can be held by the spindle in several ways; a few of the ways are shown in the figure below. On most machines, a draw bar is used to pull a spring collet into a taper in the spindle.
Spring Collets

• Locking Morse taper
Climb vs. Conventional Milling

- When milling, one should be aware of the difference between conventional, and climb milling. In conventional milling, the workpiece is fed into the rotation of the cutter. This type of cut requires lower forces and is preferred for roughing cuts.
Climb vs. Conventional Milling

- In **climb milling**, the work moves with the rotation of the cutter. This produces a better finish. It is not recommended if the workpiece cannot be held securely or cannot support high forces.
Conventional Milling vs. Climb

- Conventional

- Climb
Calculating Speeds and Feeds

• Cutting speed refers to the speed at which the tool point of the cutter moves with respect to the work measured in feet per minute. Feed is the rate at which the work moves into the cutter measured in feed per tooth revolution. Feeds and speeds affect the time to finish a cut, tool life, finish of the machined surface and power required of the machine.
Calculating Speeds and Feeds

- The cutting speed is mostly determined by the material to be cut and the material of the tool. To find the right speed for any task, refer to the Machinery's Handbook or tool manufacturer reference.
- To calculate the proper spindle speed, divide the desired cutting speed by the circumference of the tool expressed in feet. The feed rate depends on the width and depth of cut, finish desired and many other variables. To calculate the desired feed setting from the feed rate, multiply feed per tooth per revolution by number of teeth and rpm of the spindle.
MILLING SPEEDS & FEEDS
Carbide Tipped

Speeds & feeds are starting recommendations only. Factors such as machine, fixture and tooling rigidity, horsepower available, coolant application and others will affect the performance significantly. Please read machine operators instructions and use all safety shields and glasses before performing these operations. Use these charts for carbide tipped milling cutters.

<table>
<thead>
<tr>
<th>CLASS OF MATERIALS</th>
<th>MATERIAL</th>
<th>BRINELL</th>
<th>SURFACE FEET PER MINUTE (SFPM)</th>
<th>INCHES PER TOOTH (IPT)</th>
</tr>
</thead>
<tbody>
<tr>
<td>NON-FERROUS (SOFT)</td>
<td>ALUMINUM ALLOY - WROUGHT</td>
<td>30-150* (500kg)</td>
<td>1000-2000</td>
<td>.004-.008</td>
</tr>
<tr>
<td></td>
<td>MAGNESIUM ALLOY</td>
<td>50-90*</td>
<td>750-1500</td>
<td>.004-.008</td>
</tr>
<tr>
<td></td>
<td>LEAD ALLOY</td>
<td>10-20*</td>
<td>300-1000</td>
<td>.004-.008</td>
</tr>
<tr>
<td></td>
<td>NON-METAL AND PLASTIC</td>
<td>–</td>
<td>1500-3000</td>
<td>.004-.008</td>
</tr>
<tr>
<td></td>
<td>ZINC ALLOY - DIE CAST</td>
<td>80-100</td>
<td>750-1500</td>
<td>.005-.010</td>
</tr>
<tr>
<td></td>
<td>ALUMINUM BRONZE</td>
<td>40-175</td>
<td>200-600</td>
<td>.003-.006</td>
</tr>
<tr>
<td></td>
<td>BRASS ALLOY - LEADED AND FREE CUTTING</td>
<td>10-100Rb</td>
<td>400-800</td>
<td>.004-.008</td>
</tr>
<tr>
<td></td>
<td>NICKEL SILVER</td>
<td>10-100Rb</td>
<td>200-400</td>
<td>.003-.006</td>
</tr>
<tr>
<td></td>
<td>COPPER ALLOY - TOUGH</td>
<td>40-200*</td>
<td>200-500</td>
<td>.004-.008</td>
</tr>
<tr>
<td></td>
<td>DUCTILE CAST IRON - AUSTENITIC</td>
<td>120-275</td>
<td>75-150</td>
<td>.002-.004</td>
</tr>
<tr>
<td></td>
<td>DUCTILE CAST IRON - FERRITIC</td>
<td>140-270</td>
<td>250-400</td>
<td>.003-.006</td>
</tr>
<tr>
<td></td>
<td>DUCTILE CAST IRON - MARTENSITIC</td>
<td>270-400</td>
<td>200-300</td>
<td>.003-.006</td>
</tr>
<tr>
<td></td>
<td>GRAY - PEARLITIC</td>
<td>220-320</td>
<td>120-300</td>
<td>.002-.004</td>
</tr>
<tr>
<td></td>
<td>GRAY - FERRITIC</td>
<td>110-240</td>
<td>250-425</td>
<td>.003-.006</td>
</tr>
<tr>
<td></td>
<td>MALLEABLE CAST IRON - MARTENSITIC</td>
<td>200-320</td>
<td>130-225</td>
<td>.002-.004</td>
</tr>
<tr>
<td></td>
<td>LOW AND MEDIUM CARBON STEEL - FERRITIC</td>
<td>100-240</td>
<td>250-500</td>
<td>.003-.006</td>
</tr>
</tbody>
</table>

IPM = Inches Per Tooth
IPM = Inches Per Minute
RPM = Rotations Per Minute
SFPM = Surface Feet Per Minute
Cutter Diameter = Diameter of the cutter in inches
RPM = SFPM * 3.82 / CUTTER DIAMETER
IPM = IPT * RPM / # TEETH

https://littlemachineshop.com/mobile/speeds_feeds.php
Using an Edge Finder

• Before doing precise work on a milling machine, one must locate the edges of a part accurately. An edgefinder is designed to help you do this.

• An edgefinder is composed of two concentric cylinders, spring loaded together. To use it, offset the two halves slightly so that there is a wobble as it spins. Then, move the part into the tool slowly. The edge finder will center up, then break out of concentricity suddenly.

• At that point, reset the dial indicator or digital readout for that axis of the machine to a value equal the radius of the edgefinder. Repeat the process at least once.
USING AN EDGE FINDER

• The edges of a workpiece must be located before doing mill work that requires great accuracy. An edge finder helps in finding the edges.
• 800-1200 spindle RPM is recommended.
• To use an edge finder, slightly offset the two halves so they wobble as they spin.
• Slowly move the workpiece towards the edge finder.
• The edge finder will center itself, then suddenly lose concentricity.
• The digital readout tells you the position of the spindle.
Edge finder, Example

• If the diameter of the edge finder is 0.200”. So adding or subtracting half of that (0.100”) will be the tool center.

• If centering on the top left,
  – add 0.100” to the X-axis
  – subtract 0.100” from the Y-axis.

• If centering on the top right,
  – subtract 0.100” from the X-axis
  – subtract 0.100” from the Y-axis.

• Part Reference Zero is when the bit is zeroed on the X and X axes.

• A pointed edge finder is a lot easier, but not as precise. Only use a pointed edge finder if precision is not necessary.