**Cutting Speed & Feed Rates**

You should calculate RPM & Feed whenever machining with a Mill, Drill or Lathe. Experienced machinists often "fudge it" while manual machining, making changes based on the "feel" of the machine and the sound of the cut. With CNC the RPM & Feeds need to be right before you hit the green button.

**Definitions & Explanations**

- **Cutting Speed (CS)** of a material is the ideal number of Feet-per-Minute that the tool-bit should pass over the work-piece. This "Ideal" cutting speed assumes sharp tools and flood coolant. Adjustments need to be made for less than ideal cutting conditions. Different materials (High-Carbon/Low-Carbon Steels, Aluminums, Different kinds of Plastics) have different Cutting Speeds and can be worked/cut at different rates. In addition, some tools or processes (like threading, knurling, or cutting-off) will need to be worked at slower speeds than the Cutting Speed would indicate.

- **Feed Rate (Milling Machine)** refers to how fast a milling-tool moves through the material being cut. This is calculated using the **Feed Per Tooth (FPT)** to come up with the **Inches Per Minute** that a milling bit can move through a particular type of material. Thus, a Four-Flute End-Mill will cut through material at twice the speed of a Two-Flute End Mill. Feed Rates will decrease with dull tools, a lack of coolant, or deep cuts.

- **Feed Rate (Lathe)** refers to how fast a lathe-tool should move through the material being cut. This is calculated using the **Feed Per Revolution** for the particular material. Lathe tools generally have only one tooth, so in most cases the FPT and FPR will be the same. Feed rates will decrease with dull tools, a lack of coolant/lubrication, or deeper cuts.

- **Diameter** refers to the diameter of whatever is spinning: work-piece (Lathe) or cutting tool-bit (Mill/Drill). *As the diameter gets bigger use a slower RPM.*

- **RPM (Revolutions Per-Minute)** is the turning speed of whatever is spinning: On a Lathe this is the work-piece. On a Mill or a Drill it is the cutting-tool. *Using Cutting Speed and Diameter you can calculate RPM as shown further down on this page.*

When calculating spindle speed (RPM), round down to the slower speed option offered by your Lathe/Milling Machine/Drill. Operations like Threading, Knurling, or Parting-off, require much slower speeds (Generally 1/3 to 1/4 Calculated RPM for Threading, Knurling & Parting-off).

**Approximate Material Cutting Speeds & Lathe Feed-Per-Revolution:**

**Calculating RPM and Feed Rates**

<table>
<thead>
<tr>
<th>Material</th>
<th>Ballpark CS with High-Speed Tool</th>
<th>Cutting Speed High-Speed Tool</th>
<th>Cutting Speed Carbide Tool</th>
<th>Feed/Rev HSS Tool Lathe*</th>
<th>Feed/Rev Carbide Tool Lathe*</th>
</tr>
</thead>
<tbody>
<tr>
<td>SAE 1020 - Low Carbon Steel</td>
<td>100</td>
<td>80-120</td>
<td>300-400</td>
<td>.002-.020</td>
<td>.006-.035</td>
</tr>
<tr>
<td>SAE 1050 - High Carbon Steel</td>
<td>60</td>
<td>60-100</td>
<td>200</td>
<td>.002-.015</td>
<td>.006-.030</td>
</tr>
<tr>
<td>Stainless Steel</td>
<td>100</td>
<td>100-120</td>
<td>240-300</td>
<td>.002-.005</td>
<td>.003-.006</td>
</tr>
<tr>
<td>Aluminum</td>
<td>250</td>
<td>400-700</td>
<td>800-1000</td>
<td>.003-.030</td>
<td>.008-.045</td>
</tr>
<tr>
<td>Brass &amp; Bronze</td>
<td>200</td>
<td>110-300</td>
<td>600-1000</td>
<td>.003-.025</td>
<td>.008-.040</td>
</tr>
<tr>
<td>Plastics*</td>
<td>250</td>
<td>200-500</td>
<td>1000</td>
<td>.005-.050</td>
<td>.005-.050</td>
</tr>
</tbody>
</table>

*Variation in Cutting-Speed & Feed-per-Revolution will exist with different alloys, procedures, tools & desired finishes. Feed-Per-Revolution is also affected by the size of the lathe-tool, as well as the depth of cut. The cutting speed and speed of plastics will vary greatly depending upon the type of plastic.

**Link:** Cutting Speeds by Specific Alloy

**Approximate Feed Rates (Feed Per Tooth) for End Mills**

<table>
<thead>
<tr>
<th>Material</th>
<th>.050” Depth of Cut</th>
<th>.250” Depth of Cut</th>
</tr>
</thead>
<tbody>
<tr>
<td>SAE 1020 - Low Carbon Steel</td>
<td></td>
<td></td>
</tr>
<tr>
<td>SAE 1050 - High Carbon Steel</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Stainless Steel</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Aluminum</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Brass &amp; Bronze</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Plastics*</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Plain Carbon Steels
- 0.005 - 0.001
- 0.01 - 0.03
- 0.03 - 0.04
- 0.01 - 0.02
- 0.02 - 0.04

High Carbon Steel
- 0.003 - 0.001
- 0.01 - 0.03
- 0.02 - 0.04
- 0.03 - 0.01
- 0.01 - 0.04

Tool Steel
- 0.005 - 0.001
- 0.01 - 0.03
- 0.02 - 0.04
- 0.01 - 0.02
- 0.02 - 0.03

Cast Aluminum Alloy
- 0.02
- 0.03
- 0.05
- 0.03
- 0.08

Cast Aluminum - Hard
- 0.01
- 0.03
- 0.05
- 0.03
- 0.06

Brasses & Bronzes
- 0.005 - 0.001
- 0.03 - 0.04
- 0.04 - 0.06
- 0.02 - 0.03
- 0.04 - 0.06

Plastics
- * Much Variation
- 0.02
- 0.04
- 0.05
- 0.03
- 0.08

* Variation in Feed-per-Tooth exists with different Material Alloy Harness, Tool Qualities, Cutting Techniques and Desired Finishes. Contact the manufacturer if precision is important.

Twist Drill Feeds (Feed per Revolution)

<table>
<thead>
<tr>
<th>Drill Size Inches</th>
<th>Drill Feed Inches</th>
<th>Drill Size Metric</th>
<th>Drill Feed Millimeter</th>
</tr>
</thead>
<tbody>
<tr>
<td>1/8&quot; and smaller</td>
<td>0.001&quot; to 0.002&quot;</td>
<td>3mm and smaller</td>
<td>.02mm to 0.05mm</td>
</tr>
<tr>
<td>1/8&quot; to 1/4&quot;</td>
<td>0.002&quot; to 0.004&quot;</td>
<td>3mm to 6mm</td>
<td>.05mm to 0.10mm</td>
</tr>
<tr>
<td>1/4&quot; to 1/2&quot;</td>
<td>0.004&quot; to 0.007&quot;</td>
<td>6mm to 13mm</td>
<td>.05mm to 0.10mm</td>
</tr>
<tr>
<td>1/2&quot; to 1&quot;</td>
<td>0.007&quot; to 0.015&quot;</td>
<td>13mm to 25mm</td>
<td>.18mm to 0.38mm</td>
</tr>
<tr>
<td>1 to 1 1/2&quot;</td>
<td>0.015 to 0.025&quot;</td>
<td>25mm to 38mm</td>
<td>.38mm to 0.63mm</td>
</tr>
</tbody>
</table>

Quick RPM/Spindle Rate Calculations: Lathe, Mill, Drill (HSS Cutter)
(RPM changes with Cutting Speed & Diameter)

<table>
<thead>
<tr>
<th>Material</th>
<th>1/4&quot; Dia</th>
<th>1/2&quot; Dia</th>
<th>1&quot; Dia</th>
<th>1 1/2&quot; Dia</th>
<th>2&quot; Dia</th>
</tr>
</thead>
<tbody>
<tr>
<td>Low-Carbon Steel</td>
<td>1600 RPM</td>
<td>800 RPM</td>
<td>400 RPM</td>
<td>267 RPM</td>
<td>200 RPM</td>
</tr>
<tr>
<td>High-Carbon Steel</td>
<td>960</td>
<td>480</td>
<td>240</td>
<td>160</td>
<td>120</td>
</tr>
<tr>
<td>Aluminum</td>
<td>4000</td>
<td>2000</td>
<td>1000</td>
<td>667</td>
<td>500</td>
</tr>
<tr>
<td>Brass &amp; Bronze</td>
<td>3200</td>
<td>1600</td>
<td>800</td>
<td>533</td>
<td>400</td>
</tr>
</tbody>
</table>

* RPM Spindle Speed Examples calculated using Ballpark Cutting Speeds with High-Speed Steel cutting tools

Calculating RPM

Lathes, Milling Machines, Drills
RPM (Turning Spindle Speed) of the cutting tool or work piece is calculated as follows:

\[ \text{RPM} = \frac{\text{Cutting Speed} \times 4}{\text{Diameter}} \]

\[ \text{RPM} = \text{Spindle Speed} \]

\[ \text{Cutting Speed} = \text{Cutting Speed for the material being cut/worked.} \]

\[ \text{Diameter} = \text{The Diameter of whatever is turning.} \]

Example #1 Calculate RPM for turning a 1 inch diameter piece of Low-Carbon Steel with a Cutting Speed of 100 on a Lathe. (Double-check CS table)

\[ \text{RPM} = \frac{100 \times 4}{1} = 400 \text{ RPM} \]

Example #2 Calculate RPM (Spindle Speed) for a 1/2 inch High-Speed-Steel 2 flute End-Mill cutting Aluminum with a CS of 250 on a Milling Machine

\[ \text{RPM} = \frac{250 \times 4}{.5} = 2000 \text{ RPM} \]

Calculating Feed Rates

Link: Machine Shop Feed Rates Explained

Milling & Drilling

The Feed Rate in "INCHES Per MINUTE" is determined by multiplying the number of cutting teeth by the RPM, multiplying that product by the Feed per Tooth, and dividing by 3. The calculation is as follows:

\[ \text{Feed Rate} = \frac{\text{Number of Cutting Teeth} \times \text{RPM} \times \text{Feed Per Tooth}}{3} \]

Example #3 Use the information and RPM calculated in Example #2 for a Milling Machine, ball-parking the FPT (Feed per Tooth) of .005 (see table), and a cutting depth of .050"

\[ \text{Feed Rate} = \frac{2 \times 2000 \times .005}{3} = 20 \text{ inches/minute} \]

Lathes

On lathes the Feed is generally
expressed as Feed per Revolution. In practical terms this is easy. Input the Feed per Revolution for basic turning. As the RPM changes, so too will the movement of your lathe bit.

Plastic on a CNC Lathe... a greenhorn's perspective
The following, with stock held in a 3-Jaw Chuck and using a sharp HSS lathe tool, worked for me. Your results may differ:

<table>
<thead>
<tr>
<th>Material</th>
<th>CS</th>
<th>Depth of Cut</th>
<th>Feed</th>
</tr>
</thead>
<tbody>
<tr>
<td>Acetal/Delrin</td>
<td>325</td>
<td>0.035</td>
<td>0.015 - 0.010</td>
</tr>
<tr>
<td>UHMW</td>
<td>450</td>
<td>-</td>
<td>0.015</td>
</tr>
<tr>
<td>UHMW</td>
<td>250</td>
<td>-</td>
<td>0.01</td>
</tr>
</tbody>
</table>

Depth of Cut could be increased. Slower Feed Rate gives smoother finish. At 0.015 Feed Rate there were striations.

Nice finish. Slower speeds produced a rougher finish.

Parted smoothly.

When turning plastic, taking too aggressive a cut can cause the plastic to deform and pull from a 3-Jaw chuck. This has happened to me. Also, I generally use lower-speeds and higher-feeds when drilling plastic, as heat builds up and begins to melt the inside of the hole.

Quick RPM Spindle Rate Table

Speed and Feed Links
Fox Valley Tech Feeds & Speeds
Link - Suggested Cutting Speeds
Link - More Cutting Speed Charts
Link - Pierce College Cutting Speed & Feed

Top of Page

copyright 2009