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.

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

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

Material	.050" Depth of Cut			.250" Depth of Cut	
	1/8"	3/8"	1/2"	3/8"	3/4"
Plain Carbon Steels	.0005-.001	.002-.003	.003-.004	.001-.002	.002-.004
High Carbon Steel	.0003-.001	.001-.003	.002-.004	.0003-.001	.001-.004
Tool Steel	.0005-.001	.001-.003	.002-.004	.001-.002	.003-.004
Cast Aluminum Alloy	.002	.003	.005	.003	.008
Cast Aluminum - Hard	.001	.003	.005	.003	.006
Brasses & Bronzes	.0005-.001	.003-.004	.004-.006	.002-.003	.004-.006
Plastics *Much Variation	.002	.004	.005	.003	.008

**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.*

Quick RPM/Spindle Rate Calculations: Lathe, Mill, Drill

(RPM changes with Cutting Speed & Diameter)

Diameter	*1/4 Inch*	*1/2 Inch*	*1 Inch*	*1.5 Inch*	*2 Inch*
Low-Carbon Steel	1600 RPM	800 RPM	400 RPM	267 RPM	200 RPM
High-Carbon Steel	960	480	240	160	120
Aluminum	4000	2000	1000	667	500
Brass & Bronze	3200	1600	800	533	400

** 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:

RPM = (Cutting Speed x 4)/Diameter

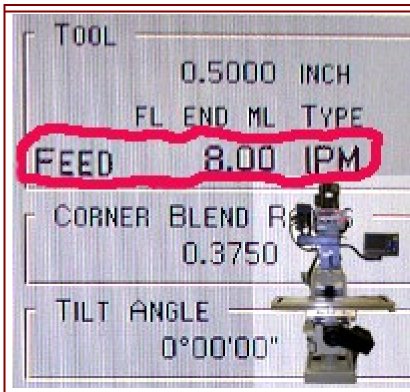
RPM = Spindle Speed
Cutting Speed = Cutting Speed for the material being cut/worked.
Diameter = The Diameter of whatever is turning.

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)

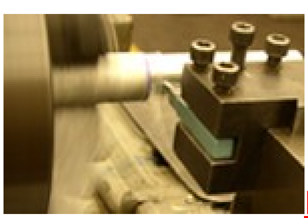
RPM = (100 x 4)/1 = 400 RPM

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

RPM = (250 x 4)/.5 = 2000 RPM



Calculating Feed Rates



Link: Machine Shop Feed Rates Explained

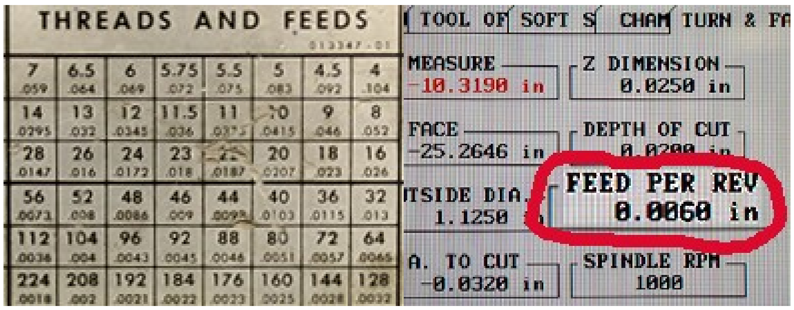
Milling & Drilling
The Feed Rate in "INCHES Per MINUTE" is determined by multiplying the

number of cutting teeth by the RPM, multiplying that product by the Feed per Tooth, and dividing by 3. The calculation is as follows:

Feed Rate = (Number of Cutting Teeth x RPM x Feed Per Tooth)

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"

Feed Rate = (2 x 2000 x .005) = 20 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.

Acetel/Delrin - Turning at CS 325, Depth of Cut 0.035, Feed 0.015 - 0.010	Depth of Cut could be increased. Slower Feed Rate gives smoother finish. At 0.015 Feed Rate there were striations.
UMHW - Turning at CS 450, Feed 0.015	Nice finish. Slower speeds produced a rougher finish.
UMHW - Part-Off at CS 250, Feed 0.01	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