

- check **MRR** to compare the efficiency of various cutting parameters.

Then... experiment. The realtime **feedrate override** available in most G-code senders is a great way to tune the chipload value and find the sweet spot for a particular job.

## Calculators

This section should have highlighted that MANY factors influence the selection of adequate feeds & speeds & DOC & WOC settings.

While predefined recommendations for common endmills and materials are very useful, at some point it becomes impossible to produce feeds & speeds charts for every possible combination of factors, and also very tedious to compute everything manually. A number of calculators have been implemented to address this, ranging from free Excel spreadsheets that basically implement the equations mentioned above, to full-fledged commercial software that embed material/tool databases, the most famous one probably being **G-Wizard**.

Whether or not you *need* a feeds & speeds calculator is debatable: most people use a limited number of combinations of material/endmill sizes anyway, in which case relying on a few good recipes for your machine is enough. The real value of calculators is in **optimizing** the feeds & speeds for a particular situation, and to see the effects of any parameter change on the rest of them.

A pretty neat feeds and speeds worksheet has been put together by @gmack on the Shapeoko forum (which he derived from an original worksheet from NYCCNC website). I have attached a version here for convenience, but you may want to check if a more recent version is available on the forum.

Cutter Manufacturer, Type, and Part Number:		Job type, owner, date, etc.: 1018 Steel with Kennametal HARV II 0.25" WOC X 1.25" LOC 5 Flute Endmill			
Imperial/Inch Cutter Information:		Spindle Manufacturer, Type, and Part Number:		CNC Machine: Shapeoko	
Cutter Material (Carbide, HSS, or Cobalt):		Makita 0701		QX Router	
Cutter Diameter (D)	1.25	HF Spindle Speed (Max-Rated for Constant Torque)	2200	Rated Input Voltage (Volts)	6.5
Cutter Length	1.25	Rated Power (Watts) @ Rated Speed	24000	Rated Input Current (Amps)	
# of Cutter Flutes/Teeth	5	Rated Speed (RPM)	24000	Rated Power (Watts)	
Shank Diameter	0.25	Maximum Speed (RPM)	24000	Efficiency	50%
Overall Stickout	1.375	Current at Rated Power (Amps)	10	Rated? Output Power (HP)	
Required Inches Per Tooth - IPT@WOC	0.00050	Available Power (HP) @ RPM	1.88	Rated? Speed (RPM)	27 000
IPT for Feed Rate Calculation (IPT@WOC = 1/20)	0.0015	Torque Constant (in-lbf / Amp)	0.7747	Available Power? (HP) @ RPM	
Year's Modulus	0.70E+07	manufacturer's recommended maximum RPM/SFM or IPT for work-piece!		CNC Machine Rigidity (Minimize spindle/router stickout), Available Feed Force, and/or Workpiece Holddown Strengths Limit Maximum Available/Usable Machine Force.	
Maximum Acceptable Deflection (in):	0.0010	SFM (>990 is AI HSM)	1000	Maximum Machine Force (Lbf):	1.8
Workpiece Material:	1018	Material K Factor from Spreadsheet:	1.88	K Factor for Power & Force Calculations	1.88
Cutter Speed (RPM):	From RPM: 2200	OR From SFM: 1000	Chosen Spindle/Router and Cutter Speed (RPM) (From Either RPM entry OR SFM):	15 200	
Cut Depth and Width	Minimum Maximum Fusion Adaptive	As Percentage of Diameter	Minimum Maximum Fusion Adaptive	Values to Use (in):	
Depth of Cut (DOC):	0% 300% 100%		0 0.75 0.25	Inches	0.250
Width of Cut (WOC):	5% 100% 20%		0.0125 0.25 0.05		0.010
CNC Feed Rate (IPM):	From IPT@WOC: 117	OR Enter IPM:	OR Enter Force (lbf): 2.0	No Entry will select from "IPT@WOC":	46
*Use climb milling to minimize rubbing. *Think thick to thin chips.*					
Estimated Results from Speeds and Feeds Calculator vs. Parameters of Spindle/Router Input Current/Power					
Calculated and Measured Results	Chipload (IPT)	MRR (in <sup>3</sup> /min)	Power (HP)	Torque (in-lbf)	% Max Deflection
Calculated Estimates	0.00023	0.11	0.061	0.25	10.39%
Measured					
HF Spindle Current Input (A) When Not Cutting					
HF Spindle Current Input (A) When Cutting					
K Factor ((in <sup>3</sup> /min)/HP)					
Deflection can be reduced without decreasing MRR by increasing RPM, reducing cutter stickout, using a carbide cutter, and/or by increasing the diameter of the cutter or its shank. Deflection, Force, and Power can be reduced at the expense of MRR by decreasing WOC, DOC, and/or IPM (at the expense of IPT).					

{% file src=".gitbook/assets/2019-08-11-speeds-and-feeds-workbook.xlsx" caption="gmack's advanced feeds and speeds worksheet" %}

1. fill-in the **specs of your router** or spindle (once).

- fill-in the **specs of the selected endmill**, and the **target chipload value** you chose (chip thinning will be taken into account automatically depending on WOC value)
- if you care about power/force analysis, look-up the **K-factor** for the material being cut (there's a list in a separate tab of the worksheet) and update it here.
- select **target RPM** value (or alternatively SFM, then RPM will be derived from it).
- select **WOC and DOC** (depending on your machining style)
- The **required feedrate** to reach the target chipload will be computed. You can alternatively choose to override it with a given feedrate value (and see what this does to chipload displayed below), OR to forget about chipload and use a given cutting force as the ultimate target. Either way, the feedrate to be used will be displayed at the right end of this line.
- You can then check the analysis of **deflection, cutting force, and power** in the lower part of the worksheet.

Then play with the input values to compare various cutting scenarios while staying within the machine's hard limits (max RPM, max feedrate, max power, and max cutting force)

**Note:**

Once you determine good feeds and speeds and confirm that it is cutting correctly, it is useful to capture a snapshot of the worksheet for that particular usecase for future reference (just duplicate the tab in the worksheet)

If you still feel overwhelmed or don't care about optimizing power, force and deflection, I derived a more basic version:

INPUTS		Endmill guidelines	Predefined constants
1	Endmill flutes Endmill diameter	1-2 0.2501	Consider using an endmill with fewer flutes if computed feedrate is above machine limit
		<b>Chipload guideline</b>	inputs outputs computed
		Endmill	Soft plastics
		1/8"	0.002" to 0.005"
		1/4"	0.004" to 0.01"
		Soft wood & hard plastics	0.001" to 0.0015"
		Hard wood & aluminium	0.0005"
		0.001" to 0.0025"	0.0005" to 0.001"
		0.001" to 0.005"	0.001" to 0.002"
		<b>RPM guideline</b>	Min RPM
		Set as high as you can tolerate (noise) and feel safe using	10000
		Decrease if computed feedrate is above machine limit	Max RPM
			30000
			Makita/Carbide3D: 1000
			Makita/Carbide3D: 3000
		<b>WOC guideline</b>	<b>DOC guideline</b>
		Milling style	Toolpath type
		roughing	recommended range
		wide and shallow for	0.1" to 0.25"
		narrow and deep for	0.0125" to 0.025"
		roughing	0.025" to 0.0625"
		finishing	0.0125" to 0.025"
		Milling style	Material
		wide and shallow in	aluminium, copper, bronze
		narrow and deep in	softer materials
			0.0125" to 0.025"
			0.025" to 0.125"
			0.25" to 0.75"
		<b>OUTPUT</b>	Shapeoko max feedrate
		Required feedrate	196.85 ipm
		61 ipm	(includes compensation for chip thinning)
		<b>(straight) plunge rate guideline</b>	
		Material	plunge rate
		aluminium, copper, bronze	6 to 18 ipm
		hard wood	18 ipm
		soft woods	24 ipm
		plastics	24 to 31 ipm
		Plunge rate	<your call>

```
{% file src="gitbook/assets/fs_worksheet_basics.xls" %}
```

- fill-in the **number of flutes** and **diameter** of your endmill
- pick a **target chipload** value from the guideline table on the right
- select an **RPM** value
- select **WOC and DOC** based on the recommended values on the right (derived from the selected endmill diameter)

This basic worksheet will just compute the required feedrate to get the desired chipload (taking chip thinning into account). If the computed feedrate turns red, it is beyond the limit of the Shapeoko, and you should select a lower RPM and/or use an endmill with a lower flute count.

## Telltale signs of wrong F&S

The most common signs of inadequate feeds and speeds are:

- sound, and specifically **chatter**: when feeds and speeds are not right for a given material/endmill/DOC/WOC, the tool tends to vibrate, and this vibration can get worse if there is resonance with another source of periodic variation elsewhere in the system (most often: the router and its RPM). This results in an ugly sound, a poor finish with marks/dents/ripples on the surface, and a reduced tool life.
- **finish quality**: even without chatter, a poor surface finish can indicate that the final cutting pass was too aggressive (too much chipload or too much deflection). Increasing RPMs may help, but the best approach is to use a finish pass with very low WOC.
- **melted material**: especially in plastics and soft metals like aluminium, if the feedrate is too low for the selected RPM, the friction will cause the material to melt rather than shear, the tool flutes will start filling with melting material, and this usually ends up with tool breakage. You will need to feed faster, and/or use an endmill with a lower flute count.
- **endmill temperature**: the endmill should not be more than slightly warm at the end of a cut: if it gets hot to the touch (careful!), the feeds and speeds are likely incorrect (too low or too high chipload), or the tool is dull and is rubbing rather than cutting. In extreme cases, the endmill color itself may change to a dark shade.
- making **dust**, instead of clearly formed chips is an indication that chipload is probably too low (MDF is an exception, you just cannot get chips anyway with this material)

## Toolpaths

A **toolpath** is the intended trajectory that the tip of the endmill will follow, to remove material and produce the desired geometry of the workpiece.

For a specific object geometry/feature defined in the CAD tool, the CAM tool will generate toolpaths as sets of lines and curves defined in X/Y/Z space, and the post-processor will then generate the corresponding G-code instructions for the selected machine.

“**2D**” toolpaths correspond to cutting a 2D feature in the design, moving only two axes of the machine at a time (typically, stepping down along Z only, then moving only in the XY plane, before proceeding with a deeper Z). Since X, Y, and Z are moved (albeit not simultaneously), this is sometimes called “**2.5D**”.