Preferred Method to Calculate Bit Feeds and Speeds for the Guild's CNC's Machining Wood

USER ASSUMES ALL RISK IN USING THIS GUIDE

This document is only a guideline to assist the user in selecting appropriate Pass Depth, Feed, and Speed settings for the Guild's CNC machines.

For bits Smaller than 0.125" or Larger than 0.50" in diameter and all **Form Bits (Roundover, Cove, Surfacing, etc.) do not rely on this guide**. For these bits consult the bit manufacturer for recommended Pass Depth, Feed Rate, and Spindle Speed for the material you are machining.

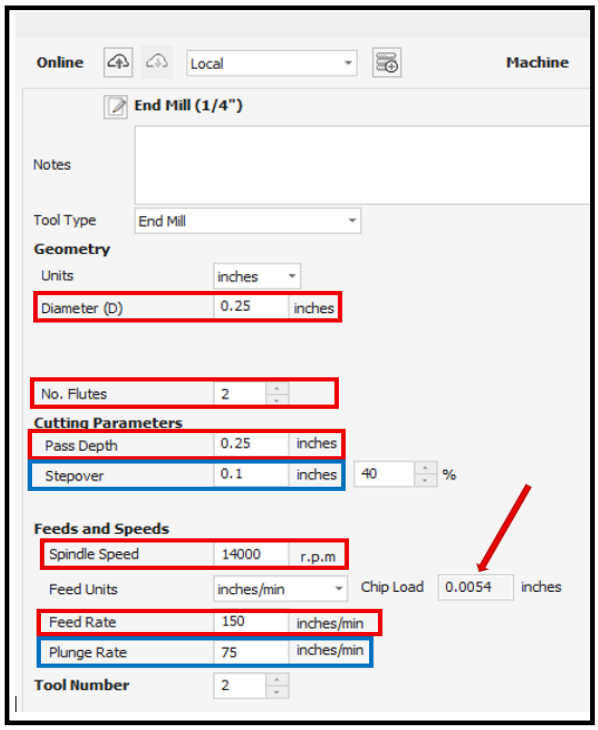
This area applies only to: Square (Flat) End Mills, Fish Tail End Mills, and Corner-Radiused End Mills.

For bits 0.125" to 0.50" in diameter:

The chart referenced (to the right) is the Chip Load Range that is recommended for the Guild CNCs. The simplest way to calculate Chip Load is to use the **Edit Tool** form in the Vectric Software. See example below.

Recommended Chip Load Range for Guild CNCs

Tool Diameter	Hardwood	Softwood / Plywood	MDF / Particle Board
0.125"	.002 - .003	.003 - .004	.003 - .004
0.250"	.004 -.007	.006 - .008	.007 - .010
0.375"	.007 - .012	.009 - .013	.011 - .015
0.500"	.008 - .014	.011 - .015	.013 -.018



Red Box Settings

1) Confirm that the bit "**Diameter**" is correct.
2) Confirm that the "**No. Flutes**" is correct.
3) "**Pass Depth**" set to 1 X bit diameter. (Note: This is different than "Cut Depth" which is set in the "Toolpath" form.)
4) Choose a "**Spindle Speed**" between 14,000 and 20,000 rpm. Usually, your Spindle Speed will be on the lower end of this range.
5) Set your "**Feed Rate**" to achieve a Chip Load in the range of that shown in the chart above.
6) "**Chip Load**" – The software will automatically calculate and display the Chip Load. **(Red Arrow)** Adjust "Feed Rate" and/or "Spindle Speed" to obtain the desired "Chip Load."

Blue Box Settings

1) "**Stepover**" is usually 40%.
2) "**Plunge Rate**" is usually ½ the Feed Rate.

For V-Bits: 0.25" – 1.0" diameter

Spindle Speed: 16,000 – 18,000 rpm; Feed Rate 60 - 80 ipm; Plunge Rate = ½ Feed Rate

On - the - Fly Adjustments to Feed and Speed while your project is machining:

If you have: a high-pitched squeal from the bit, burned edges of cuts, very small chips or dust – your chip load is too small.

Cure: Make repeated small changes to increase your Feed Rate and/or decrease you Spindle Speed.

If you have: bit chatter, scalloped machined edges, large chips or chunks – your chip load is too large.

Cure: Make repeated small changes to decrease your Feed Rate and/or increase you Spindle Speed.

On the reverse is an "Alternate Bit Feed and Speed Info." aid from Legacy Woodworking.

“Alternate Bit Feed and Speed Info.”

(Adapted from information provided by Legacy Woodworking Machinery)

Exceptions to these suggestions include:

- Surfacing bits – This depends on the hardness of the material that you will be working with.
- Specialty bits such as fret cutters, glass etching, Donek drag knife etc...

These Speeds and Feeds are conservative. This information is provided as a starting point; adjustments may be necessary depending on your material being machined. The Preferred Method to set Feeds and Speeds in on the reverse side.

Bit Speeds:

0 - 0.25 Dia bits 22,000 - 24,000rpm
0.25 - 0.5 Dia bits 18,000 - 22,000rpm
0.5 - 1.0 Dia bits 17,000 - 20,000rpm
1.0 – 1.5 Dia Bits 16,000 -18,000rpm
Anything larger than 1.5 Dia 15,000rpm

Feed Rates based on Process:

Pockets, Profiles, 3D machining, Texturing and Flutes
0 - 0.25 Dia bits 100 - 130ipm
0.25 - 0.5 Dia bits 80 - 120ipm
0.5 - 1.0 Dia bits 60 - 100ipm
1.0 – 1.5 Dia Bits 50 - 70ipm
Anything larger than 1.5 Dia 60-80ipm

V-Carve, Engraving Prism and Inlay
0 - 0.25 Dia bits 100 -110ipm
0.25 - 0.5 Dia bits 80 -100ipm
0.5 - 1.0 Dia bits 60 - 80ipm

Drilling:

0 - 0.25 Dia bits 30 - 50ipm
0.25 - 0.5 Dia bits 20 - 35ipm
Drill Bits 3,000rpm

Step Over

Step over is the percentage of the bit that is shifted over to cut new material each horizontal pass. Step over on all cutters should be at the maximum 40% of the bits’ diameter. Step over on 3D carving bits should range between 8% and 10% depending upon your personal preference and time allotted for the project. Step over on large diameter bits can be adjusted to a percentage smaller than 40 based upon your needs.

On the reverse side is the “Preferred Method to Calculate Bit Feeds and Speeds for the Guild’s CNC’s Machining Wood”