COMPLETE USER GUIDE

ToolPath Advisor

From setup wizard to perfect cut - everything you need to know.

Version 25f Neumann Guitar neumannguitar.com

What ToolPath Advisor Does

ToolPath Advisor answers the two questions that stop new CNC users cold: which bit do I use, and how fast do I run it? You tell it your machine, your spindle, and your CAM software once. After that: pick an operation, pick a material, and get a calibrated recommendation - the right bit geometry, RPM, feed rate, depth of cut, stepover, and plunge rate, formatted in your CAM software's exact field order. Everything runs in your browser. Nothing goes to a server. Your profile saves locally and comes back when you return.

Bookmark neumannguitar.com/advisor

Your machine profile, saved bits, and job history all live in your browser - not on a server. Bookmark this page now so you can return to your saved profile every time you sit down at the machine. Mac: Cmd+D Windows/Linux: Ctrl+D Mobile: tap Share then Add to Home Screen.

First Run: The Setup Wizard

On first visit, a four-step wizard collects the information that makes every recommendation specific to your setup. Takes about 60 seconds.

Step 1 - Machine + Spindle

Choose your CNC machine. The machine sets the rigidity scale - the single biggest factor in how aggressively the tool recommends you cut.

Machine	Rigidity	Notes
MPCNC	Low (2/5)	Mostly Printed CNC. Belt/printed-part limited. Conservative feeds required.
Shapeoko	Medium (3/5)	Carbide 3D. Most popular commercial hobbyist. Good all-rounder.
X-Carve	Medium (3/5)	Inventables. Large beginner ecosystem. Similar limits to Shapeoko.
Onefinity	High (4/5)	Ball-screw precision. Prosumer sweet spot. Handles full chip load well.
Carvera / Makera	High (4/5)	Enclosed ATC. Software-controlled spindle. Multi-tool workflow.
Generic / Other	You set it	Enter RPM range, wattage, frame rigidity (1-5), and collet sizes.

Step 2 - CAM Software

Choose your CAM software. This determines field labels and output order in every recommendation card.

Software	Output matches

Fusion 360	Tool Diameter, Flute Length, Spindle Speed, Feed Rate, Plunge Rate, Depth of Cut, Stepover, Coolant
Carbide Create	Cutter Dia., RPM, Feed Rate (in/min), Plunge Rate, Depth per Pass, Max Depth
VCarve / Aspire	Tool Dia., Pass Depth, Stepover %, Spindle Speed, Feed Rate, Plunge Rate, Tool Type
Easel	Bit Size, Cut Rate, Plunge Rate, Depth per Pass
Generic / Other	Drag-reorder fields to match your software

Steps 3 & 4 - Controller + Profile Summary

Select your machine controller (GRBL, LinuxCNC, Mach3/4, Marlin, FluidNC, or others). Then review your complete setup, optionally name your profile, and reorder output fields using the drag handles or up/down buttons. Hit Launch Advisor. Profile saves locally - it returns on every visit.

To edit later: the settings button in the header reopens the wizard to the Summary step. Or click any chip in the environment bar (Machine, Spindle, CAM, RPM) to jump to that step.

Tool Recommender

Operation Types

Operation	Use it for
Profile / Contour	Cutting around the outside shape of a part
Pocket	Cutting a flat-bottomed cavity into a part
Engraving / V-carve	V-bits for lettering, logos, decorative carving
Drilling	Pecking holes to a target depth
3D Surfacing	Ball-nose raster passes for carved surfaces and relief
Inlay	Small-diameter bits for pocket-and-insert inlay work

Experience Level

Level	Multiplier	Effect
Beginner	x0.75	Feed, plunge, and DOC at 75% of base. Extra margin. Start here if CNC is new to you.
Intermediate	x1.00	The calibrated default - in the cutting zone for a well-tuned hobby machine.
Experienced	x1.25	Full chip load. You know what smoke and chatter mean and how to respond.

Chip Load Gauge

Zone	Meaning	Action
Too Light	Bit rubbing, not cutting. Heat builds.	Raise feed rate or lower RPM.
OK	Target range for this material.	You're in the right zone.

Aggressive	Near the top of safe range.	Monitor for chatter. OK on rigid machines.
Danger	Overloaded. Deflection or breakage risk.	Reduce feed rate or raise RPM.

All 8 Tabs

Tool Recommender
Pick your machine and material - get the right bit geometry, size, and all CAM parameters formatted for your software.

Feeds & Speeds Calculator
Enter any bit on hand. Chip-load gauge, tuning sliders, RPM and feed rate output. Pre-populates from Tool Recommender.

Setup Sheet
Printable one-page cut summary. All parameters in one place - no digging through tabs mid-job. Print or screenshot before cutting.

Pre-Flight Checklist
33-item checklist in 7 sections. Adapts to your profile - spindle users see VFD items, router users see speed-dial items. Run before every new job.

Learn - 13 Articles
Chip load, depth of cut, feeds vs. speeds, tramming, runout, workholding, grain direction, G-code, and more. Theory behind the numbers.

G-Code Reference
Filtered to your controller dialect (GRBL, LinuxCNC, Mach3/4, Marlin, FluidNC). The codes you actually need at the machine.

Materials Library
13 categories - hardwoods, exotic tonewoods, SpectraPly, composites, HD plastics, aluminum, and more. Dust hazard ratings and CNC tips per species.

Job History
Every recommendation auto-logs. Save, search, filter by material, and reload any job. Settings stay in your browser - no account required.

Reading the Cut

Every number in ToolPath Advisor is a calibrated starting point. Your machine's actual condition - belt tension, tramming, collet runout, workholding rigidity - affects how aggressively you can cut. The numbers get you into the right zone. Your ears, your chips, and your surface tell you the rest.

Signal	Meaning
Smoke or burning smell	Feed is too slow, not too fast. The bit is rubbing. Speed up.
Squealing or chatter	DOC too deep, stepover too wide, or workholding is flexing. Back off incrementally.
Powdery dust instead of chips	Chip load is too light. Raise feed rate or lower RPM.
Clean curled chips	Cut is healthy. This is what you're aiming for.