

Speeds and Feeds

[← Back to Onefinity hub](#)

Speeds and Feeds

Use this page as a reference when setting up toolpaths in VCarve. Start conservative - increase feed rate only after successful test cuts on scrap material. Match RPM to material type and bit diameter.

1.5kW spindle limit: Avoid bits larger than ~1.5" diameter - excess rotational mass can fault the VFD.

Starter Values (Not Yet Shop-Verified)

Values below are placeholders derived from manufacturer guidance and Vectric defaults. They will be replaced after test cuts at the makerspace.

Material	Bit type	Diameter	RPM	Feed (IPM)	Depth/cut	Notes
Hardwood	Up-cut end mill	1/4"	TBD	TBD	TBD	From manual - verify at shop
Plywood	Up-cut end mill	1/4"	TBD	TBD	TBD	From manual - verify at shop
MDF	Up-cut end mill	1/4"	TBD	TBD	TBD	From manual - verify at shop
Plastic	Up-cut end mill	1/4"	TBD	TBD	TBD	From manual - verify at shop

TBD: Shop-verified speeds and feeds after test cuts. Rows will be tagged "shop-verified" when confirmed.

General Guidance

- Always confirm units (IPM vs mm/min) match your G-code post-processor
- Use VCarve’s tool database as a starting point, then verify on scrap material
- Reduce depth per pass if you hear chattering or see poor chip evacuation
- Refer to bit manufacturer recommendations for your specific cutter

Revision #4

Created 2026-06-22 21:49:53 UTC by Spencer

Updated 2026-06-22 22:24:43 UTC by Spencer