

## V2-50 Speeds and Feeds for Aluminum and Brass

When using a V2-50, you can turn up your spindle rpm but take smaller stepover cuts. Additionally, adding flutes to the endmill might add some speed to cutting but there is a loss of performance when using too many flutes because the V2-50 spindle does not have enough torque.

Below is an example formula for the V2-50 cutting aluminum with a 3 mm Datron endmill with a single flute. Notice that the RPM of the spindle is 40,000 but the Feed per Tooth is reduced to 0.00078 inches. The stepover is only 5% of the cutting diameter but since you are working in Aluminum, the depth of cut can be 127% of the tool diameter.

The screenshot displays a CNC control interface with three main panels. The left panel shows tool and feed settings, the middle panel shows tool and recipe details, and the right panel shows pass settings. Red circles highlight specific values, and red arrows indicate relationships between them.

Cutting Tool		Recipe	
Brand:	Datron	Cut Type	2D Chamfer
Part Number:	0068630E		2D Contour
Type:	Drill/Mill		Horizontal
	High Feed		3D Contour
	Chamfer		Slot
	Reamer		3D Parallel
	Corner Rounder		Adaptive
	Spot Drill		Tapping
	Drill		Adaptive (Both w
	Tap		Thread Mill
	End Mill		Bore
	Threadmill		Trace
	Engraver		Drill
	Woodruff/Saw		Other: _____
	Face Mill		
	Other: _____		
Manufacturer URI:	<a href="https://www.datron">https://www.datron</a>	Material	Aluminum
Purchase URL:	<a href="https://pocketnc.cc">https://pocketnc.cc</a>	Material Type:	6061
Price:	\$27.00	RPM:	40K
Overall Length:	40.00mm	SFM:	~1236.85
Material:	Carbide	Peak Spindle Load	N/A
	HSS	Programmed IPT	0.00078
	Cobalt	Inches per minute	31.2
	Other: _____	Radial DOC	.006" 5% diameter
Cutting Diameter:	3mm	Axial DOC	.15" 127% diameter
Shank Diameter:	0.125"	Gage Length:	N/A
Number of Flutes	1	Stickout:	1"
Flute Length:	11.5mm	Coolant	N/A
Corner Radius:	0		

**Feed & Speed Settings:**

- Spindle Speed: 10000 rpm
- Feed per Tooth: 0.003 in

**Passes Settings:**

- Stepover: 0.0875 in
- Maximum Stepdown: 0.059 in

There are always trade-offs when programming the tooling operation. If you reduce your depth of cut, then you can increase your stepover and vice versa.

The attached spreadsheet contains speeds and feeds for a 2 mm, 3 mm, and 4 mm single flute endmills from Datron. Your own speeds and feeds will vary depending on your tool's number of flutes, tool stickout, and manufacturer of the tool (depends on the tool coating).

Slow your speeds and feeds down when using multi-flute tools. Multi-flute tools remove more material per rotation but also require more torque from the spindle to rotate the tool through the material. The V2-50 spindle does not have very much torque. If your tool sticks out more than one 1 inch, you will have to also reduce your speeds and feeds because of tool deflection.

Use the sliders on the Kinetic Control Interface. Listen as the machine makes a cut and adjust the Feed Rate (or Max Velocity) slider until the machine sounds like it is making a smooth cut. Keep notes on these settings and adjust your tool operations parameters accordingly. The V2-50 spindle's torque drops off after 40,000 rpm. Increasing the spindle rpm will increase your cutting speed but if increased too much, the spindle will bog down and not make a clean cut.

When using larger endmills or tools with more flutes, use the 4 mm speeds and feeds formula in the V2-50 - Aluminum.xlsx spreadsheet and reduce the Feed per Tooth, Stepover, and Depth of Cut by about 20 to 30% as a starting point.