DENFORD

Spindle Speeds & Feed Rates

Microrouter
Microrouter Compact
Microrouter Pro
Vertical Router
Maxirouter
Micromill
Novamill
Triac
Triton

Spindle Speeds and Feedrates

These suggested values are for guidance only and may need to be adjusted from the values given. Tool wear, tool type and variations in material may affect the values. Values for spindle speeds are in rpm and values for feedrates are in mm/min.

Denford Microrouter Range

2D Programs

Material	Tool	Depth of Cut	Spindle Speed	Feed Rate
HDP/HIPS/ABS	1/8" - 1/4"	1.5mm	5000	800
Cast Acrylic	1/8" - 1/4"	1.5mm	20,000	800
MDF/Hard Wood	1/8" - 1/4"	= diameter of tool	23,000	1000
Foam/Balsa	1/8" - 1/4"	= max flute length	20,000	4000
Plywood	1/8" - 1/4"	2.5mm	18,000	2500

3D Programs (typically 10% - 40% stepover)

Material	Tool	Depth of Cut	Spindle Speed	Feed Rate
HDP/HIPS/ABS	1/8" - 1/4"	1.5mm	5000	1000
Cast Acrylic	1/8" - 1/4"	1.5mm	20,000	2000
MDF/Hard Wood	1/8" - 1/4"	= diameter of tool	23,000	2000
Foam/Balsa	1/8" - 1/4"	= max flute length	20,000	4000

(If you are cutting MDF or Plywood, Carbide cutters must be used.)

Denford Milling Machine Range

Material	Tool	Depth of Cut	Spindle Speed	Feed Rate
HDP/HIPS/ABS	2mm	1mm	1800	300
	6mm	3mm	1800	400
Cast Acrylic	2mm	1mm	1600	300
	6mm	3mm	1600	400
MDF/Hard Wood	2mm	= diameter of tool	2500	600
	6mm	= diameter of tool	2500	900
Foam/Balsa	2mm	= diameter of tool	2500	600
	6mm	= diameter of tool	2500	900
Free Cutting Aluminium	2mm	1mm	4000	75
	6mm	3mm	3000	120

Factors to consider when selecting speeds & feeds

If your CNC machine is used with incorrect spindle speeds and feedrates, your work may be machined with a poor surface finish or the workpiece or cutter could be damaged. There are many factors that would effect the decision you make to set or calculate these values, including:

- The maximum and minimum speeds and feeds available on the CNC machine.
- The condition of the CNC machine.
- The type of material being machined.
- The clamping method used to secure the billet in the CNC machine.
- The type of cutting tool used.
- The diameter of the cutting tool.
- The type of material used in the cutting tool.
- The condition of the cutting tool.

Do remember that these spindle speeds and feedrates are only approximate values, so they do not take into account factors such as tooling material types, diameters and profiles. With this in mind, we recommend performing a test cut, then changing the values according to any factors that may affect the machining of your design.

Troubleshooting with speed and feed values

Fault.	**Solution.**
Machine loses position, ie, some parts of the design are misaligned.	Lower the feedrate.
The billet material melts or fuses to cutter tip.	Lower the spindle speed or increase the feedrate.
The cutting tool tip becomes worn, blunt or burnt.	Lower the spindle speed.
The machine spindle keeps stalling (caused by low torque).	Increase the spindle speed.
Marks are produced across the surface of the design (cutter vibrates).	Increase the spindle speed

Notes:

Notes: