What feeds and speeds are achievable?

What is feed rate?

Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., aluminium, steel, wood, acrylic, etc.), the material of the cutter (carbide, high speed steel, ceramic, etc.) and many other cutting factors including desired surface and the characteristics of the CNC machine itself.

What is spindle speed?

Spindle speed, measured in RPM, is the speed at which your cutting tool rotates. This relates to the feed rate of your machine.

Climb vs. conventional milling

What is difference between climb vs. conventional milling?

The difference between these two techniques is the relationship of the rotation of the cutter to the direction of feed. In Conventional Milling, the cutter rotates against the direction of the feed. During Climb Milling, the cutter rotates with the feed.

We would recommend to use climb milling on SmartBench.

New to CNC Routing?

A general rule of thumb is to start slow and add then add speed. This is easy to do SmartBench as you can adjust the feeds and spindle speed during a job.

You will need to start somewhere, so, we have put together some recommended feeds and speeds in order to get you started.

YetiTool Ltd have tested this range of tools with a selection of materials to maintain accuracy, increase tool life longevity and achieve cleans cuts every time. Please be aware that there are multiple factors to consider, and with this in mind, the figures shown below are starting point with the ability to go up or down

where required. You should always check the condition of your cutter to ensure it is sharp and does not have damage which can affect the outcome of your finished part.

SmartBench starting feeds quick lookup table

MDF

YT Code	Diamter (mm)	Shank (mm)	Type	Maximum Step Down/Step Over (mm)	Recommended Feed rate (mm/min)	Recommended Spindle Speed (RPM)
20805	3	8	Upcut Spiral	1.5	2,000	18,000
20806	6	8	Upcut spiral	3	2,500	20,000
20807	8	8	Upcut spiral	4	3,000	22,000
20813	3.2	8	Round Nose	1.6	2,000	18,000
20814	6	8	V-Groove	2	2,000	20,000

Plywood

YT Code	Diamter (mm)	Shank (mm)	Type	Maximum Step Down/Step Over (mm)	Recommended Feed rate (mm/min)	Recommended Spindle Speed (RPM)
20805	3	8	Upcut Spiral	1.5	3,500	18,000
20806	6	8	Upcut spiral	3	3,000	20,000
20807	8	8	Upcut spiral	4	3,000	22,000
20813	3.2	8	Round Nose	1.6	2,500	18,000
20814	6	8	V-Groove	2	2,500	20,000

Document Name

Softwood

YT Code	Diamter (mm)	Shank (mm)	Type	Maximum Step Down/Step Over (mm)	Recommended Feed rate (mm/min)	Recommended Spindle Speed (RPM)
20805	3	8	Upcut Spiral	1.5	3,500	18,000
20806	6	8	Upcut spiral	3	3,000	20,000
20807	8	8	Upcut spiral	4	3,000	22,000
20813	3.2	8	Round Nose	1.6	2,500	18,000
20814	6	8	V-Groove	2	2,500	20,000

Hardwood

YT Code	Diamter (mm)	Shank (mm)	Type	Maximum Step Down/Step Over (mm)	Recommended Feed rate (mm/min)	Recommended Spindle Speed (RPM)
20805	3	8	Upcut Spiral	1.5	300	18,000
20806	6	8	Upcut spiral	3	400	20,000
20807	8	8	Upcut spiral	4	600	22,000
20813	3.2	8	Round Nose	1.6	300	18,000
20814	6	8	V-Groove	2	300	20,000

Document Name

Plastics

YT Code	Diamter (mm)	Shank (mm)	Type	Maximum Step Down/Step Over (mm)	Recommended Feed rate (mm/min)	Recommended Spindle Speed (RPM)
20805	3	8	Upcut Spiral	1.5	2,000	18,000
20806	6	8	Upcut spiral	3	2,500	20,000
20807	8	8	Upcut spiral	4	3,000	22,000
20813	3.2	8	Round Nose	1.6	2,000	18,000
20814	6	8	V-Groove	2	2,000	20,000

Aluminium

YT Code	Diamter (mm)	Shank (mm)	Type	Maximum Step Down/Step Over (mm)	Recommended Feed rate (mm/min)	Recommended Spindle Speed (RPM)
20538	6.35	6.35	Upcut Spiral	0.8	460	16,000

Composites

YT Code	Diamter (mm)	Shank (mm)	Type	Maximum Step Down/Step Over (mm)	Recommended Feed rate (mm/min)	Recommended Spindle Speed (RPM)
20815	18	8	ACM	2	800	16,000

Document Name

What to look out for

When your honing your feeds and speeds always look to make chips, not dust. Chips help in the process of taking heat away from the cutting process, thus increasing tool life and finished edge quality.

Always listen to the sound of the tool as it's cutting, if a router tool is working too hard you'll hear it.

Remember, SmartBench gives you the ability to adjust your feed speed on the fly.

Don’t worry if it seems daunting, it really isn’t, start low and build up as you gain experience. It’s better to achieve the cut part at a slightly slower speed than damage a tool or get a poor or inaccurate finish at a thousand miles an hour.

Problem solving