Calculating best feed rate for CNC routers

Definition of terms

Feedrate is defined as the velocity at which the cutter is fed. Put simply, it is the distance the tool travels during a single spindle revolution and is represented as distance per revolution.It is expressed in units of distance per revolution for turning and boring (mm/min)

The chip load is the thickness of the material that the tool removes per tooth per revolution, measured in inches or millimetres.

The spindle speed is the rotational speed of the spindle, measured in RPM

Feed rate is calculated using the following equation:
Feed = N x T x Z
Where
N = number of cutting edges (flutes)
T= chip load (chip per tooth) is the amount of material, which should be removed by each tooth of the cutter as it rotates and advances into the work. (mm per tooth)
Z= RPM, the speed at which the cutter revolves in the spindle. (Revolutions per minute)

For example
If you are running at 18000 RPM using a 25mm cutter with two flutes, and a recommended chip load of 0.1 mm/tooth:

Feed = 2 x 0.1 x 18000 = 3600 mm per min

Chip load is specific to the router bits hence one has to check the data sheet of the bits in question to come up with the right value to use for the calculations.

Suggested Feed rates

No of cutting edges (Z)	Chip Thickness (mm)	Feed rate (mm/min) at RPM		
		18000	21000	24000
1	0.1	1800	2100	2400
2	0.1	3600	4200	4800
3	0.1	5400	6300	7200
1	0.4	7200	8400	9600
2	0.4	14400	16800	19200
3	0.4	21600	25200	28800

Typical chip thickness values for various size cutters						
Cutter Diameter	Hardwood	Softwood/Ply	MDF/Particleboard	Soft Plastic	Hard Plastic	Aluminium
3mm	.08 -.13	.1 - .15	.1 - .18	.1 - .15	.15 - .2	.05 - .1
6mm	.23 - .28	.28 - .33	.33 - .41	.2 - .3	.25 - .3	.08 - .15
10mm	.38 - .46	.43 - .51	.51 - .58	.2 - .3	.25 - .3	.1 - .2
12mm and over	.48 - .53	.53 - .58	.64 - .69	.25 - .36	.3 - .41	.2 - .25

Factors to consider for good and efficient practice.

For hard material, use lower chipload than the design value. For soft material you can use a higher value.
A flute is designed to cut the same depth as the diameter of the tool, which means a 6mm tool should step down 6mm. The chip rate is designed according to that value.
You should take into consideration that if you go deeper, clearly more force will be applied on the tool, which means the tool will require more torque, making it more susceptible to snapping.
A rule of thumb is, if you step down twice, use 2/3 of the speed, if you step down 3 times, use 1/2 of the speed. It is advisable not to go any further otherwise the bit may snap.

NB this step down is not calculated from the surface of the material, rather from thickness remaining after each cut(pass).
This lead to another question, if I want to cut a board 20mm thick, can I use a 6mm tool with flute length only 6mm or 20mm+, answer is 20mm plus (thicker than the board), but you
may ask, if I only step down 6mm each time, then the rest of the flute is a waste which only increases manufacture cost and decrease the strength, In theory yes, but in reality, each pass has tolerance, maybe 0.01mm difference from the previous pass or more, then with a flute, it will cut, otherwise a solid rod will increase chance of snapping.
A good combination of feedrate(mm/min), spindle speed RPM, step down(mm) and tool itself can result in a clear cut, too fast will cause a waved edge, too slow will cause a rough edge.

KB Article by Takura Mutami