

Feeds and Speeds

Feeds and speeds is a little bit of a science, but very important to properly machining stock and also not damaging the machine. Here are some reference points to get you started

Feeds and speeds spreadsheet

<https://www.nycnc.com/speeds-feeds-excel-worksheet/>

General Rules of thumb

The following is a general starting point for regular endmills. These starting values do not apply for fly cutters, shear hogs, drill bits, taps and other special endmills

Start conservative and work you way up.

Material	IPT	Minimum IPT	Starting Depth of Cut (DOC)	Starting Width of Cut (WOC) Called Optimal load in Fusion 360 Adaptive	RPM				
Aluminum	.001	.0005	100% of the with of endmill. For example 1/4 endmill = .25 DOC	20% of tool width. For example 1/4 endmill = .05 WOC	Faster				
Steel	.001	.0005	100% of the with of endmill. For example 1/4 endmill = .25 DOC	20% of tool width. For example 1/4 endmill = .05 WOC	Slower				

Too shallow of cut is actually bad for tools. You want to make chips and not rub the tool.

Too fast or too much material will cause chatter producing a poor surface finish and damage. If the tool is chattering to the machine shaking stop immediately.

Recommended Videos

Aluminum and Steel

Steel