Speeds & Feeds

This chart provides starting parameters for calculating speeds and feeds. Actual feeds and speeds will depend on many variables including machine tool condition, rigidity, workpiece size and shape, tool extension, depth of cut, etc. If you have questions regarding a specific application please contact us.

	Workpiece Material			Examples	Hardness HB	Cutting Speed (SFM)				Feed (IPT)
						Uncoated	TiN	TiCN	TiAlN	
Steel	Carbon steels	<.25% C	Annealed	10xx, 11xx, 12xx, 12Lxx, 15xx	125	375-480	485-620	525-670	560-740	.003-.008
		>=0.25% C			190	280-360	365-470	390-500	490-660	.003-.008
		>=0.55% C			250	230-300	305-390	320-420	340-460	.003-.008
		>=0.25% C	Hardened	10xx, 15xx	250	–	300-350	340-430	390-520	.002-.006
		>=0.55% C			300	–	250-320	270-350	330-430	.002-.006
	Alloyed steels		Annealed	41xx, 43xx, 86xx	175	–	340-430	360-460	460-610	.002-.006
			Hardened	41xx, 43xx, 86xx	300	–	280-365	300-390	360-490	.002-.006
					350	–	245-315	270-340	340-460	.002-.006
	Tool & die steels		Annealed	A2, D2, H13, O1, S7	200	–	215-280	230-330	340-460	.002-.006
Stainless	Ferritic/Martensitic		Annealed	416, 420F, 430F	200	240-310	310-390	340-430	420-540	.002-.005
			Hardened	403, 410, 416	330	–	200-260	220-280	270-350	.002-.005
			PH-hardened	15-5 Ph, 17-4 Ph	330	–	180-230	200-250	240-315	.002-.005
	Austenitic		Annealed	304, 316, 321	180	160-215	–	220-300	280-380	.002-.004
Cast Iron	Gray		Pearlitic/ferritic	Class 20, 25, 30	180	225-285	–	315-400	460-620	.005-.010
			Pearlitic	Class 45, 50, 60	260	185-245	–	260-340	360-490	.005-.010
	Ductile		Ferritic	60-40-18, 80-55-06	160	420-450	550-575	600-630	460-620	.003-.008
			Pearlitic	100-70-03	250	240-310	310-400	340-430	390-520	.003-.008
Hi-temp	Titanium alloys		α+β alloys	Ti-6Al-4V		105-180	–	–	180-315	.002-.004
	High temp alloys			Inconel, Monel, Waspaloy		50-90	–	–	70-130	.002-.004
NF	Aluminum		Wrought	20xx, 50xx, 60xx, 70xx		1400-2100	–	–	–	.004-.006
	Brass, bronze, copper					780-900	–	1100-1300	–	.003-.005

General Operating Guidelines

Chip Load: Cut with an adequate chip load. Light chip loads can cause inserts to rub, rather than cut causing chatter and premature wear. Light radial cuts may require higher advances per tooth to counteract chip thinning.

Feed Rate: Reduce feed rate by 50% when entering and exiting a cut to reduce the shock of the interrupted cut.

Effective Cutting Edges: When calculating feed rate, use the effective number of inserts specified for the cutter. For staggered tooth slotting, this is 1/2 the total number of inserts.

Effective Cutter Diameter: Use an Effective Cutter Diameter when calculating RPM for tools with varying diameters like angle mills, countersinks, and corner rounding tools. Effective Cutter Diameter = (Major Diameter + Minor Diameter) divided by 2.

Cut Direction: Climb milling is recommended in most applications. Conventional milling may be necessary in older machines to minimize backlash. It can also extend insert life in sandy, scaly, welded or work hardened material.

Staggered Tooth Slotting:

- To calculate a starting radial depth of cut, multiply the cutter diameter by 0.10.
- If the cutter is engaged on both sides (as in t-slotting) reduce SFM to 75% of posted values and reduce chip loads to 50% of posted values.

Plunge Milling: When plunge milling, reduce the cutting data by 30%.

Speed Adjustments

Use lower end speeds when:	Use higher end speeds when:
Cutting harder materials	Cutting softer materials
Taking heavy cuts	Taking light cuts
Insert wear is a problem	Productivity is too low
	Finish is poor

Feed Adjustments

Use lower end speeds when:	Use higher end speeds when:
Fixturing or part is frail	Fixturing & part are sturdy
Tool overhang ratio is high	Tool overhang ratio is low
Narrow width slotting	Using wider, heavier cutter
Insert chipping occurs	Finish is not important
Finishing	Taking light radial cuts
Deep slotting	