

## 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
	Alloyed steels	Annealed	41xx, 43xx, 86xx	175	-	340-430	360-460	460-610	.002-.006	
				300	-	280-365	300-390	360-490	.002-.006	
				350	-	245-315	270-340	340-460	.002-.006	
Tool & die steels	Hardened	10xx, 15xx	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 diameter like angle mills, countersinks and corner rounding tools.  
 Effective Cutter Diameter = (Major Diameter + Minor Diameter)/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 feeds when:	Use higher end feeds when:
fixturing or part is frail	fixturing & part is 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	