STANDARD MILLING CALCULATIONS

COMMON EQUATIONS FOR OPTIMAL PERFORMANCE

The speed and motion of the cutting tool is determined by several factors. This page provides calculations to determine common industry measurements that will be required to program effective and optimized tool paths. Every application is different and may require varying feeds & speeds.

Cutting feed

The distance that the cutting tool advances during one revolution is measured in inches per revolution (IPR). Dependent on the procedure, the tool may feed into the workpiece or the workpiece may feed into the tool.

Feed rate

Feed rate is the speed of the end mill's movement correspondent to the workpiece. The feed rate is measured in inches per minute (IPM) and is the result of the cutting feed (IPR) and the spindle speed.

Speed and feed considerations are crucial for optimal results. Incorrect speeds and feeds can cause increased chatter, poor finish, hamper production, chip packing, damage the cutter, etc.

Too high of a speed or too light of a feed leads to reduction in tool life. Speed is measured in feet per minute and is referred to as cutting speed, surface speed, or peripheral speed. In the tables below, the relationship of peripheral speed to the diameter of the tool, and the rotational speed of the machine spindle are explained.

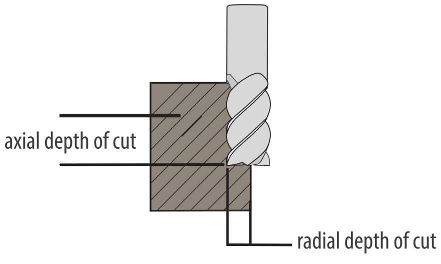
Inches per minute (IPM) is the standard for which feed is commonly measured. Feed is calculated by the number of cutting teeth in the end mill, multiplied by feed per tooth, multiplied by the revolutions per minute. Feed rates should be calculated from the chip load or feed per tooth.Regardless of the number of teeth in the tool, it is necessary that individual cutting teeth can adequately manage the feed that has been applied without breaking. Feed per tooth (FPT) affects thickness and is directly related to tool life. Maximum FPT creates longer tool life. Too high of a feed can strain the teeth causing breakage of the cutting edge. Sensible starting feeds for diameters under 0.5" range from 0.0002 to 0.002 IPT. Starting feeds for end mills over 0.5" diameter will range from 0.002 to 0.003 IPT.

Starting Points

Note that these are just starting parameters and basic information, we do not account for your particular machine or setup and there are many variables to consider. If you have any questions please do not hesitate to contact us.

KEY

ABBREVIATION	VARIABLE
D	Tool Diameter
R	Tool Radius
Z	Number of Flutes
RPM	Revolutions per Minute
SFM	Surface Feet per MInute (speed)
IPM	Inches per Minute (feed)
IPR	Inches per Revolution
FPT	Feed per Tooth
FPR	Feed per Revolution
MRR	Metal Removal Rate (Cubic Inches per Minute)
RDOC	Radial Depth of Cut
ADOC	Axial Depth of Cut
AFPT	Adjusted Feed per Tooth (Chip Thinning)
r_i	Part Radius (inside arc)
r_o	Part Radius (outside arc)



Feed Per Tooth	$= \frac{IPR}{Z}$	Feed Rate Adjustment - Outside Arc	$FPT_o = \frac{IPM \times (r_o + (R/2))}{\sqrt{r_o}}$
Inches Per Minute	$= RPM \times FPT \times Z$	Feed Rate Adjustment - Inside Arc	$FPT_i = \frac{IPM \times (r_i + (R/2))}{r_i}$
Inches Per Revolution	$= \frac{IPM}{RPM}$	IPT (Inches per Tooth)	$= \frac{(IPM / RPM)}{Z}$
Metal Removal Rate	$= RDOC \times ADOC \times IPM$	SFM (Surface Feet per Minute)	$= \frac{(RPM \times D)}{3.82}$
Revolutions Per Minute	$= \frac{SFM \times 3.82}{D}$	Ball Nose Effective Diameter	$D_{eff} = 2 \times \sqrt{R^2 - (R - ADOC)^2}$
Surface Feet Per Minute	$= RPM \times D \times .262$	Ball Nose Velocity Adjustment	$V_{adj} = \frac{SFM \times 3.82}{D_{eff}}$
Actual CLPT	$= \left(\frac{(D/2)}{RDOC}\right)^2 \times CLPT$	Adjusted Chip Load Per Tooth	$FPT_{act} = \frac{CLPT \times (D/2)}{\sqrt{(D*RDOC) - RDOC^2}}$

ADJUSTING STARTING SPEEDS AND FEEDS

Speed and feed considerations are crucial for optimal results. Incorrect speeds and feeds can cause increased chatter, poor finish, hamper production, chip packing, damage the cutter, etc. Too high of a speed or too light of a feed leads to reduction in tool life. Speed is measured in feet per minute and is referred to as cutting speed, surface speed, or peripheral speed. In the tables below, the relationship of peripheral speed to the diameter of the tool, and the rotational speed of the machine spindle are explained.

Inches per minute (IPM) is the standard for which feed is commonly measured. Feed is calculated by the number of cutting teeth in the end mill, multiplied by feed per tooth, multiplied by the revolutions per minute. Feed rates should be calculated from the chip load or feed per tooth. Regardless of the number of teeth in the tool, it is necessary that individual cutting teeth can adequately manage the feed that has been applied without breaking.

Feed per tooth (FPT) affects thickness and is directly related to tool life. Maximum FPT creates longer tool life. Too high of a feed can strain the teeth causing breakage of the cutting edge. Sensible starting feeds for diameters under 0.5" range from 0.0002 to 0.002 IPT. Starting feeds for end mills over 0.5" diameter will range from 0.002 to 0.003 IPT.

Starting Points

Note that these are just starting parameters and basic information, we do not account for your particular machine or setup and there are many variables to consider. If you have any questions please do not hesitate to contact us.

KEY

SYMBOL	ELEMENT	UNIT OF MEASUREMENT
HP	CUTTING POWER INPUT	horsepower
SFM	CUTTING SPEED	surface feet per minute
DOC	DEPTH OF CUT	inches
D	END MILL DIAMETER	inches
IPR	FEED PER REVOLUTION	inches per revolution
IPT	FEED PER TOOTH	inches per tooth
IPM	MACHINE FEED RATE	inches per minute
K	POWER CONSTANT	horsepower/cubic inch/minute
RPM	ROTATIONAL SPEED	revolutions per minute
WOC	WIDTH OF CUT	inches

SPEED ADJUSTMENTS

USE LOWER SPEEDS FOR:	USE HIGHER SPEEDS FOR:
hard materials	softer materials
tough materials	better finishes
abrasive materials	small diameter mills
heavy cuts	light cuts
minimum tool wear	frail workpiece or set-ups
maximum mill life	maximum production rates
	non metallics

FEED ADJUSTMENTS

USE HIGHER FEEDS FOR:	USE LIGHTER FEEDS FOR:
heavy roughing cuts	light and finishing cuts
rigid set-ups	frail set-ups
easy to machine work materials	hard to machine work materials
rugged heavy duty mills	deep slots
high tensile strength materials	frail and small diameter mills
coarse tooth mills	low tensile strength materials
abrasive materials	

MILLING CORRECTIONS

TROUBLE	CORRECTIVE ACTION
lack of rigidity	increase speed, decrease feed
excessive abrasion of the tool	decrease speed, increase feed
chipping of the cutting edge	decrease feed per tooth
burning of the cutting edge	decrease speed
chatter	use other combinations of speed and feed

SPEED AND FEED CALCULATIONS

FOR CALCULATING:	KNOWN VALUES	FORMULAE
CUTTING POWER INPUT - HP	width of cut, WOC depth of cut, DOC machine feed rate, IPM workpiece material constant, K	HP = WOC x DOC x IPM x K
FEED PER REVOLUTION - IPR	machine feed rate, IPM	IPR = IPM / RPM
FEED PER TOOTH - IPT	machine feed rate, IPM rotational speed, RPM number of teeth, T	IPT = IPM / (RPM x T)
MACHINE FEED RATE - IPM	rotational speed, RPM number of flutes (Teeth), T feed per tooth, IPT	IPM = T x IPT x RPM
PERIPHERAL CUTTING SPEED – SFM	mill diameter, D rotational speed RPM	SFM = 0.262 x RPM x D SFM estimated = (RPM x D) / 4
ROTATIONAL SPEED – RPM	peripheral cutting speed, SFM mill diameter, D	RPM = SFM /(0.262 x D) RPM estimated = (4 x SFM) / D

CONSTANTS

WORKPIECE MATERIAL	CONSTANT (K)	WORKPIECE MATERIAL	CONSTANT (K)	WORKPIECE MATERIAL	CONSTANT (K)
ALUMINUM	.3	**HIGH TEMP. ALLOYS**		**HIGH TENSILE ALLOYS**	
MAGNESIUM	.3	Ferritic	1.7	180,000 - 220,000 psi	2.0
COPPER	.5	Austenitic	2.0	220,000 - 260,000 psi	2.5
BRASS	.4	Nickel Base	2.5	260,000 - 300,000 psi	3.3
BRONZE	.5	Cobalt Base	2.5	**TITANIUM**	
CAST IRONS		**STEEL**		under 100,000 psi	1.3
FERRITIC	.7	up to 150 Brinell	1.4	100,000 - 135,000 psi	1.7
PEARLITIC	1.0	up to 300 Brinell	1.7	135,000 psi & over	2.5
CHILLED	1.7	up to 400 Brinell	2.0	**STAINLESS STEEL**	
MALLEABLE IRON	1.0	up to 500 Brinell	2.5	Free Machining	1.0
				Other	1.7