Technical Information

MANCHESTER CUTTING TOOLS
LOWER COST PER CUT

www.manchestertools.com

OTM TECH SUPPORT

The technical section of this catalog is intended to provide sufficient basic information to apply indexable cutting tools, specifically OTM products. The majority of requests for technical support that OTM receives are for speeds and feeds. If you need technical help beyond what is contained in this section, contact our technical staff.

APPLICATION CONSIDERATIONS

Feed and Speeds

Proper cutting speeds and feeds are obviously important considerations. OTM has developed some recommended starting points through testing in our shop and in the field.

Rigid Setups

No tool will perform at maximum potential in a poor setup. Workpieces must be held securely, and tool extension should be held to a minimum. Tombstones must be mounted properly, and workholding must be adequate. Worn jaws can introduce chatter, and possibly mis-locate parts, or even release parts during the cut. The machine itself must be in good condition, and **tool gage lengths should be as short as possible**. Shorter gage lengths result in higher feed capabilities with less deflection and chatter, and longer insert life.

Machine Horsepower vs. Spindle Horsepower

There can be a great difference between the spindle motor rating (machine horsepower) and the horsepower at the tool tip (spindle horsepower). Typically, spindle horsepower is around 75% of rated horsepower.

DRILLING BASICS

Proper drilling involves proper machine setup, coolant delivery, insert selection, and a basic understanding of the drilling process. Indexable drills are not the same as twist drills in application or performance.

BASIC DRILLING

Drilling a hole from solid stock is a tough job for a variety of reasons. Chip and heat evacuation are tough, the actual cutting action is hidden from view, and the surface footage varies from the programmed speed at the outer cutting edge to zero at the center of the tool. A drill doesn't actually cut at the center, it pushes the material out of the way. Cutting forces are high and other factors such as coolant delivery and machine alignment can combine to cause a drill to deflect and cut off-size, or fail catastrophically.

MACHINE SETUP

This is an important consideration when using indexable drills. In drilling, shorter is always better. Use the shortest drill and the shortest adapter possible. This will always improve performance. The machine spindle and bearings should be in good condition, and the fixturing must be adequate and rigid.

In lathes, *alignment is crucial.* The tool must run within 0.003" TIR to the spindle, or insert failure could occur. **If the center insert continually chips, check the alignment.** Chuck jaws should be in good condition as well and provide adequate clamping pressure. The workpiece should not extend from the chuck excessively either.

COOLANT DELIVERY

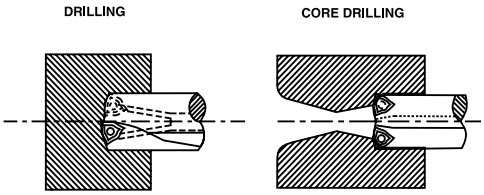
Coolant thru-the-spindle should be used wherever possible. Coolant improves tool life by lubricating the cut and removing heat and chips from the work area. Chip evacuation is very important when drilling.

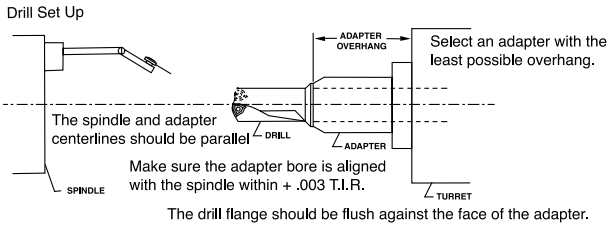
INDEXABLE VS. TWIST DRILLS

Indexable drills are very different than twist drills. Replaceable inserts eliminate drill regrinding. Because of higher surface footages, indexable drills generally out-feed twist drills.

DRILLS VS. CORE DRILLS

OTM drills are designed to put holes in solid material. OTM core drills are used to open a pre-existing hole. Core drills, because of their design, can attain two times the feed rate of insert drills. As a result, core drills provide an excellent means enlarging an existing hole.





DRILL APPLICATION HOLESHOT® INDEXABLE DRILL

STATIONARY

For stationary (lathe) applications, the drill body should be mounted concentric with the spindle center line within + .003". Longitudinal alignment should be within .005" in six inches. Insert failure can result if alignment is too far above or below the centerline. Flats on the shank should be precisely aligned so that the cutting edges are parallel to the x-axis, which will help chip flow.

A disc is normally produced as the drill breaks through the hole. Although the disc is usually minimal with HOLESHOT®, adequate guarding should be provided for and in place.

HOLESHOT® drills can be slightly offset .004" - .006", on lathe applications, to drill other than nominal sizes from solid. Offset can only be performed for plus sizing in the axis which increases/decreases cutting diameter. Offset in the other axis changes above/below center cutting conditions, resulting in breakage. Select an adapter with the least possible overhang.

ROTATING

Make certain the spindle is rigid with minimal runout. Since both machine and fixture rigidity are key factors, make sure the workpiece is rigidly fixtured and secured. Mount drill for the least possible overhang and make sure the drill flange is flush against the face of the adapter.

Coolant - Chip removal and tool life are enhanced by feeding coolant through the drill. 30 P.S.I. minimum coolant pressure is recommended for horizontal applications. Vertical position requires a higher coolant pressure (40 to 60 P.S.I.) to flush chips properly. HOLESHOT® accepts standard coolant glands.

Through the tool coolant is preferred for HOLESHOT® drilling, but due to the unique flute design, coolant deficiencies can be overcome. Especially in smaller, lower horsepower machine tools, strong flood coolant can be utilized with excellent results. When flooding the cut, direct coolant directly into the drilling area.

FEED / SPEED CALCULATIONS

UNIT CONVERSIONS

Feed and speed calculations use the following abbreviations:

FPT = Feed Per Tooth (Drills have one effective tooth, coredrills have two effective teeth)
FR = Feed Rate
IPM = Inches Per Minute
IPR = Inches Per Revolution
RPM = Revolutions Per Minute
SFM = Surface Feet Per Minute

The formulas are:

FPT = (IPR x RPM) / effective teeth
IPM = IPR x RPM or FPT x effective teeth x RPM
IPR = FPT x effective teeth
RPM = 3.82 x (SFM / drill diameter)
SFM = 0.262 x drill diameter x RPM

Cutting cycle time = length of cut (inches) / IPM

SURFACE SPEED PER MINUTE

$$SFM = .262 \times DIA \times RPM$$

REVOLUTIONS PER MINUTE

$$RPM = \frac{3.82 \times SFM}{DIA}$$

FEEDRATE (inches/minute)

$$IPM = IPR \times RPM$$

FEEDRATE (inches/revolution)

$$IPR = \frac{t_{chip}}{\cos \alpha}$$

METAL REMOVAL RATE

3 x Dia x IPR x SFM for Drilling
12 x WOC x IPR x SFM for Core Drilling

TIME IN CUT (seconds)

$$T = \frac{15.7 \times DIA \times LOC}{SFM \times IPR}$$

or

$$T = \frac{60 \times LOC}{IPM}$$

DEFINITION OF TERMS

DIA = DIAMETER OF THE DRILL (INCHES)
DOC = DEPTH OF CUT (INCHES)
EFF = MACHINE EFFICIENCY
HPM = HORSEPOWER REQUIRED AT MOTOR
HPS = HORSEPOWER REQUIRED AT SPINDLE
IPM = FEEDRATE (INCHES PER MINUTE)
IPR = FEEDRATE (INCHES PER REVOLUTION)
FR = FEEDRATE (SEE IPM AND IPR)
LOC = LENGTH OF CUT (INCHES)
Q = METAL REMOVAL RATE (CU. INCHES PER MINUTE)
RPM = REVOLUTIONS PER MINUTE
SFM = SURFACE FEET PER MINUTE
T = TIME (IN SECONDS)
t_{chip} = CHIP THICKNESS (INCHES)
UHP = UNIT HORSEPOWER FACTOR
a = LEAD ANGLE

DRILLING SPEEDS AND FEEDS
HOLESHOT® DRILLS AND COREMASTER™ COREDRILLS

NOTE: Drills are one-flute effective tools. Coredrills are two flute effective tools. Feedrates should be calculated accordingly.

MACHINING RECOMMENDATIONS

Material	Condition	Insert Grade*	SFM	FPT	Comments
Aluminum	6061	6N	1800-2000	.008-.010	
Cast Iron, Gray	<=150 BHN	10, 90C	400-800	.005-.011	
Cast Iron, Nodular (ductile)	150-250 BHN (<=25 Rc)	10, 66C	300-500	.004-.010	
Cast Iron, Malleable	250-350 BHN (25-38Rc)	10, 66C	350-700	.004-.010	
Inconel	718	66C	150	.0010-.0015	
Stainless Steel	15-5	66C	200-300	.0025	
	17-4	66C	450	.003	
	200-300 Series	66C	250-500	.003-.005	
	304	66C	425-550	.0040-.0055	
	400 Series	66C	350-600	.003-.006	
	410	66C	340-600	.004-.006	
Steel	No/Low Alloy CRS, HRS	66C	800-900	.0015-.0035	"Pleated", or W-shaped chips can be catastrophic here. Reduce feedrate to create smooth chipping.
	Carbon, <=38 RC	66C 90C	400-800	.005-.009	
	Tool, Med Alloy, 12-38 Rc	66C 90C	250-750	.004-.010	
	High-Temp Alloy, 12-38 Rc	66C 90C	100-350	.003-.006	
Stellite L605		66C	150	.0025	
Titanium		66C	100-210	.0025-.0030	

**Unless noted, use the same grade insert in all pockets of the drill.*

DRILLING SPEEDS AND FEEDS
STINGER™ SMALL DIAMETER DRILLS

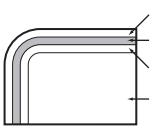
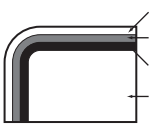
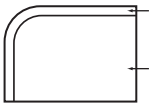
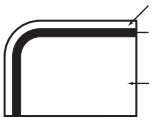
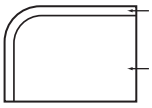
NOTES: 1. Do not overtighten the insert screws in these drills. Over-tightening the screws will break the inserts.
2. Do not drill stacked plates with these drills.

MACHINING RECOMMENDATIONS

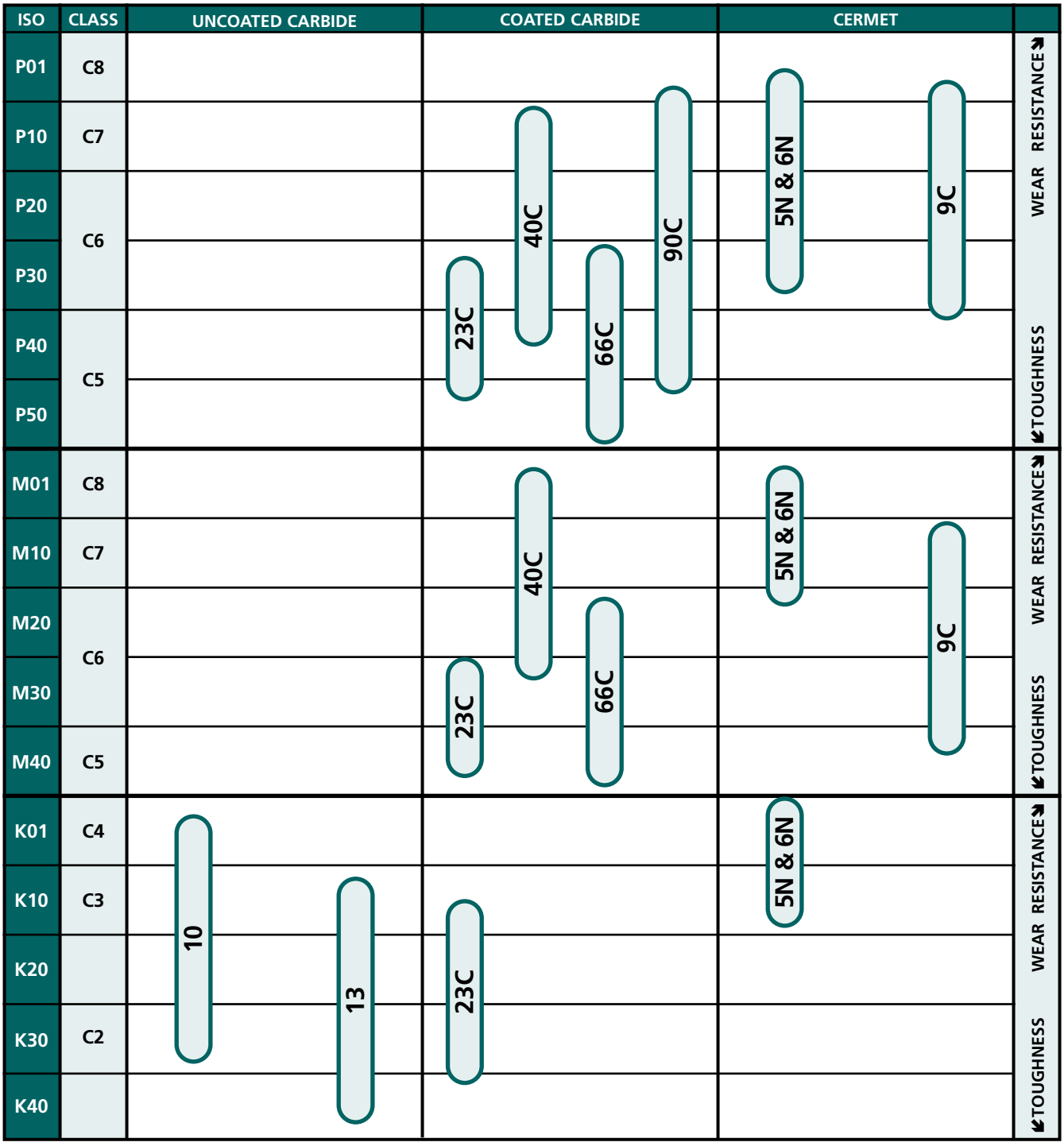
Material	Condition	Insert Grade*	SFM	FPT
Copper		5N	900-2000	001-.003
		13	800-1900	.001-.003
High Temp Alloys		13	100-175	.0005-.0008
Inconel	600	13	250	.0008-.0012
	718	13	150-250	.0015-.0025
Stainless Steel	17-4	13	450	.0009-.0012
	300 Series	23C	300-500	.0012-.0015
		40C	400-600	.0012-.0015
	304	23C	300-600	.0008-.0012
		40C	400-700	.0008-.0012
	400 Series	23C	400-600	.0012-.0015
		40C	500-700	.0012-.0015
Steel, Low Carbon	Annealed	5N	600-900	.0012-.0018
		40C	500-800	.0012-.0018
	Carburized, 35-50 Rc	5N	300-400	.0012-.0018
		40C	250-350	.0012-.0018
Steel, Medium Carbon	17-32 Rc	5N	500-700	.0012-.003
		40C	400-600	.0012-.003
Titanium Alloys		13	150-250	.0008-.0012
Tool Steel	Annealed	5N	350-450	.001-.0015
		5N	150-250	.0007-.0012
4130, 4140, 4150	Annealed	5N	500-700	.0012-.003
	35-50 Rc	40C	400-600	.0012-.003
6150	Annealed	5N	500-600	.0012-.0015
		40C	400-600	.0012-.0015
Aluminum		5N	1000+	.002-.006
		13	900-1800	.002-.006
Brass		5N	900-1500	.001-.003
		13	800-1400	.001-.003
Bronze		5N	500-900	.001-.003
		13	400-600	.001-.003
Cast Iron	Gray	5N	500-900	.001-.003
		13	300-500	.001-.003
		23C	300-500	.001-.003
	Nodular (Ductile)	5N	500-700	.001-.003
		13	300-500	.001-.003
		23C	350-450	.001-.0022
	Malleable	5N	400-800	.001-.003
		13	300-500	.001-.003
		23C	300-500	.001-.003

**Unless noted, use the same grade insert in all pockets of the drill.*

INSERT GRADES FOR DRILLING

	GRADES	COMPOSITION & COATINGS	APPLICATION RECOMMENDATIONS
COATED CARBIDE GRADES	23C		Tough CVD coated carbide grade for demanding applications in steel, stainless steel and steel castings. Takes shock and interruptions well.
	40C		CVD coated. Thick wear resistant coating layers over a cobalt enriched substrate provide high wear resistance with good edge security, drilling in most steels.
	66C		Very tough PVD coated grade for drilling alloy steel, stainless and tougher materials.
	90C		CVD coated grade for drilling applications in steel requiring toughness with good wear resistance.
UNCOATED CARBIDE	10		Uncoated carbide grade for drilling in gray cast iron and non-ferrous materials.
	13		Micrograin uncoated carbide for drilling heat resistant and titanium alloys.
CERMET GRADES	5N		Tough cermet with good wear resistance for drilling in most steels. Not for center pocket. (Except for the Stinger Drills)
	6N		Tough cermet with good wear resistance for drilling in most steels. Not for center pocket.
	9C		PVD coated cermet for roughing to finishing of steels. Not recommended for center pocket.

CLASSIFICATION CHART FOR DRILLING GRADES



ISO DESIGNATIONS	
P	STEELS, TOOL STEELS
M	STAINLESS STEELS
K	CAST IRON & NON-FERROUS METALS

C CLASSIFICATIONS	
C-1 through C-4	C-5 through C-8
Abrasion resistant grades. Machining cast iron & short chip materials.	Crater resistant grades. Machining steel & continuous chip materials.
C-1 Roughing C-2 General Purpose C-3 Semi-Finishing C-4 Finishing & Precision Boring	C-5 Roughing & General Purpose C-6 Semi-Finishing C-7 Light Finishing C-8 Finishing & Precision Boring

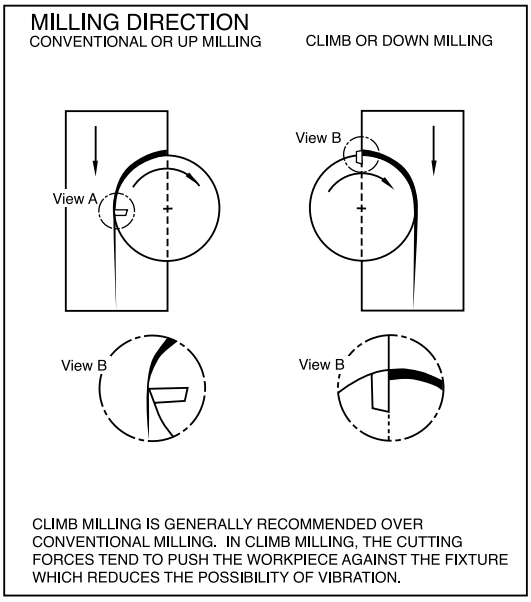
MILLING BASICS

BASIC MILLING

Most machining benefits from rigid set-ups. Milling applications benefit greatly. Short adapters and rigid fixturing will greatly increase tool life and feed capabilities.

CLIMB MILLING VS. CONVENTIONAL MILLING

Climb mill whenever possible. The inserts enter the cut at full feed per tooth, and exit as the chip thins to zero. Less heat is generated and work hardening is minimized. Today's inserts can easily take the forces generated, and free-cutting positive geometries won't shake the machine to pieces. Exceptions to this are manual machines and worn out CNC machines, where conventional milling is necessary because of backlash.



POSITIVE VS. NEGATIVE GEOMETRY

Traditional indexable mills have used negative inserts extensively. Today's mills use positive inserts. Positive inserts have only half, or less, usable edges than negative inserts, but the free cutting action of positives usually outweighs the effects of edges lost. Today's insert compositions and edge geometries make strong, free-cutting edges that hold up in the cut. Feedrates are higher than negative inserts, horsepower requirements are lower, and tool life is greater. In the end, the cost per part produced is generally lower using positive inserts cutting most materials.

FACE MILLING

Whenever possible, face with a 30-degree lead mill. This will cut more freely, lowering horsepower requirements, and increasing tool life. The only time face milling with a 90-degree lead mill is appropriate is when a square corner is required along a cut.

PROFILE MILLING

Profile milling is a common milling application. The speeds below apply to profile milling. The feedrates can be increased, in some cases dramatically. The reason is that when the width of cut is less than half the cutter diameter, chip thinning occurs. Higher feed rates are actually required to create the necessary feed per tooth.

SLOTTING

Slotting is the most difficult milling application. The cutter begins cutting a thin chip, continues into a thick chip, then exits the cut through a thin chip again. The chip has one-half revolution to get free of the flute. Chip evacuation is critical, and chips can pile up behind the cut until they fall back into the mill and get re-cut. Re-cut chips will destroy insert edges quickly, leading to possible catastrophic tool failure. Deeper slots are even more difficult. Compressed air should be used to move all chips out of the work area, and feedrates must be appropriate for the material but not excessive.

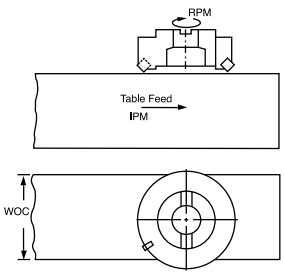
LEFT-HAND MILLS

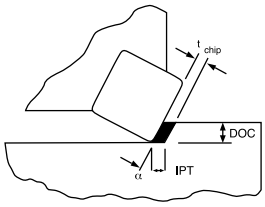
OTM has a large selection of standard left-hand mills designed specifically to utilize the unused portions of the inserts from right-hand mills. Economy is the driving force of this design.

TIPS AND TRICKS

When milling around sharp corners, whenever possible interpolate the corner, using the corner as the center of the arc and the cutter radius as the arc radius. This will keep the cutter loaded through the corner, eliminate the exit and entrance cuts, and not leave a burr on the corner. Insert life will improve, and it's also a shorter distance to travel.

When making multiple passes to get to depth, don't divide the depth into equal increments. Instead, vary the DOC with each pass. This will increase insert life by reducing notch wear of the insert. (Notch wear occurs at the point the insert intersects the top edge of the cut.)





MILLING CHECKLIST

1. Properly identify the milling task process by reviewing part print parameters by dimension and material to be machined.
2. Identify the best machine tool and process for optimum performance.
3. Review the machine capabilities by:
    a. Rigidity of set-up
    b. Horsepower
    c. Axis travel
    d. Speed and feed limits
    e. Overall condition
    f. Spindle configuration
4. Select a cutter:
    a. By style (facing, chamfering, etc.)
    b. By task:
        1) Cut diameter
        2) Lead Angle
        3) Mounting method
        4) Hand
    c. Select inserts by proper grade, geometry and top prep.
    d. Select machining data:
        1) Recommended surface speed (ft/min)
        2) Calculate RPM
        3) Select feed rate per tooth (when radial depth of cut is less than 50% of the diameter, chip load should be increased accordingly)
        4) Check K factor (effective number of teeth in cut)
        5) Calculate table feed
        6) Calculate metal removal rate and power consumption

OTHER CONSIDERATIONS

Coolants

Coolants are **not recommended for milling with cermet and ceramic inserts.** Coolant exposes the inserts to damage from thermal shock caused by the intermittent heating and cooling cycles. Directed air pressure or sometimes an aerated mist can aid in chip flow.

Cutter Handling

A cutter should be maintained in good condition, making sure the cutter is properly cleaned and changing spare parts.

Insert Care, Mounting and Indexing

With the exception of single point cutters, each milling insert is dependent on the others in the cutter. Failure of one insert increases chip load on the others, usually resulting in severe breakage of the rest.

To insure proper chip load on all inserts, care must be taken to seat them properly into the pocket.

When mounting new inserts or indexing, the pockets and/or cartridges must be thoroughly cleaned of any loose material.

Milling inserts must be eventually replaced due to failure by wear or fracturing. Observation of insert condition during use, and the nature of it, will indicate satisfactory or unsatisfactory performance.

MOUNTING CONSIDERATIONS

Integral shank cutters offer the best rigidity. Use appropriate style end mill holder or collet chuck with non-pull collets. Special continuous clamping milling chucks usually result in the most rigid set-up. On shell mills, make sure arbor is proper size, type and in good repair. Run-out, chatter, poor surface finish and poor tool life can sometimes be traced to poor or improper mounting.

MILLING METHOD CONSIDERATIONS

Down milling (climb milling), or milling in the direction of the feed, should always be the first choice. In down milling, the insert enters through the un-milled surface, and produces a chip that becomes progressively thinner as the insert pushes through the part, which helps in materials that tend to work-harden. Down-milling also places downward pressure on the workpiece and forces the work in the direction of the feed. The thin chip exit helps insert edge security, better tool life and of considerable importance in milling heat resistant materials.

FEED / SPEED CALCULATIONS

UNIT CONVERSIONS

Feed and speed calculations use the following abbreviations:

DEFINITION OF TERMS

WOC = WIDTH OF CUT (INCHES)
DIA = CUTTER DIAMETER (INCHES)
DOC = DEPTH OF CUT (INCHES)
EFF = MACHINE EFF
f = FEEDRATE (SEE IPM, IPR, AND IPT)
HPM = HORSEPOWER REQUIRED AT THE MOTOR (HP)
HPS = HORSEPOWER REQUIRED AT THE SPINDLE (HP)
IPM = FEEDRATE (INCHES PER MINUTE)
IPR = FEEDRATE (INCHES PER REVOLUTION)
IPT = FEEDRATE (INCHES PER TOOTH)
LOC = LENGTH OF CUT (INCHES)
N = NUMBER OF EFFECTIVE TEETH IN CUTTER
Q = METAL REMOVAL RATE (CUBIC INCHES PER MINUTE)
RPM = REVOLUTIONS PER MINUTE
SFM = SURFACE FEET PER MINUTE
T = TIME (SECONDS)
t_{chip} = CHIP THICKNESS (INCHES)
UHP = UNIT HORSEPOWER FACTOR
α = LEAD ANGLE

Cutting cycle time=length of cut (inches)/IPM
Cubic inches removed per minute =IPM x WOC x DOC

SURFACE SPEED

$$SFM = .262 \times DIA \times RPM$$

REVOLUTIONS PER MINUTE

$$RPM = \frac{3.82 \times SFM}{DIA}$$

FEEDRATE (INCHES/REVOLUTION)

$$IPM = IPT \times N \times RPM$$

FEEDRATE (INCHES/REVOLUTION)

$$IPR = \frac{IPM}{RPM}$$

FEEDRATE (INCHES/TOOTH)

$$IPT = \frac{t_{chip}}{\cos\alpha}$$

METAL REMOVAL RATE

$$Q = WOC \times DOC \times IPM \ (in^3/min)$$

TIME IN CUT (IN SECONDS)

$$T = \frac{15.7 \times DIA \times LOC}{SFM \times IPT \times N}$$

or

$$T = \frac{60 \times LOC}{IPM}$$

OPTIMUM FEED RATES

The following formula can be used to determine optimum feed rates when chip thinning occurs:

$$IPM = \frac{\text{desired FPT}}{\sqrt{\frac{(\text{cutter DIA} - WOC) \times WOC}{\text{cutter RADIUS}}}} \times \text{effective teeth} \times RPM$$

The **lead angle** of the cutter also affects chip thinning. Only when using a 90-degree lead mill is the chip thickness equal to the feed advance per tooth. When the lead angle is 30 degrees, the chipload is actually only 87% of the feed advancement, resulting once again in less than optimum metal removal rates. Using the cosine of the lead angle the correct FPT can be determined.

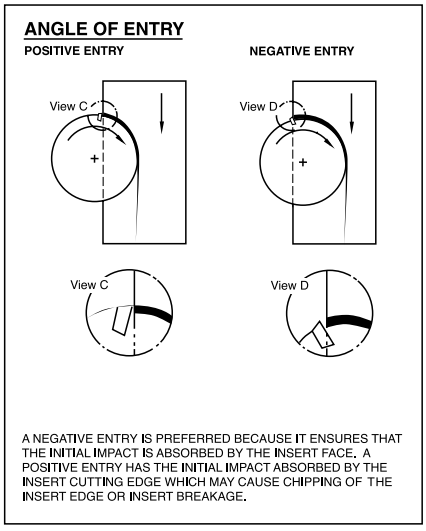
FPT = desired FPT / COS (lead angle)

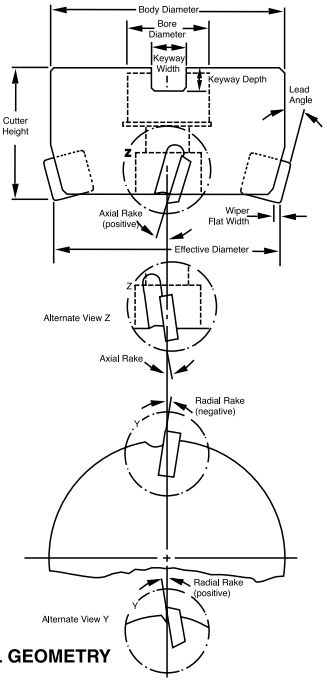
These two formulas appear to be a lot of trouble over relatively small gains, but when proper feedrates are applied, tool life will increase, cycle time will drop, and better part quality will result.

CHIP THINNING

In milling, the feedrate is also affected by the width of cut (WOC). As long as the WOC equals or exceeds the radius of the cutter, the FPT will equal the chip thickness. Whenever the WOC is less than the radius of the cutter, chip thinning occurs, meaning the chip thickness is less than the FPT (feed advance per tooth).

This results in lower metal removal rates, less efficient machining, higher cutting temperatures, and early insert failure.





TOOL GEOMETRY

MILLING FEEDS AND SPEEDS

MACHINING RECOMMENDATIONS

Material	Condition	Insert Grade	SFM	FPT	Comments
Aluminum	Low Silicon (< 8%)	5, 6, 13	1500-1800	.005-.007	
		12	2000-4/5000 SFM or 10K RPM or so max	.005-.020	Use Coolant
Aluminum, Aluminum-Bronze	High Silicon (> 8%)	66C	800-1100	.005-.008	
Bronze		1, 5, 6	1000-1100	.005-.008	
Bronze	Malleable	1, 5, 6, 10	400-700	.003-.006	
	Gray	10, 23C, 93C	600-1200	.004-.016	
	Nodular (Ductile)	1, 5, 6, 10	600-850	.004-.008	
Copper		1, 5, 6	1200	.007	
CPM 9V	9% Vanadium, 5.25% Chromium	23C	310-400	.005-.006	Flood Coolant
D2	Annealed	1, 5, 6	400	.002	
Inconel		93C	100	.003	0.060 DOC Max
	718	10	100-200	.001-.002	0.100 DOC Max
Invar 36	30% Nickel	93C	100-250	.003-.004	
Stainless Steel	Ferritic Martensitic	5, 6, 23C, 66C, 93C	500-800	.002-.006	
	Cast	5, 6, 23C, 66C, 93C	500-800	.003-.006	
	304	5, 6, 23C, 66C, 93C	400	.003	
	316L	5, 6, 23C, 66C, 93C	300	.003-.004	
Steel, Unalloyed		1, 5, 6	800-1400	.003-.006	
Steel, Low Alloy	Annealed	1, 5, 6	600-1200	.003-.006	
	Hardened	1, 5, 6	400-700	.002-.005	
Steel, High Alloy	Annealed	1, 5, 6	400-700	.003-.006	
	Hardened Rc	1, 5, 6	250	.002	
Steel, Cast	Low Alloy	1, 5, 6	600-1200	.004-.008	
	High Alloy	1, 5, 6	400-700	.003-.006	
Titanium		93C, 23C, 13	100-210	.002-.003	
Tungsten		6	400	.002-.004	

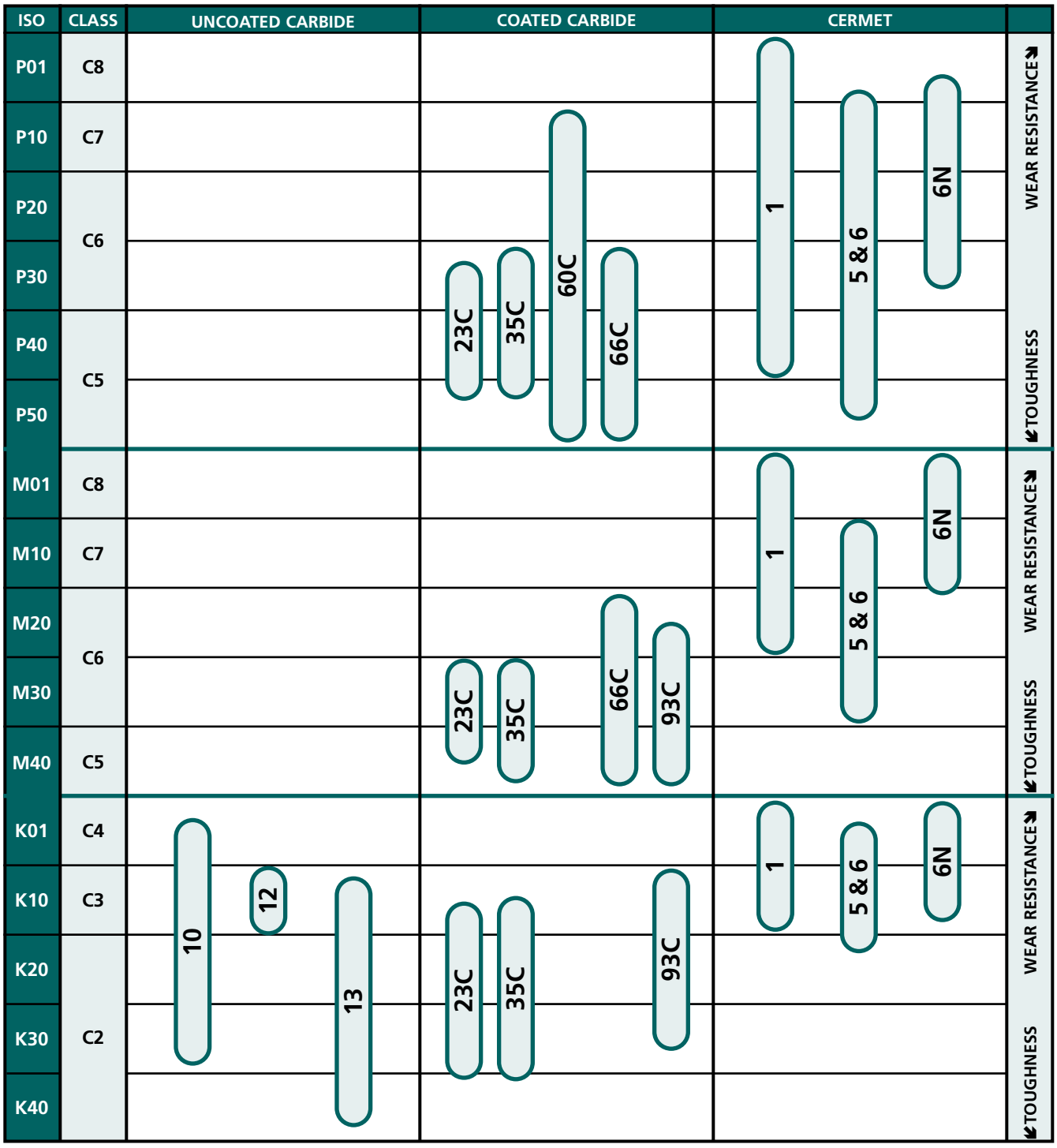
NOTE: *Do not use coolant on Grades 1, 5, or 6 unless specifically noted.*

LONG EDGE MILLING

Slot milling with long edge mills is not recommended. When using long edge mills to circular interpolate inside diameters, allow sufficient room for chip clearance. Use of forced, directed air may be required in all cases. Depth of cut must be carefully considered relative to the machine horsepower.

Speeds and feeds shown above are suggested starting points. Actual speed and feeds may vary due to material, machine and operating conditions. Test cuts are recommended, checking machine and holding rigidity prior to cutting at the high range of recommended speed and feed. As depth of cut increases and/or full slotting, modify speed and feed downward accordingly. All speeds and feeds that apply to cermets should be reduced 10-20% when using carbide grades.

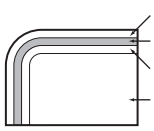
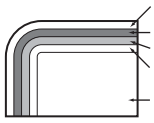
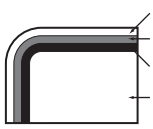
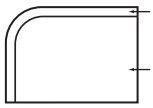
CLASSIFICATION CHART FOR MILLING GRADES



ISO DESIGNATIONS	
P	STEELS, TOOL STEELS
M	STAINLESS STEELS
K	CAST IRON & NON-FERROUS METALS

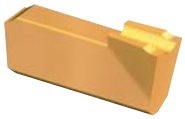
C CLASSIFICATIONS	
C-1 through C-4	C-5 through C-8
Abrasion resistant grades. Machining cast iron & short chip materials.	Crater resistant grades. Machining steel & continuous chip materials.
C-1 Roughing C-2 General Purpose C-3 Semi-Finishing C-4 Finishing & Precision Boring	C-5 Roughing & General Purpose C-6 Semi-Finishing C-7 Light Finishing C-8 Finishing & Precision Boring

INSERT GRADES FOR MILLING

	GRADES	COMPOSITION & COATINGS	APPLICATION RECOMMENDATIONS
COATED CARBIDE GRADES	23C		Tough CVD coated carbide grade for demanding applications in steel, stainless steel and steel castings. Takes shock and interruptions well.
	35C		CVD coatedgrade for milling steel, stainless steel and cast steel.
	60C		Tough CVD coated grade for roughing and semi-roughing at low to medium speeds in steel.
	66C		PVD coated grade as a first choice in milling tough to machine stainless steels and other tough materials.
	93C		PVD coated micrograin carbide for sharp edge applications when wear resistance is required. Recommended for tough stainless steel, high temp and titanium alloys.
UNCOATED CARBIDE	10		Micrograin uncoated carbide recommended as first choice in milling gray cast iron. Works well in most aluminum and non-ferrous materials, and as an alternate choice for high nickel/high temp alloy materials.
	12		Micrograin uncoated carbide primarily for milling aluminum.
	13		Micrograin uncoated carbide recommended first heat resistant and titanium alloys; also aluminum and non-ferrous materials.
CERMET GRADES	1		A newly developed "super cermet" grade for rough milling of steel, tool steel and some stainless steels. Secondary application in cast iron, aluminum and other non-ferrous materials. Has toughness and shock resistance paralleling Grade 6, but with better wear resistance.
	5		Tough cermet, but also exhibits good wear resistance. Recommended primarily for steel, tool steels and some stainless steels. Also shows some versatility in a variety of other materials.
	6		Toughest cermet with good shock resistance for rough milling in a wide range of materials. Primarily recommended for tool steel applications
	6N		Toughness combined with good wear resistance. Covers a wide range of applications and handles interruptions well.

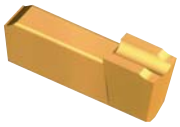
CHOOSING THE APPROPRIATE SEPARATOR® CUTOFF INSERT

Separator® Classic



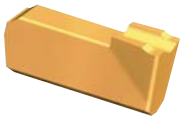
A good general purpose insert for carbon steels, alloy steels and most stainless steels. The Separator Classic chip breaker is designed to perform well at moderate to slow speeds and feeds. The Classic offers standard high lead angles and sharp corners making it the first choice when choosing an insert for nib free cutoff.

Separator® F²



This insert offers superior flatness and finish on a wide variety of materials. Ideal for thick wall parts or cutting off larger diameter parts to center. The Separator F² performs well at slow to moderate speeds and feeds.

Separator® G²



This insert has a corner radius and a slightly more open chip breaker. It's an ideal general purpose insert for CNC machine cutoff applications. Performs well at moderate speeds and feeds on carbon steels, alloy steels and most stainless steels.

Separator® D²

This insert has a high positive dish style chipbreaker. The chip breaker geometry has excellent edge strength for high feed rate applications. The neutral (0 degree lead) version can be used for grooving applications.

Separator® S²

This high positive rake with more open chip breaker allows for increased speeds and feeds for the moderate to high speed applications. The geometry also includes wipers and a corner radius that provides superior flatness and finish. This insert is also available with sharp corners. This geometry works well on a variety of materials although it's greatest strengths can be seen on Stainless Steels and soft gummy steels.

Separator® S²-Ultra

The S²-Ultra is an enhanced version of the S² and is ideal for 300 series stainless, Nickel Based alloys, Tool Steel, Inconel and Titanium at moderate to high speeds and feeds.

Separator® X²

This insert has the same geometry as the MTC-SX™. This chip control geometry offers the widest range of speed and feed capability and provides excellent flatness and finish. This chip breaker cuts with the least amount of tool pressure, extends tool life. The geometry also includes wipers and a corner radius. This geometry works well on a variety of materials.

Separator® X²-Ultra

This insert has the same geometry as the MTC-SX™. The X²-Ultra is an enhanced version of the X² and is ideal for stainless steels, Nickel Based alloys, Tool Steel, Inconel and Titanium.

The inserts listed thus far are single end inserts that go into standard componentized and integral shank holders with up to 3" bar capacity capability. Greater depths of cut and bar capacity can be achieved through customizing and/or special systems. If necessary these Separator inserts can be customized for a particular application.

Separator® PL

This insert is Pressure Locked into a blade style system that allows for varying depths of cut depending on the blade extension from the holder. This insert has a good general purpose chip breaker and is available in sharp corner or corner radius. Ideal for medium to slow speed conditions on a wide variety of materials.

MTC-SX™

This double ended insert is the latest Separator design. The chip control geometry offers the widest range of speed and feed capability and provides excellent flatness and finish. This chip breaker cuts with the least amount of tool pressure, extends tool life. The longer insert and double V offer maximum insert stability. Operator friendly insert indexing. This insert fits into standard and componentized holders capable of up to 25mm depth of cut.

MTC-SX™-Ultra

The SX-Ultra is an enhanced version of the SX and is ideal for 300 series stainless, Nickel Based alloys, Tool Steel, Inconel and Titanium at moderate speeds and feeds.

To improve chip control:

- Use an insert with aggressive chip control geometry. See Table 1, or call Tech. Support.
- Adjust feed rate up or down to accommodate chip formation.
- Use a 0° or smallest lead available.
- Use ample amounts of well-directed coolant. *(See Illustration A.)*
- Maintain sharp cutting edge and corners.

Table 1: Insert Selection For Aggressive Chip Control Requirements

Width	Insert No.	Grade
3/32"	507-207	M45
3mm	507-314	M45
1/8"	507-224	M45
4mm	507-225	M45
3/16"	507-226	M45

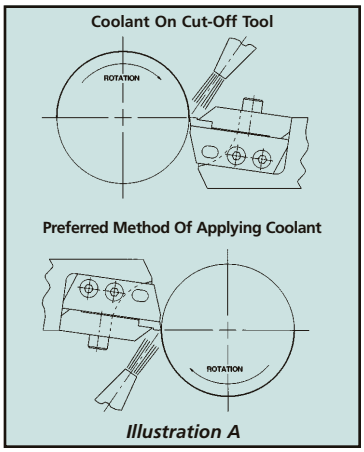


Illustration A

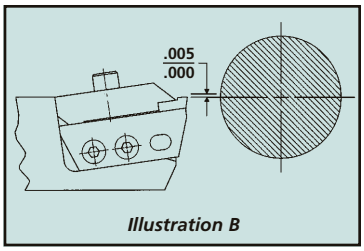


Illustration B

To improve flatness of cutoff surfaces:

- Maintain 90° (perpendicular alignment) position between cutoff tool and work piece.
- For low to moderate speed (sfpm) use Separator F².
- For moderate to high-speed (sfpm) use Separator S² or SX.
- Use strongest tool holder system possible.
- Use 0° lead angle inserts when possible. If lead angle inserts are needed, reduce the feed rate.
- Check for minimum overhang of holder and blade.
- Set up for minimum work piece overhang (distance out of chuck).
- Reduce feed rate.
- Maintain sharp edge and corners on cutoff insert.
- Increase speed (rpm).
- Use ample amounts of well-directed coolant. *(See Illustration A.)*
- Maintain proper tool center height .000" to .005" above center. *(See Illustration B.)*.

To improve surface finish:

- For low to moderate speed (sfpm) use Separator F².
- For moderate to high-speed (sfpm) use Separator S² or SX.
- Avoid overly aggressive chip control.
- Increase speed.
- Reduce lead angle.
- Decrease the feed rate .
- Corner radius too large or small.
- Use a coated grade.
- Use coolant. *(See Illustration A.)*

To minimize edge chipping:

- Check to see if tool is significantly above or below center.
- Reduce feed prior to part drop off.
- Use Separator S², SX or D².
- Choose the proper speed associated with the insert grade used.
- Call Tech. Support to see if a larger hone size is needed.
- Eliminate chatter.
- Avoid chip re-cutting.
- Check for these part and machine problems:
  - Slide is loose.
  - Slide travel is irregular.
  - Bar/tube I.D. and/or O.D. is out of round.
  - Bar/tube is bent.
  - Thin wall collapses (deforms) in the cut.
  - Part is unstable.
  - Cutoff through unturned stock.
  - Excessive tool overhang.
  - Bent or partly attached flash ring.

To eliminate chatter:

- Minimize tool blade and holder overhang.
- Minimize part overhang.
- Use strongest tool holder system.
- Use a narrower width of insert.
- Chip breaker might be too aggressive, call Tech. Support.
- Adjust speed up or down.
- Adjust feed rate up or down.
- Have work piece held rigidly.
- With a longer part, support with steady rest or live center.
- Avoid machine dwell.
- Use S² or SX to reduce cutting forces.

To reduce the cutoff nib on a solid bar or I.D. burr on tubing:

- Check tool height. Insert cutting edge should be on center to .002" above centerline of work piece.
- To reduce nib on part, use a high lead angle type insert. Lead angle on a cutoff insert reduces the nib, which remains on the work piece. *Caution: The higher the lead the more tool side deflection.*
- Use the narrowest possible cutoff insert. This will minimize the cutoff burr length.
- Reduce feed rate at the end of a cut.
- On most tubing type parts a 4° or 5° lead angle will be sufficient.
- Add support to a long slender type part.
- Maintain proper sub-spindle alignment.
- If nib or burr persist, call Tech. Support about reducing hone size.
- Use small or no corner radius.

To eliminate built-up edge:

- Select proper grade for insert.
- Increase speed (rpm).
- Use ample amounts of well-directed coolant. *(See Illustration A on H3.)*

Chamfer and cutoff operations:

- Use Separator G², S², SX and D².
- Groove or breakdown work piece surface being machined.
- Machine the chamfer.

For jobs requiring a chamfer on both ends of the part, begin by plunging to a depth just beyond the depth of the chamfers. Then, return to the part O.D. and profile each chamfer individually. Finish the cut off after completion of the second chamfer.

- Cutoff the work piece. *(See Illustration C.)*

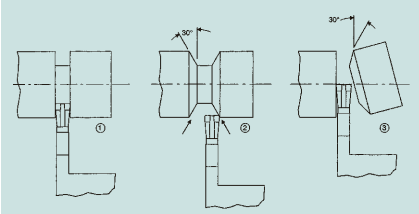
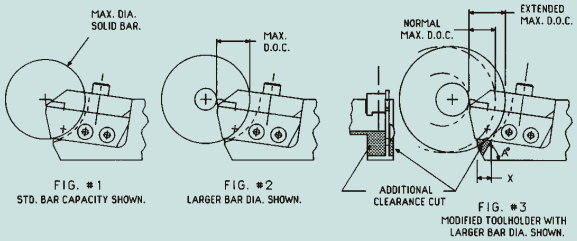


Illustration C

Modifications For Increased Depths Of Cut



CAPACITY CHART FOR 2-1/4" DIA. BAR CAPACITY TOOLING.

BAR DIA.	2.5	3.0	3.5	4.0	4.5	5.0	6.0	NOTE.
MAX. D.O.C.	0.94	0.75	0.62	0.56	0.50	0.47	0.44	WITH NO MODIFICATION ON TOOLHOLDER.
	1.12	1.03	0.97	0.91	0.87	0.84	0.78	WITH MODIFICATION X = .40 ON TOOLHOLDER. A = 60°

CAPACITY CHART FOR 3.0" DIA. BAR CAPACITY TOOLING.

BAR DIA.	3.5	4.0	4.5	5.0	6.0	NOTE.
MAX. D.O.C.	1.12	1.00	0.88	0.78	0.69	WITH NO MODIFICATION ON TOOLHOLDER.
	1.44	1.37	1.31	1.25	1.12	WITH MODIFICATION X = .38 ON TOOLHOLDER. A = 55°

Separator P/L

The Separator P/L insert should be placed in the blade finger tight, then properly seated by lightly tapping with either a rubber or plastic hammer. If none is available, a wood block should be placed over the cutting edge prior to seating with the hammer. *(See Illustration D.)*

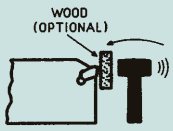


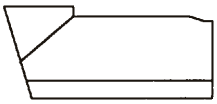
Illustration D

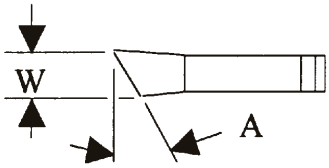
CUTOFF FLAT TOP SEPARATOR® INSERTS

Flat Top Separator® inserts have no chip breaker, no center channel and no corner radii. The flat top Separator® insert mounts into all standrd Manchester Separator® cutoff toolholders.

Advantages a Flat Top Separator® gives you:

- Less cutting tool pressure than a Separator® Classic. This helps when trying to obtain a nib free part.
- 507-189 and 507-122 can be used to produce OD grooves with a flat bottom.
- Flat top inserts provide better finishes on brass, bronze, and most non-ferrous materials than an insert with a chip breaker.





Standard Flat Top Separator® Inserts							INCH
				Grades			
Part Number	Width W	Lead Angle A	Hand	C2	C5	M20	M40
507-189	1/16	0°	N				•
507-141	3/32	4°	R	•	•		
507-179*	3/32	18°	R				•
507-122	1/8	0°	N	•			
507-180*	1/8	18°	R			•	

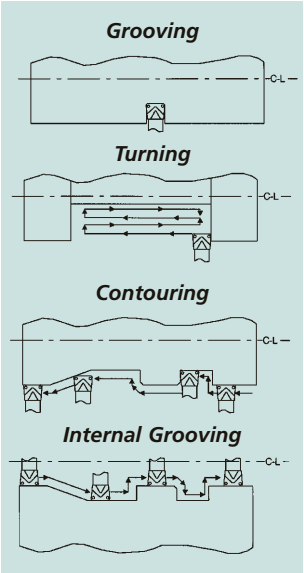
Standard Flat Top Separator® Inserts				METRIC
				Grades
Part Number	Width W mm	Lead Angle A	Hand	M20
507-338	2.4	0°	N	•
507-339	2.4	4°	R	•
507-340	2.4	4°	L	•
507-341	2.4	12°	R	•
507-342	2.4	12°	L	•
507-343	3	0°	N	•
507-344	3	4°	R	•
507-345	3	4°	L	•
507-346	3	12°	R	•
507-347	3	12°	L	•

**Has no back taper clearance on lead side for improved flatness and finish.*

•Indicates grade availability.

Application Information:

- Insure that the machine and the workpiece setups are as rigid as possible.
- The first choice for rigidity in toolholder selection is the use of integral shank holders.
- Use the holder with the shortest possible depth of cut for the application. This is extremely important in applications that involve significant amounts of turning.
- Toolholder overhang out of the block should be as short as possible.
- Inserts should cut as close to center as possible. They should never be more than .010" above center. If measurements are in doubt, a facing cut will corroborate center line position.
- For turning and profiling applications, use 1/2 the insert width as a starting point recommendation for depth of cut. Adjust based on material machinability and feed rate. Select a smaller corner radius to improve chip control.
- The MTC-PT™ offers the widest range of chip control in the Plunge and Turn operation.
- Chipmaker inserts are effective in the plunge mode up to .008 IPR and operate efficiently in the turning mode up to .012 IPR.
- Chipmaker 95 inserts are most effective in the plunge mode at heavier feed rates up to .015 IPR. CM-95 is capable of delivering chip control with corner radii of .060" and larger.



- Use cutoff guidelines, where applicable, for finish, chatter, and edge chipping related problems.
- When changing inserts, make sure the new insert locates securely against the positive stop.
- Plunge grooving with the MTC-PT™ system yields excellent chip control in most materials at productive speeds and feeds.
- The Manchester grooving system is sufficiently rigid to turn in both directions at the same feed rates. This allows continuous cutting when roughing a wide groove as shown.
- Manchester grooving system rigidity allows profiling cuts to be made at productive speeds and feeds along with excellent surface finishes. The MTC-PT™ and Chipmaker® permit chip control in all modes of operation.

Application Information:

- When changing inserts, be sure the new insert locates against the positive stop located on the clamp.
- Never tighten the insert clamping screw without an insert in the pocket. Permanent damage to the clamp could occur.
- Toolholder projection length out of the tool block should be as short as possible to maintain rigidity.
- Slower speeds and feeds are recommended compared to O.D. grooving

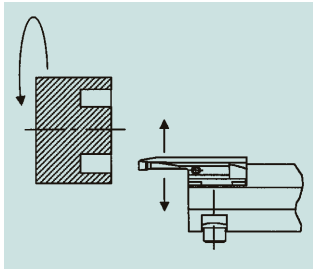
Face Grooving Ranges Per Setting

Given Diameter Setting	Plunge Range At Diameter Setting	
	Smallest O.D.	Largest O.D.
2-1/4	2-1/4	2-3/8
2-1/2	2-3/8	2-5/8
2-3/4	2-9/16	2-15/16
3.0	2-5/8	3-3/8
3-1/2	3-1/16	3-15/16
4.0	3-1/2	4-1/2
5.0	4-1/4	5-3/4
6.0	5.0	7.0
8.0	6-1/2	9-1/2
10.0	8.0	11.0
11.0-16.0	9.0	16.0

Note: This chart is a general guide for face groove entry at outside diameters both smaller and larger than each given OD setting on the tool.
Example: If the tool is adjusted for 4.0 inch OD, plunge cuts from 3-1/2 inch OD to 4-1/2 inch OD can be made without changing the 4.0 inch OD setting.

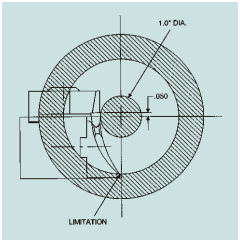
Widening A Face Groove

Additional clearance is generated on the workpiece after the first groove cut. Without further adjustment, the tool may then be used to widen the groove toward the center, or toward the O.D. of the workpiece.



Small Diameter Face Grooving Clearances

The cutting edge of the small diameter face grooving system is +.050" above center. This is done to improve cutting clearances. This tool should not be repositioned on center. When facing toward center, this system does not have sufficient clearance to cut at diameters of less than 1".

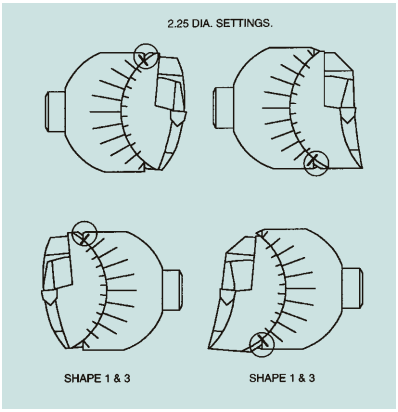


Adjusting Information for Ranger™ Tooling

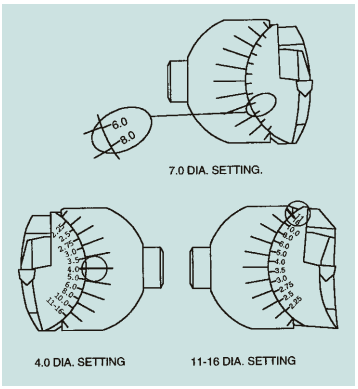
The following instructions are for style 1 Ranger tools. Instructions for style 2 tools are in [brackets].

- Appropriate diameter range setting can be accomplished as follows:

Step 1 Loosen the support blade locking screw and rotate the support blade so that the 2.25 mark is above the top line on the toolholder. [Below the line on toolholder for style 2.]



Step 2:
Slowly rotate the support blade down until the 2.25 mark is aligned with the top line of the toolholder. [Rotate the support blade up until the 2.25 mark is aligned with the bottom line on the toolholder for style 2.] At this point, the support blade assembly is properly aligned to cut face grooves at 2.250 inch OD.



For diameters larger than 2.25 OD, continue to rotate the support blade in the same direction until the desired diameter range has been aligned.
Example: The 7.0 inch diameter setting falls between the 6.0 inch and the 8.0 inch diameter settings.

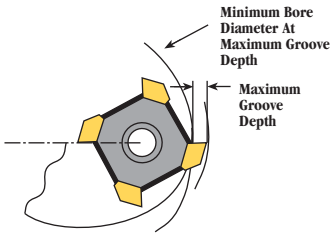
Step 3: Tighten the support blade screw. Inspect the scale to ensure that the desired diameter range is aligned.

Note: It is important that these instructions are followed. Failure to do so may result in damage to the tool and the workpiece.

Limits With Standard Inserts

INTERNAL GROOVING

Internal Groove Depth vs. Insert Interference



NOTE: Internal grooving depth limits are a function of insert clearance versus bore diameters.

Left Hand	Max. Groove Depth	Min. Bore Diameter	Right Hand
571-122	0.050	0.950	**571-131**
571-194	0.125	1.050	**571-138**
571-128	0.125	1.050	**571-141**
571-130	0.125	1.050	**571-143**
571-147	0.125	1.450	**571-170**
571-153	0.180	1.750	**571-177**
571-154	0.180	1.750	**571-183**
571-163	0.180	1.750	**571-185**
571-164	0.190	1.750	**571-188**

Adjustable Helix Threading System

1. SELECT PRODUCTION METHOD

The choice of production method is dependent upon the part's shape, the part's stability and the flow of chips.

Normal Threading

The tool is fed in the direction toward the chuck. This method is the most common and provides for the greatest stability.

Reverse Threading

The tool is fed away from the chuck. This method is less common and requires sufficient clamping of workpiece and rigidity of set up. An advantage of this method is that it provides superior chip evacuation in internal threading.

This method requires the use of a N3.0 or N1.5 negative helix.

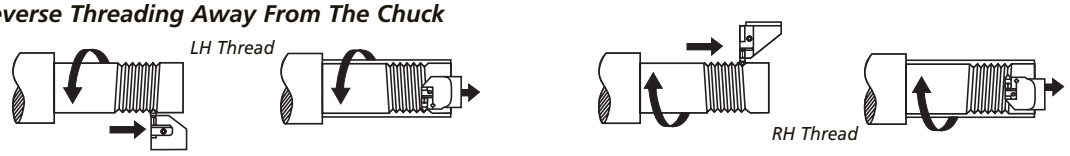
2. SELECT APPROPRIATE CARTRIDGE

To determine the correct threading helix angle, refer to the chart below. The grid triangle on the left side of the chart provides appropriate helix selection for reverse threading. The grid triangle on the right side of the chart provides appropriate helix selection for normal threading.

Use the chart to cross-reference the thread diameter and the Pitch/TPI being machined. Apply the appropriate helix angle to the Adjustable Helix Threading System shown on page E4.

Adjustable Helix Threading System — Helix Angle Chart

Reverse Threading Away From The Chuck



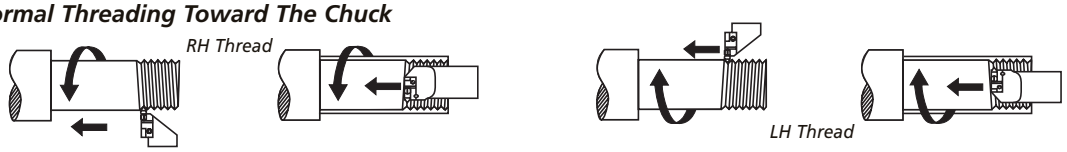
RH Toolholder, RH Neg. Helix — *LH Bar, LH Neg. Helix* — *LH Toolholder, LH Neg. Helix.* — *RH Bar, RH Neg. Helix*

MM	TPI	1/4	1/2	3/4	1	1-1/4	1-1/2	1-3/4	2	2-1/4	2-1/2	2-3/4	3	3-1/4	3-1/2	3-3/4	4	TPI	MM
5.0	5	—	—	—	—	—	N3.0	N3.0	N1.5	N1.5	N1.5	N1.5	N1.5	N1.5	N1.5	N1.5	N1.5		
4.0	6	—	—	—	N3.0	N3.0	N3.0	N1.5	N1.5	N1.5	N1.5	N1.5	N1.5	N1.5	N1.5	N1.5	N1.5		
3.5	7	—	—	—		N3.0	N1.5	N1.5	N1.5	N1.5	N1.5	N1.5	N1.5	N1.5	N1.5	N1.5			
3.0	8	—	—	N3.0	N3.0	N1.5	N1.5	N1.5	N1.5	N1.5	N1.5	N1.5	N1.5	N1.5	N1.5				
	9	—	—	N3.0	N1.5	N1.5	N1.5	N1.5	N1.5	N1.5	N1.5	N1.5	N1.5	N1.5					
2.5	10	—	—	N3.0	N1.5	N1.5	N1.5	N1.5	N1.5	N1.5	N1.5	N1.5	N1.5				P1.5	56	
2.0	11	—	N3.0	N3.0	N1.5	N1.5	N1.5	N1.5	N1.5	N1.5	N1.5	N1.5	N1.5			P1.5	P1.5	48	.5
	12	—	N3.0	N3.0	N1.5	N1.5	N1.5	N1.5	N1.5	N1.5	N1.5	N1.5			P1.5	P1.5	P1.5	44	
1.75	13	—	N3.0	N1.5	N1.5	N1.5	N1.5	N1.5	N1.5	N1.5	N1.5			P1.5	P1.5	P1.5	P1.5	40	
1.5	14	—	N3.0	N1.5	N1.5	N1.5	N1.5	N1.5	N1.5	N1.5	N1.5			P1.5	P1.5	P1.5	P1.5	36	
	16	—	N3.0	N1.5	N1.5	N1.5	N1.5	N1.5	N1.5	N1.5			P1.5	P1.5	P1.5	P1.5	P1.5	32	.75
1.25	18	—	N1.5	N1.5	N1.5	N1.5	N1.5	N1.5	N1.5			P1.5	P1.5	P1.5	P1.5	P1.5	P3.0	28	
1.0	20	—	N1.5	N1.5	N1.5	N1.5	N1.5	N1.5				P1.5	P1.5	P1.5	P1.5	P1.5	P3.0	24	1.0
	24	N3.0	N1.5	N1.5	N1.5	N1.5	N1.5	N1.5			P1.5	P1.5	P1.5	P1.5	P1.5	P1.5	—	20	1.25
.75	28	N3.0	N1.5	N1.5	N1.5	N1.5	N1.5			P1.5	P1.5	P1.5	P1.5	P1.5	P1.5	P1.5	—	18	
	32	N1.5	N1.5	N1.5	N1.5	N1.5			P1.5	P1.5	P1.5	P1.5	P1.5	P1.5	P1.5	P3.0	—	16	1.5
	36	N1.5	N1.5	N1.5	N1.5				P1.5	P1.5	P1.5	P1.5	P1.5	P1.5	P1.5	P3.0	—	14	1.75
	40	N1.5	N1.5	N1.5	N1.5			P1.5	P1.5	P1.5	P1.5	P1.5	P1.5	P1.5	P1.5	P3.0	—	13	
.5	44	N1.5	N1.5	N1.5			P1.5	P1.5	P1.5	P1.5	P1.5	P1.5	P1.5	P1.5	P3.0	P3.0	—	12	2.0
	48	N1.5	N1.5			P1.5	P1.5	P1.5	P1.5	P1.5	P1.5	P1.5	P1.5	P1.5	P3.0	P3.0	—	11	
	56	N1.5				P1.5	P1.5	P1.5	P1.5	P1.5	P1.5	P1.5	P1.5	P1.5	P3.0	—	—	10	2.5
					P1.5	P1.5	P1.5	P1.5	P1.5	P1.5	P1.5	P1.5	P1.5	P1.5	P3.0	—	—	9	
				P1.5	P1.5	P1.5	P1.5	P1.5	P1.5	P1.5	P1.5	P1.5	P1.5	P3.0	P3.0	—	—	8	3.0
			P1.5	P1.5	P1.5	P1.5	P1.5	P1.5	P1.5	P1.5	P1.5	P1.5	P3.0	P3.0	—	—	—	7	3.5
		P1.5	P1.5	P1.5	P1.5	P1.5	P1.5	P1.5	P1.5	P1.5	P1.5	P3.0	P3.0	P3.0	—	—	—	6	4.0
		P1.5	P1.5	P1.5	P1.5	P1.5	P1.5	P1.5	P1.5	P1.5	P3.0	P3.0	—	—	—	—	—	5	5.0
MM	TPI	4	3-3/4	3-1/2	3-1/4	3	2-3/4	2-1/2	2-1/4	2	1-3/4	1-1/2	1-1/4	1	3/4	1/2	1/4	TP1	MM

▲ REVERSE THREADING (NEGATIVE HELIX)

▼ NORMAL THREADING (POSITIVE HELIX)

Normal Threading Toward The Chuck



RH Toolholder, RH Pos. Helix — *LH Bar, LH Pos. Helix* — *LH Toolholder, LH Pos. Helix* — *RH Bar, RH Pos. Helix*

Adjustable Helix Threading System

3. SELECT APPROPRIATE THREADING INSERT

Full Profile Inserts

The advantage of full profile threading inserts is that they provide a perfect thread with a complete profile without burrs.

Partial Profile Inserts

The advantage of partial profile inserts is that single insert may be utilized to cover a variety of thread pitches and can be utilized for external and internal applications.

4. SELECT INSERT GRADE/CUTTING SPEED

To determine recommended cutting speed (SFM) and grade, refer to chart below.

AISI Material Designation	*Recommended Cutting Speed Ft/Min*			
	M40	M45	M50	M93
Low Carbon Steels 1010, 1015, 1018, 1022, 1025, 1140 12L13, 12L14, 1035, 1040, 1042	100-350	125-400	350-600	450-750
High Carbon Steels 1095, 4130, 4140, 6150, 8620	100-300	125-350	250-400	350-500
High Alloy Steels O1, S1, W1, 4340, H13 A2, D2, M2, M7, M35	100-225	100-250	250-350	300-450
Free Machining Stainless Steels 303, 410, 410S, 416, 430	100-300	125-350	300-400	300-400
Difficult Machining Stainless Steels 304, 348, 420, 440C, 15-5PH, 17-4PH 310, 316, 316L, 317, 318, 321, 660	100-200	125-250	—	200-400
Cast Iron, Grey Iron, Alloy Cast Iron A48-20B, A47-32510, A48-25B, 60/40/18 A48-50B, , A48-60B	250-400	250-400	400-600	400-700
Nickel, Cobalt and Iron Based Superalloys under 38 Rc	100-200	100-250	125-250	150-300
Brass	200-500	250-500	400-700	400-700
Bronze	200-500	250-500	300-400	350-500
Copper	200-500	250-500	—	225-300
Aluminum	200-500	500-900	350-500	500-900
Titanium	75-175	100-200	200-300	150-250

Preferred selection shown in shaded box. If two selections are shown, check for stock status.

6. SELECT NUMBER OF PASSES/INFEED PER PASS

The following tables provide basic recommendations for threading in free machining steels with a hardness of below 320 HB (32Rc). The total number of passes may be adjusted slightly higher or lower for optimum performance as refined by trial.

If insert breakage should occur, the number of passes should be increased. If flank wear is excessive, the number of passes should be reduced.

UN Threads, External

Pitch TPI	5	6	7	8	9	10	11	12	13	14	16	18	20	24	28	32
Tot. depth in.	0.130	0.107	0.092	0.082	0.072	0.065	0.060	0.055	0.051	0.047	0.041	0.037	0.033	0.028	0.024	0.021
Pass 1 inch	0.017	0.014	0.014	0.012	0.011	0.011	0.011	0.011	0.010	0.009	0.009	0.009	0.008	0.007	0.007	0.007
2	0.016	0.013	0.013	0.011	0.010	0.010	0.010	0.010	0.009	0.009	0.008	0.008	0.007	0.007	0.006	0.006
3	0.014	0.011	0.010	0.010	0.008	0.008	0.008	0.008	0.007	0.007	0.006	0.006	0.006	0.006	0.004	0.005
4	0.012	0.009	0.009	0.008	0.007	0.007	0.007	0.007	0.007	0.006	0.006	0.006	0.005	0.005	0.004	0.003
5	0.010	0.009	0.008	0.007	0.007	0.006	0.006	0.006	0.006	0.005	0.005	0.005	0.004	0.003	0.003	
6	0.009	0.008	0.007	0.006	0.006	0.006	0.006	0.005	0.005	0.004	0.004	0.003	0.003			
7	0.008	0.007	0.007	0.006	0.006	0.005	0.005	0.005	0.004	0.004	0.003					
8	0.008	0.006	0.006	0.005	0.005	0.005	0.004	0.003	0.003	0.003						
9	0.007	0.006	0.006	0.005	0.005	0.004	0.003									
10	0.007	0.006	0.005	0.005	0.004	0.003										
11	0.007	0.005	0.004	0.004	0.003											
12	0.006	0.005	0.003	0.003												
13	0.005	0.004														
14	0.004	0.004														

UN Threads, Internal

Pitch TPI	5	6	7	8	9	10	11	12	13	14	16	18	20	24	28	32
Tot. depth in.	0.118	0.097	0.084	0.074	0.065	0.059	0.054	0.049	0.045	0.042	0.037	0.033	0.030	0.025	0.022	0.019
Pass 1 inch	0.017	0.014	0.013	0.012	0.011	0.011	0.011	0.011	0.010	0.009	0.009	0.009	0.008	0.007	0.007	0.007
2	0.015	0.013	0.013	0.011	0.010	0.010	0.009	0.009	0.008	0.007	0.007	0.007	0.006	0.006	0.005	0.005
3	0.013	0.010	0.009	0.009	0.007	0.007	0.007	0.007	0.006	0.006	0.006	0.006	0.005	0.005	0.004	0.004
4	0.011	0.008	0.008	0.007	0.006	0.006	0.006	0.006	0.005	0.005	0.005	0.005	0.004	0.004	0.003	0.003
5	0.009	0.007	0.007	0.006	0.006	0.005	0.005	0.005	0.005	0.004	0.004	0.003	0.004	0.003	0.003	
6	0.008	0.006	0.006	0.005	0.005	0.005	0.005	0.004	0.004	0.004	0.003	0.003	0.003			
7	0.007	0.006	0.006	0.005	0.005	0.004	0.004	0.004	0.004	0.004	0.003					
8	0.007	0.006	0.005	0.004	0.004	0.004	0.004	0.003	0.003	0.003						
9	0.006	0.005	0.005	0.004	0.004	0.004	0.003									
10	0.006	0.005	0.005	0.004	0.004	0.003										
11	0.005	0.005	0.004	0.004	0.003											
12	0.005	0.004	0.003	0.003												
13	0.005	0.004														
14	0.004	0.004														

THREADING TECHNICAL RECOMMENDATIONS

Selection Guidelines

ISO Metric Threads, External

Pitch TPI	5.0	4.5	4.0	3.5	3.0	2.5	2.0	1.75	1.5	1.25	1.0	.75	.50
Tot. depth in.	0.126	0.113	0.100	0.088	0.076	0.063	0.049	0.044	0.037	0.032	0.026	0.019	0.014
Pass 1 inch	0.016	0.015	0.013	0.013	0.011	0.011	0.009	0.008	0.008	0.008	0.008	0.006	0.005
2	0.015	0.013	0.013	0.012	0.010	0.009	0.009	0.008	0.008	0.007	0.006	0.006	0.004
3	0.013	0.011	0.010	0.010	0.008	0.008	0.007	0.007	0.007	0.006	0.005	0.004	0.003
4	0.011	0.009	0.009	0.008	0.007	0.007	0.006	0.006	0.006	0.004	0.004	0.003	0.002
5	0.010	0.009	0.008	0.008	0.007	0.006	0.006	0.005	0.005	0.004	0.003		
6	0.009	0.008	0.007	0.007	0.006	0.006	0.005	0.004	0.003	0.003			
7	0.009	0.008	0.007	0.006	0.006	0.005	0.004	0.003					
8	0.008	0.007	0.006	0.006	0.005	0.004	0.003	0.003					
9	0.007	0.007	0.006	0.006	0.005	0.004							
10	0.007	0.006	0.005	0.005	0.004	0.003							
11	0.006	0.006	0.005	0.004	0.004								
12	0.006	0.005	0.004	0.003	0.003								
13	0.005	0.005	0.004										
14	0.004	0.004	0.003										

ISO Metric Threads, Internal

Pitch TPI	5.0	4.5	4.0	3.5	3.0	2.5	2.0	1.75	1.5	1.25	1.0	.75	.50
Tot. depth in.	0.117	0.104	0.092	0.081	0.070	0.058	0.045	0.041	0.033	0.030	0.024	0.018	0.012
Pass 1 inch	0.017	0.015	0.013	0.013	0.011	0.010	0.009	0.009	0.008	0.007	0.007	0.006	0.004
2	0.016	0.013	0.012	0.012	0.010	0.010	0.008	0.007	0.007	0.007	0.006	0.005	0.003
3	0.013	0.011	0.009	0.009	0.008	0.007	0.007	0.006	0.006	0.006	0.004	0.004	0.003
4	0.010	0.009	0.008	0.007	0.006	0.006	0.006	0.005	0.005	0.004	0.004	0.003	0.002
5	0.009	0.008	0.007	0.007	0.006	0.005	0.005	0.004	0.004	0.003	0.003		
6	0.008	0.007	0.007	0.006	0.005	0.005	0.004	0.004	0.003	0.003			
7	0.007	0.006	0.006	0.006	0.005	0.004	0.003	0.003					
8	0.006	0.006	0.005	0.005	0.004	0.004	0.003	0.003					
9	0.006	0.006	0.005	0.005	0.004	0.004							
10	0.006	0.005	0.005	0.004	0.004	0.003							
11	0.005	0.005	0.004	0.004	0.004								
12	0.005	0.005	0.004	0.003	0.003								
13	0.005	0.004	0.004										
14	0.004	0.004	0.003										

Selection Guidelines

7. SELECT INFEED METHOD

In practice, always consider the workpiece material, the machine tool and the thread pitch in determining the preferred method of infeed.

For short chipping materials and when threading fine pitches, the infeed method selected is less critical.

When threading long chipping materials and coarse pitches, the infeed method selected becomes more critical.

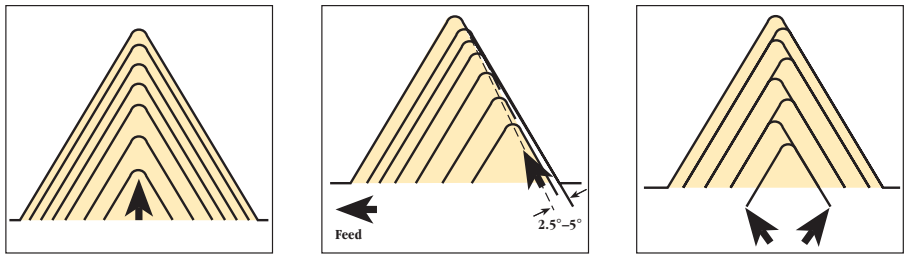
Following are shown three basic selections of infeed method.

Radial Infeed is the only method available on manual machines. It is the preferred method in work hardening materials.

Modified Flank Infeed is the preferred method on all CNC Turning Centers. If this method cannot be applied use Standard Flank Infeed.

Incremental Infeed is less common and requires a special CNC program. This method is most suitable for long part run applications.

Radial Infeed *Modified Flank Infeed* *Incremental Infeed*



HELIX SELECTION FOR COMMON EXTERNAL THREADS

The following charts may be used for Quick Reference in selecting the Correct External Threading Helix for Common External Threads when threading toward the chuck, using the "normal threading" method.

Helix Angle Chart For Standard External UN Threads

PART DIAMETER (inches)	PITCH (TPI)													
	32	28	24	20	18	16	14	13	12	11	10	9	8	7
1/4	P3.0	P3.0												
5/16	P1.5		P3.0											
3/8	P1.5		P3.0			P3.0								
7/16		P1.5		P3.0			P3.0							
1/2		P1.5		P1.5				P3.0						
9/16			P1.5		P1.5				P3.0					
5/8			P1.5		P1.5					P3.0				
3/4				P1.5		P1.5					P3.0			
7/8				P1.5			P1.5					P3.0		
1				P1.5					P1.5				P3.0	
1-1/8					P1.5				P1.5					P3.0
1-1/4					P1.5				P1.5					P3.0

Helix Angle Chart For Standard External ISO Metric Threads

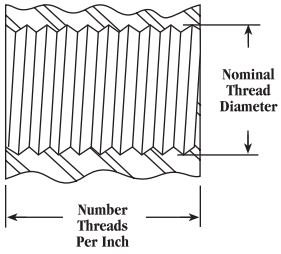
PART DIAMETER (mm)	PITCH (mm)							
	1.0	1.25	1.5	1.75	2.0	2.5	3.0	3.5
6	P3.0							
7	P3.0							
8	P3.0	P3.0						
10		P3.0	P3.0					
12		P1.5		P3.0				
14			P1.5		P3.0			
16			P1.5		P3.0			
18			P1.5			P3.0		
20			P1.5			P3.0		
22			P1.5			P3.0		
24					P1.5		P3.0	
27					P1.5		P3.0	

THREADING TECHNICAL RECOMMENDATIONS

Selection Guidelines

Limits With Standard Inserts

INTERNAL THREADING
Pitch Range vs. Internal Diameter



Unified "J" Series Thread

The controlled root radius thread form (MIL-S-8879A) is defined for the external thread only.

To machine the corresponding internal thread, choose any internal full profile UN threading insert (per desired pitch).

Refer to Tables II-VII in MIL-S-8879A for the correct "J" thread minor diameter values on internal thread.

**** Requires insert and internal bar modification. A more coarse pitch per nominal thread size may be attained dependent on thread form and type of material.***

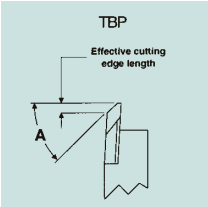
Nominal Thread Size	Insert Style 3/Range of TPI
1"	32-24
1-1/16"	32-18
1-1/8"	32-14
1-1/4"	32-12
1-5/16"	32-10
1-3/8"	32-10
1-7/16"	32-10
1-1/2"	32-8
1-9/16"	32-8
1-5/8"	32-8
1-3/4"	32-8
1-7/8"	32-8
2"	32-8
2-1/4"	32-8
2-1/2"	32-8
2-3/4"	32-8
3"	32-8

SWISS TECHNICAL RECOMMENDATIONS

Swiss Style Tool Systems

The effective cutting edge length for backturning inserts is shown below.

Effective Cutting Edge Length
TBP55 = 2.8mm
TBP60 = 3.5mm
TB32 = 2.5mm
TB43 = 3.5mm

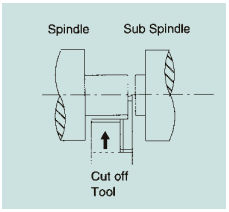


Recommended values for maximum depth of cut (mm) per pass for each insert design are listed below.

Work Material	*TBP55*	*TBP60*	*TB32*	*TB43*	*Grades*
Steels	1.96	2.45	1.75	2.45	M40
	N/A	N/A	1.25	1.75	Cermet
Stainless Steels	1.68	2.10	1.50	2.10	M40
	N/A	N/A	1.00	1.40	Cermet
Non-Ferrous	2.52	3.15	2.25	3.15	M40
	N/A	N/A	2.00	2.80	Cermet
Non-Metal	2.52	3.15	2.25	3.15	M40
	N/A	N/A	2.25	3.15	Cermet

Sub-Spindle Swiss CTP Cutoff

Part transfer to sub-spindle for machining and final cutoff, use CTP-K, CTPA-K cutoff inserts. These inserts provide opposite hand lead angle for the hand of tool.



APPLICATIONS

Material	Grade	Cutoff (Separator)		Grooving		Face Grooving	
		SFM	IPR	SFM	IPR	SFM	IPR
Carbon Steels	M40	100-350	.001-.004	100-350	.002-.004*	100-350	.001-.004
Low Carbon (less than .30% C)	M43	250-700	.001-.004	250-700	.002-.007		
AISI 1000, 1100 and 1200 Series	M45	125-400	.001-.004	125-400	.002-.004		
	M50	250-700	.001-.004	250-700	.002-.004	250-400	.001-.004
	M53			250-700	.004-.010*		
	M74			700-900	.001-.004		
	M93	300-800	.001-.004	300-900	.002-.007		
	C5	250-500	.001-.004	250-500	.002-.004	250-400	.001-.004
	GC	350-700	.001-.004	350-700	.002-.004	350-400	.001-.004
Carbon Steels	M40	100-350	.002-.005	100-350	.002-.005*	100-350	.002-.005
High Carbon (Greater than .30%C)	M43	250-700	.003-.007	250-700	.002-.005		
AISI 1000, 1100 and 1200 Series	M45	125-400	.002-.005	125-400	.002-.005		
	M50	250-700	.003-.007	250-500	.003-.006	250-400	.003-.006
	M53			250-600	.004-.012*		
	M74			700-900	.001-.004		
	M93	300-800	.003-.007	300-800	.003-.008		
	C5	250-450	.003-.006	250-450	.003-.006	250-400	.003-.006
	GC	350-700	.003-.005	250-550	.003-.006	350-400	.003-.005
Alloy Steels	M40	100-350	.001-.005	100-350	.002-.005*	100-350	.001-.005
Low Carbon (Less than .30%C)	M43	250-700	.002-.007	250-700	.003-.007		
AISI 1300, 4000, 5000, 6000,	M45	125-400	.001-.005	125-400	.002-.005		
8000 and 9000 Series	M50	250-600	.002-.007	250-600	.002-.007	250-400	.002-.006
	M53			250-600	.004-.012*		
	M74			700-900	.001-.004		
	M93	300-800	.002-.007	300-900	.003-.007		
	C5	250-500	.002-.006	250-500	.002-.006	250-400	.002-.006
	GC	350-600	.002-.005	300-600	.002-.005	300-400	.002-.005
Alloy Steels	M40	100-300	.002-.005	100-300	.002-.005*	100-300	.002-.005
High Carbon (Greater than .30%C)	M43	250-600	.003-.006	250-600	.003-.008		
AISI 1300, 4000, 5000, 6000,	M45	125-400	.002-.005	125-400	.002-.005		
8000 and 9000 Series	M50	250-600	.003-.006	250-500	.003-.005	250-400	.003-.006
	M53			250-600	.004-.012*		
	M74			700-900	.001-.004		
	M93	300-800	.003-.006	300-800	.003-.008		
	C5	250-450	.003-.005	250-450	.003-.006	250-400	.003-.005
	GC	350-600	.003-.005	250-550	.003-.005	350-400	.003-.005
Tool Steels	M40	100-200	.002-.005	100-200	.002-.005*	100-200	.002-.005
Example A-2, D-2, M-2, H-13	M43	150-300	.003-.006	150-350	.003-.007		
	M45	125-250	.002-.005	125-250	.002-.005		
	M50	150-300	.003-.006	150-300	.003-.005	150-300	.003-.006
	M53			250-350	.004-.008*		
	M74			400-600	.002-.004		
	M93	200-370	.003-.006	250-450	.003-.007		
	C5	180-250	.003-.006	180-250	.003-.005	180-250	.003-.006
	GC	220-300	.003-.005	220-300	.003-.005	220-300	.003-.005

APPLICATIONS

Material	Grade	Cutoff (Separator)		Grooving		Face Grooving	
		SFM	IPR	SFM	IPR	SFM	IPR
Martensitic Stainless Steels	M40	100-300	.001-.004	100-300	.002-.004*	100-300	.001-.004
400 Series	M43	250-450	.002-.007	250-450	.003-.007		
	M45	125-350	.002-.004	125-350	.002-.004		
	M50	250-500	.002-.007	250-500	.002-.007	250-400	.002-.006
	M53			350-500	.004-.012*		
	M74			400-600	.002-.004		
	M93	300-500	.002-.007				
	C5	250-400	.002-.006	250-400	.002-.006	250-400	.002-.006
	GC	350-500	.002-.006	350-500	.002-.005	350-400	.002-.006
Austenitic Stainless Steels	M40	100-200	.001-.005	100-200	.002-.005*	100-200	.001-.005
300 Series	M43	150-300	.002-.005	150-300	.003-.006		
	M45	125-250	.001-.005	125-250	.002-.005		
	M53			200-350	.004-.007		
	M74			300-500	.001-.003		
	M93			200-350	.002-.005		
	C2	200-250	.0025-.004	200-250	.0025-.004	200-250	.0025-.004
Low Machinability Alloys	M40	50-125	.002-.005	50-125	.002-.005	50-125	.002-.005
Iron, Nickel and Cobalt based	M43	70-200	.002-.005				
Example: Inconel™, Hastelloy™	M45	70-150	.002-.005	70-125	.002-.005		
	M93	90-240	.002-.004				
	C2	70-175	.003-.004	70-175	.003-.004	70-175	.003-.004
Non Ferrous Free Machining	M40	400+	.002-.010	400+	.002-.010	400+	.002-.010
Materials—Aluminum., Copper	M43	450+	.002-.010	450+	.002-.010		
And Zinc Based	M45	400+	.002-.010	400+	.002-.010		
	M93	500+	.002-.010				
	C2	500+	.002-.010	500+	.002-.010	500+	.002-.010
Titanium	M40	100-150	.002-.005	100-150	.002-.005	100-150	.002-.005
	M43	150-225	.002-.005				
	M45	125-175	.002-.005	125-175	.002-.005		
	M93	170-250	.002-.004				
	C2	150-250	.002-.004	150-250	.002-.004	150-250	.002-.004
Cast Irons	M40	400-500	.002-.010	400-600	.002-.010	400-500	.002-.010
Grey, Soft (20-30 Rc)	M43	400-600	.002-.010	500-800	.002-.010		
	M45	400-500	.002-.010	450-700	.002-.010		
	M93	800-1000	.002-.008				
	C2	500-600	.002-.008	500-800	.002-.008	500-600	.002-.008
	AlOx			800-1200	.002-.010	800-1200	.002-.008

**Feed rates higher for Chipmaker® 95*

Inconel is a trademark of Huntington Alloys.
Hastelloy is a trademark of Cabot Corporation.

SWISS APPLICATIONS

Material	Grade	SFM	IPR (Cutoff)	IPR (Turning)
Carbon, Alloy, and	M40	150-300	.001-.003	.002-.010
Free Cutting Steels	M74	150-600	.001-.002	.002-.005
Austenitic Stainless Steel	M40	150-250	.001-.0025	.002-.008
	M74	150-450	.001-.0025	.002-.005
Non Ferrous Metals	M40	150-700	.002-.006	.005-.010

CBN GROOVING APPLICATIONS

Material	Grade	SFM	IPR
Gray Cast Iron (180-270 Bhn)	CBN-CI	1500-3000	.002-.010
Hard Cast Iron (>400 Bhn)	CBN-CI	250-500	.002-.010
Hardened Steel (>450 Bhn)	CBN-HT	300-450	.002-.006
Superalloys (>350 Bhn)	CBN-CI	550-800	.002-.010
Sintered Iron	CBN-CI	300-600	.002-.010

CERAMIC GROOVING APPLICATIONS

Material	Grade	SFM	IPR
Gray Cast Iron (180-270 Bhn)	M70	500-2000	.002-.008
Hard Cast Iron (>400 Bhn)	M70	100-800	.002-.008
Hardened Steel	M70	100-400	.002-.005
Superalloys	M70	400-800	.002-.007

PRODUCT/PAGE INDEX

PRODUCT/PAGE INDEX

PRODUCT/PAGE INDEX

PRODUCT/PAGE INDEX

PRODUCT/PAGE INDEX

Part Number	Page Number	Part Number	Page Number	Part Number	Page Number	Part Number	Page Number	Part Number	Page Number
507-119	B2	507-232	B4	507-298	B5	507-374	B7	508-308	C3
507-120	B2	507-233	B4	507-301	B5	507-378	B5	508-308	C5
507-122	H5	507-234	B4	507-305	B7	507-379	B5	508-309	C3, C17
507-124	B2	507-235	B4	507-306	B7	507-380	B5	508-310	C3, C17, D19
507-125	B2	507-236	B4	507-307	B7	507-382	B8	508-311	C3, D19
507-126	B2	507-237	B4	507-308	B7	507-383	B8	508-312	C3
507-127	B2	507-238	B4	507-309	B7	507-384	B8	508-313	C3, C17
507-128	B2	507-239	B4	507-310	B7	507-386	C5, C15	508-314	C3
507-129	B2	507-240	B3	507-314	H3	507-387	C5, C15	508-315	C3, C17
507-131	B2	507-241	B3	507-316	B2	507-388	C5, C15	508-316	C3, C8, D19
507-132	B2	507-242	B3	507-317	B2	507-389	C5, C15	508-317	C3, D19
507-133	B2	507-243	B3	507-318	B2	507-393	B9	508-318	C3
507-134	B2	507-244	B3	507-319	B2	507-394	B9	508-319	C4, C17
507-140	B2	507-245	B3	507-320	B2	507-395	B9	508-320	C4
507-141	H5	507-246	B3	507-321	B2	507-396	B9	508-321	C4, C17
507-143	B2	507-247	B3	507-329	B6	507-397	B9	508-322	C4, C17, D19
507-144	B2	507-248	B3	507-330	B6	507-398	B9	508-323	C4, D19
507-146	B2	507-249	B3	507-331	B6	507-399	B9	508-324	C4
507-148	B2	507-250	B3	507-332	B6	507-400	B9	508-325	C4, C17
507-149	B2	507-251	B3	507-333	B6	508-208	C2, D18	508-326	C4
507-150	B2	507-252	B3	507-334	B6	508-238	C2, C16	508-327	C4, C17
507-151	B2	507-253	B3	507-338	H5	508-239	C2, C16	508-328	C4, C17, D19
507-152	B2	507-254	B3	507-339	H5	508-240	C2, C16, D18	508-329	C4, D19
507-154	B2	507-255	B3	507-340	H5	508-241	C2	508-330	C4
507-155	B2	507-256	B3	507-341	H5	508-242	C2, D18	508-331	C4, C17
507-161	B2	507-257	B3	507-342	H5	508-243	C2	508-332	C4
507-172	C16	507-258	B3	507-343	H5	508-244	C2, C16	508-333	C4, C17
507-176	B2	507-259	B3	507-344	H5	508-245	C2	508-334	C4, C17, D19
507-179	H5	507-260	B3	507-345	H5	508-246	C2, C16	508-335	C4, D19
507-180	H5	507-261	B3	507-346	H5	508-247	C2, C16, D18	508-336	C4
507-182	B28, C5, C15	507-262	B3	507-347	H5	508-248	C2, D18	508-337	C17
507-183	B28, C5, C15	507-263	B3	507-348	B6	508-249	C2	508-338	C17
507-185	B28, C5, C15	507-267	B4	507-349	B6	508-250	C16	508-339	C17
507-186	B28, C5, C15	507-268	B4	507-350	B6	508-251	C16	508-340	C3, D19
507-189	H5	507-269	B4	507-351	B6	508-252	C16	508-341	C3, D19
507-196	B2	507-275	B5	507-352	B6	508-256	C2, C16	508-403	C5
507-197	B2	507-276	B5	507-353	B6	508-257	C2, C16	508-404	C5
507-200	C16	507-277	B5	507-354	B8	508-258	C2	508-405	C5
507-201	B2	507-278	B5	507-355	B8	508-259	C2	508-412	C5
507-202	B2	507-279	B5	507-356	B8	508-260	C2, C16	508-413	C5
507-203	B2	507-280	B5	507-357	B8	508-261	C2, C16	508-422	C5, D18
507-204	B2	507-281	B5	507-358	B8	508-262	C2, C16, D18	508-423	C5, D18
507-205	B2	507-282	B5	507-359	B8	508-263	C2, C16, D18	508-424	C5, D18
507-206	B2	507-283	B5	507-363	B7	508-264	C16	508-435	C5, D18
507-207	H3	507-288	B3	507-364	B7	508-265	C16	508-436	C5, D18
507-213	B2	507-289	B3	507-365	B7	508-276	C16	508-446	C3
507-214	B2	507-290	B3	507-366	B8	508-279	C2, D18	508-447	C3
507-224	H3	507-291	B3	507-367	B8	508-301	C3, C17	508-448	C3
507-225	H3	507-292	B3	507-368	B8	508-302	C3	508-449	C3
507-226	H3	507-293	B3	507-369	B8	508-303	C3, C17	508-450	C3
507-227	B4	507-294	B3	507-370	B8	508-304	C3, C17	508-451	C3
507-228	B4	507-295	B5	507-371	B8	508-305	C3, D19	508-452	C3
507-229	B4	507-296	B5	507-372	B7	508-306	C3	508-453	C3
507-231	B4	507-297	B5	507-373	B7	508-307	C3, C17	508-454	C3

PRODUCT/PAGE INDEX

PRODUCT/PAGE INDEX

PRODUCT/PAGE INDEX

PRODUCT/PAGE INDEX

Subsidiary Federal Signal Corporation

Corporate Headquarters
5142 Manchester Road, Akron, OH 44319 USA
Phone (330) 644-8853 • Fax (330) 644-6139
Customer Service (800) 237-8789 • Technical Support (800) 282-1331
sales@manchestertools.com • www.manchestertools.com

ISA
Linking Channel Partners

MEMBER

Form No. 444