

Application and Technical Information



Thread Milling System (TMS)

Minimum Bore Diameters for Thread Milling

UN-ISO-BSW

cutter	tpi	48	32	24	20	16	12	10	8	7	6						4.5			4						
	pitch mm	0,5	0,75	1,0	1,25	1,5	2,0	2,5	3,0	3,5	4,0	4,5	5,0	5,5			6,0									
	cutter dia. (D1)	minimum bore diameter (D) (inches)																								
K035TM1RW050-STN10	.35	.374	.394	.421	.449																					
K045TM1RW050-STN11N	.45	.472	.492	.520	.547	.571																				
K049TM1RW037LT11S	.49	.512	.531	.559	.587	.610																				
K061TM1RW062-STN16T	.61	.630	.650	.667	.705	.728	.768																			
K067TM2RW075-STN11D	.67	.693	.717	.748	.772	.787	.827																			
K075TM1RW075-STN16T	.75	.776	.803	.827	.850	.866	.906																			
K079TM1RW075-STN16N	.79	.815	.843	.866	.890	.906	.945																			
K087TM1RW100-STN16L	.87	.893	.921	.945	.969	.984	1.024																			
K102TM2RW100-STN16D	1.02	1.051	1.079	1.102	1.130	1.154	1.193																			
K118TM1RW100-STN27N	1.18	1.209	1.236	1.260	1.291	1.319	1.362	1.441	1.535	1.654	1.772	1.890														
K146TM1RW125-STN27N	1.46	1.496	1.520	1.555	1.591	1.614	1.654	1.732	1.830	1.929	2.047	2.185														
K165TM2RW125-STN27D	1.65	1.701	1.724	1.772	1.811	1.831	1.866	1.929	2.047	2.146	2.268	2.401														
	1.38 (UN)	—	—	—	—	—	—	—	—	—	1.969	—	1.843	—	1.756	—	2.228									
	1.38 (ISO)	—	—	—	—	—	—	—	—	—	1.969	2.102	1.673	1.969	—	2.264	—									
	1.38 (BSW)	—	—	—	—	—	—	—	—	—	1.961	—	1.831	—	1.866	—	—									

Applications in this area may be machined with special inserts and cutter bodies. Quoted on request.

Cutting Data Recommendation

workpiece material	cutting speed (sfm)			feed rate per revolution
	grade KC610M	grade KC620M	grade KC635M	
carbon steels < 187 HB	250 - 600	—	300 - 700	.004 - .008
carbon steels 187 to 220 HB	250 - 400	—	300 - 500	.004 - .006
alloy steel 200 - 250 HB	150 - 350	—	200 - 425	.004 - .006
alloy steel 250 - 325 HB	100 - 250	—	150 - 300	.004 - .006
stainless steel, austenitic < 210 HB	250 - 375	—	300 - 450	.004 - .006
stainless steel, martensitic < 321 HB	200 - 400	—	250 - 350	.002 - .006
stainless steel, ferritic < 245 HB	250 - 450	—	350 - 550	.002 - .004
cast steel < 140 HB	300 - 450	—	350 - 550	.002 - .006
cast steel 220 - 302 HB	200 - 300	—	225 - 425	.002 - .004
titanium alloys	100 - 200	100 - 200	200 - 400	.001 - .003
high-temperature (nickel and iron base)	50 - 100	50 - 100	75 - 150	.001 - .002
high-temperature (cobalt base)	35 - 75	35 - 75	50 - 100	.001 - .002
cast iron	—	150 - 300	250 - 350	.002 - .006
malleable iron	—	150 - 350	250 - 400	.001 - .003
bronze/copper	—	390 - 590	—	.004 - .008
aluminum	—	660 - 1400	—	.004 - .012

NOTES:

- All thread milling inserts are full profile or cresting type.
- Inserts are designed to mill full thread depth in one revolution or pass.
- When machining difficult materials, two passes may be desired. A 60% thread depth on the first pass, and a 40% thread depth on the second pass is recommended.
- Thread relief grooves in blind holes are not necessary.

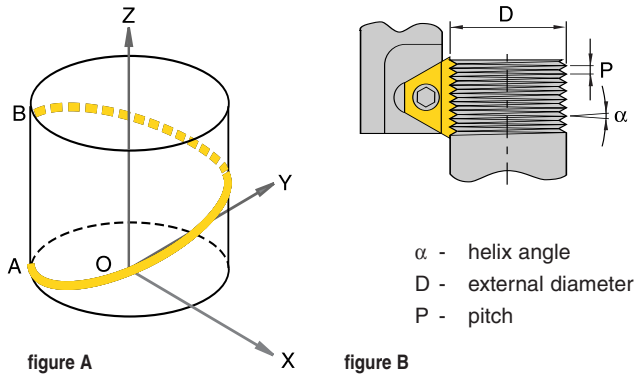
- Thread milling large parts requires considerably less horsepower compared to other threading methods.
- Thread milling produces short chips compared to stringy chips of other threading methods.
- One holder is suitable for many different thread pitches.
- PVD coated inserts provide maximum tool life for a wide variety of materials.



Thread Milling System (TMS)

Understanding Thread Milling

In order to perform a thread milling operation, a milling machine with three-axis control, capable of helical interpolation, is required. Helical interpolation is a CNC function producing tool movement along a helical path. This helical travel combines circular movement in one plane with a simultaneous linear motion in a plane perpendicular to the first. For example, the path from point A to point B (figure A) on the envelope of the cylinder combines a circular movement in the X and Y plane with a linear movement in the Z direction.



On most CNC systems, this function can be executed in two different ways:

- **G02: helical interpolation in a clockwise direction**
- **G03: helical interpolation in a counterclockwise direction**

The thread milling operation (figure B) consists of circular rotation of the tool about its own axis together with an orbiting motion along the bore or workpiece circumference.

During one such orbit, the tool will move vertically one pitch length. These movements, combined with the insert geometry, create the required thread form.

There are three acceptable ways of approaching the workpiece with the tool to initiate production of the thread:

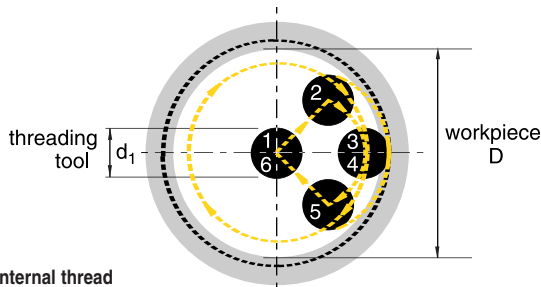
1. **along a tangential arc**
2. **radially**
3. **along a tangential straight line**

NOTE: Climb milling is preferred.

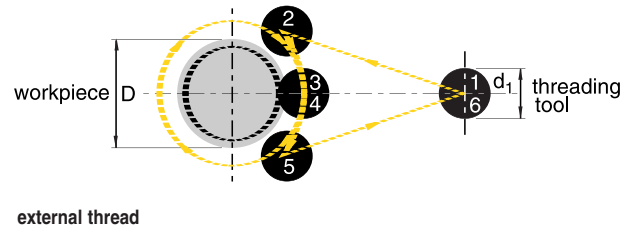
Tangential Approach (arc)

With this method, the tool enters and exits the workpiece smoothly. No marks are left on the workpiece and there is no vibration, even with harder materials.

Although it requires slightly more complex programming than the radial approach method (see next page), this is the method recommended for machining the highest quality threads.

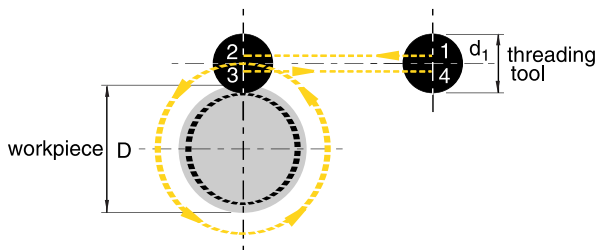


1 - 2: rapid approach
2 - 3: tool entry along tangential arc, with simultaneous feed along Z-axis
3 - 4: helical movement during one full orbit (360°)
4 - 5: tool exit along tangential arc, with continuing feed along the Z-axis
5 - 6: rapid return



Tangential Approach (line)

This method is very simple, and has all of the advantages of the tangential arc method. However, it is applicable only to external threads.



1 - 2: radial entry with simultaneous feed along the Z-axis
2 - 3: helical movement during one full orbit (360°)
3 - 4: radial exit



Thread Milling System (TMS)

Radial Approach

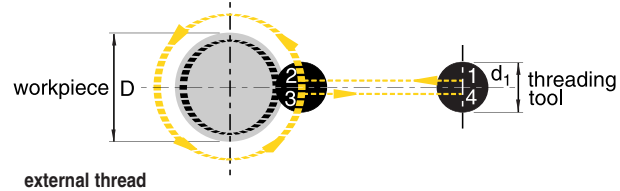
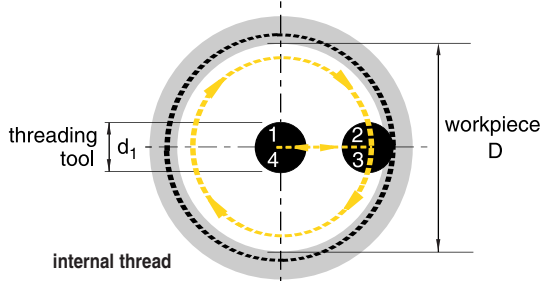
- 1 - 2: radial entry
- 2 - 3: helical movement during one full orbit (360°)
- 3 - 4: radial exit

This is the simplest method. There are two characteristics worth noting about the radial approach:

a. A small vertical mark may be left at the entry (and exit) point. This is of no significance to the thread itself.

b. When using this method with very hard materials, there may be a tendency for the tool to vibrate as it approaches the full cutting depth.

NOTE: Radial feed during entry to the full profile depth should be only 1/3 of the subsequent circular feed!



Preparing for the Thread Milling Operation

calculation of feed rates at the cutting edge

The first step is to calculate the tool feed rate at the cutting edge.

$$F_1 = ipt \times nt \times rpm$$

where:

- F_1 = tool feed rate at the cutting edge (in./min.)
- ipt = inch per tooth (feed rate)
- nt = number of effective inserts in cutter
- rpm = rotational speed (spindle rpm)

The rotational speed (rpm) is calculated by the following formula:

$$rpm = \frac{12 \times sfm}{\pi \times d_1}$$

where:

- sfm = cutting speed, surface feet per minute
- d_1 = cutter diameter, over insert
- π = 3.1416

calculation of program feed rate

On most CNC machines, the feed rate required for programming is at the centerline of the tool. When dealing with linear tool movement, the feed rate at the cutting edge and the centerline are identical. But, with

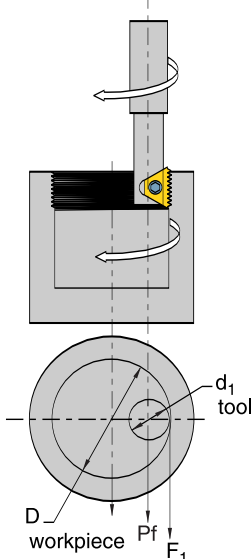
circular tool movement this is not the case. The following equations define the relationship between feed rates at the cutting edge and at the tool centerline.

internal thread

$$P_f = F_1 - \frac{(F_1 \times d_1)}{D}$$

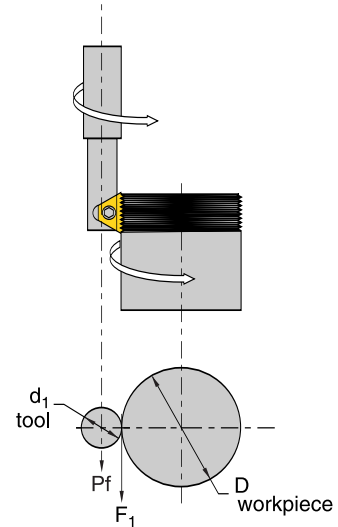
where:

- P_f = program feed rate (in./min.)
- D = major diameter (external thread)
- D = minor diameter (internal thread)
- d_1 = cutting diameter, over insert



external thread

$$P_f = F_1 + \frac{(F_1 \times d_1)}{D}$$



Inserts
 Face Mills
 End Mills
 Die and Mold
 Slotting
 Thread Milling
 Widia Cutters
 Vintage Cutters
 Accessories
 Technical Data
 Mat'l Database
 Index

Application and Technical Information



Step-by-Step Thread Milling Example

thread: internal right hand 1 1/4 x 16 UN-2B-RH(21)
material: AISI 4140 (300 HB)
thread diameters: D (minimum bore dia.) = 1.182
 D_o (nominal dia.) = 1.25 inch
thread length: .50 inch

Define the Appropriate Cutter:

For best thread quality, the cutter with the largest d₁ (cutter diameter) possible should be used. This cutter diameter can be found in the table on page M370, as a function of pitch and minimum bore diameter "D". The result for the above example is that any cutter diameter 1.02 inch or smaller can be utilized.

A cutter with a smaller d₁ will perform the thread milling operation in less time. The smaller d₁ may result in less tool rigidity, so it should be used with caution on very tough materials.

Find the appropriate "normal-length shank cutter" on page M368. Use the minimum bore diameter table above.

Find the appropriate normal-length shank cutter diameter on page M368.

pitch (tpi)	24	20	16	12
pitch mm	1,0	1,25	1,5	2,0
cutter dia. d ₁	minimum bore diameter D			
.67	.748	.772	.787	.827
.75	.827	.850	.866	.906
.79	.866	.890	.906	.945

figure B: cutter selected: K079TMIRW075STN16N
 outer dimensions: d₁ = .79, R_t (radius of tool) = d₁ ÷ 2 = .395

Choosing Insert Size

The insert IC is defined by the selected cutter (STN16). Use the appropriate insert table on pages M363-M366.

insert size IC	a inch (mm)	pitch (tpi)	internal thread	b inch thread length	number of teeth	grade		external thread	b inch thread length	number of teeth	grade		cutter type
			catalog number			KC610M	KC620M	catalog number			KC610M	KC620M	
3/8	.63 (16)	32	STN16 32UN-I	.59	19	<input type="checkbox"/>	<input type="checkbox"/>	STN16 32UN-E	.59	19	<input type="checkbox"/>	<input type="checkbox"/>	STN16
		28	STN16 28UN-I	.57	16	<input type="checkbox"/>	<input checked="" type="checkbox"/>	STN16 28UN-E	.57	16	<input type="checkbox"/>	<input type="checkbox"/>	
		27	STN16 27UN-I	.56	15	<input type="checkbox"/>	<input checked="" type="checkbox"/>	STN16 27UN-E	.56	15	<input checked="" type="checkbox"/>	<input type="checkbox"/>	
		24	STN16 24UN-I	.55	14	<input type="checkbox"/>	<input type="checkbox"/>	STN16 24UN-E	.58	14	<input checked="" type="checkbox"/>	<input type="checkbox"/>	
		20	STN16 20UN-I	.55	11	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	STN16 20UN-E	.55	11	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
		18	STN16 18UN-I	.56	10	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	STN16 18UN-E	.56	10	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
		16	STN16 16UN-I	.56	9	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	STN16 16UN-E	.56	9	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
		14	STN16 14UN-I	.57	8	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	STN16 14UN-E	.57	8	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
		13	STN16 13UN-I	.54	7	<input type="checkbox"/>	<input type="checkbox"/>	STN16 13UN-E	.54	7	<input type="checkbox"/>	<input type="checkbox"/>	
		12	STN16 12UN-I	.58	7	<input type="checkbox"/>	<input checked="" type="checkbox"/>	STN16 12UN-E	.58	7	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	

insert selected: STN16 16UN-I
 ■ stock standard □ non-stock standard

Cutting Data Recommendation

Choose the carbide grade, define the cutting speed and select the proper feed.

workpiece material	cutting speed (sfm)		feed rate per revolution	mini thread mill STN10
	grade KC610M	grade KC620M		
carbon steels (< 187 HB)	520 - 900	—	.004 - .006	.002 - .004
carbon steels (187 to 220 HB)	550 - 860	—	.004 - .006	.002 - .004
alloy steel (200 - 250 HB)	390 - 600	—	.004 - .006	.002 - .004
alloy steel (250 - 325 HB)	350 - 525	—	.004 - .006	.002 - .004
stainless steel austenitic (< 210 HB)	450 - 660	—	.004 - .006	.002 - .004

selected carbide grade: KC610M
 selected cutting speed: 500 sfm
 selected feed: .004 ipt

Inserts
 Face Mills
 End Mills
 Die and Mold
 Slotting
 Thread Milling
 Width Cutters
 Vintage Cutters
 Accessories
 Technical Data
 Mat'l Database
 Index



Step-by-Step Thread Milling Example (cont'd.)

Inserts

Calculate the feed rates:

First, find the rpm.

$$\text{rpm} = \frac{12 \times \text{sfm}}{\pi \times d_1} = \frac{12 \times 500}{3.14 \times .79} = 2418 \text{ rpm}$$

Face Mills

Next, calculate the feed rate at the insert cutting edge (F₁):

(using the chosen feed per tooth of .004.)

$$F_1 = \text{ipt} \times \text{nt} \times \text{rpm} = .004 \times 1 \times 2418 = 9.67 \text{ in./min.}$$

End Mills

Finally, calculate the feed rate at the cutter centerline (F₂):

$$F_2 = \frac{F_1 \times (D - d_1)}{D} = \frac{9.67 \times (1.182 - .79)}{1.182} = 3.207 \text{ in./min.}$$

Die and Mold

Select the thread milling method.

[Selected method...climb milling (preferred), refer to page 712.]

Calculate the radius of the tangential arc R_e:

$$R_e = \frac{(R_i - C_L)^2 + R_o^2}{2R_o} = \frac{(.591 - .02)^2 + .625^2}{2 \times .625}$$

$$R_e = .573333 \text{ in.}$$

Slotting

Calculate the angle (β):

$$\beta = 90^\circ + \arcsin\left(\frac{R_o - R_e}{R_e}\right)$$

$$\beta = 90^\circ + \arcsin\left(\frac{.625 - .573333}{.573333}\right)$$

$$\beta = 90^\circ + 5.17^\circ = 95.17^\circ = 95^\circ 10'$$

Thread Milling

Calculate the movement along the Z-axis during the entry approach from point "A" to point "B" (Z_α).

(NOTE: P = pitch)

$$Z_\alpha = P(\text{in.}) \times \frac{\alpha^\circ}{360^\circ} = .0625 \times \frac{95.17}{360} = .0156 \text{ in., since } \alpha = 90^\circ$$

Widia Cutters

Calculate the "X" and "Y" values at the start of the entry approach.

$$X = 0 \quad Y = -R_i + C_L = -.591 + .02 = -.571 \text{ in.}$$

Accessories

Define Z-axis location at the start of the entry approach.

(NOTE: L = length of thread)

$$Z = -(L + Z_\alpha) = -(.50 + .0156) = -.5156 \text{ in.}$$

Technical Data

Define the starting point.

$$X_a = 0$$

$$Y_a = 0$$

Mat'l Database

Index

CNC Program (Fanuc 11M)

```

%
N10G90G00G57X0.000Y0.000
N20G43H10Z0.M3S2417
N30G91G00X0.Y0.Z-.5156
N40G41D60X0.000Y-.5710Z0.
N50G03X0.6250Y0.5710Z0.0156R0.5733F3.206
N60G03X0.Y0.Z0.0625I-.0625J0.
N70G03X-.0625Y0.5710Z0.0156R0.5733
N80G00G40X0.Y-.5710Z0.
N90G49G57G00Z8.0M5

```

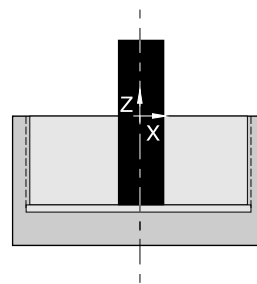
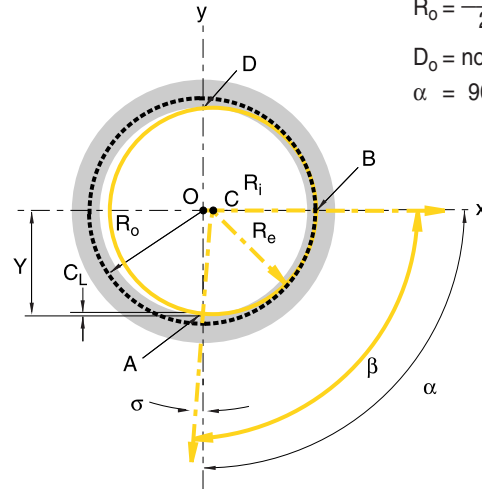
$$R_i = \frac{D}{2}$$

D = minor diameter

$$R_o = \frac{D_o}{2}$$

D_o = nominal diameter

$$\alpha = 90^\circ$$





Thread Milling System (TMS)

Appendix A

Derivation of Formulas for Internal Thread Milling

R_e , β , and X can be found by a geometric analysis of the entry path.

This entry path is defined by the tool traveling along a circular path, with a radius of R_e about the point "C".

$$\text{derivation of } R_e = \frac{(R_i - C_L)^2 + R_o^2}{2R_o}$$

Triangle OAC enables us to simply solve for R_e . Note that OAC is a right angle triangle, and that:

$$OA = R_i - C_L$$

$$CA = R_e$$

$$OC = R_o - R_e$$

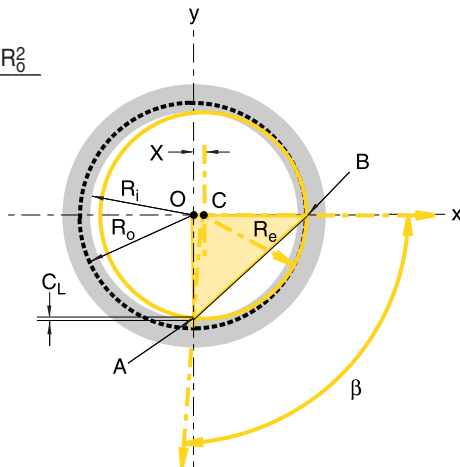
$$\text{Pythagoras' law states: } OA^2 + OC^2 = AC^2$$

Replacing actual values, we get:

$$(R_i - C_L)^2 + (R_o - R_e)^2 = R_e^2$$

Simplifying, we get:

$$R_e = \frac{(R_i - C_L)^2 + R_o^2}{2R_o}$$



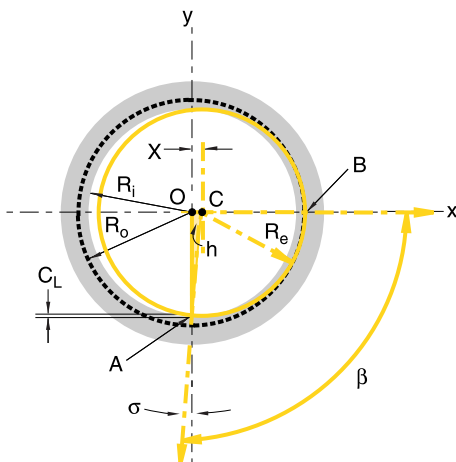
Find the angle β .

$$\beta = 90^\circ + \sigma$$

$$\sin \sigma = \frac{OC}{CA} = \frac{(R_o - R_e)}{R_e}$$

$$\sigma = \arcsin \left(\frac{R_o - R_e}{R_e} \right)$$

$$\text{Therefore, } \beta = 90^\circ + \arcsin \left(\frac{R_o - R_e}{R_e} \right)$$



Appendix B

Derivation of Formulas for External Thread Milling

R_e , β , and X can be found by a geometric analysis of the entry path.

This entry path is defined by the tool traveling along a circular path, with a radius of R_e about the point "C".

$$\text{derivation of } R_e = \frac{(R_o + C_L)^2 + R_i^2}{2R_i}$$

Triangle OAC enables us to simply solve for R_e . Note that OAC is a right angle triangle, and that:

$$OA = R_o + C_L$$

$$CA = R_e$$

$$OC = R_e - R_i$$

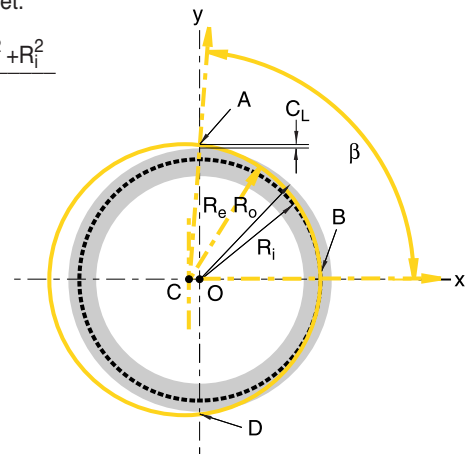
$$\text{Pythagoras' law states: } OA^2 + OC^2 = AC^2$$

Replacing actual values, we get:

$$(R_o + C_L)^2 + (R_e - R_i)^2 = R_e^2$$

Simplifying, we get:

$$R_e = \frac{(R_o + C_L)^2 + R_i^2}{2R_i}$$

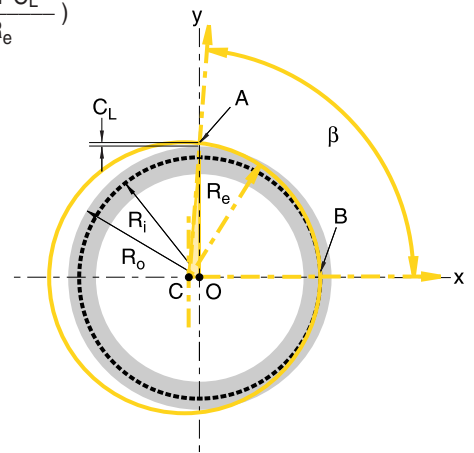


Find the angle β .

β can be easily found using the same triangle:

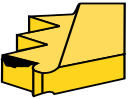
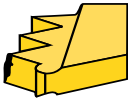
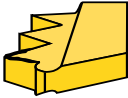
$$\sin \beta = \frac{AO}{AC} = \frac{(R_o + C_L)}{R_e}$$

$$\beta = \arcsin \left(\frac{R_o + C_L}{R_e} \right)$$





Thread Milling System (TMS) Thread Milling Troubleshooting

problem	possible cause	solution
excessive insert flank wear 	• cutting speed too high	• Reduce cutting speed.
	• chip is too thin	• Increase feed rate.
	• insufficient coolant	• Increase coolant quantity/pressure.
chipping of cutting edge 	• chip is too thick	<ul style="list-style-type: none"> • Reduce feed rate. • Use the tangential arc method of entrance. • Increase rpm.
	• vibration	• Check rigidity.
material build-up on the cutting edge 	• cutting speed too slow	• Increase cutting speed.
	• chip thickness too small	• Increase feed rate.
chatter / vibration	• feed rate is too high	• Reduce the feed.
	• profile is too deep (coarse pitch threads)	<ul style="list-style-type: none"> • Execute two passes, each with increased cutting depth. • Execute two passes, each cutting only half the thread length.
	• thread length is too long	• Execute two passes, each cutting only half of the thread length.
insufficient thread accuracy	• tool deflection	<ul style="list-style-type: none"> • Reduce feed rate. • Execute a "zero" cut.

Insert Tolerance Classes

thread designation	standard designation	tolerance class
UN	ANSI B 1.174	2A/2B
*UNJ	MIL-S-8879A	3A/3B
ISO	R262 (DIN 13)	6g/6H
NPT	USAS B2.1 : 1968	standard NPT
NPTF	ANSI B 1.20.3-1976	standard
BSW	B.S. 84 : 1956, DIN 259, ISO 228/1 : 1982	medium class A
BSPT	B.S. 21 : 1985	standard BSPT
*ACME	ANSI B1/5 : 1988	3G
*PG	DIN 40430	standard
*TR	DIN 103	7e/7H

*NOTE: Denotes thread forms quoted on request as non-stock standards (4-6 weeks delivery).

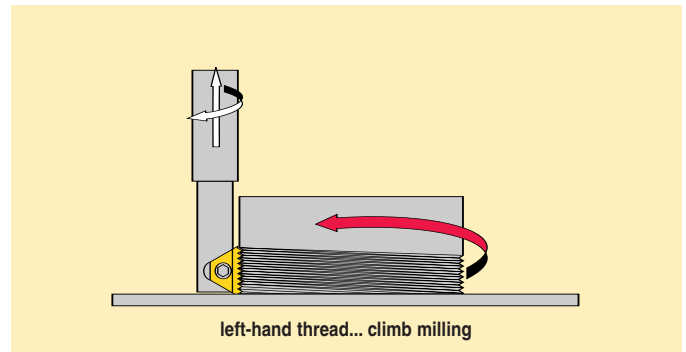
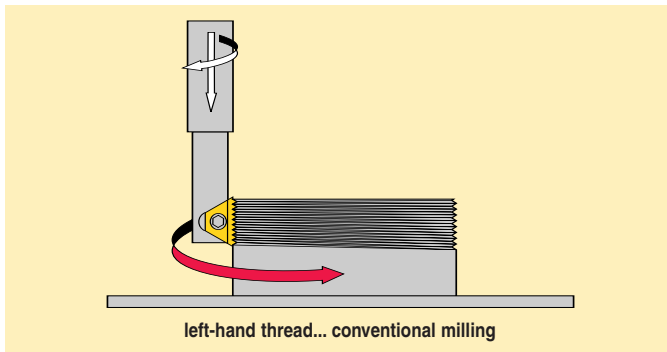
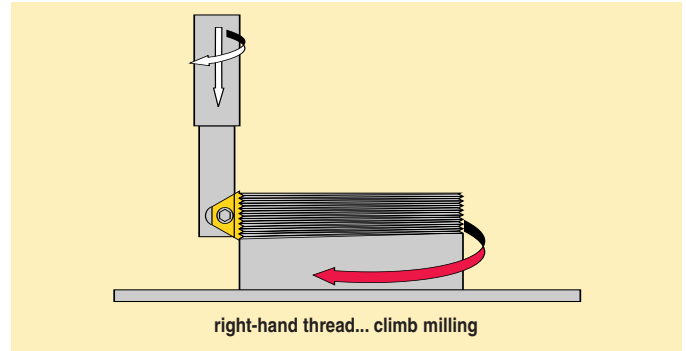
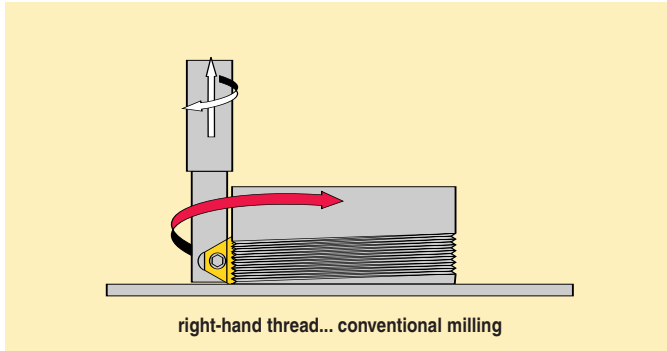


Thread Milling System (TMS)

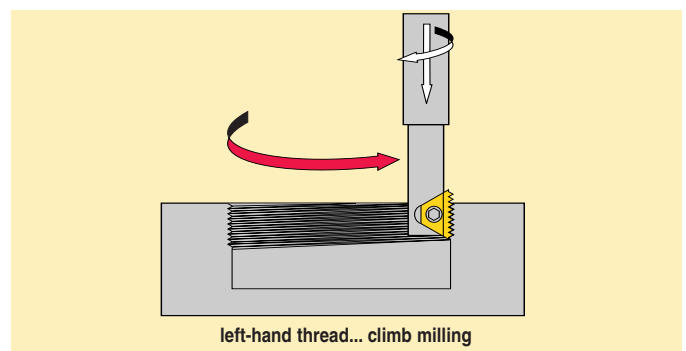
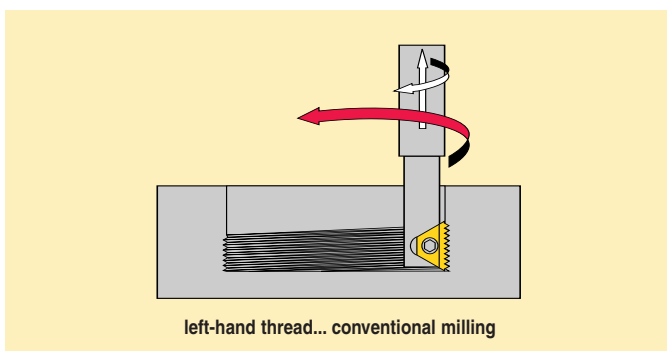
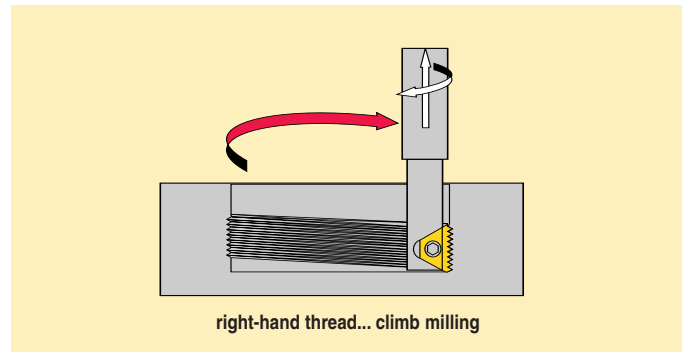
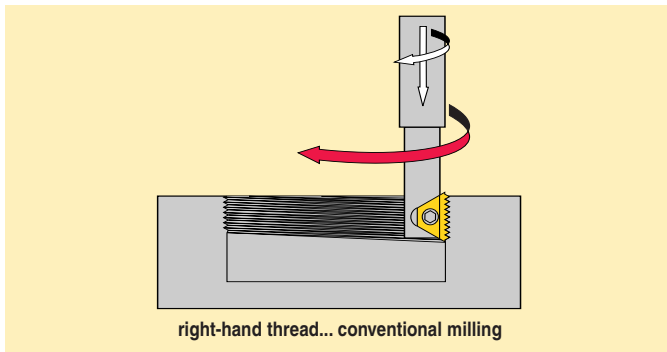
The following are a few thread milling methods (work directions).

NOTE: Climb milling results in lower cutting forces, better chip development, higher thread surface quality and longer insert life. Therefore, it should be used whenever possible. But, in the case of some hardened materials or when milling certain difficult-to-machine exotic materials, conventional milling may be preferred.

Methods of External Thread Milling



Methods of Internal Thread Milling



Inserts

Face Mills

End Mills

Die and Mold

Slotting

Thread Milling

Widia Cutters

Vintage Cutters

Accessories

Technical Data

Mat'l Database

Index