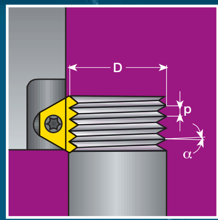
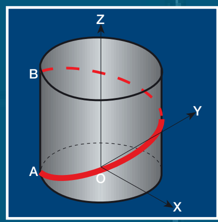
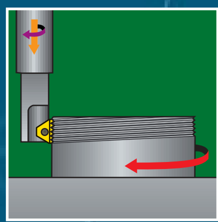
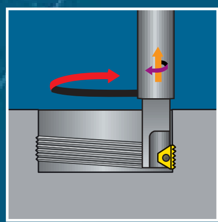
THREAD MILLING TECHNICAL DATA

Contents

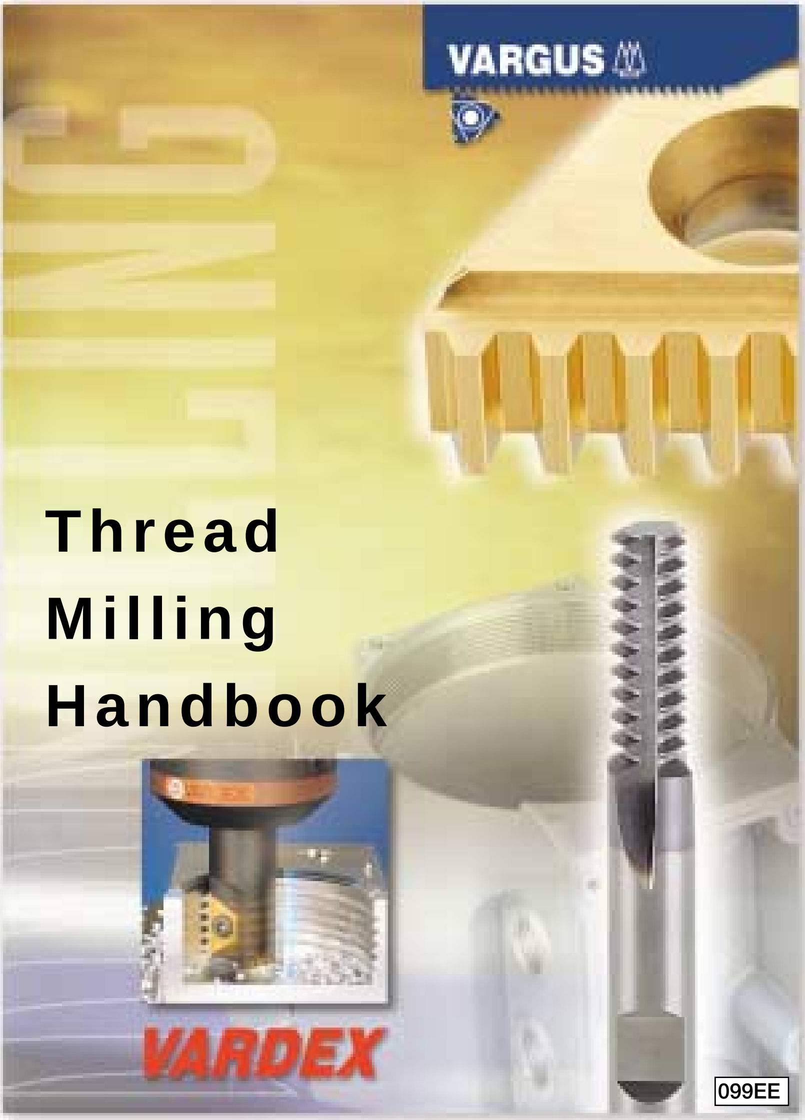








For more Technical Data see our TM Handbook



About 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 motion 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 (Fig. A) on the envelope of the cylinder combines a circular movement in the xy plane with a linear displacement in the z direction.
On most CNC systems this function can be executed in two different ways:

GO2: Helical interpolation in a clockwise direction
GO3: Helical interpolation in a counter-clockwise direction

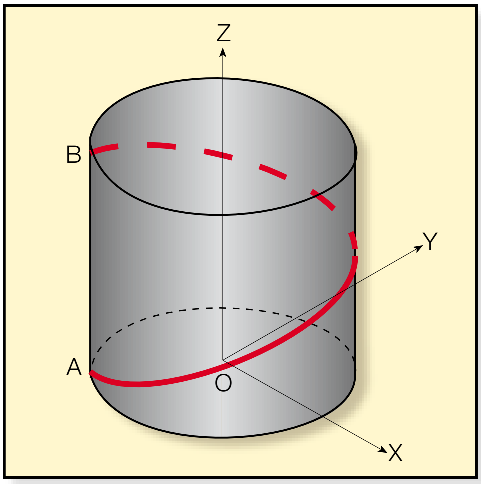


Fig. A

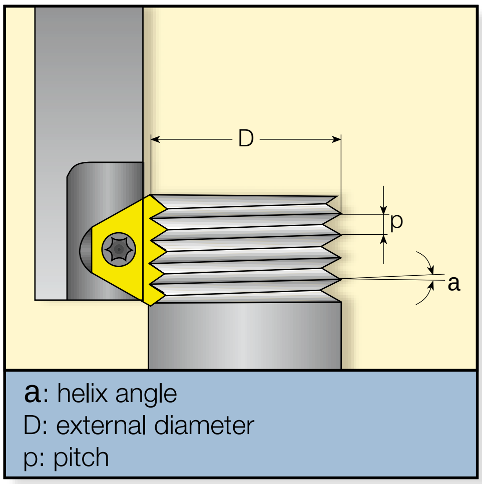


Fig. B

The thread milling operation (Fig. B) consists of circular rotation of the tool around its own axis together with an orbiting motion along the bore or workpiece circumference.
During one such orbit, the tool will shift 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. Tangential Arc Approach

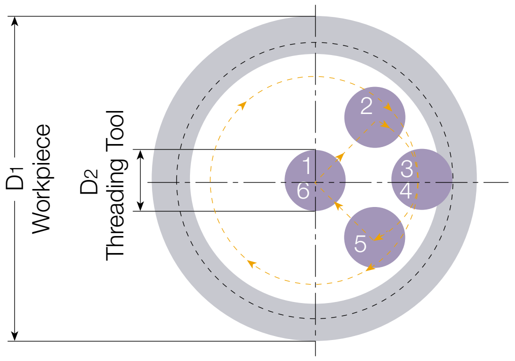
2. Radial Approach

3. Tangential Line Approach

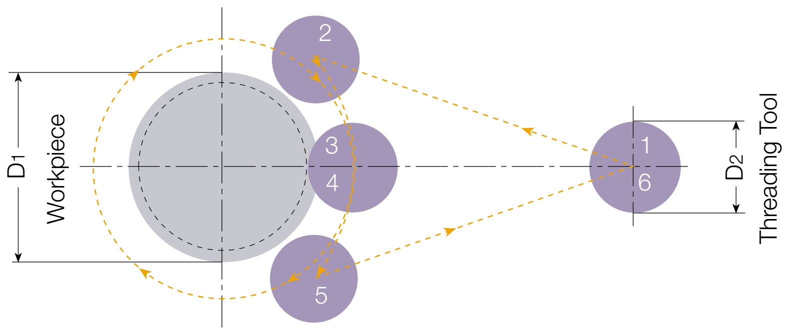
1. Tangential Arc Approach

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 (see below), this is the method recommended for machining the highest quality threads.

Internal Thread



External Thread

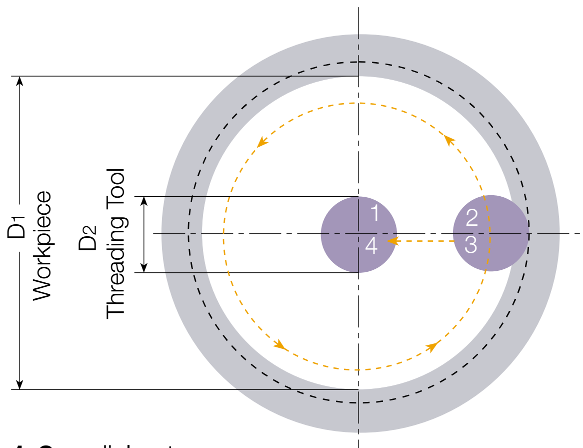


1-2: rapid approach
2-3: tool entry along tangential arc, with simultaneous feed along z-axis
3-4: helical movement during one full orbit (360º).
4-5: tool exit along tangential arc, with continuing feed along z-axis
5-6: rapid return

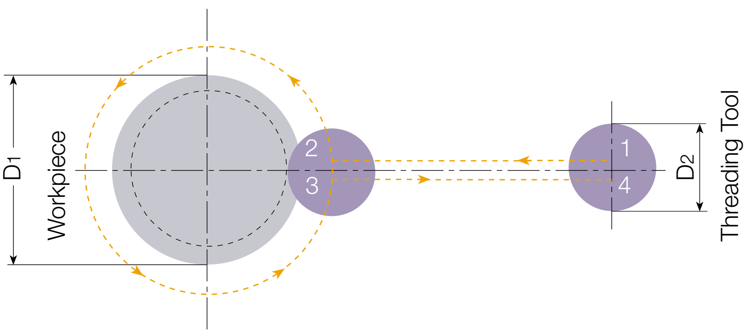
2. Radial Approach

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 of the tool to vibrate as it approaches the full cutting depth.
Note: Radial feed during entry to the full profile depth should only be 1/3 of the subsequent circular feed !

Internal Thread



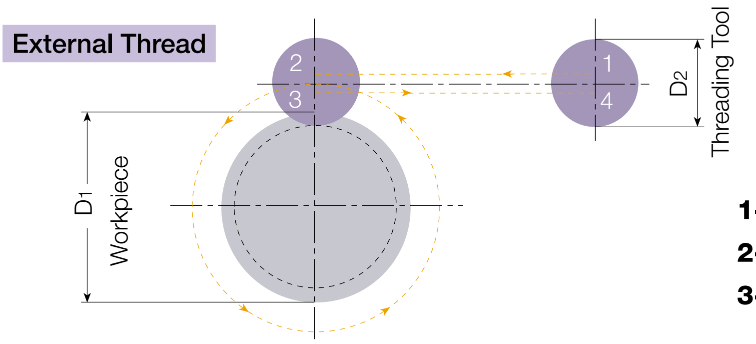
External Thread



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

3. Tangential Line Approach

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



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

Preparing for the Thread Milling Operation

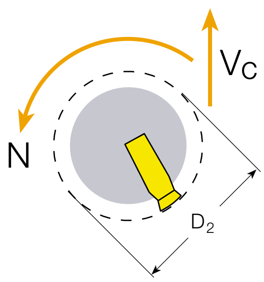
1. Calculation of Rotational Velocity and Feed at the Cutting Edge

$$N = \frac{1000 \times V}{\pi \times D_2}$$

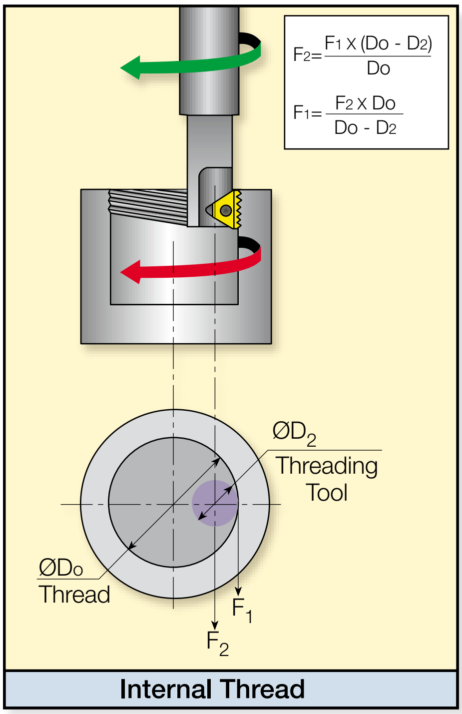
$$V = \frac{N \times \pi \times D_2}{1000}$$

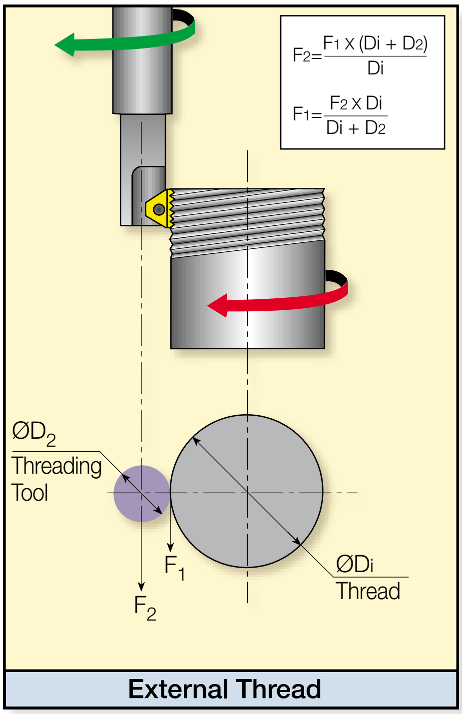
$$F_1 = N \times z \times f$$

N - Rotational Velocity [R.P.M.]
V - Cutting Speed [m/min]
D_2 - Toolholder Cutting Dia. [mm]
F_1 - Tool Feed Rate at the Cutting Egde [mm/min]
z - No. of Cutting Edges
f - Feed per Tooth per Rotation [mm/rev]



2. Calculation of Feed Rates at the Tool Center Line





On most CNC machines, the feed rate required for programming is that of the center-line of the tool. When dealing with linear tool movement, the feed rate at the cutting edge and the center line are identical, but with circular tool movement such is not the case.
The equations define the relationship between feed rates at the cutting edge and at the tool center line.

Appendix C

List of “G” Codes (ISO) for CNC Program

Code	Description
%	Recognition code (ISO or EIA), +End of tape
G00	Fast feed linear positioning
G01	Linear interpolation
G02	Circular/Helical interpolation CW
G03	Circular/Helical interpolation CCW
G40	Cutter radius compensation cancel
G41	Cutter radius compensation left
G42	Cutter radius compensation right
G43	Tool length compensation +
G49	Tool length compensation cancel
G57	Work coordinate system selection
G90	Absolute command relative to work coordinate origin
G91	Incremental command relative to tool position
F	Feed mm/min
S	Spindle speed RPM

Code	Description
H	Tool length compensation number
D	Tool radius compensation number
X	X coordinate
Y	Y coordinate
Z	Z coordinate
R	Radius of travel
I	X coordinate to center of starting arc travel
J	Y coordinate to center of starting arc travel
M3	Spindle forward rotation
M5	Spindle stop
M30	Program end & rewind
O	Program number
N	Block number (can be avoided)
(	Start of comment
)	End of comment

Grades and their Applications

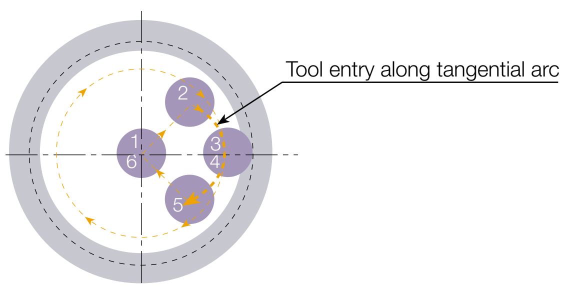
Grade	Application	Sample
VBX	**First Choice for steel and cast iron.** A tough sub-micron substrate with TiCN coating. Provides good fracture toughness and excellent wear resistance.	
VTX	**First Choice for stainless steel.** A tough sub-micron substrate with TiAlN coating. Provides good fracture toughness and excellent wear resistance.	
VK2	Uncoated grade for machining cast iron & nonferrous metals.	
VTS	TM Solid, straight flute grade. Excellent for general use. TiAlN coated.	
VTH	TM Solid, helical flute grade. Excellent for general use. TiCN coated.	

Recommended Grades, Cutting Speeds Vc [m/min] and Feed f [mm/tooth]

Material			Hardness Brinell HB	Vc[m/min]					Feed f [mm/tooth] *		
				Indexable Inserts			Solid carbide		Indexable Inserts	Solid Helical Flute	Solid Straight Flute
				Coated		Uncoated	Coated				
				VBX	VTX	VK2	VTH	VTS			
P	Unalloyed steel	Low carbon (C=0.1-0.25 %)	125	100-210	90-180		80-250	50-180	0.05-0.3	0.03-0.15	0.01-0.1
P		Medium carbon (C=0.25-0.55 %)	150	100-180	90-170		80-230	50-140	0.05-0.25	0.03-0.1	0.01-0.08
P		High carbon (C=0.55-0.85 %)	170	100-170	90-160		80-200	50-120	0.05-0.2	0.03-0.08	0.01-0.05
P	Low alloy steel (alloying elements ≤ 5%)	Non hardened	180	90-160	90-155		60-180	60-170	0.05-0.25	0.03-0.1	0.03-0.07
P		Hardened	275	80-150	80-160		60-170	60-160	0.05-0.2	0.03-0.07	0.03-0.07
P		Hardened	350	70-140	70-150		60-160	60-150	0.05-0.15	0.01-0.03	0.005-0.01
P	High alloy steel (alloying elements > 5%)	Annealed	200	60-130	70-115		40-100	40-90	0.05-0.2	0.03-0.05	0.01-0.03
P		Hardened	325	70-110	60-100		30-80	30-70	0.05-0.1	0.01-0.03	0.005-0.01
P	Cast steel	Low alloy (alloying elements <5%)	200	100-170	100-170	100-150	80-250	70-200	0.05-0.15	0.03-0.1	0.01-0.03
P		High alloy (alloying elements >5%)	225	70-120	70-130	60-130	60-170	60-150	0.05-0.1	0.01-0.03	0.005-0.01
M	Stainless steel Feritic	Non hardened	200	100-170	120-180		60-150	50-140	0.05-0.15	0.04-0.1	0.01-0.05
M		Hardened	330	100-170	120-180		60-120	50-110	0.05-0.1	0.01-0.05	0.005-0.01
M	Stainless steel Austenitic	Austenitic	180	70-140	100-140		60-140	60-130	0.05-0.15	0.04-0.1	0.007-0.02
M		Super austenitic	200	70-140	100-140		60-130	50-120	0.05-0.1	0.04-0.1	0.007-0.02
M	Stainless steel Cast feritic	Non hardened	200	70-140	100-140		60-160	50-150	0.05-0.15	0.04-0.1	0.01-0.03
M		Hardened	330	70-140	100-140		60-110	50-100	0.05-0.1	0.03-0.05	0.005-0.01
M	Stainless steel Cast austenitic	Austenitic	200	70-120	100-120		60-150	50-140	0.05-0.15	0.04-0.1	0.01-0.03
M		Hardened	330	70-120	100-120		60-100	50-90	0.05-0.1	0.03-0.05	0.005-0.01
M	High temperature alloys	Annealed (Iron based)	200	20-45	20-40	20-30	30-60	30-50	0.05-0.1	0.04-0.1	0.007-0.02
M		Aged (Iron based)	280	20-30	20-30	15-25	20-50	20-40	0.02-0.05	0.01-0.03	0.005-0.01
M		Annealed (Nickel or Cobalt based)	250	15-20	15-20	15-20	15-35	15-30	0.02-0.05	0.01-0.03	0.005-0.01
M		Aged (Nickel or Cobalt based)	350	10-15	10-15	10-15	15-30	15-25	0.02-0.05	0.01-0.03	0.005-0.01
M	Titanium alloys	Pure 99.5 Ti	400Rm	70-140	70-120	40-60	40-80	30-70	0.02-0.05	0.03-0.05	0.007-0.02
M		a+b alloys	1050Rm	20-50	20-50	20-40	20-50	20-45	0.02-0.05	0.03-0.05	0.007-0.02
K	Extra hard steel	Hardened & tempered	55HRc	20-45	20-45		15-45	15-35	0.01-0.03	0.005-0.01	0.003-0.006
K	Malleable cast iron	Ferritic (short chips)	130	60-130	100-120		70-160	60-150	0.02-0.08	0.01-0.03	0.007-0.02
K		Pearlitic (long chips)	230	60-120	80-100		60-150	100	0.02-0.05	0.03-0.05	0.005-0.01
K	Grey cast iron	Low tensile strength	180	60-130	80-100		70-160	50-140	0.05-0.15	0.05-0.1	0.007-0.02
K		High tensile strength	260	60-100	80-100		40-120	40-110	0.05-0.1	0.03-0.05	0.005-0.01
K	Nodular SG iron	Feritic	160	60-125	80-100		40-110	40-100	0.05-0.15	0.05-0.1	0.007-0.02
K		Pearlitic	260	50-90	60-90		40-100	40-90	0.05-0.1	0.03-0.05	0.005-0.01
K	Aluminium alloys Wrought	non aging	60	100-250		200-300	200-300	150-250	0.1-0.4	0.1-0.25	0.05-0.15
K		Aged	100	100-180		60-110	150-250	100-220	0.1-0.3	0.1-0.2	0.03-0.1
K	Aluminium alloys	Cast	75	150-400		60-120	100-200	80-150	0.1-0.3	0.1-0.2	0.05-0.15
K		Cast & aged	90	150-280		60-100	120-220	90-160	0.05-0.25	0.1-0.15	0.03-0.1
K	Aluminium alloys	Cast Si 13-22%	130	80-150		20-50	200-300	150-250	0.1-0.3	0.1-0.2	0.05-0.15
K	Copper and copper alloys	Brass	90	120-210	100-200	50-70	200-300	150-250	0.1-0.3	0.1-0.25	0.05-0.15
K		Bronze and non leaded copper	100	120-210	100-200	50-70	150-250	100-220	0.05-0.25	0.1-0.2	0.03-0.1

Recommendation:
At tool entry, set the Feed f [mm/tooth] to 70% lower than the threading Feed.

Example:
Threading Feed: 0.3[mm/tooth]
Tool entry Feed: 0.09[mm/tooth]



Minimum Bore Diameters for Thread Milling

Pitch mm		0.5	0.6	0.7	0.75 0.80	0.9	1.0	1.25	1.5	1.75	2.0		2.5	3.0	3.5	4.0	4.5	5.0	5.5		6.0	
Pitch tpi		48	44	36	32	28	26 24	20 19	18 16	14	13 12	11.5 11	10	9 8	7	6		5		4.5		4
Toolholder Ordering Code	D2	Minimum Bore Diameter Di mm																				
TMMC 12-6.0	9.0	9.5	9.7	9.9	10.0	10.4	10.7	11.4	12.0													
TMMC 20-6.0	9.0	9.5	9.7	9.9	10.0	10.4	10.7	11.4	12.0													
TMMC 20-6.0 124/003	9.0	9.5	9.7	9.9	10.0	10.4	10.7	11.4	12.0													
TMC 12-2	11.5	12.0	12.2	12.4	12.5	12.9	13.2	13.9	14.5	15.1												
TMC 20-2	11.5	12.0	12.2	12.4	12.5	12.9	13.2	13.9	14.5	15.1												
TMLC 25-2	11.5	12.0	12.2	12.4	12.5	12.9	13.2	13.9	14.5	15.1												
TMSC 10-2	12.5	13.0	12.6	13.6	13.5	13.9	14.2	14.9	15.5	16.1												
TMOC 20-2	14.5	15.1	15.2	15.3	15.4	16.0	16.4	17.0	17.8	18.6												
TMNC 16-3	15.5	16.0	16.2	16.4	16.5	16.9	17.2	17.9	18.5	19.0	19.5	20.0										
TMC 16-3 124/001	15.5	16.0	16.2	16.4	16.5	16.9	17.2	17.9	18.5	19.0	19.5	20.0										
TMC 16-3	17.0	17.6	17.8	18.0	18.2	18.70	19.0	19.6	20.0	20.5	21.0	21.5										
BTMC 16-3B	17.0	17.6	17.8	18.0	18.2	18.7	19.0	19.6	20.0	20.5	21.0	21.5										
TM2C 20-2	17.0	17.6	17.8	18.0	18.2	18.7	19.0	19.6	20.0	20.5												
BTMC 20-3B	19.0	19.7	20.0	20.2	20.4	20.8	21.0	21.6	22.0	22.5	23.0	23.5										
TMNC 20-3	19.0	19.7	20.0	20.2	20.4	20.8	21.0	21.6	22.0	22.5	23.0	23.5										
TMC 20-3	20.0	20.7	21.0	21.2	21.4	21.8	22.0	22.6	23.0	23.5	24.0	24.5										
TMOC 20-3	20.0	20.7	21.0	21.2	21.4	21.8	22.0	22.6	23.0	23.5	24.0	24.5										
BTMWC 25-3B	22.0	22.7	23.0	23.2	23.4	23.8	24.0	24.6	25.0	25.5	26.0	26.5										
BTMLC 25-3B	22.0	22.7	23.0	23.2	23.4	23.8	24.0	24.6	25.0	25.5	26.0	26.5										
TMLC 25-3	22.0	22.7	23.0	23.2	23.4	23.8	24.0	24.6	25.0	25.5	26.0	26.5										
TMC 25-5 124/004	25.0	25.7	26.0	26.2	26.4	26.8	27.0	27.7	28.2	28.7	29.2	29.7	31.3	33.7	36.7	39.7	42.7					
TM2C 25-3	26.0	26.7	27.0	27.2	27.4	27.8	28.0	28.7	29.3	29.8	30.3	30.8										
BTM2C 25-3B	26.0	26.7	27.0	27.2	27.4	27.8	28.0	28.7	29.3	29.8	30.3	30.8										
TMC 25-5	30.0	30.7	31.0	31.2	31.4	31.8	32.0	32.8	33.5	34.1	34.6	35.6	36.6	39.0	42.0	45.0	48.0					
TMLC 25-5	30.0	30.7	31.0	31.2	31.4	31.8	32.0	32.8	33.5	34.1	34.6	35.6	36.6	39.0	42.0	45.0	48.0					
TMOC 25-5	30.0	30.7	31.0	31.2	31.4	31.8	32.0	32.8	33.5	34.1	34.6	35.6	36.6	39.0	42.0	45.0	48.0					
TMC 32-6B	35.0								38.5	39.1	39.6	40.6	42.0	44.0	47.0	50.0	53.4	42.5	50.0	44.6	57.5	56.6
TMC 32-5	37.0	38.0	38.2	38.4	38.6	39.1	39.5	40.4	41.0	41.5	42.0	43.0	44.0	46.5	49.0	52.0	55.5					
TMLC 32-5	37.0	38.0	38.2	38.4	38.6	39.1	39.5	40.4	41.0	41.5	42.0	43.0	44.0	46.5	49.0	52.0	55.5					
TMNC 32-5	37.0	38.0	38.2	38.4	38.6	39.1	39.5	40.0	41.0	41.5	42.0	43.0	44.0	46.5	49.0	52.0	55.5					
TMSH D38-16-2	38.0	38.5	38.7	38.9	39.0	39.6	40.0	41.0	42.0	43.0												
TM2C 32-5	42.0	43.2	43.4	43.6	43.8	44.5	45.0	46.0	46.5	47.0	47.4	48.2	49.0	52.0	54.5	57.5	61.0					
TMVC 32-5	46.0																				62.5	
TMC 40-6B	46.0								49.5	50.1	50.6	51.6	53.0	55.0	55.2	55.6	55.0	52.5	54.0	54.5	57.5	56.6
TMLC 40-6B	46.0								49.5	50.1	50.6	51.6	53.0	55.0	55.2	55.6	55.0	52.5	54.0	54.5	57.5	56.6
TMSH D50-22-2	50.0	50.5	50.7	50.9	51.0	51.6	52.0	53.0	54.0	54.5												
TMSH D50-22-3	50.0	50.5	50.7	50.9	51.0	51.6	52.0	53.0	54.0	54.5	55.0	55.5										

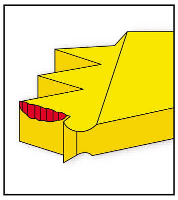
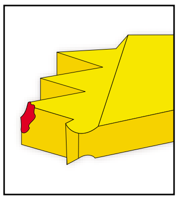
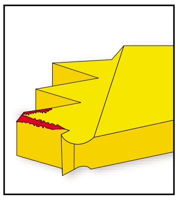
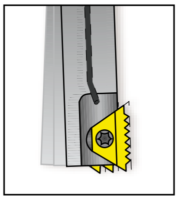
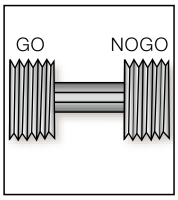
Minimum Bore Diameters for Thread Milling (Con't)

Pitch mm		0.5	0.6	0.7	0.75 0.80	0.9	1.0	1.25	1.5	1.75	2.0		2.5	3.0	3.5	4.0	4.5	5.0	5.5		6.0	
Pitch tpi		48	44	36	32	28	26 24	20 19	18 16	14	13 12	11.5 11	10	9 8	7	6		5		4.5		4
Toolholder Ordering Code	D2	Minimum Bore Diameter Di mm																				
TM2C 40-6B	52.0								56.0	56.2	56.5	57.0	59.0	61.5		63.0	64.0	66.0	67.0	67.6	69.0	70.0
TMSH D63-22-3B	63.0	63.5	63.7	63.9	64.0	64.6	65.0	66.0	67.0	67.5	68.0	69.0										
TMSH D63-22-5	63.0	63.5	63.7	63.9	64.0	64.6	65.0	66.0	67.0	67.5	68.0	69.0	70.0	72.0	73.0	74.0	75.0					
TMSH D63-22-6B	63.0								67.0	67.5	68.0	69.0	70.0	72.0	73.0	74.0	75.0	77.0	78.0	78.6	80.0	81.0
TMSH D80-27-5	80.0	80.5	80.7	80.9	81.0	81.6	82.0	83.0	84.0	84.5	85.0	86.0	87.0	89.0	90.0	91.0	92.0					
TMSH D80-27-6B	80.0								84.0	84.5	85.0	86.0	87.0	89.0	90.0	91.0	92.0	94.0	95.0	95.6	97.0	98.0
TMSH D100-32-5	100.0	100.5	100.7	100.9	101.0	101.6	102.0	103.0	104.0	104.5	105.0	106.0	107.0	109.0	110.0	111.0	112.0					
TMSH D100-32-6B	100.0								104.0	104.5	105.0	106.0	107.0	109.0	110.0	111.0	112.0	114.0	115.0	115.6	117.0	118.0
TMSH D125-40-5	125.0	125.5	125.7	125.9	126.0	126.6	127.0	128.0	129.0	129.5	130.0	131.0	132.0	134.0	135.0	136.0	137.0					
TMSH D125-40-6B	125.0								129.0	129.5	130.0	131.0	132.0	134.0	135.0	136.0	137.0	139.0	140.0	140.6	142.0	143.0

Coarse Pitch Tooling:

This table is not applicable to the Coarse Pitch system, which can thread mill bores smaller than those listed above. See the Coarse Pitch section of the various thread standards.

Troubleshooting

	Problem	Possible Cause	Solution
	**Increased insert flank wear**	Cutting speed too high Chip is too thin Insufficient coolant	Reduce cutting speed/use coated insert Increase feed rate Increase coolant flow rate
	**Chipping of cutting edge**	Chip is too thick Vibration	Reduce feed rate/ Use the tangential arc method/ Increase RPM Check stability
	**Material Build up on the Cutting edge**	Incorrect cutting speed Unsuitable carbide grade	Change cutting speed Use a coated carbide grade
	**Chatter / Vibration**	Feed rate is too high Profile is too deep Thread length is too long	Reduce the feed Execute two passes, each with increased cutting depth/ Execute two passes, each cutting only half the thread length Execute two passes, each cutting only half the thread length
	**Insufficient thread accuracy**	Tool deflection	Reduce feed rate/ Execute a “zero” cut