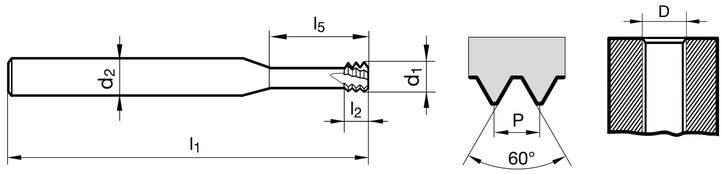
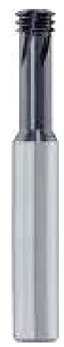
Micro-thread milling cutters

3xD

Tool material	Solid carbide
Surface	A
Type	MTMH3 SP
Internal cooling	
Shank form	HA

P		Cutting data page 60
M		
K		
N		
S	○	
H	•	for hard machining 45-65 HRC





Company std. Article no. **4227**

D	P	d1	d2	l1	l2	l5	Z	Code no.	Availability
	mm	mm	mm	mm	mm	mm			
M2	0.400	1.550	3.000	39.000	1.200	6.000	4	2.000	●
M2,5	0.450	1.950	3.000	39.000	1.400	7.500	4	2.500	●
M3	0.500	2.350	6.000	58.000	1.500	9.500	4	3.000	●
M4	0.700	3.100	6.000	58.000	2.100	12.500	4	4.000	●
M5	0.800	3.800	6.000	58.000	2.400	16.000	4	5.000	●
M6	1.000	4.800	6.000	58.000	3.000	20.000	4	6.000	●
M8	1.250	5.950	6.000	58.000	3.800	24.000	4	8.000	●
M10	1.500	7.800	8.000	64.000	4.500	23.000	4	10.000	●
M12	1.750	9.000	10.000	73.000	5.300	26.000	5	12.000	●

Recommendations for Thread Milling Cutters

Material Group	Hardness		SFM (in/min)	Feed Rate (inch/tooth) for Thread Mill Diameter						
	Rc	Brn		#10	1/4”	3/8”	1/2”	5/8”	3/4”	1”
Structural Steels		<180	260-400	0.0008	0.0015	0.0020	0.0030	0.0035	0.0035	0.0040
Free cutting steels		<180	260-400	0.0008	0.0015	0.0020	0.0030	0.0035	0.0035	0.0040
Unalloyed case hardened steels	<20	<230	260-400	0.0008	0.0015	0.0020	0.0030	0.0035	0.0035	0.0040
Unalloyed heat-treatable steels	<25	<250	260-400	0.0008	0.0015	0.0020	0.0030	0.0035	0.0035	0.0040
Alloyed case hardened steels	<25	<250	200-260	0.0004	0.0008	0.0012	0.0016	0.0020	0.0025	0.0030
Alloyed heat-treatable steels	<30	<280	200-260	0.0004	0.0008	0.0012	0.0016	0.0020	0.0025	0.0030
Alloyed tool steels	<35	<320	200-260	0.0004	0.0008	0.0012	0.0016	0.0020	0.0025	0.0030
High speed tool steels	<38	<380	200-260	0.0004	0.0008	0.0012	0.0016	0.0020	0.0025	0.0030
Hardened Steel (55RHC Max)	<55	<560	130-170	0.0003	0.0006	0.0009	0.0012	0.0016	0.0020	0.0024
Stainless -- sulpheric		<180	160-230	0.0008	0.0015	0.0020	0.0030	0.0035	0.0035	0.0040
Stainless - austenitic	<25	<250	160-230	0.0008	0.0015	0.0020	0.0030	0.0035	0.0035	0.0040
Stainless - martensitic	<30	<280	130-230	0.0008	0.0015	0.0020	0.0030	0.0035	0.0035	0.0040
Structural Steels	<20	<230	260-330	0.0008	0.0015	0.0020	0.0030	0.0035	0.0035	0.0040
Case hardened steels	<25	<250	260-330	0.0008	0.0015	0.0020	0.0030	0.0035	0.0035	0.0040
Heat-treatable steels	<25	<250	260-330	0.0008	0.0015	0.0020	0.0030	0.0035	0.0035	0.0040
Nitriding steels	<30	<280	260-330	0.0008	0.0015	0.0020	0.0030	0.0035	0.0035	0.0040
Al wrought alloys		<150	400-750	0.0020	0.0030	0.0040	0.0050	0.0060	0.0070	0.0080
Al cast alloys <10% Si			500-1000	0.0020	0.0030	0.0040	0.0050	0.0060	0.0070	0.0080
Al cast alloys >10% Si			330-660	0.0020	0.0030	0.0040	0.0050	0.0060	0.0070	0.0080
Cast iron - Grey	<25	<240	330-500	0.0020	0.0030	0.0040	0.0050	0.0050	0.0060	0.0060
Cast iron - ductile (alloyed)	<30	<280	260-400	0.0020	0.0030	0.0040	0.0050	0.0050	0.0060	0.0060
Cast iron - malleable	<32	<300	260-400	0.0020	0.0030	0.0040	0.0050	0.0050	0.0060	0.0060
Non-ferrous metals, copper alloys		<180	260-500	0.0020	0.0028	0.0035	0.0045	0.0065	0.0080	0.0100
Brass, short-chipping		<180	200-300	0.0020	0.0028	0.0035	0.0045	0.0065	0.0080	0.0100
Plastics, Carbon/glass reinforced			330-660	0.0020	0.0028	0.0035	0.0045	0.0065	0.0080	0.0100
Titanium and Ti-alloys	<35	<320	130-200	0.0004	0.0008	0.0012	0.0016	0.0020	0.0025	0.0030
Ni-alloys	<35	<320	130-200	0.0004	0.0008	0.0012	0.0016	0.0020	0.0025	0.0030
	<55	<560	130-170	0.0003	0.0006	0.0009	0.0012	0.0016	0.0020	0.0024

Drilling Parameters for DTMC Thread Mills

Material Group	Hardness		SFM (in/min)	Feed Rate (inch/tooth) for Thread Mill Diameter				
	Rc	Brn		#10	1/4”	3/8”	1/2”	5/8”
Al wrought alloys		<150	400-750	0.0020	0.0040	0.0060	0.0080	0.0100
Al cast alloys <10% Si			500-1000	0.0020	0.0060	0.0090	0.0120	0.0160
Al cast alloys >10% Si			330-660	0.0020	0.0040	0.0070	0.0100	0.0120
Cast iron - Grey	<25	<240	330-500	0.0020	0.0040	0.0060	0.0080	0.0100
Cast iron - ductile (alloyed)	<30	<280	260-400	0.0020	0.0030	0.0040	0.0060	0.0080
Cast iron - malleable	<32	<300	260-400	0.0020	0.0030	0.0040	0.0060	0.0080
Non-ferrous metals, copper alloys		<180	260-500	0.0020	0.0060	0.0100	0.0120	0.0160
Brass, short-chipping		<180	200-300	0.0020	0.0040	0.0060	0.0080	0.0100
Plastics, Carbon/glass reinforced			330-660	0.0020	0.0060	0.0090	0.0120	0.0160

Recommendations for Micro-Thread Milling Cutters

Material Group	Hardness Rc	Hardness Brn	SFM (in/min)	Feed Rate (inch/tooth) for Thread Mill Diameter 1 mm	1.5 mm	2 mm	3 mm	4 mm	5 mm	6 mm	7 mm	8 mm	9 mm	10 mm	12 mm	14 mm
Structural Steels		<180	230-400	0.0015	0.0015	0.0019	0.0019	0.0024	0.0024	0.0028	0.0028	0.0031	0.0035	0.0035	0.0040	0.0047
Free cutting steels		<180	230-400	0.0015	0.0015	0.0019	0.0019	0.0024	0.0024	0.0028	0.0028	0.0031	0.0035	0.0035	0.0040	0.0047
Unalloyed case hardened steels	<20	<230	230-400	0.0015	0.0015	0.0019	0.0019	0.0024	0.0024	0.0028	0.0028	0.0031	0.0035	0.0035	0.0040	0.0047
Unalloyed heat-treatable steels	<25	<250	230-400	0.0015	0.0015	0.0019	0.0019	0.0024	0.0024	0.0028	0.0028	0.0031	0.0035	0.0035	0.0040	0.0047
Alloyed case hardened steels	<25	<250	200-300	0.0012	0.0012	0.0015	0.0015	0.0019	0.0019	0.0024	0.0024	0.0028	0.0028	0.0031	0.0035	0.0040
Alloyed heat-treatable steels	<30	<280	200-300	0.0012	0.0012	0.0015	0.0015	0.0019	0.0019	0.0024	0.0024	0.0028	0.0028	0.0031	0.0035	0.0040
Alloyed tool steels	<35	<320	200-300	0.0012	0.0012	0.0015	0.0015	0.0019	0.0019	0.0024	0.0024	0.0028	0.0028	0.0031	0.0035	0.0040
High speed tool steels	<38	<380	200-260	0.0010	0.0010	0.0013	0.0013	0.0016	0.0016	0.0020	0.0020	0.0024	0.0024	0.0026	0.0030	0.0035
Hardened Steel (55RHC Max)	<55	<560	130-170	0.0004	0.0007	0.0007	0.0012	0.0012	0.0014	0.0014	0.0015	0.0018	0.0018	0.0019	0.0019	0.0024
Stainless -- sulpheric		<180	130-260	0.0008	0.0008	0.0008	0.0012	0.0012	0.0015	0.0019	0.0019	0.0019	0.0024	0.0024	0.0024	0.0028
Stainless - austenitic	<25	<250	130-260	0.0008	0.0008	0.0008	0.0012	0.0012	0.0015	0.0019	0.0019	0.0019	0.0024	0.0024	0.0024	0.0028
Stainless - martensitic	<30	<280	130-260	0.0008	0.0012	0.0012	0.0015	0.0015	0.0019	0.0019	0.0019	0.0024	0.0028	0.0028	0.0031	0.0035
Structural Steels	<20	<230	230-400	0.0015	0.0015	0.0019	0.0019	0.0024	0.0024	0.0028	0.0028	0.0031	0.0035	0.0035	0.0040	0.0047
Case hardened steels	<25	<250	200-300	0.0015	0.0015	0.0019	0.0019	0.0024	0.0024	0.0028	0.0028	0.0031	0.0035	0.0035	0.0040	0.0047
Heat-treatable steels	<25	<250	230-400	0.0012	0.0012	0.0015	0.0015	0.0019	0.0019	0.0024	0.0024	0.0028	0.0028	0.0031	0.0035	0.0040
Nitriding steels	<30	<280	200-300	0.0012	0.0012	0.0015	0.0015	0.0019	0.0019	0.0024	0.0024	0.0028	0.0028	0.0031	0.0035	0.0040
Al wrought alloys		<150	260-500	0.0015	0.0019	0.0019	0.0024	0.0024	0.0028	0.0028	0.0031	0.0035	0.0040	0.0043	0.0047	0.0055
Al cast alloys <10% Si			500-1000	0.0015	0.0019	0.0019	0.0024	0.0024	0.0028	0.0028	0.0031	0.0035	0.0040	0.0043	0.0047	0.0055
Al cast alloys >10% Si			330-650	0.0015	0.0015	0.0019	0.0019	0.0024	0.0024	0.0028	0.0028	0.0031	0.0035	0.0035	0.0040	0.0047
Cast iron - Grey	<25	<250	325-500	0.0015	0.0015	0.0019	0.0019	0.0024	0.0024	0.0028	0.0028	0.0031	0.0035	0.0035	0.0040	0.0047
Cast iron - ductile (alloyed)	<30	<280	260-500	0.0015	0.0015	0.0019	0.0019	0.0024	0.0024	0.0028	0.0028	0.0031	0.0035	0.0035	0.0040	0.0047
Cast iron - malleable	<35	<320	260-500	0.0015	0.0015	0.0019	0.0019	0.0024	0.0024	0.0028	0.0028	0.0031	0.0035	0.0035	0.0040	0.0047
Non-ferrous metals, copper alloys		<180	260-500	0.0015	0.0019	0.0019	0.0024	0.0024	0.0028	0.0028	0.0031	0.0035	0.0040	0.0043	0.0047	0.0055
Brass, short-chipping		<180	200-300	0.0015	0.0019	0.0019	0.0024	0.0024	0.0028	0.0028	0.0031	0.0035	0.0040	0.0043	0.0047	0.0055
Plastics, Carbon/glass reinforced			200-650	0.0019	0.0019	0.0024	0.0028	0.0028	0.0031	0.0035	0.0035	0.0040	0.0043	0.0047	0.0051	0.0059
Titanium and Ti-alloys		140-300	65-130	0.0007	0.0007	0.0007	0.0012	0.0012	0.0015	0.0019	0.0019	0.0019	0.0024	0.0024	0.0024	0.0028
		300-380	200-300	0.0012	0.0012	0.0015	0.0015	0.0019	0.0019	0.0024	0.0024	0.0028	0.0028	0.0031	0.0035	0.0040
Ni-alloys	<32	<300	200-300	0.0012	0.0012	0.0015	0.0015	0.0019	0.0019	0.0024	0.0024	0.0028	0.0028	0.0031	0.0035	0.0040
	<55	<560	130-170	0.0004	0.0007	0.0007	0.0012	0.0012	0.0014	0.0014	0.0015	0.0018	0.0018	0.0019	0.0019	0.0024

Thread production with thread milling cutters and their advantages

Thread milling is like thread cutting in that it is a chip forming production process. A major benefit of thread milling is the size of the thread to be produced, as the cost of taps in large dimensions can question the economic efficiency of the production process. Furthermore, with larger diameters thread cutting requires more power from the machine tool. The possibility of machining materials with a higher tensile strength or hardness can also be a decisive factor in choosing thread milling.

With conventional thread cutting the thread is produced from the image of the tool profile, conditional on the chamfer lead for the workpiece. In contrast, with thread milling the thread is produced via a series of cutting paths by the milling cutter, whereby the pitch is generated by the machine. The thread mill makes numerous passes through the axis section of a thread during a spiral motion in axial tool-workpiece direction and thereby interpolates the contour of the thread.

A major factor is that cutting speeds and feed rates can be chosen independently of each other. Chip formation and tool loading can be considerably influenced via these setting parameters. A feature of the process is the formation of short, comma shaped chips in contrast to thread cutting.

To detach the chips, the direction of rotation of the machine spindle does not need to be reversed. The tools applied have a thread profile without pitch. Initially, the thread milling cutter is lowered along the hole axis to the required thread depth. In the approach cycle the thread milling cutter is plunged to the nominal diameter of the thread.

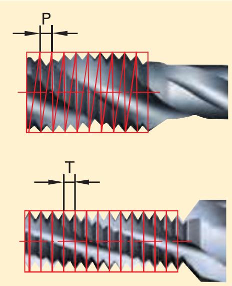
The thread is produced by a 360° circular interpolation. Finally, in the exit cycle the thread milling cutter is radially returned to the hole axis and then axially withdrawn from the thread.

A thread milling cutter can produce threads of varying diameters (or tolerances) with the same pitch. Right- or left-handed threads can be produced with the one tool. As thread milling produces only short chips, chip evacuation does not cause any problems.

With thread milling, an identical tool geometry can be applied for a multitude of materials. This considerably limits the multiplicity of tools required. In contrast to thread cutting, thread milling produces a complete thread which is virtually the total length of the applied tool.

Comparison of tool design between taps/thread milling cutters

In contrast to a tap, which basically consists of a single spiral shaped tooth, the series of teeth of a thread milling cutter do not form a spiral but are configured without pitch. This fundamental difference in tool design is attributed to the different processes which have already been described in an earlier chapter.



Tap
The pitch P is produced by the tap/fluteless tap.

Milling cutter
The pitch P is produced by CNC control circular program)

Dimensions and cutting section measurements

Apart from the thread pitch of the tool, the design of a thread milling cutter is principally very similar to that of a tap. Thread milling cutters are also characterized by dimensions and the size of the cutting section. The thread length l_2 and the total length l_1 are also part of the dimensions.

The different design forms incorporate milling cutters with or without collar as well as with or without countersinking chamfer. The cutting section sizes of a thread milling cutter consist of the flute length l_4, the flute profile, the tooth with Z_b and the relief S. As with a tap, the flute length also incorporates the run-out of the flutes. They do not have to be as large as the flutes of a tap, as this machining process produces smaller chips. The chips do not remain in the flutes during the process and do therefore not restrict further chip development. The width of the tooth is therefore larger than with taps. The relief grinding helps to create the clearance angle required for milling cutters.

Thread milling process and technology

Machining combinations (reverse rotation/synchr. milling)

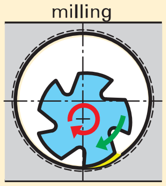
As the thread milling cutters are designed for right hand cutting, the direction of rotation is generally clockwise. By altering the axial direction of feed, reverse rotation or synchronous milling, all thread combinations can be produced.

The application conditions, such as blind or through hole, horizontal or vertical machining methods as well as the type of lubrication and chip removal determine the correct choice of milling procedure.

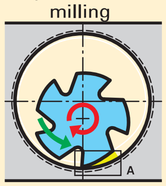
For thread milling, synchronous milling should be applied whenever possible, in order to achieve lower cutting forces, improved chip formation, a better tool life and surface quality.

Synchronous milling can be identified when the cutting edge emerges with chip thickness h = 0.

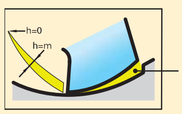
reverse rotation milling

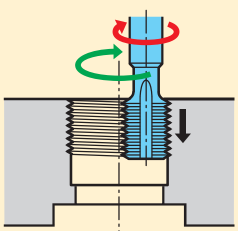


synchronous milling

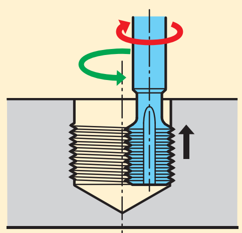


Synchronous thread milling can be identified when the cutting edge emerges with a chip thickness h=0

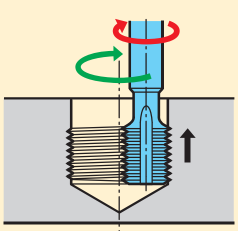




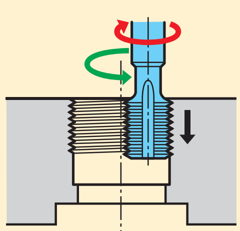
RH cutting in through hole: reverse rotation clockwise circle, downward pitch



RH cutting in blind hole: synchronous rotation anticlockwise circle, upward pitch



LH cutting in blind hole: reverse rotation clockwise circle, upward pitch



LH cutting in through hole: synchronous rotation anticlockwise circle, downward pitch

thread milling cutter direction of rotation
axial direction of feed
pitch

Interference and feed ratio

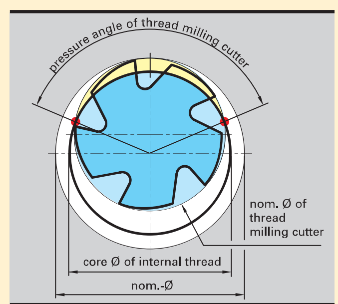
If the milling cutter diameter to the nominal thread diameter ratio of 70° is adhered to, a profile distortion, irrespective of the profile depth of the thread, should not occur. This factor is well proven.

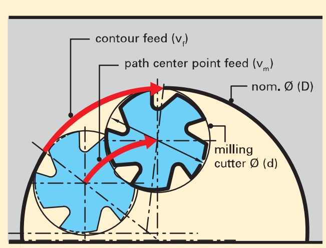
This drawing illustrates that the diameter of the thread milling cutter and the profile depth determine the pressure angle to the thread diameter.

The feed at the cutting edge of the thread milling cutter is calculated by the cutting speed revolutions) and the feed rate per tooth.

With linear movement, the feed rate at the cutting edge is identical to that at the tool center. However, the helical interpolation follows the path of a circle in the plane. As the machine tool always calculates to the tool center, a command must be programmed for converting the cutting speed (contour related program). If such a command does not exist or the central point is programmed, the feed rate must be first converted.

The interactive control at the control panel always indicates the speed at the center point of the tool. When running with no load this is simple to check. If disregarded, the milling cutter runs at a speed many times faster than the feed which generally leads to tool breakages.





Formula of calculation

$$v_c = \frac{d \cdot \pi \cdot n}{1000} \quad [\text{m/min}]$$

$$n = \frac{v_c \cdot 1000}{d \cdot \pi} \quad [\text{min}^{-1}]$$

$$v_f = n \cdot z \cdot f_z \quad [\text{mm/min}]$$

$$v_m = \frac{v_f \cdot (D - d)}{D} \quad [\text{mm/min}]$$

$$v_b = n \cdot f_b \quad [\text{mm/min}]$$

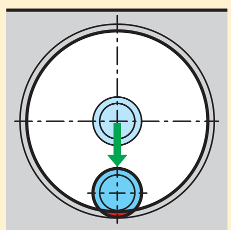
v_c = cutting speed
v_f = contour feed
v_m = center point path feed
n = revolutions
z = number of teeth
f_z = feed per tooth
f_b = feed for drill per revolution*
v_b = drill feed rate*
D = Ø nom. of thread [mm]
d = milling cutter nom. Ø [mm]
* for drill/thread milling

Thread milling process and technology

Thread milling cutter entry cycles

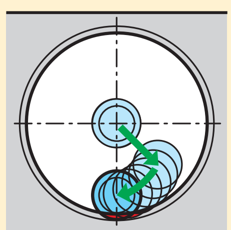
Linear plunging

With linear plunging of the thread milling cutter into the material, a very large angle of contact is created at the milling cutter which leads to long chips and a high loading on the tool. This is particularly the case when the difference in diameter between the hole size and the milling cutter is small. In addition, this method produces a small delay mark. This method is not suitable for accurate and small threads.



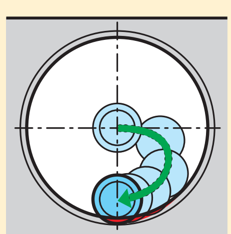
90° quarter circle entry cycle

A 90° entry cycle with a small difference in diameter between the tool and the thread removes a large part of chip volume during the linear section of the entry cycle. This method is therefore only recommended for relatively large differences in diameter between hole size and thread milling cutter (thread milling cutter TMU). The advantage using this entry method lies in the simple programming and the relatively short entry path.



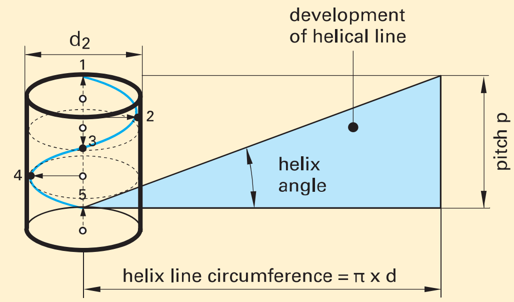
180° semicircle entry cycle

With a 180° entry cycle, the loading on the tool is the lowest when plunging, as the angle of contact is relatively small during the complete entry cycle. This method requires a little more sophistication in programming but has shown to be the most costefficient when thread milling with the TM, TMC and DTMC thread milling cutter.



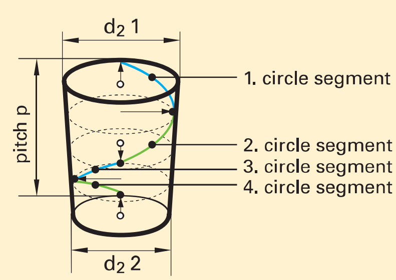
Helical interpolation (cyl. thread)

Helical interpolation is the overlaying of circular and linear movement. Different threads can be produced by the form of overlaying the direction of pitch and the direction of rotation of the circular movement.



Helical interpolation (conical thread)

In order to produce a perfectly round thread with a NPT thread milling cutter, it is necessary to take into account the pitch when NC-progamming. In contrast to cylindrical threads, the machining path is not a 360° circle but four segments of a circle. With every one of the four segments the taper is corrected inwards.



Technical

Thread milling cutters w/o countersinking step Type TM SP

Machine example

Coating:	TiCN
Thread:	M12
Pitch:	1.75 mm
Thread depth:	24 mm / 2 x D

Tool material:	Steel
Cutting speed:	100 m/min
Feed per tooth:	0.08 mm
Cutting time:	2.7 s

Programming example:

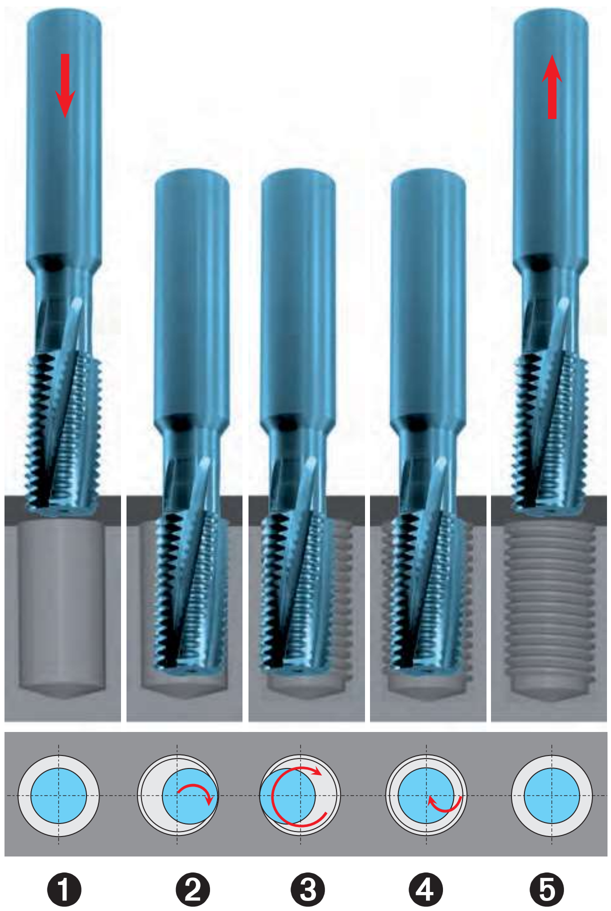
	CNC Code:	Plain text
	N10 M6T1	Tool call
	N20 G90 G54 G00 X0.000 Y0.000	Work offset
❶	N30 Z2.000 S3199 M3 D1	Positioning centered on start position above tapping size hole and spindle speed call-up
	N40 G00 Z-21.725	Rapid movement to thread milling start position centered in tapping size hole
	N50 G91	Switch to incremental
	N60 G42 G01 X0.000 Y4.975 F1000	Cutter radius compensation on
❷	N70 G02 X0.000 Y-10.975 I0.000 J-5.488 Z-0.263 F87	180° entry cycle to profile depth, start thread milling process
❸	N80 G02 X0.000 Y0.000 I0.000 J6.000 Z-1.750 F175	360° thread milling cycle with axial movement of the thread pitch in Z-direction
❹	N90 G02 X0.000 Y10.975 I0.000 J5.488 Z-0.263 F350	180° withdrawal cycle to the thread center, end of thread milling
	N100 G40 G01 X0.000 Y-4.975 F1000	Cutter radius compensation off
	N110 G90	Switch to absolute
❺	N120 G80 G53 G00 Z2.000	Withdrawal from hole to start position centered above tapping size hole
	N130 M30 M95	End

Universal thread milling cutter
Type TMU SP - 1 milling cycle

Machine example

Coating:	bright
Thread:	M24
Pitch:	1.5 mm
Thread depth:	24 mm / M16x1.5

Tool material:	Cast Aluminum
Cutting speed:	220 m/min
Feed per tooth:	0.15 mm
Cutting time:	1.7 s



Programming example:

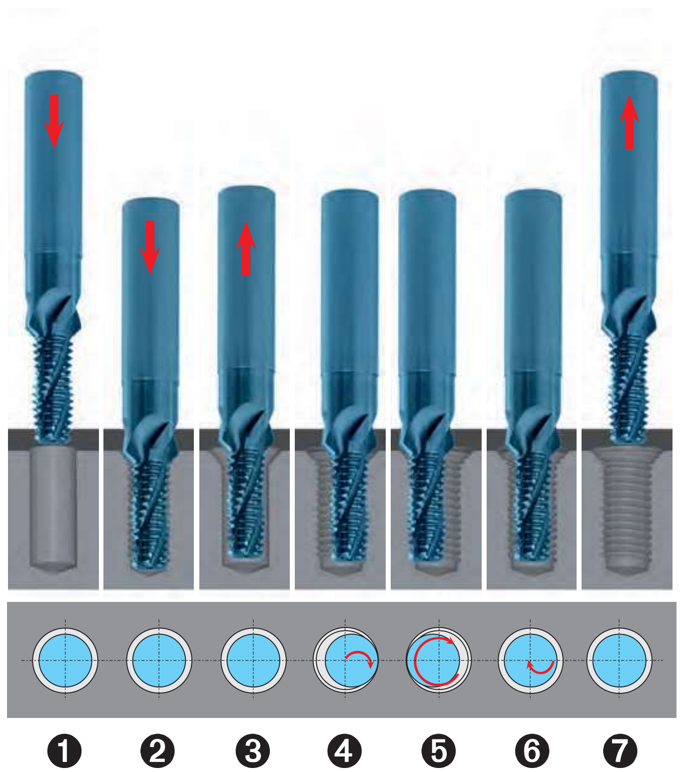
	CNC Code:	Plain text
	N10 M6 T1	Tool call
	N20 G90 G54 G00 X0.000 Y0.000	Work offset
❶	N30 Z2.000 S3199 M3 D1	Positioning centered on start position above tapping size hole and spindle speed call-up
	N40 G00 Z-21.725	Rapid movement to thread milling start position centered in tapping size hole
	N50 G91	Switch to incremental
	N60 G42 G01 X0.000 Y4.975 F1000	Cutter radius compensation on
❷	N70 G02 X0.000 Y-10.975 I0.000 J-5.488 Z-0.263 F87	180° entry cycle, start of thread milling
❸	N80 G02 X0.000 Y0.000 I0.000 J6.000 Z-1.750 F175	360° thread milling cycle with axial movement of the thread pitch in Z-direction
❹	N90 G02 X0.000 Y10.975 I0.000 J5.488 Z-0.263 F350	180° withdrawal cycle to the thread center, end of thread milling
	N100 G40 G01 X0.000 Y-4.975 F1000	Cutter radius compensation off
	N110 G90	Switch to absolute
❺	N120 G80 G53 G00 Z2.000	Withdrawal from hole to start position centered above tapping size hole
	N130 M30 M95	End

Thread milling cutters with countersinking step Type TMC SP

Machine example

Coating:	TiCN
Thread:	M16
Pitch:	1.5 mm
Thread depth:	40 mm / M16x1.5

Tool material:	5120 Alloyed Steel
Cutting speed:	100 m/min
Feed per tooth:	0.06 mm
Cutting time:	6.4 s



Programming example:

	CNC Code:	Plain text
	N10 M6T1	Tool call
	N20 G90 G54 G00 X0.000Y0.000	Work offset
❶	N30 Z2.000 S497 M3 D1	Positioning centered on start position above tapping size hole and spindle speed call-up
	N40 G00 X0.000Y0.000 Z-41.300	Rapid movement to countersinking start position
❷	N50 G01 X0.000Y0.000 Z-43.200 F119	Countersinking of 90° chamfer
❸	N60 G00 Z-38.050 S2487	Rapid movement to thread milling start position centered in tapping size hole
	N70 G91	Switch to incremental
	N80 G42 G01 X0.000Y6.400 F1000	Cutter radius compensation on
❹	N90 G02 X0.000Y-14.400 I0.000 J-7.200 Z-0.225 F60	180° entry cycle, start of thread milling
❺	N100 G02 X0.000Y0.000 I0.000 J8.000 Z-1.500 F119	360° thread milling cycle with axial movement of the thread pitch in Z-direction
❻	N110 G02 X0.000Y14.400 I0.000 J7.200 Z-0.225 F239	180° withdrawal cycle to the thread center, end of thread milling
	N120 G40 G01 X0.000Y-6.400 F1000	Cutter radius compensation off
	N130 G90	Switch to absolute
❼	N140 G80 G53 G00 Z2.000	Withdrawal from hole to start position centered above tapping size hole
	N150 M30 M95	End

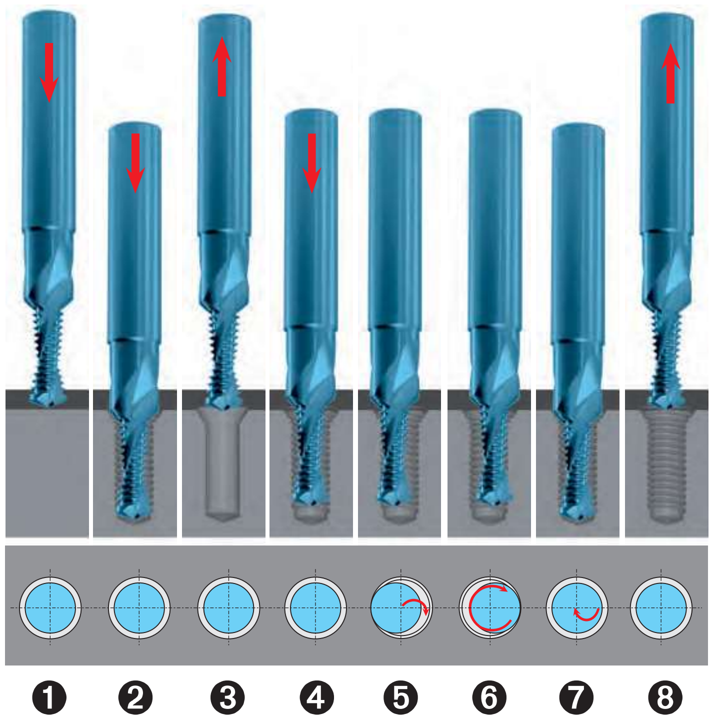
Technical

Drill/thread milling cutter
Type DTMC SP

Machine example

Coating:	bright
Thread:	M8
Pitch:	1.25 mm
Thread depth:	16 mm / 2 x D

Tool material:	Cast Iron
Cutting speed:	100 m/min
Feed per tooth:	0.06 mm
Cutting time:	5.3 s



Programming example:

	CNC Code:	Plain text
	N10 M6 T1	Tool call
	N20 G90 G54 G00 X0.000 Y0.000	Work offset
❶	N30 Z2.000 S5013 M3 D1	Positioning centered on start position above tapping size hole and spindle speed call-up
	N40 G01 X0.000 Y0.000 Z-1.000 F251	Centering at half the feed rate
❷	N50 X0.000 Y0.000 Z-19.825 F501	Drilling the tapping size hole and countersinking 90° chamfer
❸	N60 G00 X0.000 Y0.000 Z0.000 S5013	Withdrawal of tool from the hole for pecking
❹	N70 Z-14.375	Rapid movement to thread milling start position centered in tapping size hole
	N80 G91	Switch to incremental
	N90 G42 G01 X0.000 Y3.175 F1000	Cutter radius compensation on
❺	N100 G02 X0.000 Y-7.175 I0.000 J-3.588 Z-0.188 F62	180° entry cycle, start of thread milling
❻	N110 G02 X0.000 Y0.000 I0.000 J4.000 Z-1.250 F124	360° thread milling cycle with axial movement of the thread pitch in Z-direction
❼	N120 G02 X0.000 Y7.175 I0.000 J3.588 Z-0.188 F248	180° withdrawal cycle to the thread center, end of thread milling
	N130 G40 G01 X0.000 Y-3.175 F1000	Cutter radius compensation off
	N140 G90	Switch to absolute
❽	N150 G80 G53 G00 Z2.000	Withdrawal from hole to start position centered above tapping size hole
	N160 M30 M95	End

Available free of charge at www.guhring.com:

CNC THREAD MILL PROGRAM GENERATOR

You enter the information - it writes the program.

At www.GUHRING.com, under the Technical tab