Xactform

YOUR PARTNER IN THREADING AND THREAD MILLING

TECHNICAL GUIDE

YOUR EXPERIENCE & OUR KNOW-HOW: THE BEST TOOLS FOR YOUR MACHINING NEEDS!

SWISS MADE

Introduction

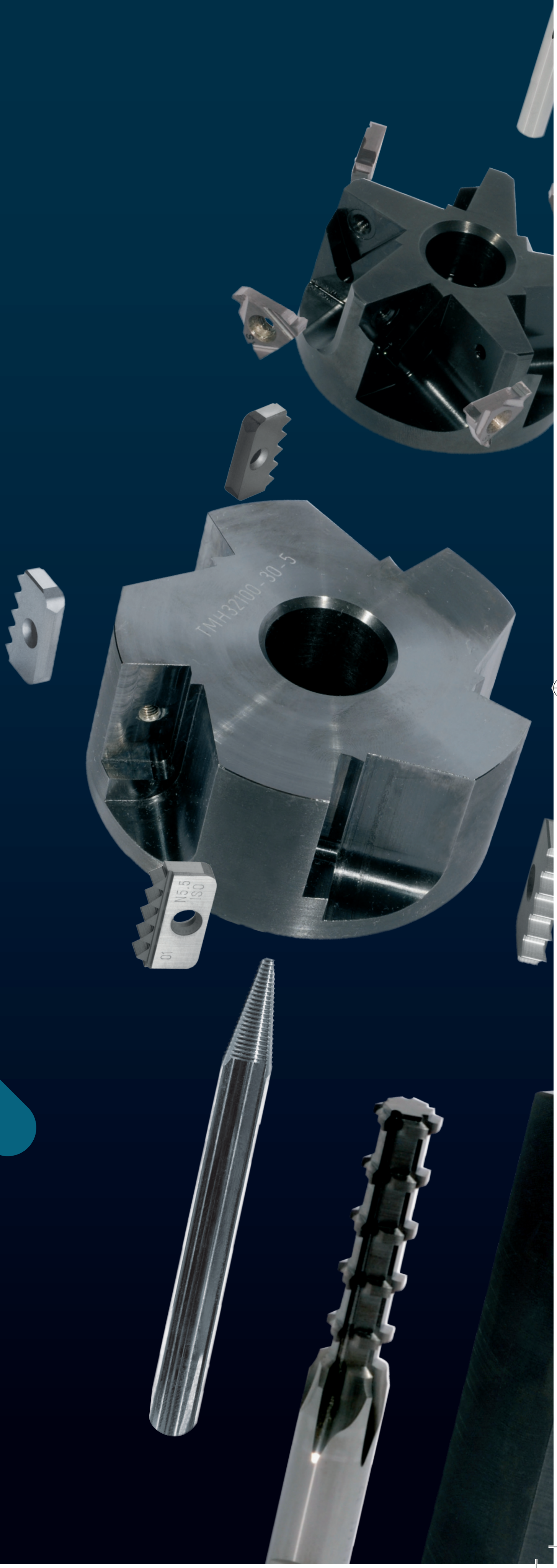
Xactform SA is a leading Swiss manufacturer of hard metal cutting tools. Specialising in machine threading, it offers both standard ranges and made to measure tools.

A pioneer in this specialised field, its expertise enables it to produce the highest-quality tools for all threading applications, able to meet the most demanding technical and economic requirements across five continents.

Purpose of the technical guide

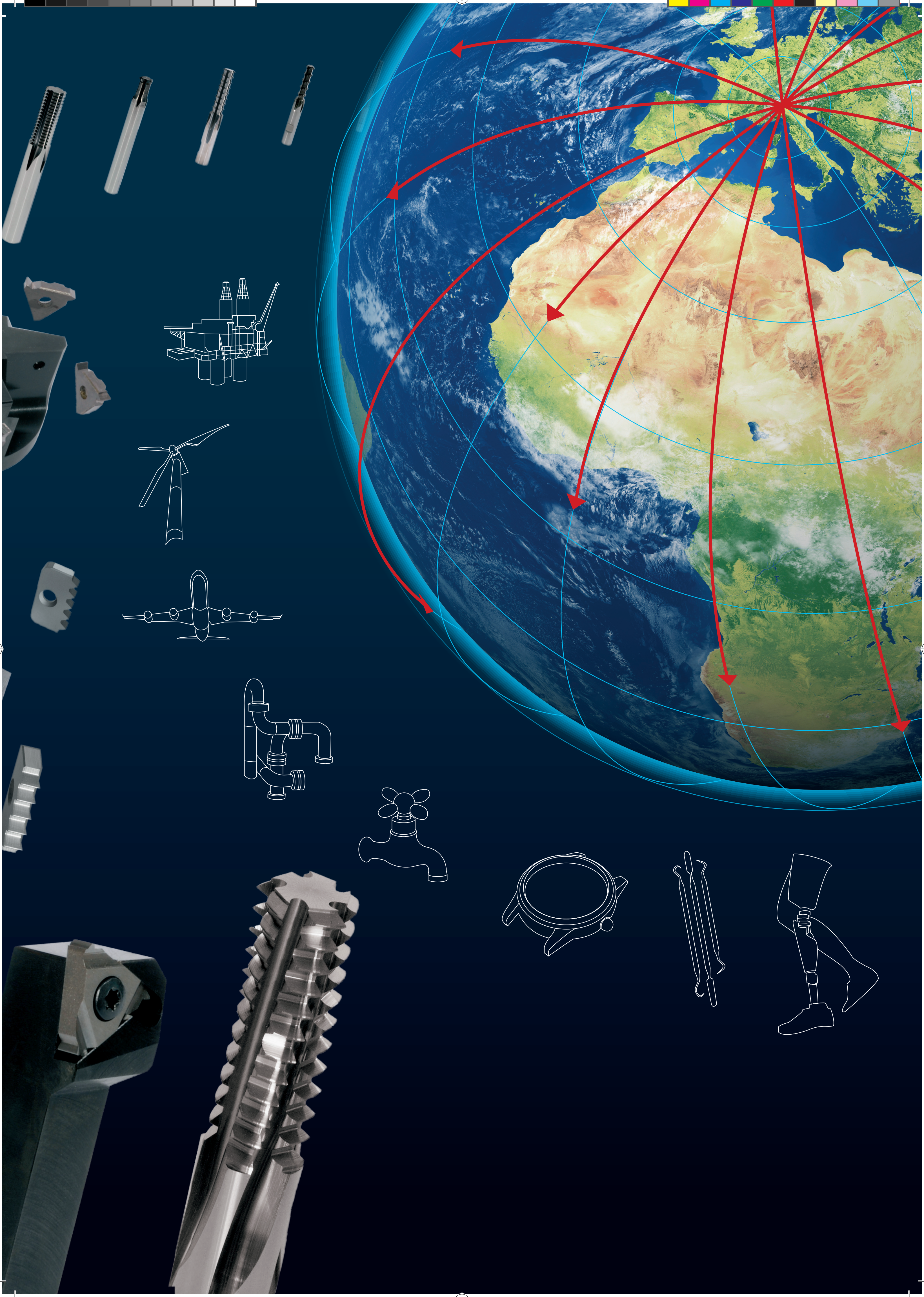
The field of thread milling is a relatively new domain of which many industrial players still have scant knowledge. The purpose of this guide is to explain its foundations and operating principles to provide users with the comprehensive understanding required to implement the technique.

Created exclusively for our customers, this brochure addresses all of the questions they may have concerning the application of the thread milling technique. The guide contains general and technical data (tool selection methods, cutting conditions, programming principles, etc.) to enable customers to get the best use out of the tools we offer.



Information and downloads

www.xactform.com



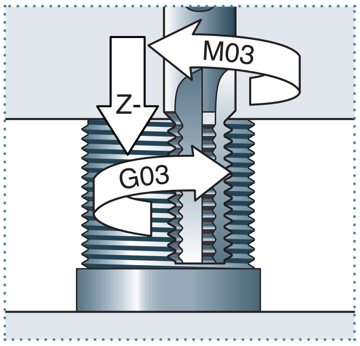
Index

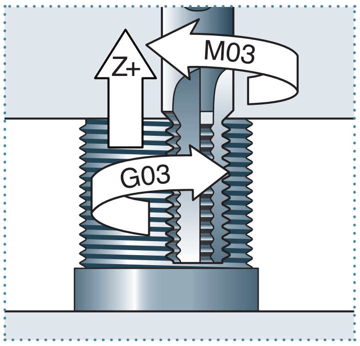
(1)

Thread milling

(1.1) Principle

The concept of thread milling was developed several years ago following the introduction of competitively-priced CNC machining centres capable of circular interpolation (helical movement in 3 simultaneous axes). This method is used to machine threads by employing the basic milling principle, and to replace the use of traditional taps, with all the associated benefits.





(1.2) Operation

Example: machining a straight internal thread. A rotating tool (thread mill) is inserted at a rapid feedrate to the required depth in a hole pre-drilled to the minor diameter of the desired thread. Via circular interpolation (simultaneous movement in 3 axes X, Y and Z), the tool penetrates the material up to a line tangential to the thread diameter, then rotates fully with simultaneous movement in the Z axis equivalent to the thread pitch. The cycle is then complete and the tool is simply withdrawn from the hole at a rapid feedrate.

In most cases, the thread is created by a single rotation along the entire working length. In certain cases, two or more rotations may be required:

- Small diameters and/or materials which are very hard to machine
- Deep thread machined using a mill with a release mechanism
- Tool holder with indexed insert on which the thread is executed in steps

Bottom to top milling is the most common type. The two thread directions – right- or left-hand – can be obtained through a simple program modification. The same applies to thread milling with multiple entry points: the thread tolerance is dependent on the accuracy of the machine, not just the tool.

(1.3) Main benefits

Reduces unit stresses on the thread
While taps produce only 65 to 75% of the standardised thread height, thread mills machine 100% of the height. The stress is therefore distributed over a larger surface area, with its unit value reduced. However, there is one prerequisite: the diameter of the pre-drilled hole must be the same as the thread minor diameter. This means that the drilling tables for tap threading cannot be used for thread milling!

Eliminates thread grooves
Precise positioning close to a shoulder or in a blind hole eliminates the need for thread grooves. This saves time, thereby increasing productivity.

Maximum positioning precision, independent of the roughing operation.
For example, this means there is no need to drill a tapered hole to machine an NPT tapered thread: a simple cylindrical hole is sufficient, as the thread mill does the rest.

Low drive power requirement
Most current CNC machining centres do not provide the necessary torque to tap large diameter threads, particularly of the tapered type. These operations must therefore often be performed on special tapping machines.
Thread milling allows all the threads to be machined using the same machine, irrespective of the diameter or pitch, with the added benefit of significantly reduced production costs.

Small dimensions available
UGV machining centres surpass taps in every way, even for cutting diameters of a few millimetres. They are more economical and enable significant increases in productivity.

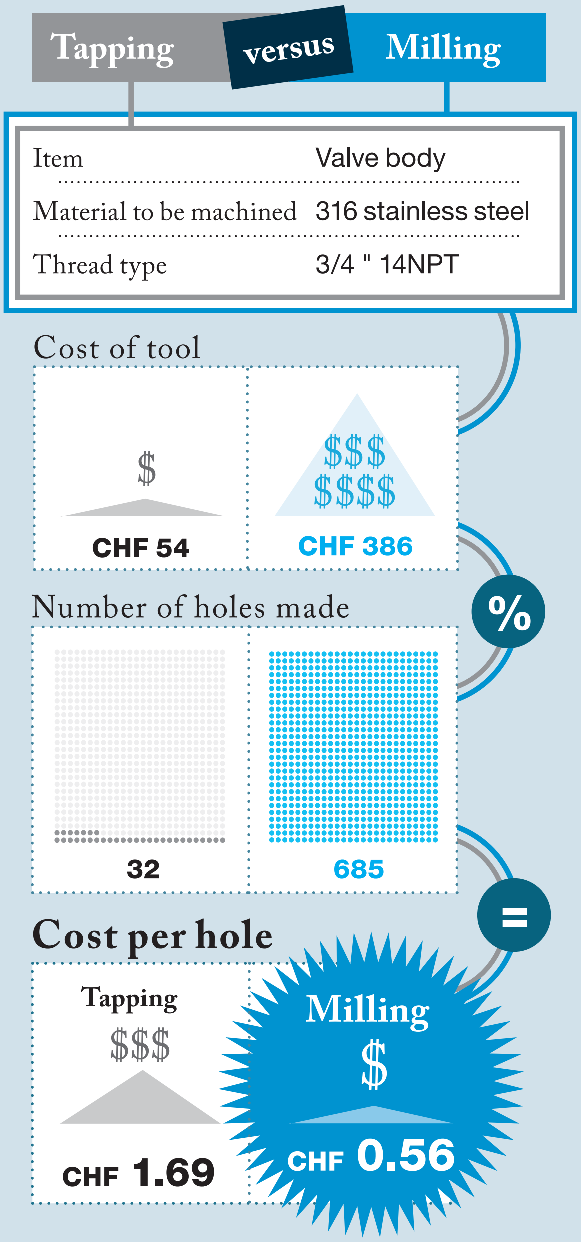
Exceptional quality finish
There is no comparison between the surface finish of threads created by tapping and milling. This means there is no need for deburring.

Significantly increased tool service life
Thread milling is a very fast process. The service life of a thread mill (number of holes machined) is much longer and more consistent than that of a tap. If the milling cutter breaks - which is very rare - the thread can be re-milled immediately. In the tapping process, time is wasted extracting the broken end of the tap from the workpiece.

Increased versatility
A single tool or tool holder can be used to machine threads of different diameters, left- or right-hand threads, threads with single or multiple entry points, and sometimes even both external and internal threads. This greatly reduces tooling costs.

Significantly increased productivity
In most cases, milling a thread is around 2 or 3 times faster than tapping. Add to this simplified preparation operations, fewer tool changes (see previous point) and increased tool service life. The result is threading operations that are often 3 or 4 times more productive than tapping operations.

COMPARISON
Thread milling and tapping



Item	Valve body
Material to be machined	316 stainless steel
Thread type	3/4 " 14NPT

(1.4) Thread standards

Threading is a very specialised field governed by numerous standards which vary depending on the area of application. Below are the most common.

The main thread standards by area of application

Standard	Designation	Area of application
Metric ISO	NF ISO 965-1 ISO 5864	General mechanical Automotive Watchmaking, etc.
UN (US standard in inches)	ISO 5864	General mechanical Automotive
NiHS	SN 280 605	Watchmaking industry
Metric MJ	DIN ISO 5855-1	Aerospace construction Aeronautical
UNJ (US standard in inches)	ISO 3161	
UNJ (US standard in inches)		Medical implants Prostheses
NPT **National Pipe Taper**	NF E29-684 ANSI/ASME B1.20.1	Pipes Tube assembly Valves, etc.
NSPM (US standard in inches) **National Pipe Straight Mechanical**	ANSI/ASME B1.20.1	
BSW (US standard in inches) **British Straight Whitworth**	DIN 6630 DIN ISO 228-1 NF E 03-005	
BSP **British Straight Pipe**	DIN 6630 DIN ISO 228-1 NF E 03-005	
BSPT **British Straight Pipe Taper**	DIN 3858 DIN ISO 7-1 NF E 03-004	
W **Whitworth**	DIN 477	Pipes Gas canisters
Rd **Rund**	DIN 405	Food manufacturing facilities
Pg **Stahlpanzerrohrgewinde**	DIN 40430	Mountings for electrical wiring Electrical units
Tr **Trapezgewinde**	DIN 103 NF ISO 2901	Guide screws Drives
API **American Petroleum Institute**	API RD API BUTTRESS	Oil pipelines
SPLK **Spirallock**		Firefighting systems Army, aerospace, automotive Self-locking threads, etc.

(1.5) Tool selection

(1.5.1) Solid carbide mills or indexed inserts: reciprocal benefits

While this is not an issue in thread turning operations, Xactform offers two ranges for thread milling applications. How do they differ and for which applications are they designed?

Xactform has sought to develop a versatile solution for sub-contracting workshops producing small runs of threads, who wish to take advantage of the thread milling process. In such cases, a complete range of solid carbide mills is difficult to keep profitable. This is where thread milling indexed inserts and tool holders come in (TM).

The principle is simple: with numerous tool holders and a range of indexed inserts with different pitches, users can mill the majority of threads in various dimensions with limited investment. In addition, the entire range of inserts can be replaced and/or enhanced at any time. However, there is one limitation: thread diameters of less than 12 mm cannot be produced with this range of tools.

For all other applications, in particular very large runs, the superior performance of solid carbide mills makes a real difference, both increasing productivity and lowering costs. These mills therefore remain the most commonly used.

Finally, for certain specific applications, in particular the oil industry, Xactform has combined thread milling and turning. It is currently the only company to offer tool holders for thread milling with triangular inserts.

(1.6) Coatings and applications

Depending on the material to be machined, the thread cutting tools may be used either coated or uncoated. The purpose of coating is to increase the tool's service life, however it can reduce the cutting quality. It therefore needs to be limited to cases where it is strictly necessary.

Other special coatings are available on request. Please contact us for more information.

The different types of coating and their usage conditions

Coating		Operating temperature (°C)		Colour	Applications
V1	TiAlN	900°C	Futura Nano	Purplish-grey	General machining
V2	AiCrN	1100°C	Alnova	Light grey	Special machining
V3	TiAlN+WC/C	800°C	Hardlube	Dark grey	
V4	TIN	700°C	Balinit A	Golden yellow	

General and special machining, see cutting conditions table (p. 16 to 19)

(2) Solid carbide range

(2.1) Listing

Reference								
TMSC	03	010	N	0.30	ISO	T2	HA	WCH

TMSC
Straight flutes

TMHE
Helical flutes

TBR/L
Whirl cutter

D
Shank diameter

D1
Working diameter

N
Internal

E
External

NE
Internal/external

Pitch (mm, inches)

Standard

T2, T3
Every other/every third tooth

2, 25
L1 = 2xD, 2.5xD

2A
L1 = 2xD + 60° chamfer

2T, 3T
2 teeth, 3 teeth

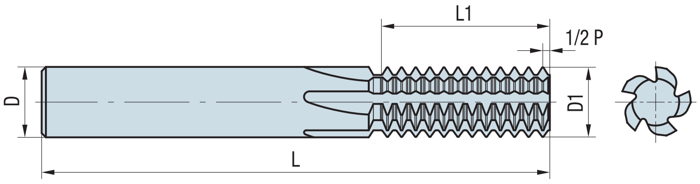
HA
Smooth cylindrical

HB
Weldon 6535-HB

HE
Weldon 6535-HE

WCH
Central lubrication

WCG
With lubrication grooves



L
Total tool length

L1
Working length

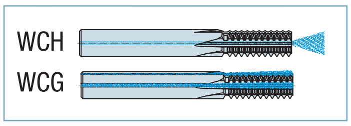
NF
Number of teeth

Z
Number of flutes

H
Profile height

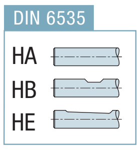
Standard	Reference	L	L1	D	D1	NF	Z	H
M8	TMSC 06050 N 1.25 ISO	57	12.500	6.00	5.00	10	3	0.734

Available with lubrication from Ø6 mm



WCH Central lubrication

WCG With lubrication grooves in the flutes



HA Smooth cylindrical

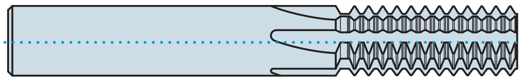
HB Weldon 6535-HB

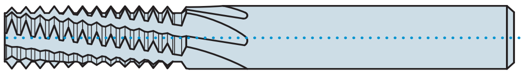
HE Weldon 6535-HE

(2.2) Tool selection

(2.2.1) Straight flutes vs. helical flutes

Most standard thread mills are available with helical or more economical straight flutes. The helix angle removes swarf in a specific direction, which means helical flute versions are recommended for machining materials susceptible to jamming. In other cases, **a tool with straight flutes guarantees superior precision of the machined thread and reduces production costs.**

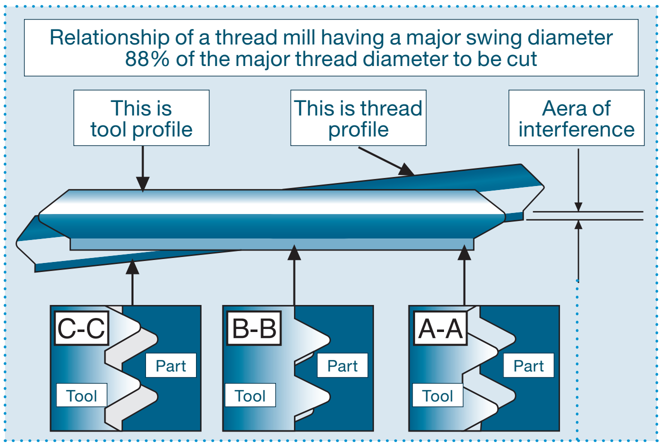


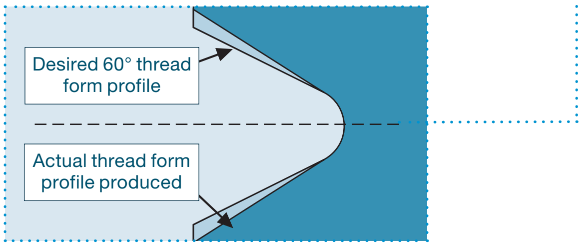


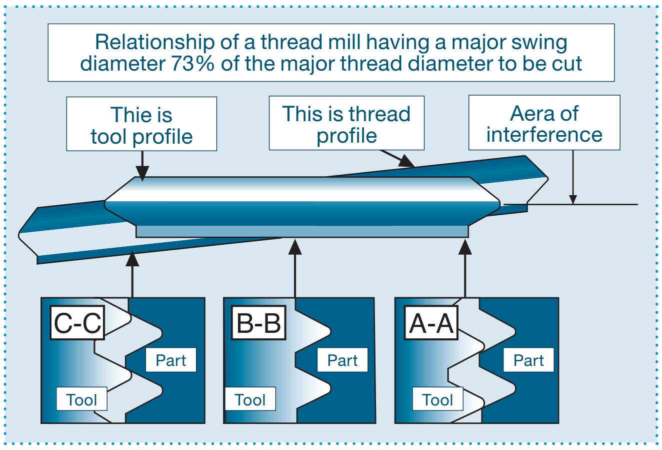
(2.2.2) Dimensions

Diameter

Mills and inserts enabling machining in line with the main thread standards are available as standard in our catalogue.
However, for non-standard diameters, the tool diameter must not in principle exceed 75% of the thread diameter to prevent ploughing. For certain special applications, this value can rise to 80%. However, if the thread's helix angle is greater than 4°, the diameter must be reduced to prevent ploughing. Please contact us for more information.

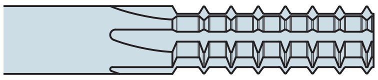




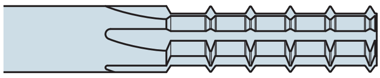


Length

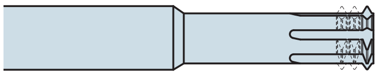
The standardised depth of a thread is 1.5 x the diameter. For most materials, it is not possible to increase this depth, as the screw head breaks before the thread.
For special applications where the depth is greater, Xactform recommends specific versions:



- **Depth 2.0 x diameter:** T2 version.
The thread is machined by two successive interpolations.



- **Depth 3.0 x diameter:** T3 version.
The thread is machined by three successive interpolations.

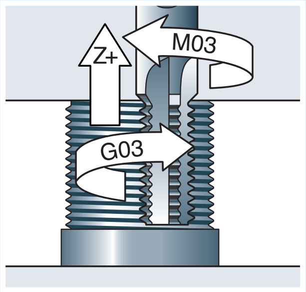


- **Depth > 3.0 x diameter:** whirl cutter.
With no more than 3 teeth, the thread is machined using multiple interpolations.

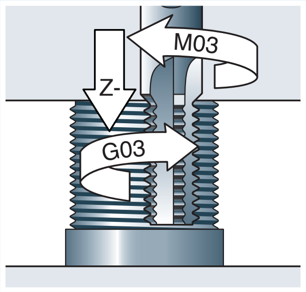
Left-hand and right-hand thread

The thread direction does not affect the choice of tool. The thread direction is obtained by circular interpolation:

bottom to top for right-hand thread.



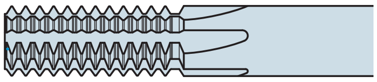
top to bottom for left-hand thread.



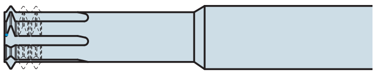
(2.2.3) Tool types for thread milling

Several types of tool can be selected depending on the application:

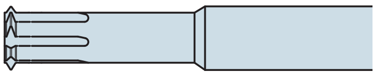
- **Thread mills**
These fulfil most requirements in both straight and helical versions.



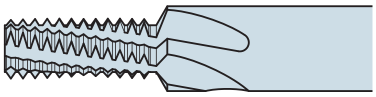
- **Whirl cutters**
These are designed to machine deep threads or for certain special applications. One-, two- and three-tooth versions are available depending on the dimensions.



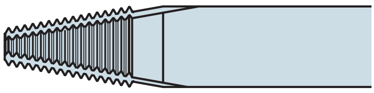
- **Partial profile**
Xactform's standard range includes so-called "partial profile" tools (whirl cutters and triangular inserts) in 60° and 55° versions. With a single tooth, they offer great flexibility and can be used to create a range of different pitches. However, as they can only machine one thread at a time, they are only suitable for very small runs (prototypes, etc.).



- **Multi-purpose tools**
These allow several machining operations with the same tool, for example threading, chamfering and/or drilling.



- **Special tools**
If the standard tools cannot be used to produce the desired thread, Xactform has the resources to very quickly develop special tools enabling it to be created. In principle, all standards and dimensions can be produce, as well as non-standard threads.
Visit http://www.xactform.ch/fr/pages/form.php for a quick quote!



(2.3) Cutting conditions

Speed and feedrate

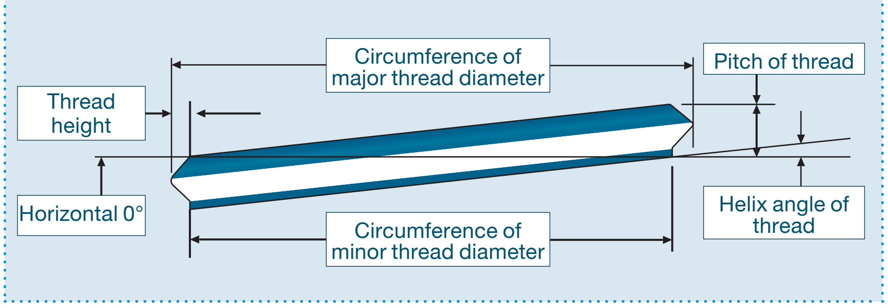
Below are the basic data, which must be fine-tuned in line with specific conditions.

Helix angle

This is calculated as follows:

$$\mathbf{tg\beta} = \frac{\text{Pitch}}{\pi * \text{thread } \varnothing}$$

For a 60° tooth shape (ISO), as the mill does not have a helix angle, if the helix angle of the thread does not exceed 4° and the tool diameter is 75% of the diameter of the thread being machined, tool ploughing is avoided. Most standard threads have helix angles of less than 4°; however, for threads with multiple entry points, for example, the angle may be greater than this, which means a standard tool is no longer suitable, as ploughing deforms the thread and damages the surface finish.



For all applications requiring a helix angle greater than 4°, please contact us; we will recommend the tool to use or, if it does not exist, provide you with a quote to quickly produce a specially-adapted tool.

Feedrate

This is calculated as follows:

Where:

F	programmed feedrate [mm/min]
Vf	feedrate at working diameter of tool [mm/min]
D	thread major diameter
D1	tool working diameter

Internal thread

$$\mathbf{F} = \frac{Vf * (D - D1)}{D}$$

External thread

$$\mathbf{F} = \frac{Vf * D}{D - D1}$$

The tables on pages 17 to 20 show the cutting speeds and feedrate per tooth for the most common materials

Cutting conditions
Thread mills with straight and tapered flutes

Materials to be machined		Coating	Cutting speed	
			HM	V1/V2
			Vc (m/min)	
Unalloyed steel / Low alloyed steel	<600N/mm2	C1	70 - 100	90 - 110
Unalloyed steel / Low alloyed steel	>600N/mm2	C1	40 - 60	70 - 90
Lead alloyed cutting steel		C2	70 - 100	90 - 110
High alloyed steel / Stainless steel	400 - 700N/mm2	C2	40 - 60	70 - 90
High alloyed steel or cast iron / Heat resisting stainless steel	700 - 1500N/mm2	C2	30 - 45	40 - 55
Special alloys		C2	15 - 30	25 - 35
Grey cast iron / Nodular iron pearlitic	<250 HB	C2	70 - 100	90 - 110
Cast iron / Nodular iron pearlitic	>250 HB	C2	40 - 70	70 - 90
Nodular ferritic cast iron / Malleable cast iron		C2	70 - 100	90 - 110
Titanium, Titanium alloy		C2 (C3)	30 - 45	40 - 60
Copper alloy (brass, bronze)		C2 (C3)	140 - 160	200 - 220
Copper alloy / Aluminium bronze	(CuAlFe)	C2	120 - 140	170 - 190
Aluminium alloy / Magnesium alloy		C2 (C3)	180 - 220	230 - 270
Aluminium cast iron	if < 8%	C2 (C3)	240 - 260	300 - 340
Aluminium cast iron	if > 8%	C2	140 - 160	210 - 230
Plastic		C2 (C4)	240 - 260	300 - 340
Gold, silver		C2 (C3)	140 - 160	200 - 220

$$n\ (rpm) = \frac{Vc\ (m/min) \times 1000}{\pi \times D1(mm)} \qquad Vf\ (mm/min) = n\ (rpm) \times fz\ (mm) \times z$$

Feed per flute Av/d (mm)									
Cutting diameter (D1)									
0,80-1,50	1,5-2,00	2,00-2,50	2,50-3,00	3,00-4,00	4,00-6,00	6,00-8,00	8,00-10,00	10,00-12,00	12,00-16,00
0,008 - 0,010		0,008 0,020	0,010 0,020	0,012 0,030	0,016 0,040	0,024 0,060	0,030 0,070	0,040 0,080	0,050 0,110
0,008 - 0,010				0,009 0,020	0,012 0,030	0,018 0,040	0,020 0,050	0,030 0,060	0,040 0,080
0,008 0,015	0,008 0,020	0,010 0,030	0,013 0,030	0,015 0,040	0,020 0,,060	0,030 0,080	0,040 0,100	0,050 0,120	0,060 0,160
0,008 - 0,010				0,009 0,020	0,012 0,030	0,018 0,040	0,020 0,050	0,030 0,060	0,040 0,080
0,006 - 0,010			0,008 0,010	0,009 0,020	0,012 0,030	0,018 0,040	0,020 0,050	0,030 0,060	0,040 0,080
0,006 - 0,010			0,009 0,020	0,012 0,030	0,018 0,040	0,020 0,050	0,030 0,060	0,040 0,080	0,040 0,080
0,008 - 0,010		0,008 0,020	0,010 0,020	0,012 0,030	0,016 0,040	0,024 0,060	0,030 0,070	0,040 0,080	0,050 0,110
0,008 - 0,010				0,009 0,020	0,012 0,030	0,018 0,040	0,020 0,050	0,030 0,060	0,040 0,080
0,008 0,010	0,008 0,015	0,008 0,020	0,010 0,020	0,012 0,030	0,016 0,040	0,024 0,060	0,030 0,070	0,040 0,080	0,050 0,110
0,008 - 0,010				0,009 0,020	0,012 0,030	0,018 0,040	0,020 0,050	0,030 0,060	0,040 0,080
0,008 0,015	0,008 0,020	0,010 0,030	0,013 0,030	0,015 0,040	0,020 0,060	0,030 0,080	0,040 0,100	0,050 0,120	0,060 0,160
0,006 - 0,010				0,009 0,020	0,012 0,030	0,018 0,040	0,020 0,050	0,030 0,060	0,040 0,080
0,008 0,015	0,008 0,020	0,010 0,030	0,013 0,030	0,015 0,040	0,020 0,060	0,030 0,080	0,040 0,100	0,050 0,120	0,060 0,160
0,008 0,015	0,008 0,020	0,010 0,030	0,013 0,030	0,015 0,040	0,020 0,060	0,030 0,080	0,040 0,100	0,050 0,120	0,060 0,160
0,008 0,015	0,008 0,020	0,010 0,030	0,013 0,030	0,015 0,040	0,020 0,060	0,030 0,080	0,040 0,100	0,050 0,120	0,060 0,160
0,008 0,023	0,011 0,030	0,014 0,040	0,018 0,040	0,021 0,060	0,028 0,09	0,042 0,120	0,060 0,150	0,070 0,180	0,080 0,240
0,008 0,015	0,008 0,020	0,010 0,030	0,013 0,030	0,015 0,040	0,020 0,060	0,030 0,080	0,040 0,100	0,050 0,120	0,060 0,160

Cutting conditions
Thread mills with straight and tapered flutes

Materials to be machined		Coating	Cutting speed	
			HM	V1/V2
			Vc (m/min)	
Unalloyed steel / Low alloyed steel	<600N/mm2	C1	70 - 100	90 - 110
Unalloyed steel / Low alloyed steel	>600N/mm2	C1	40 - 60	70 - 90
Lead alloyed cutting steel		C2	70 - 100	90 - 110
High alloyed steel / Stainless steel	400 - 700N/mm2	C2	40 - 60	70 - 90
High alloyed steel or cast iron / Heat resisting stainless steel	700 - 1500N/mm2	C2	30 - 45	40 - 55
Special alloys		C2	15 - 30	25 - 35
Grey cast iron / Nodular iron pearlitic	<250 HB	C2	70 - 100	90 - 110
Cast iron / Nodular iron pearlitic	>250 HB	C2	40 - 70	70 - 90
Nodular ferritic cast iron / Malleable cast iron		C2	70 - 100	90 - 110
Titanium, Titanium alloy		C2 (C3)	30 - 45	40 - 60
Copper alloy (brass, bronze)		C2 (C3)	140 - 160	200 - 220
Copper alloy / Aluminium bronze	(CuAlFe)	C2	120 - 140	170 - 190
Aluminium alloy / Magnesium alloy		C2 (C3)	180 - 220	230 - 270
Aluminium cast iron	if < 8%	C2 (C3)	240 - 260	300 - 340
Aluminium cast iron	if > 8%	C2	140 - 160	210 - 230
Plastic		C2 (C4)	240 - 260	300 - 340
Gold, silver		C2 (C3)	140 - 160	200 - 220

$$n\ (\text{rpm}) = \frac{Vc\ (\text{m/min}) \times 1000}{\pi \times D1(\text{mm})} \qquad Vf\ (\text{mm/min}) = n\ (\text{rpm}) \times fz\ (\text{mm}) \times z$$

Feed per flute Av/d (mm)									
Cutting diameter (D1)									
0,80-1,50	1,5-2,00	2,00-2,50	2,50-3,00	3,00-4,00	4,00-6,00	6,00-8,00	8,00-10,00	10,00-12,00	12,00-16,00
0,008 - 0,010		0,008 0,020	0,010 0,020	0,012 0,030	0,016 0,040	0,024 0,060	0,030 0,070	0,040 0,080	0,050 0,110
0,008 - 0,010				0,009 0,020	0,012 0,030	0,018 0,040	0,020 0,050	0,030 0,060	0,040 0,080
0,008 0,015	0,008 0,020	0,010 0,030	0,013 0,030	0,015 0,040	0,020 0,,060	0,030 0,080	0,040 0,100	0,050 0,120	0,060 0,160
0,008 - 0,010				0,009 0,020	0,012 0,030	0,018 0,040	0,020 0,050	0,030 0,060	0,040 0,080
0,006 - 0,010			0,008 0,010	0,009 0,020	0,012 0,030	0,018 0,040	0,020 0,050	0,030 0,060	0,040 0,080
0,006 - 0,010			0,009 0,020	0,012 0,030	0,018 0,040	0,020 0,050	0,030 0,060	0,040 0,080	0,040 0,080
0,008 - 0,010		0,008 0,020	0,010 0,020	0,012 0,030	0,016 0,040	0,024 0,060	0,030 0,070	0,040 0,080	0,050 0,110
0,008 - 0,010				0,009 0,020	0,012 0,030	0,018 0,040	0,020 0,050	0,030 0,060	0,040 0,080
0,008 0,010	0,008 0,015	0,008 0,020	0,010 0,020	0,012 0,030	0,016 0,040	0,024 0,060	0,030 0,070	0,040 0,080	0,050 0,110
0,008 - 0,010				0,009 0,020	0,012 0,030	0,018 0,040	0,020 0,050	0,030 0,060	0,040 0,080
0,008 0,015	0,008 0,020	0,010 0,030	0,013 0,030	0,015 0,040	0,020 0,060	0,030 0,080	0,040 0,100	0,050 0,120	0,060 0,160
0,006 - 0,010				0,009 0,020	0,012 0,030	0,018 0,040	0,020 0,050	0,030 0,060	0,040 0,080
0,008 0,015	0,008 0,020	0,010 0,030	0,013 0,030	0,015 0,040	0,020 0,060	0,030 0,080	0,040 0,100	0,050 0,120	0,060 0,160
0,008 0,015	0,008 0,020	0,010 0,030	0,013 0,030	0,015 0,040	0,020 0,060	0,030 0,080	0,040 0,100	0,050 0,120	0,060 0,160
0,008 0,015	0,008 0,020	0,010 0,030	0,013 0,030	0,015 0,040	0,020 0,060	0,030 0,080	0,040 0,100	0,050 0,120	0,060 0,160
0,008 0,023	0,011 0,030	0,014 0,040	0,018 0,040	0,021 0,060	0,028 0,09	0,042 0,120	0,060 0,150	0,070 0,180	0,080 0,240
0,008 0,015	0,008 0,020	0,010 0,030	0,013 0,030	0,015 0,040	0,020 0,060	0,030 0,080	0,040 0,100	0,050 0,120	0,060 0,160

(2.4) Programming principles

(2.4.1) Straight thread

Working length and pitch

For this M12 example, tool TMSC 08079 N 1.75 ISO is used. According to the data in the Xactform catalogue, its pitch is 1.75 mm and its working length L1 is 19.25 mm. The latter must be at least equal to the length of the thread to be produced.

If the pitch is in inches, convert it to mm using the following formula:

p [mm]= 25.4/P [inch]

Properties of the material to be machined

The properties of the material to be machined – in this case E295 – determine the cutting speed (Vc) and feedrate (Vf). This information is available in the tables on pages 8 to 11 of the Xactform catalogue and provides the following calculations:

$$\textbf{Tool rotation n} = \frac{Vc\ [mm/min]}{\pi * D1\ [mm]} = \frac{70 * 1'000}{\pi * 7.9} = 2820\ rpm$$

$$\textbf{Feedrate Vf} = n * Z * fz = 2820 * 5 * 0.036 = 507\ mm/min$$

NB: the programmed feedrate must be that for the diameter over which the tool moves. The formula to obtain the programmed feedrate is as follows:

$$\textbf{F} = Vf\ [mm/min] * (D – D1) / D = 507 * (12 – 7.90) / D = 173\ mm/min$$

Where:
D = thread Ø major diameter [mm]
D1 = tool working Ø [mm]

Z axis compensation

The Z axis compensation is the distance travelled by the tool in the Z axis during interpolation to penetrate the material and reach the thread major diameter, then withdraw from the material at the end of machining. It corresponds to one eighth of the pitch and is expressed in [mm].
In our example: 1.75 (pitch) / 8 = 0.219 mm

Absolute value

The absolute value corresponds to the tool position in [mm] centred at the base of the thread, i.e. (D – D1)/2 = 2.05 mm

Incremental value

The incremental value in [mm] or the tool position at the start of interpolation to penetrate the material equates to: (D – D1)/4 = 1.025 mm

Drilling diameter

The drilling diameter equivalent to 100% of the thread height is calculated as follows:

D – (H*2), where H is the full depth of a thread = 9.9 mm

Drilling table

Ø Drilling diameter

Given that the thread mill produces 100% of the thread height, the diameter of the pre-drilled hole must be less than that of a hole drilled for a tap.

Example
M6 thread ‡ profile height 1.00 ISO = 0.587 mm
Pre-drilled hole for a tap: Ø 5 mm
Pre-drilled hole for a milled thread: Ø 4.82 mm = 6 – (2 * 0.587)

Table of drilling diameters for milled threads

Drilling Ø for other standards available on request

	Pitch													
	0.08	0.09	0.10	0.125	0.15	0.175	0.20	0.225	0.25	0.30	0.35	0.40	0.45	0.50
thread Ø														
0.3	0.21													
0.35		0.25												
0.4			0.29											
0.5				0.36										
0.6					0.44									
0.7						0.51								
0.8							0.58							
0.9								0.66						
1							0.78		0.73					
1.1							0.88							
1.2							0.98		0.93					
1.4							1.18		1.13	1.08				
1.6							1.38		1.33		1.22			
1.7							1.48		1.43		1.32			
1.8							1.58		1.53		1.42			
2							1.78		1.73		1.62	1.57		
2.2							1.98		1.93		1.82		1.71	
2.3							2.08		2.03		1.92		1.81	
2.5							2.28		2.23		2.12		2.01	
2.6							2.38		2.33		2.22		2.11	
3							2.78		2.73		2.62			2.46
3.5							3.28		3.23		3.12			2.96
4							3.78		3.73		3.62			3.46
4.5							4.28		4.23		4.12			3.96
5							4.78		4.73		4.62			4.46
5.5							5.28		5.23		5.12			4.96
6							5.78		5.73		5.62			5.46
6.5							6.28		6.23		6.12			5.96
7							6.78		6.73		6.62			6.46
7.5							7.28		7.23		7.12			6.96
8							7.78		7.73		7.62			7.46
8.5							8.28		8.23		8.12			7.96
9							8.78		8.73		8.62			8.46
9.5							9.28		9.23		9.12			8.96
10							9.78		9.73		9.62			9.46
10.5											10.12			9.96
11											10.62			10.46
11.5											11.12			10.96
12											11.62			11.46
12.5											12.12			11.96
13											12.62			12.46
13.5											13.12			12.96
14											13.62			13.46
14.5											14.12			13.96
15											14.62			14.46
15.5											15.12			14.96
16											15.62			15.46
17											16.62			16.46
18											17.62			17.46
19											18.62			18.46
20											19.62			19.46
21											20.62			20.46
22											21.62			21.46
23											22.62			22.46
24											23.62			23.46
25											24.62			24.46
26											25.62			25.46
27											26.62			26.46
28											27.62			27.46
29											28.62			28.46
30											29.62			29.46
31											30.62			30.46
32											31.62			31.46
33											32.62			32.46
34											33.62			33.46
35											34.62			34.46
36											35.62			35.46
37											36.62			36.46
38											37.62			37.46
39											38.62			38.46
40											39.62			39.46
42											41.62			41.46
44														
45											44.62			44.46
48											47.62			47.46
50											49.62			49.46
…														…
64														63.46
														…

🟦 Metric standard ISO (DIN 13-1)

🟪 Metric standard ISO Fine Pitch (DIN 13-2 to DIN 13-9)

🟩 Metric standard ISO for Ø<1mm (DIN 14)

	0.60	0.70	0.75	0.80	1.00	1.25	1.50	1.75	2.00	2.50	3.00	3.50	4.00	5.00	6.00
	2.85														
		3.24													
			3.69												
			4.19	4.13											
			4.69												
			5.19		4.92										
			5.69												
			6.19		5.92										
			6.69		6.42										
			7.19		6.92	6.65									
			7.69		7.42										
			8.19		7.92	7.65									
			8.69		8.42										
			9.19		8.92	8.65	8.38								
			9.69		9.42										
			10.19		9.92		9.38								
			10.69		10.42										
			11.19		10.92	10.65	10.38	10.11							
			11.69		11.42										
			12.19		11.92		11.38								
			12.69		12.42										
			13.19		12.92		12.38		11.84						
			13.69		13.42										
			14.19		13.92		13.38								
			14.69												
			15.19		14.92		14.38		13.84						
			16.19		15.92		15.38		14.84						
			17.19		16.92		16.38		15.84						
			18.19		17.92		17.38		16.84						
			19.19		18.92		18.38		17.84	17.29					
			20.19		19.92		19.38		18.84						
			21.19		20.92		20.38		19.84						
			22.19		21.92		21.38		20.84						
			23.19		22.92		22.38		21.84		20.75				
			24.19		23.92		23.38		22.84						
			25.19		24.92		24.38		23.84						
			26.19		25.92		25.38		24.84						
			27.19		26.92		26.38		25.84		24.75				
			28.19		27.92		27.38								
			29.19		28.92		28.38		27.84		26.75	26.21			
			30.19		29.92		29.38								
			31.19		30.92		30.38		29.84		28.75				
			32.19		31.92		31.38		30.84		29.75				
			33.19		32.92		32.38		31.84		30.75				
			34.19		33.92		33.38		32.84		31.75				
			35.19		34.92		34.38		33.84		32.75		31.67		
			36.19		35.92		35.38								
			37.19		36.92		36.38		35.84		34.75				
			38.19		37.92		37.38		36.84		35.75				
			39.19		38.92		38.38		37.84		36.75		35.67		
			41.19		40.92		40.38		39.84		38.75		37.67		
					42.92										
			44.19		43.92		43.38		42.84		41.75		40.67		
			47.19		46.92		46.38		45.84		44.75		43.67	42.59	
			49.19		48.92		48.38		47.84		46.75		45.67		
			…		…		…		…		…		…		
			63.19		62.92		62.38		61.84		60.75		59.67		57.51
			…		…		…		…		…		…		

(2.4.2) Straight thread programming

ISO program

Data required for thread milling

Thread specification	**M12**	①
Tool selected	**TMSC08079-N-1.75 ISO**	①
Thread major diameter	**12 mm**	①
Pitch	**1.75 mm**	①
Thread length	**18 mm**	①
Material	**E295**	②
Cutting speed (Vc)	**70 m/min**	②
Feedrate per tooth (Av/d)	**0.036 mm**	②
Number of flutes (Z)	**5**	①
Tool rotation diameter or rotation diameter (D1)	**7.90 mm**	①
Speed (rpm)	**2820 rpm**	②
Feedrate Vf (mm/min)	**507 mm/min**	②
Programmed feedrate (F)	**173 mm/min**	③
Z axis compensation (1/8 pitch) penetration and withdrawal	**0.219 mm**	④
Absolute value (Maj. diam. - D1) / 2	**2.05 mm**	⑤
Incremental value (Maj. diam. - D1) / 4	**1.025 mm**	⑥
Drilling diameter to obtain 100% of the thread	**9.90 mm**	⑦

① Tool TMSC08079-N-1.75 ISO was selected with reference to the Xactform catalogue entitled "Solid carbide thread mill, ISO Metric 60° internal, straight flute" and based on straight thread programming. This tool has a pitch of 1.75 mm (if the pitch is in inches, convert it: 25.4 / P) and a working length of 19.25 (L1 must be at least equal to the thread length).

② The properties of the material, E295, determine the tool feedrate. This information can be found in the cutting conditions tables.
Speed = 70 x 1000 / ϖ x 7.90 = 2820 rpm
Feedrate Vf = 2'820 x Z x 0.036 = 507 mm/min

③ The programmed feedrate is the rate which is correct for the diameter over which the tool moves. This operation is one of the critical points for successful thread milling. The formula is found in "Calculating the feedrate for an internal thread". 507 x (12-7.90) / 12 = 173 mm/min.

④ The Z axis compensation is the distance moved by the tool in the Z axis during interpolation to penetrate the material and arrive at a line tangential to the major diameter. 0125 x 1.75 = 0.219 mm

⑤ The absolute value, or the value of the tool position at the base of the thread. (12 - 7.90) / 2 = 2.05 mm

⑥ The incremental value, or the value of the tool position at the start of interpolation to penetrate the material. (12 - 7.90) / 4 = 1.025 mm

⑦ The drilling diameter to obtain 100% of the thread height, (drilling table for taps, max. result 75% of the thread height) 12 - (2 x 1.027) = 9.90 mm (tap Ø = 10.20 mm)

CNC programming

Programming for:
Example: straight thread

Program with absolute values
S28220 M03 G90 X0 Y0 Z0
G01 X0 Y0 Z-19.094 F2500
G41 X1.025 Y-1.025 D1
G03 X2.05 Y0 Z-8.875 I0 J1.025 F173
G03 X2.05 Y0 Z-17.125 I-2.05 J0
G03 X1.025 Y1.025 Z-16.25 I1.025 J0 F259
G01 G40 X0 Y0 F2500
G0 Z0

Program in incremental values
S2820 M03 G90 X0 Y0 Z0 **G91**
G01 X0 Y0 Z-19.094 F2500
G41 X1.025 Y-1.025 D1
G03 **X1.025 Y1.025 Z0.219** I0 J1.025 F173
G03 **X0** Y0 **Z1.75** I-2.05 J0
G03 X1.025 Y1.025 **Z0.219** I1.025 J0 F259
G01 G40 **X-1.025 Y-1.025** F2500
G90 G0 Z0

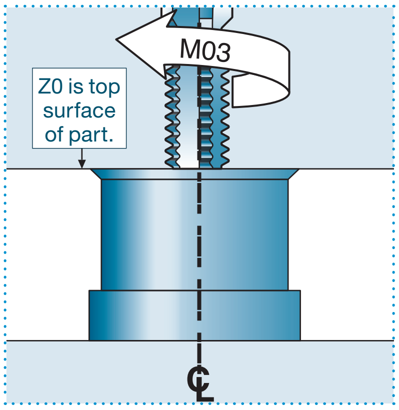
Detailed program with absolute values

① ③
S2820 M03

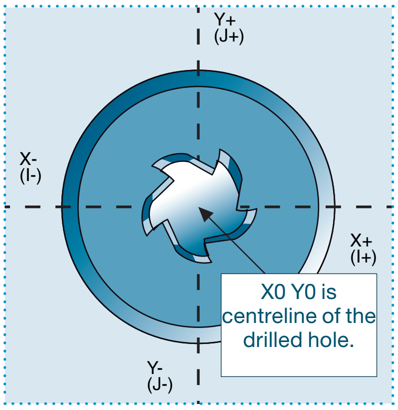
① Spindle rotation speed in rpm.
③ Clockwise spindle rotation.

② ⑦
G90 X0 Y0 Z0

② Absolute programming (the coordinates always start from the workpiece origin).
⑦ In this example, the origin is in the centre of the hole, on the surface of the workpiece.



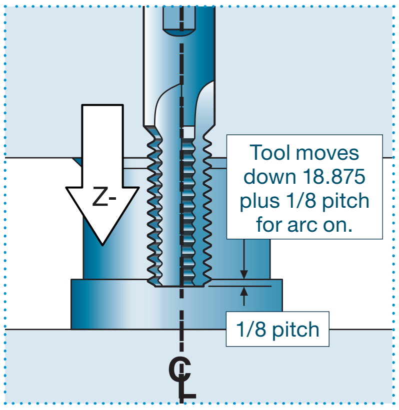
Z0 is top surface of part.



X0 Y0 is centreline of the drilled hole.

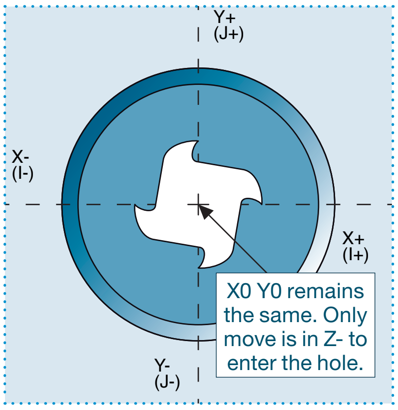
① ③ ②
G01 X0 Y0 Z-19.094 F2500

①Register programmed feedrate.
③ Z axis movement of tool to begin thread milling. Thread length + half pitch (distance between the edge of the tool during interpolation to penetrate the material).
② Rapid feedrate to enter the hole (this can be modified at the start to ensure a slower feedrate in order to check that the hole position is correct).



Tool moves down 18.875 plus 1/8 pitch for arc on.

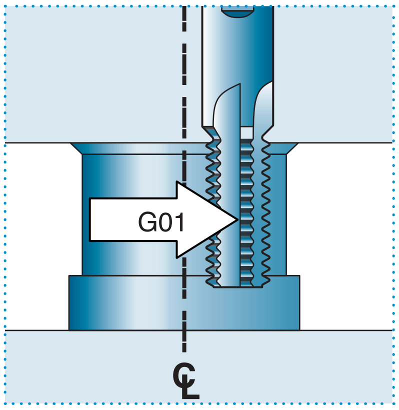
1/8 pitch

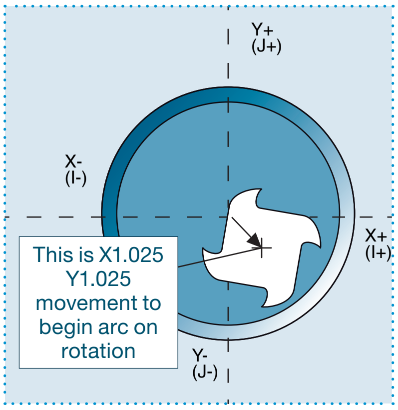


X0 Y0 remains the same. Only move is in Z- to enter the hole.

① ③ ②
G41 X1,025 Y-1,025 D1

① Tool correction registered.
③ Tool movement on a circular arc to penetrate the material.
②Tool correction parameter No. used to check the thread diameter.



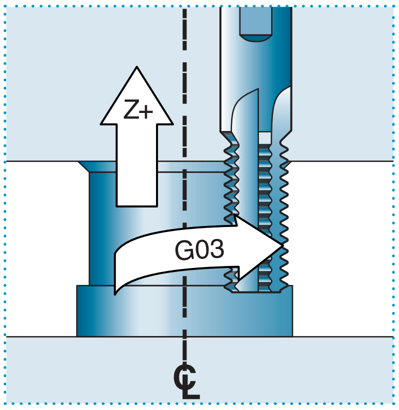


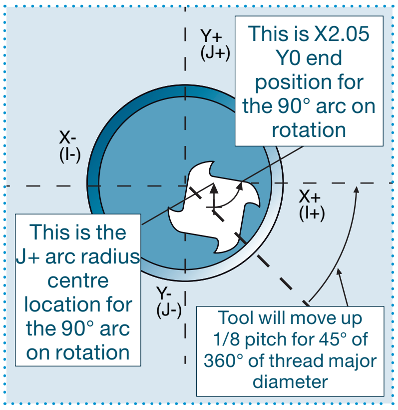
This is X1.025 Y1.025 movement to begin arc on rotation

① ③ ② ⑦ ⑥

G03 X2.05 Y0 Z-18.875 I0 J1.025 F173

① Tool correction registered.
③ Tool movement on a circular arc to penetrate the material.
② Tool correction parameter No. used to check the thread diameter.
⑦ Circular interpolation centre position.
⑥ Programmed feedrate.



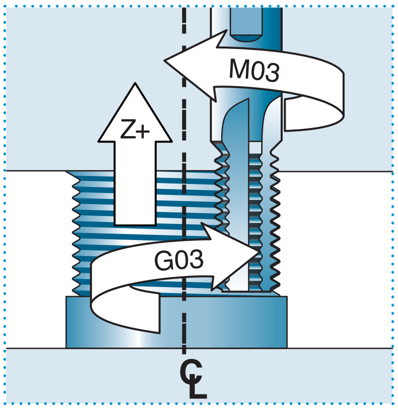


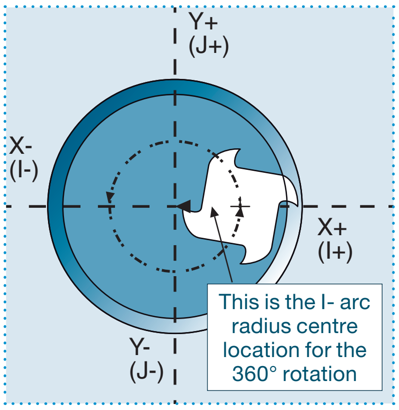
① ③ ②

*** G03 X2.05 Y0 Z-17.125 I-2.05 J0**

① Final position following 360° interpolation.
③ Thread length - the pitch.
② Interpolation centre position.

* for T2 add this line 1x=(2x)
* for T3 add this line 2x=(3x)

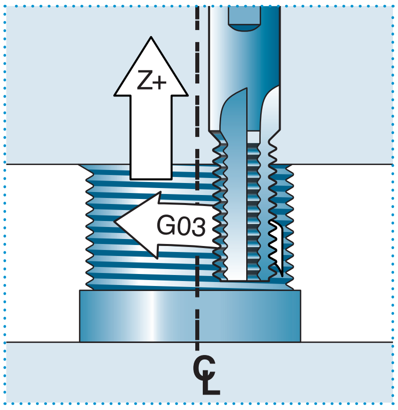


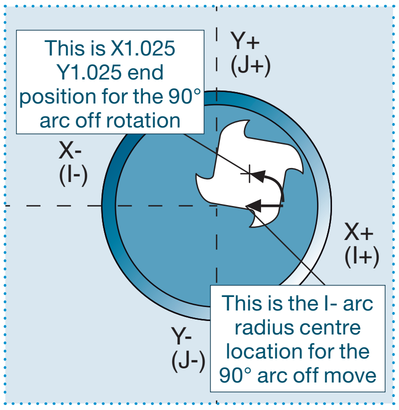


① ③ ②

G03 X1.025 Y1.025 Z-16.25 I1.025 J0 F259

Tool interpolation to allow withdrawal from the material.
① Movement of 1/8 pitch, (to remain in the same helix angle).
③ Interpolation centre.
② Feed to withdraw from material approximately 1.5x programmed feedrate.

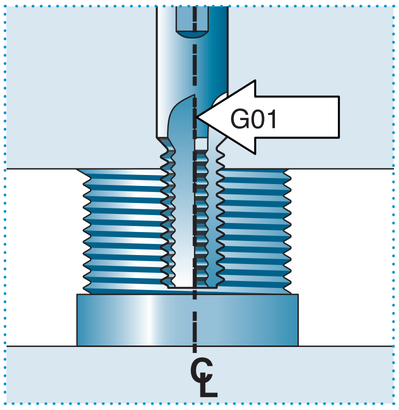


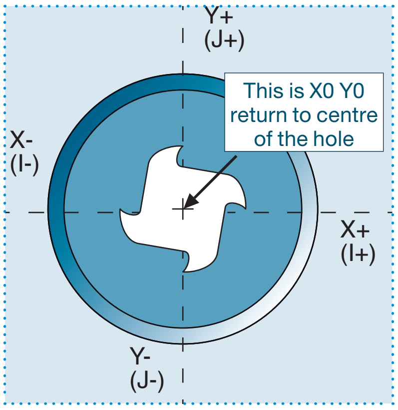


① ③ ②

G01 G40 X0 Y0 F2500

- ① Cancellation of tool correction.
- ③ Repositioning in the centre of the hole.
- ② Rapid feedrate (can be adjusted in G0).



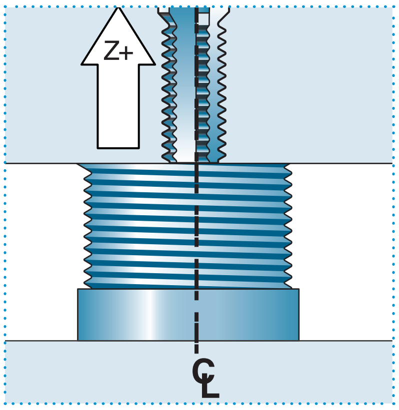


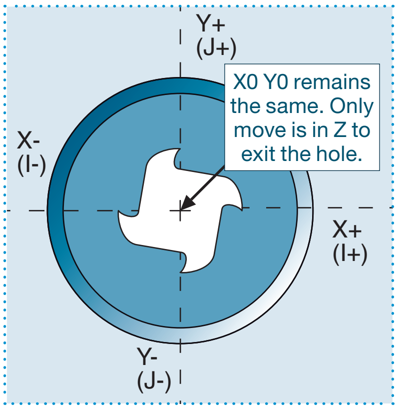
This is X0 Y0 return to centre of the hole

①

G0 Z0

- ① Return to origin with rapid feedrate.





X0 Y0 remains the same. Only move is in Z to exit the hole.

To programme with incremental values, change the values **in bold** from absolute to incremental and add the values G90, G03 and G01 **in bold** (see table at the bottom of page 24).

(2.4.3) Multiple pass threading

Roughing and finishing

Some materials require the thread to be machined in successive passes, often a roughing pass and a finishing pass. In most cases, the roughing pass only machines around 85-90% of the tooth depth, and the finishing pass machines to the final dimensions.
However, to prevent premature tool wear, we recommend reserving at least 0.05 mm on the finishing pass for a maximum tool working diameter of 6 mm and 0.10 mm for larger diameters.

Example: For an M12 thread

Two-pass program (incremental)

```
S2820 M03
G91 X0 Y0 Z0
G01 Z-19.094 F2500
G41 X0.902 Y-0.902
G03 X0.902 Y0.902 Z0.219 I0 J0.902 F152
(the feedrate is also reduced proportionally)
G03 X0 Y0 Z1.750 I-1.804 J0
G03 X-0.902 Y0.902 Z0.219 I-0.902 J0 F228
G01 G40 X-0.902 Y-0.902 F2500
G01 Z-2.188
(Descend from Z axis movement P+(2*1/8 P) = 2.188)
G41 X1.025 Y-1.025 D1
G03 X1.025 Y1.025 Z0.219 I0 J1.025 F173
G03 X0 Y0 Z1.750 I-2.050 J0
G03 X-1.025 Y1.025 Z0.219 I-1.025 J0 F260
G01 G40 X-1.025 Y-1.025 F2500
G90 G0 Z0
```

T2, T3 threads and whirl cutters

Because tools T2 and T3 have only every other tooth (every third for T3), the circular interpolation must be doubled (or tripled) to obtain the complete thread.

Example: For an M12 thread

Two-pass program (incremental)

```
S2820 M03
G91 X0 Y0 Z0
G01 Z-19.094 F2500
G41 X1.025 Y-1.025 D1
G03 X1.025 Y1.025 Z0.219 I0 J1.025 F173
G03 X0 Y0 Z1.750 I-2.050 J0
G03 X0 Y0 Z1.750 I-2.050 J0
(once more for a T2)
G03 X0 Y0 Z1.750 I-2.050 J0
(twice more for a T3)
G03 X-1.025 Y1.025 Z0.219 I-1.025 J0 F260
G01 G40 X-1.025 Y-1.025 F2500
G90 G0 Z0
```

The principle is the same for whirl cutters: the length of the thread is divided by the pitch, and the result gives the number of interpolations to be made (i.e. the number of executions on the line).

Internal thread data	
Thread (M)	M 1.6
Drilling diameter (dp) for 100%	1.47 mm
Thread length	4.8 mm
No thread (p)	0.35 mm
Int. thread height 0.5413*p (H1)	0.189 mm
Tool reference	TBR 03012 N 0.35 ISO
VC	50 m/min
Tool cutting diameter (D1)	1.2 mm
Number of tool flutes (Z)	3
Feedrate per tooth	0.008 mm
Linear feedrate	318 mm/min
RPM. (n)	13.263 rpm
Circular feedrate (fcl)	80 mm/min
1st tooth distance (C x p)	0.175 mm
Programming data	
Z compensation (ztg) (entry; withdrawal) 45° tangential = (pitch/8)	0.044 mm
Compensated depth (zl) Thread length + 1st tooth distance + (entry; withdrawal) tangential	5.019 mm
Absolute value	0.2 mm
Incremental value	0.1 mm

Tool offset radius = 0.00

G	G	X	Y	Z	I	J	F
G 90		0.000	0.000	20.000			
				0.000			
G 91							
	G 1			-5.019			500
G 41	G 1	0.100	-0.100				318
	G 3	0.100	0.100	0.044	0.000	0.100	80
	G 3	0.000	0.000	0.350	-0.200	0.000	
	G 3	-0.100	0.100	0.044	-0.100	0.000	
G 40	G 1	-0.100	-0.100				318
G 90	G0			20.000			

Repeat red line = Thread length/pitch

Expl. 4.8/0.35 = 14

Internal thread data	
Thread (M)	M 5
Drilling diameter (dp) for 100%	4.31 mm
Thread length	7.5 mm
No thread (p)	0.80 mm
Int. thread height 0.5413*p (H1)	0.433 mm
Tool reference	TMSC 06036 N 0.80 ISO
VC	70 m/min
Tool cutting diameter (D1)	3.6 mm
Number of tool flutes (Z)	3
Feedrate per tooth	0.012 mm
Linear feedrate	233 mm/min
RPM. (n)	6189 rpm
Circular feedrate (fcl)	62 mm/min
1st tooth distance (1/2x p)	0.4 mm
Programming data	
Z compensation (ztg) (entry; withdrawal) 45° tangential = (pitch/8)	0.1 mm
Compensated depth (zl) Thread length + 1st tooth distance + (entry; withdrawal) tangential	8.0 mm
Absolute value	0.7 mm
Incremental value	0.35 mm

Tool offset radius = 0.00

G	G	X	Y	Z	I	J	F
G 90		0.000	0.000	20.000			
				0.000			
G 91							
	G 1			-8.000			500
G 41	G 1	0.350	-0.350				223
	G 3	0.350	0.350	0.100	0.000	0.350	62
	G 3	0.000	0.000	0.800	-0.700	0.000	
	G 3	-0.350	0.350	0.100	-0.350	0.000	
G 40	G 1	-0.350	-0.350				318
G90	G0			20.000			

Internal thread data	
Thread (M)	M 6
Drilling diameter (dp) for 100%	4.92 mm
Thread length	9.0 mm
No thread (p)	1.00 mm
Int. thread height 0.5413*p (H1)	0.541 mm
Tool reference	TMSC 06040 N 1.00 ISO
VC	70 m/min
Tool cutting diameter (D1)	4.0 mm
Number of tool flutes (Z)	3
Feedrate per tooth	0.012 mm
Linear feedrate	201 mm/min
RPM. (n)	5570 rpm
Circular feedrate (fcl)	67 mm/min
1st tooth distance (1/2x p)	0.5 mm
Programming data	
Z compensation (ztg) (entry; withdrawal) 45° tangential = (pitch/8)	0.125 mm
Compensated depth (zl) Thread length + 1st tooth distance + (entry; withdrawal) tangential	9.625 mm
Absolute value	1.0 mm
Incremental value	0.5 mm

Tool offset radius = 0.00

G	G	X	Y	Z	I	J	F
G 90		0.000	0.000	20.000			
				0.000			
G 91							
	G 1			-9.625			500
G 41	G 1	0.500	-0.500				201
	G 3	0.500	0.500	0.125	0.000	0.500	67
	G3	0.000	0.000	1.000	-1.000	0.000	
	G 3	-0.500	0.500	0.125	-0.500	0.000	
G 40	G 1	-0.500	-0.500				318
G90	G0			20.000			

Internal thread data	
Thread (M)	M 8
Drilling diameter (dp) for 100%	6.31 mm
Thread length	24.0 mm
No thread (p)	1.125 mm
Int. thread height 0.5413*p (H1)	0.677 mm
Tool reference	TMSC 06050 N 1.25 ISO T3
VC	70 m/min
Tool cutting diameter (D1)	5.0 mm
Number of tool flutes (Z)	3
Feedrate per tooth	0.012 mm
Linear feedrate	160 mm/min
RPM. (n)	4456 rpm
Circular feedrate (fcl)	60 mm/min
1st tooth distance (1/2x p)	0.625 mm
Programming data	
Z compensation (ztg) (entry; withdrawal) 45° tangential = (pitch/8)	0.156 mm
Compensated depth (zl) Thread length + 1st tooth distance + (entry; withdrawal) tangential	24.781 mm
Absolute value	1.5 mm
Incremental value	0.75 mm

Tool offset radius = 0.00

G	G	X	Y	Z	I	J	F	
G 90		0.000	0.000	20.000				
				0.000				
G 91								
	G 1			-24.781			500	
G 41	G 1	0.750	-0.750				160	
	G 3	0.750	0.750	0.156	0.000	0.750	60	
	G 3	0.000	0.000	1.250	-1.500	0.000		1p
	G3	0.000	0.000	1.250	-1.500	0.000		2p
	G 3	0.000	0.000	1.250	-1.500	0.000		3p
	G 3	-0.750	0.750	0.156	-0.750	0.000		
G 40	G 1	-0.750	-0.750				160	
G90	G0			20.000				

Internal thread data	
Thread (M)	M 70
Drilling diameter (dp) for 100%	64.05 mm
Thread length	60.0 mm
No thread (p)	5.50 mm
Int. thread height 0.5413*p (H1)	2.977 mm
Tool reference	TM40 N 5.50 ISO
VC	120 m/min
Tool cutting diameter (D1)	50.0 mm
Number of tool flutes (Z)	1
Feedrate per tooth	0.05 mm
Linear feedrate	38 mm/min
RPM. (n)	764 rpm
Circular feedrate (fcl)	11 mm/min
1st tooth distance (C x p)	4.125 mm
Programming data	
Z compensation (ztg) (entry; withdrawal) 45° tangential = (pitch/8)	0.688 mm
Compensated depth (zl) Thread length + 1st tooth distance + (entry; withdrawal) tangential	64.813 mm
Absolute value	10.0 mm
Incremental value	5.0 mm

Tool offset radius = 0.00 Tool holder **TMH04050-2-40**

G	G	X	Y	Z	I	J	F	
G 90		0.000	0.000	20.000				
				0.000				
G 91								
	G 1			-64.813			500	
G 41	G 1	5.000	-5.000				38	step 1
	G 3	5.000	5.000	0.688	0.000	5.000	11	
	G3	0.000	0.000	5.500	-10.000	0.000	11	
	G 3	-5.000	5.000	0.688	-5.000	0.000	38	
G 40	G 1	-5.000	-5.000				38	
	G 1			26.124			500	
G 41	G 1	5.000	-5.000				38	step 1
	G 3	5.000	5.000	0.688	0.000	5.000	11	
	G 3	0.000	0.000	5.500	-10.000	0.000	11	
	G 3	-5.000	5.000	0.688	-5.000	0.000	38	
G 40	G 1	-5.000	-5.000				38	
G 90	G 1			20			500	

(2.4.4) Tapered thread

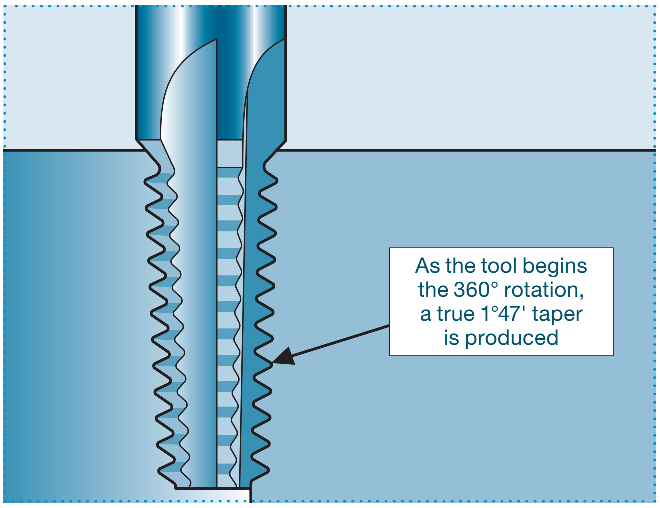
The programming for a tapered thread is only marginally more complex than that for a straight thread. In theory, the X and Y movements should be modified at each degree, which some CNC machines allow; however, in practice, compensation is only applied every 90°.

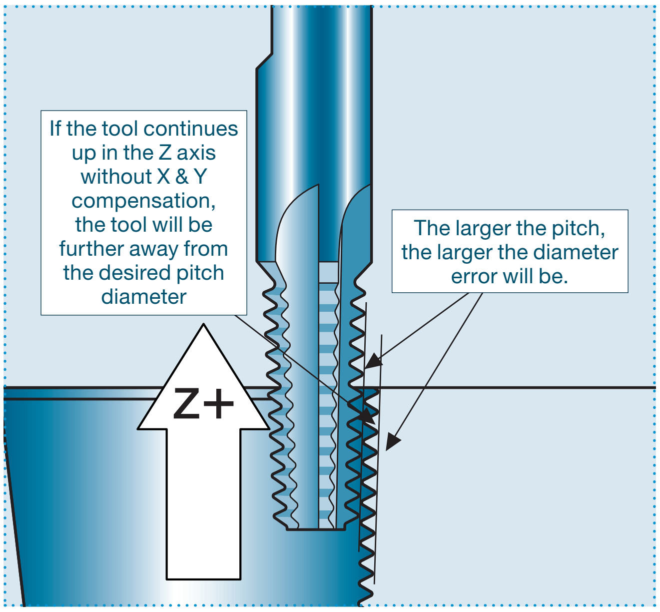
Compensation calculation example (valid for all tapered threads):

Pitch: 8”NPT (⇢ 3.175mm)
Taper: 1:16 (⇢ß = 1°47’24”)

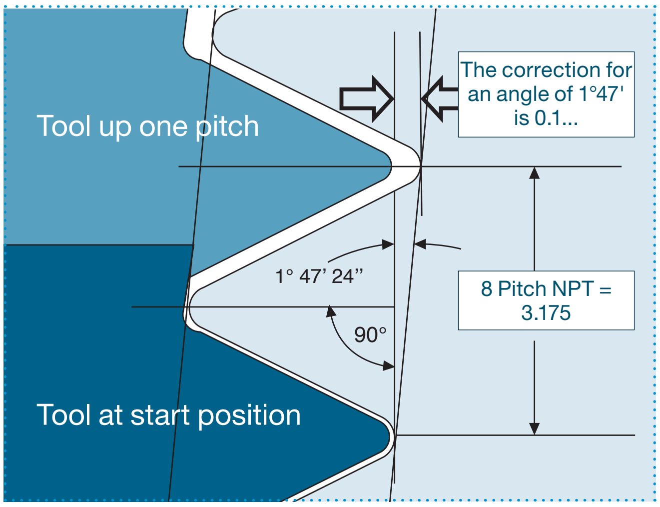
Formula: **compensation [mm] = pitch * tan (ß)**

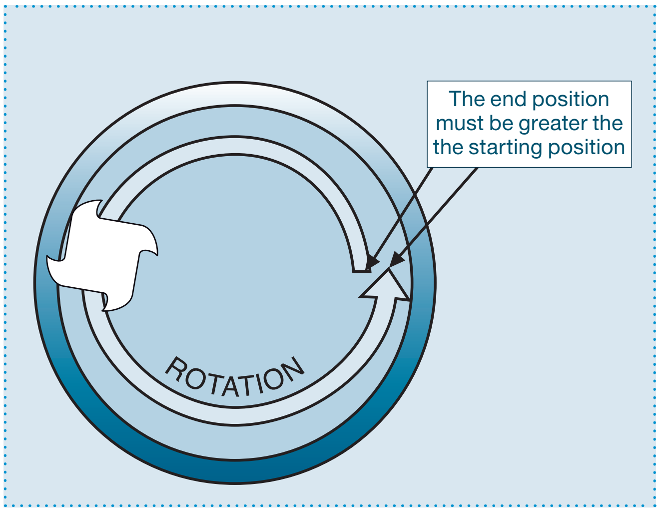
This compensation value equates to a complete pitch; so, for adjustment every 90°, the value must be divided by 4. In our example:
3.175 * tan (1°47'24") / 4 = 0.025 mm

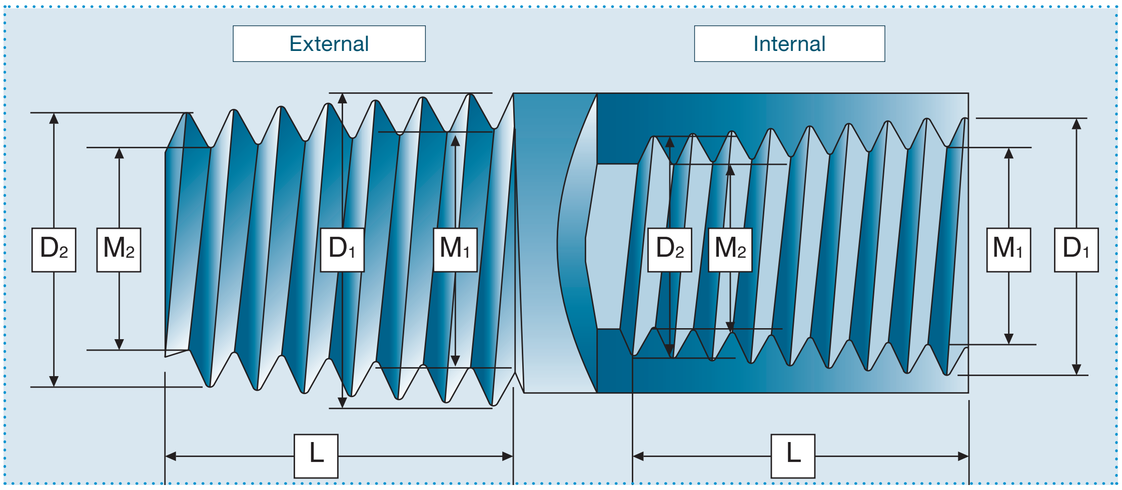

As the tool begins the 360° rotation, a true 1°47' taper is produced


If the tool continues up in the Z axis without X & Y compensation, the tool will be further away from the desired pitch diameter

The larger the pitch, the larger the diameter error will be.


The correction for an angle of 1°47' is 0.1...


The end position must be greater the the starting position



Dimensions for NPT and NPTF tapered threads

(data in mm for standard external and internal threads)

Standard	Pitch	D1	D2	L	M1	M2
1/16" – 27"	0.941	7.823	7.417	6.883	6.401	5.994
1/8" -27"	0.941	10.160	9.754	6.934	8.738	8.331
1/4" – 18"	1.411	13.538	12.903	10.033	11.379	10.770
3/8" -18"	1.411	16.967	16.332	10.338	14.834	14.199
1/2" -14"	1.814	21.133	20.295	13.564	18.364	17.526
3/4" – 14"	1.814	26.492	25.603	14.046	23.724	22.835
1" 11.5	2.209	33.147	32.080	16.789	29.794	28.727
1-1/4" - 11.5"	2.209	41.910	40.818	17.297	38.557	37.465
1-1/2" - 11.5"	2.209	47.955	46.888	17.297	44.602	43.536
2" - 11.5"	2.209	59.995	58.903	17.704	56.642	55.550
2-1/2" – 8"	3.175	72.619	70.942	26.848	67.793	66.116
3" – 8"	3.175	88.519	86.716	28.981	83.693	81.890
3-1/2" – 8"	3.175	101.219	99.238	31.750	96.393	94.412
4" – 8"	3.175	113.894	111.811	33.020	109.068	106.985
5" – 8"	3.175	140.868	138.633	35.712	136.042	133.807

(2.4.5) Tapered thread programming

To prepare a milling program for tapered threads, the data must be assembled in the same way as for straight threads. In this example, a program for a tapered thread over a quarter of the workpiece perimeter is written in ASCII format in line with the ISO standard. The thread characteristics are tapered, 1"- 11.5 NPT and the material is 316 stainless steel.

The characteristics for this and all NPT and NPTF threads are shown on page 33.

Data sheet for NPT thread milling

Thread characteristics	**1" -11.5 NPT**	①
Tool selected	**TMX 5/8x0.620-11.5NPT**	②
Main thread diameter	**1.305"**	③
Pitch	**11.5 TPI = 1 ÷ 11.5 =0.08696"**	①
Thread length	**0.661" + 1/2 PITCH**	③
Material	**316 stainless steel**	④
Cutting speed	**525**	④
Flute penetration	**0.0010"**	④
Number of flutes	**6**	②
Tool rotation diameter	**620"**	②
Spindle rpm	**3.234**	④
Penetration in PPM, actual	**19.4**	⑤
Penetration in PPM, programmed	**10.18**	⑥
Z axis compensation (1/8 pitch) for start/stop arc	**0.0109"**	⑦
Absolute value (Thread maj. diam. -tool diam.) + 2	**0.3425"**	⑧
Incremental value (Thread maj. diam. -tool diam.) + 4	**0.1713"**	⑧
X and Y comp. for quarter circle	**0.0007"**	⑨

Calculating the data

① The thread characteristics determine the choice of tool ②, the main thread diameter ③, the pitch expressed in threads per inch and the length of the thread to be milled. This information is provided on the workpiece printout and in the thread characteristics for standard NPT sizes, as shown on page 33.

Tool ②, TMX 5/8 x 0.620-11.5NPT, was chosen in reference to the Xactform catalogue of solid carbide thread mills complying with the NPT National Pipe standard, and by verifying the mill recommended for a 1" NPT thread size. The characteristics indicated for this tool show that it has a pitch configuration of 11.5 NPT and a total cutting length of 1.043", far exceeding the length of 0.661" ③ required for this operation.
Xactform solid carbide thread mills are designed to produce sizes of 11.5 NPT using the same mill.

④ The characteristics of the material, 316 stainless steel, determine the tool's speed and penetration. This information was obtained using the cutting speed and feedrate table on pages 16 to 19 of this technical guide. The rpm values were calculated using the formula shown at the top of page 17. (3.81972 x 525) ÷ the cutting tool diameter of 0.620 = 3.234.

⑤ The actual PPM value of the tool was calculated using the standard formula: rpm x penetration per tooth per flute x number of flutes or 3.234 x 0.0010” x 6 = 19.4 IPM.

⑥ The programmed PPM value is the corrected penetration rate for the tool shaft. This calculation is particularly critical to ensure successful milling operations. The programmed feedrate or penetration speed was determined using the mill feedrate formula from the table on pages 16-17 of this guide.
F2 = PPM x (+ large thread diam. –cutting diam.).
Main thread diameter:
or (19.4 x 1.305" – 0.620") / 1.305 = 10.18 PPM programmed.

⑦ The Z axis compensation value, or the raising value for the 1/8 arc movement during feed and withdrawal was calculated by dividing the pitch (0.08696") by 8, or 0.08696" / 8 = 0.0109".

⑧ The absolute value, or the value of the tool shaft position at the complete thread length, was calculated by subtracting the cutting tool diameter from the main thread diameter and dividing by 2, or (1.305” - 0.620) / 2 = 0.0950". The incremental value, or the value of the tool shaft position at the start of the arc movement, was calculated by subtracting the cutting tool diameter from the main thread diameter and dividing by 4, or (1.305 - 0.620) / 4 = 0.0475".

⑨ The compensation of angles X and Y for the 11.5 NPT thread was calculated using the table on page 33.

Program for 1"-11.5 NPT

```
S3234 M03
G90 X 0 Y 0 Z 0
G01 X 0 Y 0 Z.-0.7154 F50
G41 X 0.1713 Y-0.1713
G03 X 0.3425 Y 0 Z-0.7045 I 0 J 0.1713 FI0.I8
G03 X 0 Y 0.3432 Z-0.6828 I-0.3432 J 0
G03 X-0.3439 Y 0 Z-0.6611 1 0 J-0.3439
G03 X 0 Y-0.3446 Z-0.6394 I 0.3446 J 0
G03 X 0.3453 Y 0 Z-0.6177 I 0 J 0.3453
G03 X 0.1727 Y 0.1727 Z-0.6068 I-0.1727 J 0 F 20.00
G01
G40 X 0 Y 0 F 50
G00 Z 0
```

The 1"-1.5 NPT program shown on the left comprises a cycle with a single pass and one rotation. The main difference between the 1/2 -20 UN internal program shown in the straight thread programming section and the NPT example is the three extra lines added to move the tool in 4 separate 90° stages, instead of the single line which moves over 360° as executed in the 1/2"-20 program.
In addition, the movements in the Z axis were added to the pitch in 1/4 increments, since the tool moves in an arc over 90° or per quarter of the hole perimeter. This value was calculated by dividing the pitch by 4 and subtracting the value from the previous line.
The X, Y, I and J values are 0.0007" added to each line. This is the taper compensation for each 90° arc movement. The illustrations on p. 36 show each line and explain the program in detail.

Internal thread data	
Thread (NPT)	1.000"
thread end Ø	32.08 mm
Drilling diameter (dp) for 100%	28.73 mm
Thread length	16.78 mm
Pitch (Pp)	11.5"
No thread (p)	2.2087 mm
Int. thread height 0.7589*p (H1)	1.67618 mm
Tool reference	TMSC 16159 NE 11.5 NPT
VC	70 m/min
Tool cutting diameter (D1)	14.3 mm
Number of tool flutes (Z)	6
Feedrate per tooth	0.012 mm
Linear feedrate	112 mm/min
RPM. (n)	1558 rpm
Circular feedrate (fcl)	62 mm/min
1st tooth distance (C x p)	1.1043 mm
Mill angle	1.79 mm
Programming data	
Z compensation (ztg) (entry; withdrawal) 90° tangential = (pitch/4)	0.552 mm
Taper Z compensation (zc) 90°	0.552 mm
Compensated depth (zl) Thread length + 1st tooth distance + (entry; withdrawal) tangential	18.437 mm
Entry tangential radius (Rf)	14.452 mm
Entry movement (g)	1.589 mm
Y start	14.364 mm
X start	16.040 mm
Radius taper offset (dc)	0.017 mm
Withdrawal tangential radius (Rfs)	14.521 mm
Withdrawal movement (gs)	1.594 mm
Y1 final	14.433 mm
X2 final	16.109 mm

Tool offset radius = 7.15

G	G	X	Y	Z	I	J	F
G90		0.000	0.000	20.000			
				0.000			
G91							
	G1			-18.437			500
G41	G1	0.000	-14.364				112
	G3	16.040	14.364	0.552	1.589	14.364	62
	G3	-16.040	16.057	0.552	-16.040	0.017	
	G3	-16.075	-16.057	0.552	-0.017	-16.057	
	G3	16.075	-16.092	0.552	16.057	-0.017	
	G3	16.109	16.092	0.552	0.017	16.092	
	G3	-16.109	14.433	0.552	-14.515	0.000	112
	G1		-14.433				500
G40	G90			20.000			

④

S3234 M03

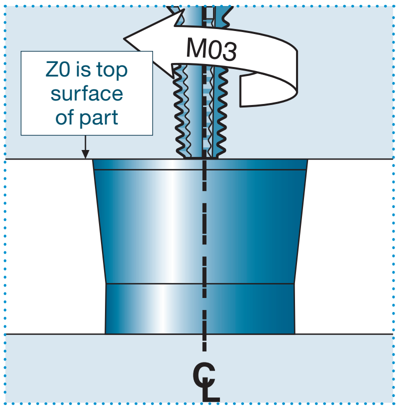
④ S indicates the speed in rpm

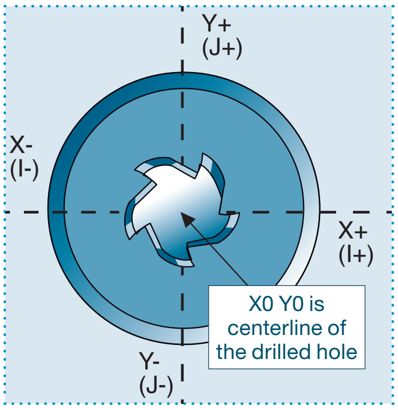
M03 indicates the spindle clockwise rotation. All Xactform thread mills turn in M03 mode.

G90 X0 Y0 Z0

G90 indicates an absolute program. This is the start position or origin of the program and all other positions will refer to this zero point.

In this case, the zero point is the hole axis at X and Y, and is located on the workpiece's top surface.





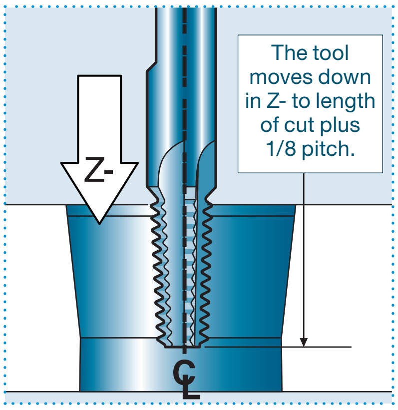
⑦

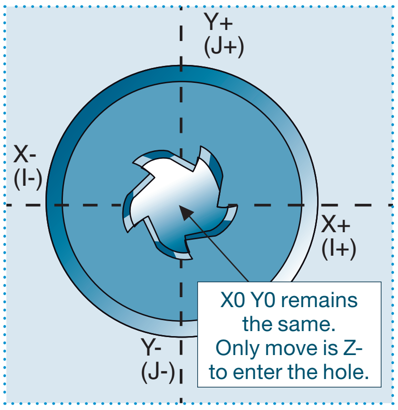
G01 X 0 Y 0 Z-0,7154 F 50

③ & ⑦ The tool now makes a negative Z movement into the bore, for a distance equal to the thread length plus 1/2 pitch plus 1/8 pitch. (0.661" + 0.0435 + 0.0109 = 0.7154.

NB: The characteristics for 1"-11.5 NPT require a minimum thread length of 0.661". An extra 1/2 pitch (0.0435") was added to take into account the distance from the end of the mill to the top (apex) of the first tooth.

F50 is the speed of the tool movement in inches/minute.



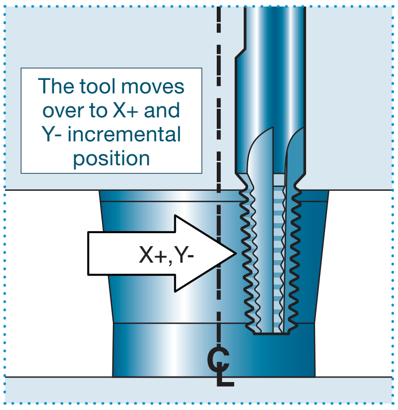


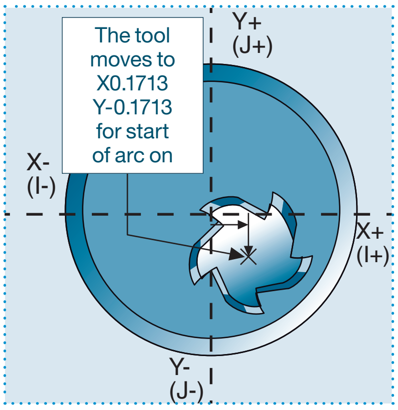
⑧

G4I X 0,1713 Y-0,I7I3

⑧ The tool now moves towards the start position to start the arc for the entire depth of the thread. Code G41 refers to the cutting tool compensation. ⑧ Note that the tool moves an equal distance in the X and Y axes. This is the incremental value calculated in the programming data sheet.

NB: A compensation register for the cutting tool must be added to this line. Generally a D number, this is used for the final sizing and to compensate for the tool wear. The current register must be zero for the first cycle, with adjustments being made after the final inspection (sizing) is complete. A negative sign in the register will increase the thread diameter, while a positive sign will reduce the thread size.





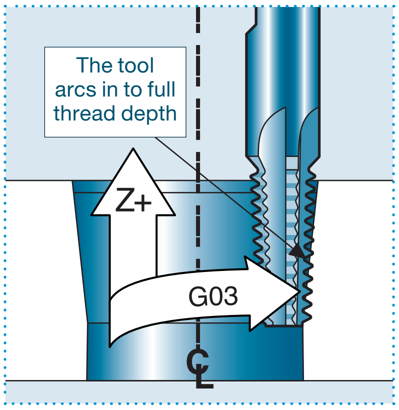
⑧ ③ ⑧ ⑥

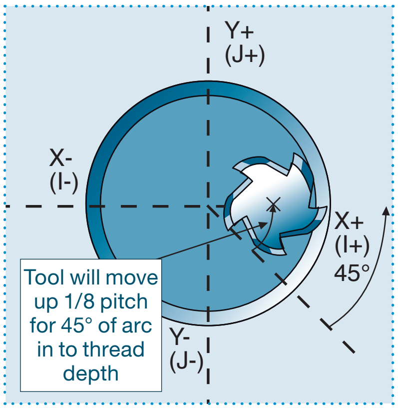
G03 X 0,3425 Y 0 Z-0,7045 I 0 J 0,1713 F10,I8

The absolute position ⑧ X0.3425 Y0 calculated using the data sheet is the milling end of travel position once the arc movement is complete.

Position Z ③ is the required complete thread length minus 1/8 pitch (0.0109").

The position ⑧ for the centre of the I-J arc is the incremental value and the programmed feedrate ⑥ is the compensated feedrate, not the normal PPM value for the feedrate calculated for straight milling.





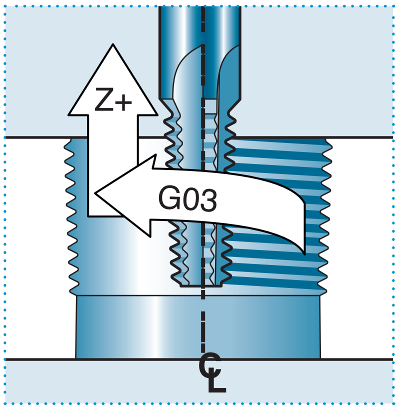
⑧&⑨ ③ ⑧

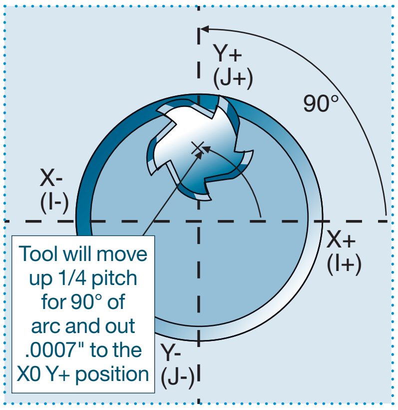
G03 X 0 Y 0,3432 Z-0,6828 I-0,3432 J 0 F10,18

The value X0 Y0.3432 is the absolute position ⑧ calculated using the data sheet plus ⑨ the compensation value 0.0007" for the arc movement once the tool has completed its first 90° arc movement.

Position Z ③ is the previous thread length minus 1/4 of pitch ①, or 0.7045 minus 0.0217 = 0.6828.

Position ⑧ of the I-J axis arc is the absolute value plus the X and Y compensation value 0.0007". The arc radius axis is the hole axis





Tool will move up 1/4 pitch for 90° of arc and out .0007" to the X0 Y+ position

⑧&⑨ ③&① ⑧

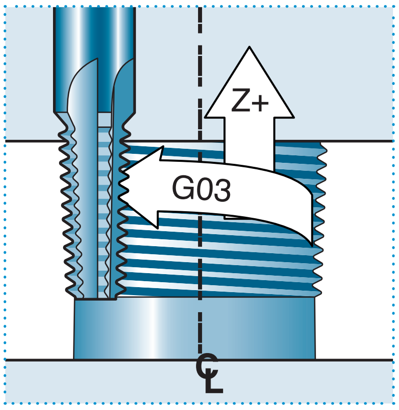
G03 X-0,3439 Y 0 Z-0,6611 I 0 J-0,3439 F10,18

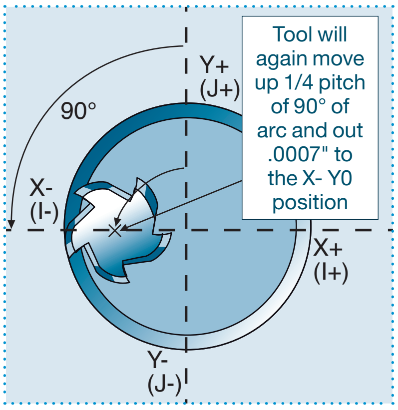
The value X-0.3439 Y0 is the absolute position ⑧ from the data sheet plus ⑨ the compensation value 0.0007" of the arc movement multiplied by 2 when the tool has completed two 90° arc movements or 180° of the total arc.

Position Z ③ is the previous thread length minus ¼ of pitch ①, or 0.6828 minus 0.0217 = 0.6611.

The position of the I-J arc axis ⑧ is the value I of the previous line plus the X and Y compensation value 0.0007".

Note that the value of I or J is equal to the corresponding value of X or Y.





Tool will again move up 1/4 pitch of 90° of arc and out .0007" to the X- Y0 position

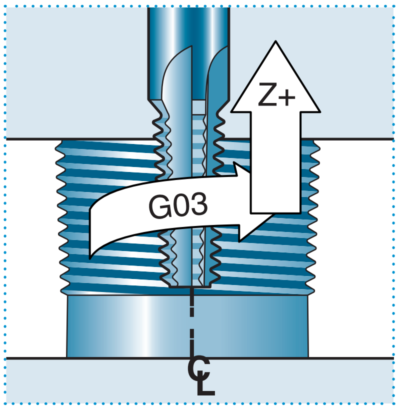
⑧&⑨ ③&① ⑧

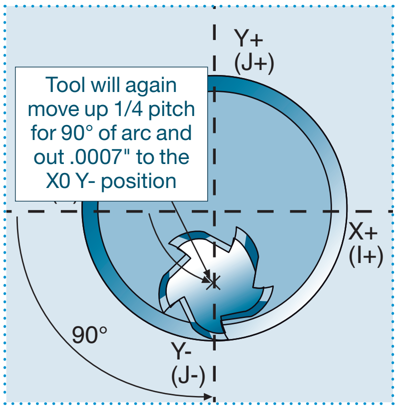
G03 X 0 Y-0,3446 Z-0,6394 I 0,3446 J 0 F10,18

The tool now moves towards position X0 Y-0.3446. 0.0007" is again added to the previous X value. ⑧ and ⑨ The compensation value 0.0007" for the arc movement multiplied by 3 once the tool has completed three 90° arc movements, or 270° of the total arc.

Position Z ③ is the previous thread length minus 1/4 of pitch minus an additional 1/4 of the pitch ①, or 0.6611 minus 0.0217 = 0.6394.

The position of the I-J arc axis ⑧ is the value J of the previous line plus another X and Y compensation value 0.0007".





Tool will again move up 1/4 pitch for 90° of arc and out .0007" to the X0 Y- position

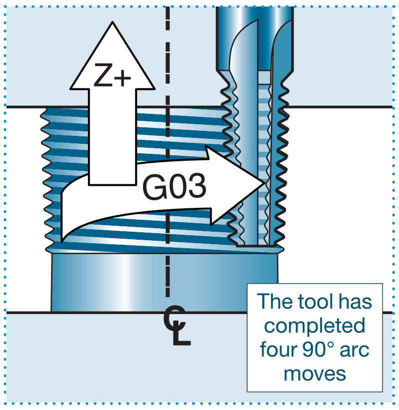
⑧&⑨ ③&① ⑧

G03 X 0,3453 Y 0 Z-0,6177 I 0 J 0,3453 F10,18

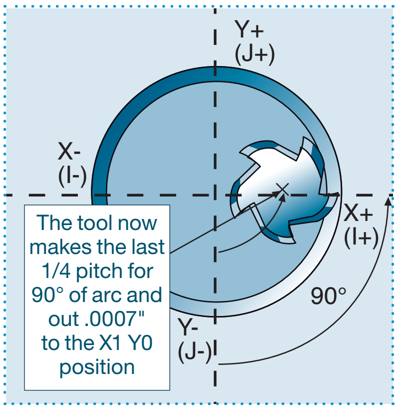
To complete 360°, position X0,3453 Y0 includes the last 0.0007" added to the previous Y value. ⑧ and ⑨ The compensation value of 0.0007" for the arc movement multiplied by 4 once the tool has completed its 360° rotation.

Position Z ③ is the previous thread length minus 1/4 of pitch ①, or 0.6394 minus 0.0217 = 0.6177.

The position of the I-J arc axis ⑧ is the value I of the previous line plus another 0.0007" as the X and Y compensation value.



The tool has completed four 90° arc moves



The tool now makes the last 1/4 pitch for 90° of arc and out .0007" to the X1 Y0 position

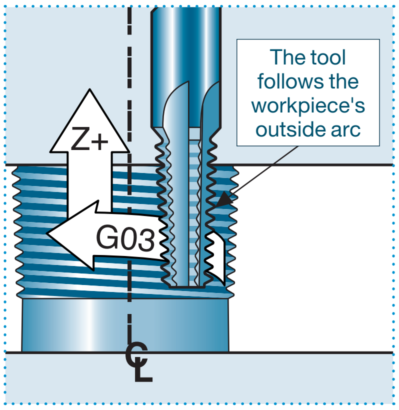
⑧&⑨ ③&⑦ ⑧

G03 X 0,1727 Y 0,1727 Z-0,6068 I-0,1727 J 0 F20,00

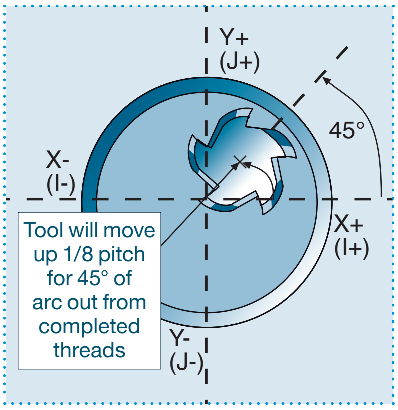
The tool now follows the arc on the outside, away from the completed threads. The value X0.1727 Y0.1727 ⑧ and ⑨ is half of the last absolute position X0.3453 at the end of the 360° arc position line.

Position Z ③ is the last Z position minus 1/8 of pitch ⑦, or 0.6177 minus 0.0109 = 0.6068. **NB: the outside arc must include a Z axis upward movement to ensure that the thread flanks are not deformed when the tool is withdrawn.**

The position of the **I-J** arc axis ⑧ is the value X and Y. A feedrate of 20"/min. is used when the tool is no longer cutting.



The tool follows the workpiece's outside arc



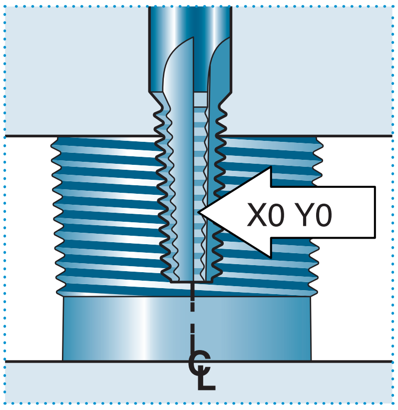
Tool will move up 1/8 pitch for 45° of arc out from completed threads

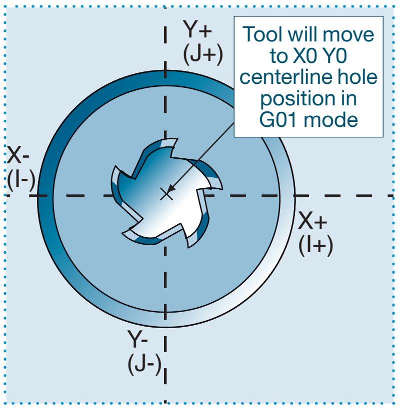
G01

G40 X 0 Y 0 F50

The tool now returns to the centre of the hole in a straight-line movement or mode G01. To ensure that the tool is centred, a G40 or cutting tool compensation code is used to delete any value which could have been used to correct the tool size.

Now that the tool is away from the threads, the movement can be executed at a speed of F50, or 50 inches per minute.



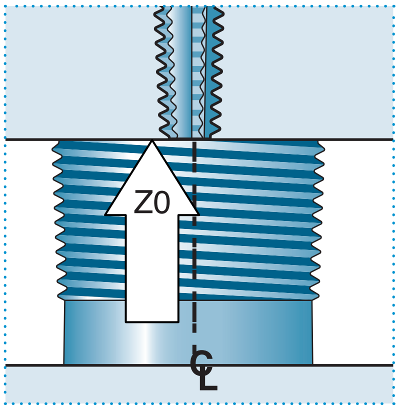


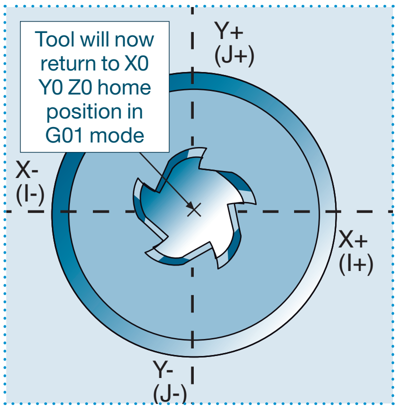
Tool will move to X0 Y0 centerline hole position in G01 mode

G00 Z 0

The tool now returns to the starting point or the original position in the Z axis only. Command G00 is the final phase of the program, and returns the tool at high speed to its original position.

This program is written in absolute values. Each base point refers to the original position or position X0 Y0 Z0 .





Tool will now return to X0 Y0 Z0 home position in G01 mode

(3)
TM insert range

(3.1) Listing

Reference

TM21 N 1.50 ISO -S-SXP

TM
Insert type

L
Insert length

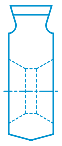
N
Internal

E
External

NE
Internal/external

Pitch (mm, inches)

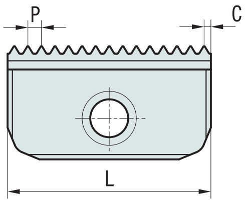
Standard



2 chip breakers unless specified

S
Single work side

SXP
single work side with release mechanism on the other



L
Total insert length

NF
Number of teeth

C
Centring

H
Profile height

Pitch	Reference	L	NF	C	Tool holder	H
0.50	TM14 E 0.50 ISO	14	28	0	TMH....-...-.-14	0.316

(3.2) Attachment and lubrication

Attachment

This is defined based on the gripper system fitted to the machining centre in question.

Lubrication

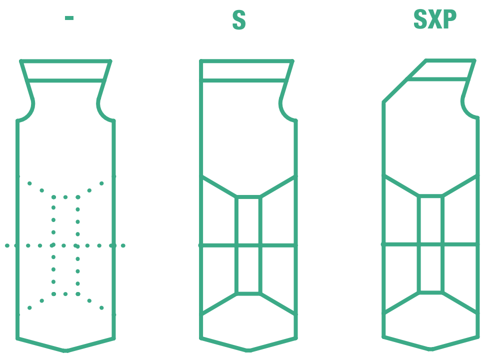
Xactform tool holders include one or more lubrication channels to optimise the cutting conditions depending on the material being machined. Versions without lubrication channels are available on request.

(3.3) Tool selection

Insert types

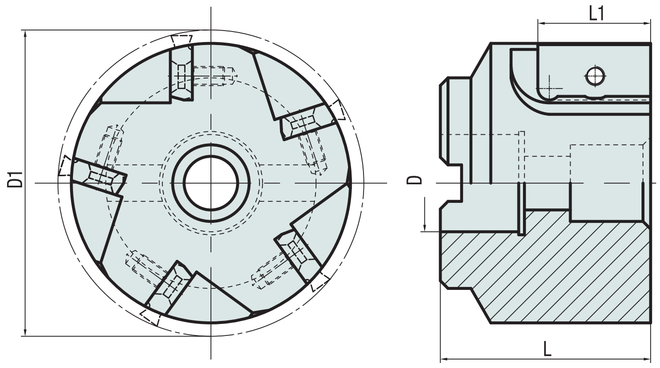
In certain dimensions, inserts are available in both standard as well as -S and -SXP versions:

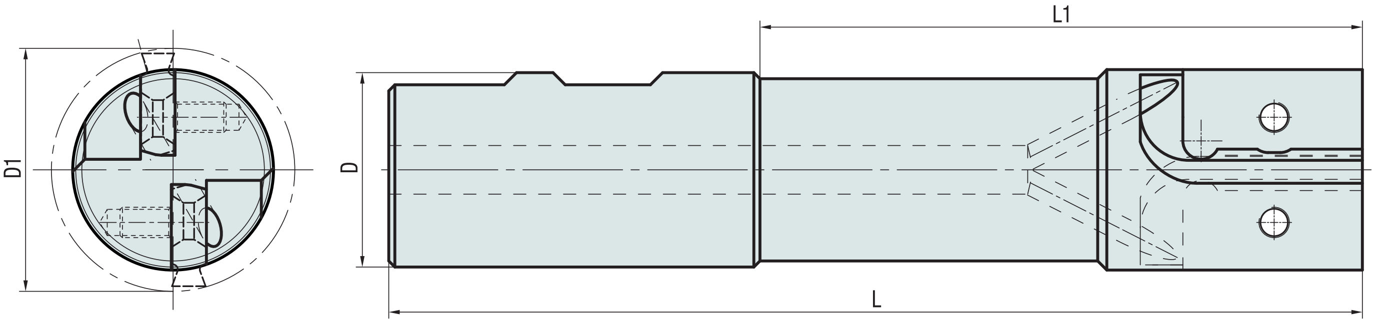
- the standard version includes a chip breaker on each side. The insert can therefore be reversed to double its service life.
- the S version (single chip breaker) is designed for machining hard metals.
- the SXP version (single chip breaker + release behind the tooth) is reserved for helix angles greater than 4°.



Number of inserts

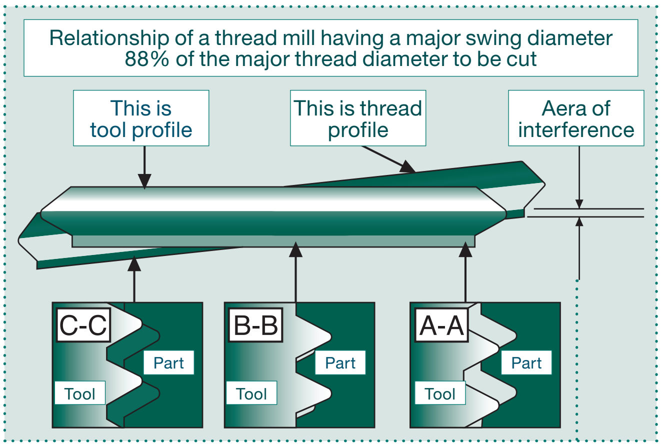
The number of inserts corresponds to the number of cutting edges; the higher the number, the quicker the thread is machined.

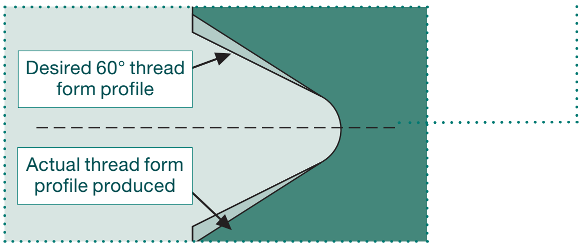


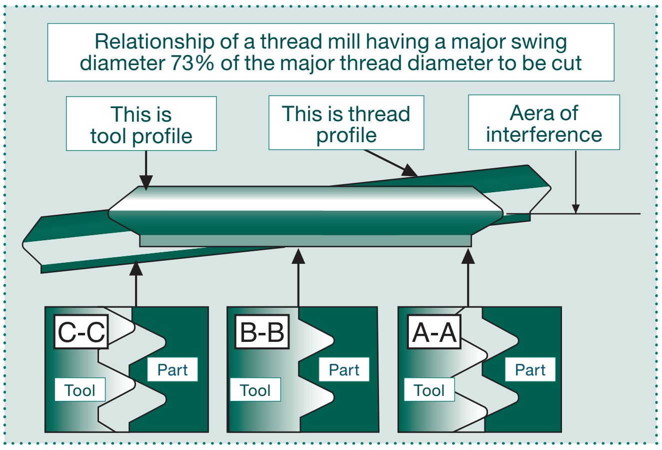


Insert sizes

As with solid carbide mills, when machining non-standard diameters, the tool's cutting diameter must not in principle exceed 75% of the thread diameter. For certain special applications, this value can rise to 80%. However, if the helix angle is greater than 4°, the diameter must be reduced to prevent ploughing. Contact us for more information.







Length

The standardised depth of a thread is 1.5 x the diameter. This depth cannot be increased, as the screw head will break before the thread.

For special applications with a greater depth, use the Xactform Long tool holder.

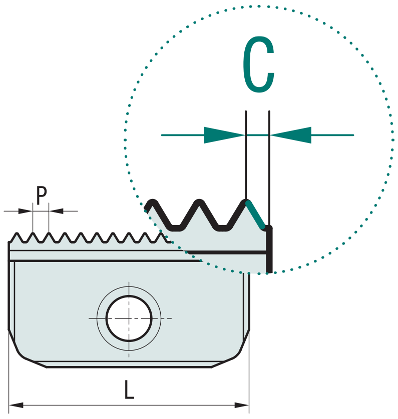
Left-hand and right-hand thread

Tool holders for standard indexed thread inserts are not designed for left-hand threads. Please contact us if you require more information.

Special tools

If the standard tools cannot be used to produce the desired thread, Xactform has the resources to very quickly develop special inserts or tool holders enabling it to be created.

Go to
http://www.xactform.ch/fr/pages/form.php
for a quick quote!



Centring the inserts

The insert's cutting profile is centred on the insert's length L, which allows it to be reversed without offsetting the cutting profile.

The centred position is the offset between the edge of the insert and the start of the first pitch.

(3.4) Cutting conditions

See table on pages 14-17.

$$n\ (rpm) = \frac{Vc\ (m/min) \times 1000}{\pi \times D1(mm)} \qquad Vf(mm/min) = n\ (rpm) \times fz\ (mm) \times z$$

Preparing for thread milling

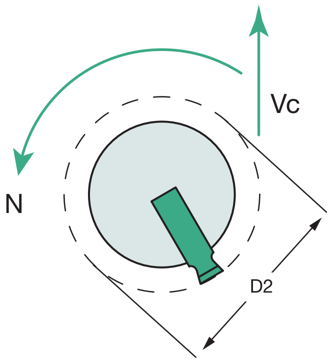
1. Calculating the cutting feed rate and rotation speed.

$$N = \frac{1000 \times V}{\pi \times D2}$$

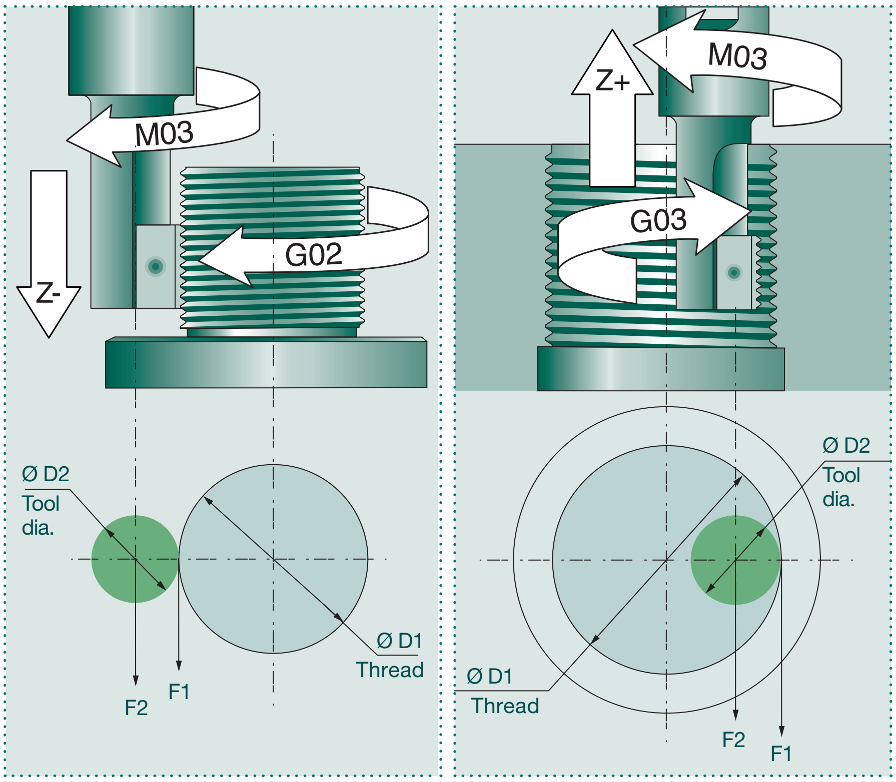
$$V = \frac{N \times \pi \times D2}{1000}$$

$$F1 = N \times z \times f$$

N: Rotation speed [RPM]
V: Cutting speed [m/min]
D2: Tool cutting diameter [mm] [mm]
F1: Tool working feedrate [mm/min]
z: Number of cutting edges
f: Feedrate per tooth per rotation [mm/tooth]



2. Calculating the working feedrate on the tool shaft


External thread
Internal thread

On most CNC machines, the feedrate required for programming is that of the tool shaft. When the tool receives a linear movement command, the feedrate on the cutting edge and the shaft are identical, however this is not the case when the tool movement is circular. The equations define the relationship between the feedrates on the cutting edge and the tool shaft.

(3.5) Programming principles

Straight and tapered threads

Refer to page 24 on solid carbide mills.

Thread milling in steps

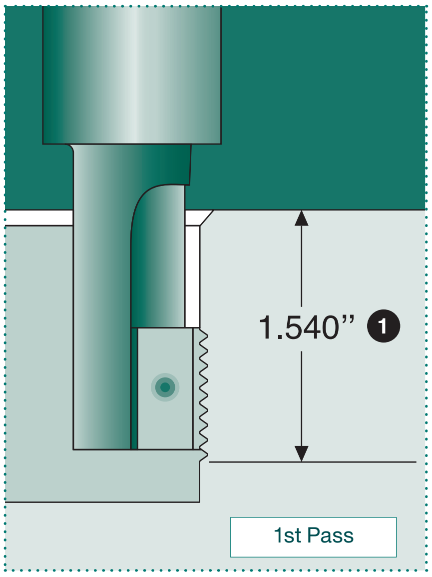
In cases where the thread depth will exceed the length L of the insert, one or more additional interpolations are made, each offset by the height of the insert.

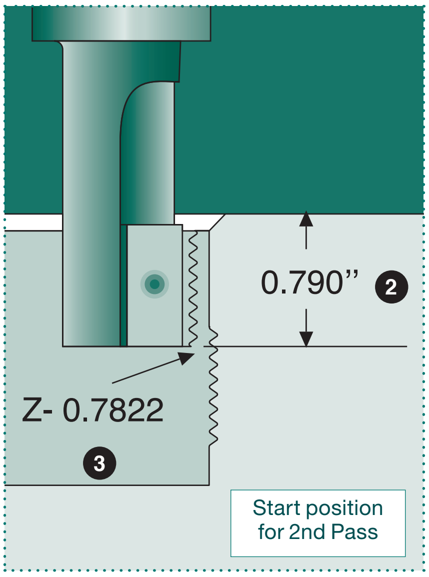
Calculating the depth (Z axis)

Example:
M56 x 5.50 ISO, working depth A = 96mm
Insert: TM40 N 5.50 ISO -> length L = 40mm
-> centring C = 0.75 mm

step 1: $Z = A + 1/8\,P + C + 1/2\,P$

step 2: $Z = L - (P + (2 \times 1/8\,P) + 2C)$





(4)

Thread turning insert range

(4.1) Listing

Reference

11 ER 0.35 ISO 3M CM

L
Theoretical length of insert side.

ER
Right-hand external

EL
Left-hand external

NR
Right-hand internal

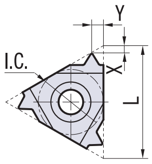
NL
Left-hand internal

Pitch (mm, inches)

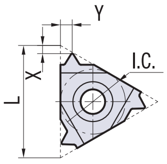
Standard

Number of teeth

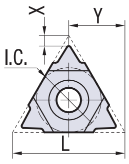
CM
Special grade (ceramic - metal) for special applications



Right-hand internal (NR)



Left-hand internal (NL)



U

IC
Insert recorded diameter

X
Profile position vertical reference

Y
Profile position horizontal reference

IC	Pitch	Right	Left	L	X	Y	Anvil		Tool holder
1/4"	0.35	11NR 0.35 ISO	11NL 0.35 ISO	11	0.8	0.3	-	-	PO**-**-11N*

(4.2) Tool selection

Determining the sub-insert

For standard threads, the helix angle is approximately 1°30'. Xactform tool holders are manufactured with this angle.

For non-standard threads, the helix angle must be calculated using the following formula:

$$\text{tg}\beta = \frac{\text{Pitch}}{\pi^{*}\ \text{thread Ø}}$$

The angle for positioning the insert in the tool holder (10° for external tool holders and 15° for internal tool holders) creates a **lateral clearance angle**. If the helix angle is greater than 1°30', the positioning must be corrected to prevent lateral friction on the insert. This is done by adding a sub-insert.

Table of sub-inserts

		Helix angle							
		4.5°	**3.5°**	**2.5°**	**1.5°**	**0.5°**	**0°**	**-0.5°**	**-1.5°**
Insert	**Holder**	**Sub-insert command reference**							
3-3/8"-16	ER NL	YE3-3P	YE3-2P	**YE3-1P**	YE3	YE3-1N	YE3-1.5N	YE3-2N	YE3-3N
3-3/8"-16	EL NR	YI3-3P	YI3-2P	YI3-1P	YI3	YI3-1N	YI3-1.5N	YI3-2N	YI3-3N
4-1/2"-22	ER NL	YE4-3P	YE4-2P	YE4-1P	YE4	YE4-1N	YE4-1.5N	YE4-2N	YE4-3N
4-1/2"-22	EL NR	YI4-3P	YI4-2P	YI4-1P	YI4	YI4-1N	YI4-1.5N	YI4-2N	YI4-3N
5-5/8"-27	ER NL	YE5-3P	YE5-2P	YE5-1P	YE5	YE5-1N	YE5-1.5N	YE5-2N	YE5-3N
5-5/8"-27	EL NR	YI5-3P	YI5-2P	YI5-1P	YI5	YI5-1N	YI5-1.5N	YI5-2N	YI5-3N
4U-1/2"U-22U	ER NL	YE4U-3P	YE4U-2P	YE4U-1P	YE4U	YE4U-1N	YE4U-1.5N	YE4U-2N	YE4U-3N
4U-1/2"U-22U	EL NR	YI4U-3P	YI4U-2P	YI4U-1P	YI4U	YI4U-1N	YI4U-1.5N	YI4U-2N	YI4U-3N
5U-5/8"U-27U	ER NL	YE5U-3P	YE5U-2P	YE5U-1P	YE5U	YE5U-1N	YE5U-1.5N	YE5U-2N	YE5U-3N
5U-5/8"U-27U	EL NR	YI5U-3P	YI5U-2P	YI5U-1P	YI5U	YI5U-1N	YI5U-1.5N	YI5U-2N	YI5U-3N

Example

Item: M20 x 3

Helix angle: 2°44'

Tool selection: PO12-12-16ER / 16ER 3.00 ISO

Selecting the sub-insert YE3-1P

Y
Sub-insert

E/I
External/internal version

3
Insert size
(4U and 5U = U insert)

1
1° difference compared to standard angle (1°30')

P / N
Positive/Negative

Cutting conditions

$$\textbf{Rotation of the machined part n}\ [\text{rpm}] = \frac{\text{Vc [mm/min]}}{\pi^{*}\ \text{D1 [mm]}}$$

(4.3) Cutting conditions

$$n\ (rpm) = \frac{Vc\ (m/min) \times 1000}{\pi \times D1(mm)}$$

Turning cutting conditions

Materials to be machined		HM		Cutting speed per coating			
		STD	P3O	V1	V2	V3	V4
	N/mm2	Vc (m/min)					
Non-alloy steel/Low-alloy steel	<400 <500 <600		80 - 150	95 - 190 80 - 175 70 -155	140 - 300 120 - 280 110 - 280		
Non-alloy steel/Low-alloy steel	<600 <900 <1200		70 -130	65 - 145 55 - 140 50 - 135	100 - 230 90 - 200 80 - 180		
Leaded bar turning steel			80 -140	100 - 175	120 - 180		
High-alloy steel/Stainless steel	400 - 700			60 -140	70 - 150		
High-alloy steel or cast iron/High-temperature stainless steel	700 - 1500			40 - 120	40 -120		
Special alloys (Inconel, Nimonic, Hastelloy)				15 - 30	15 - 30		
Grey cast iron/Pearlitic spheroidal graphite cast iron	<600 <900			70 - 130 60 - 115	70 - 130 60 - 120		
Cast iron alloy/Pearlitic spheroidal graphite cast iron	<550 <900			125 - 160 90 - 120	120 - 170 90 - 120		
Malleable cast iron/Ferritic spheroidal graphite cast iron	<450 <800			70 -160 60 - 145	70 - 170 60 - 150		
Titanium, titanium alloy	400 1050	60 - 100 40 - 60		140 - 170 50 - 70	140 - 170 50 -70		
Copper alloy (brass, bronze)	<350	70 - 170		80 - 225	80 - 200	80 - 200	
Copper alloy (aluminium bronze)	<370	70 - 170		80 - 225	80 - 200		
Aluminium alloy/Magnesium alloy	<370	100 - 600		80 - 365	80 - 240	100 - 240	
Cast aluminium	<350	70 - 120		200 - 400		200 - 400	200 - 1000
Cast aluminium	<450	50 - 120		60 - 180	60 - 180		
Plastic		80 - 160		200 - 400			200 - 400
Gold, silver		70 - 170		80 - 200		80 - 220	

For small profiles, we recommend selecting the minimum speed.

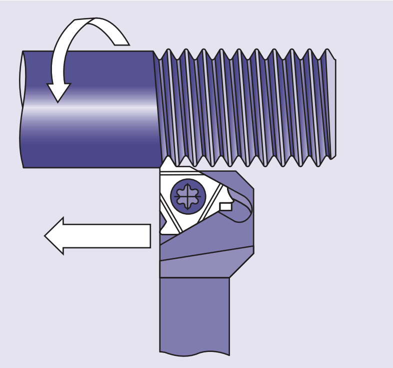
Recommended number of passes

Pitch	mm	0.50	0.75	1.00	1.25	1.50	1.75	2.00	2.50	3.00	3.50	4.00	4.50	5.00	5.50	6.00	8.00
	tpi	48	32	24	20	16	14	12	10	8	7	6	5.5	5	4.5	4	3
No. of passes		4-6	4-7	4-8	5-9	6-10	7-12	7-12	8-14	9-16	10-18	11-18	11-19	12-20	12-20	12-20	15-24

(4.4) Machining methods

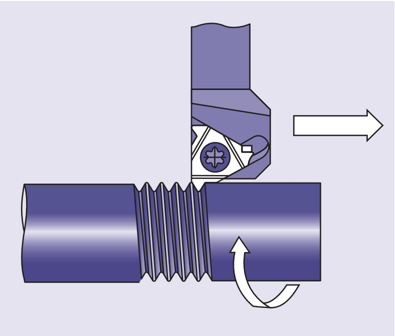
1. Right-hand external thread

Right-hand version (ER)



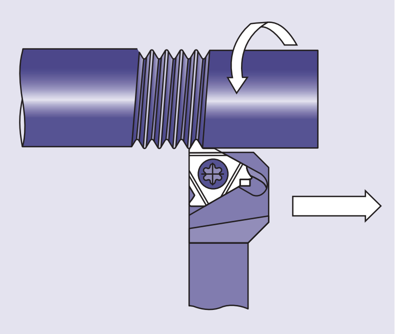
2. Right-hand external thread

Left-hand version (EL)



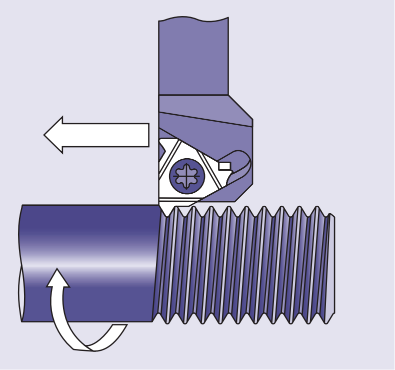
3. Left-hand external thread

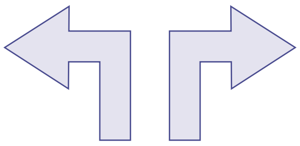
Right-hand version (ER)



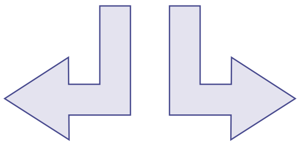
4. Left-hand external thread

Left-hand version (EL)



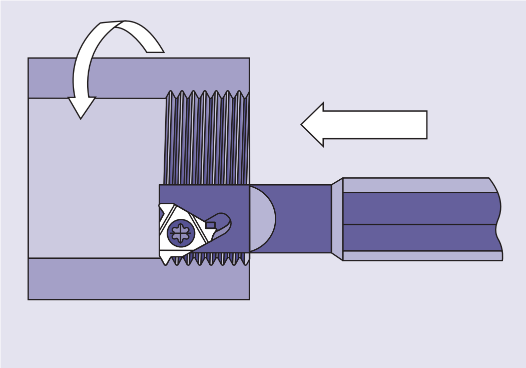


Thread milling with inverted helix angle: use a sub-insert with a negative angle



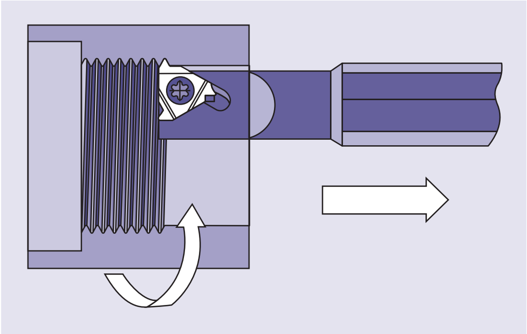
5. Right-hand internal thread

Right-hand version (NR)



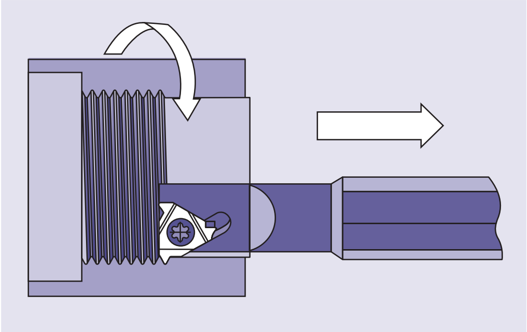
6. Right-hand internal thread

Left-hand version (NL)



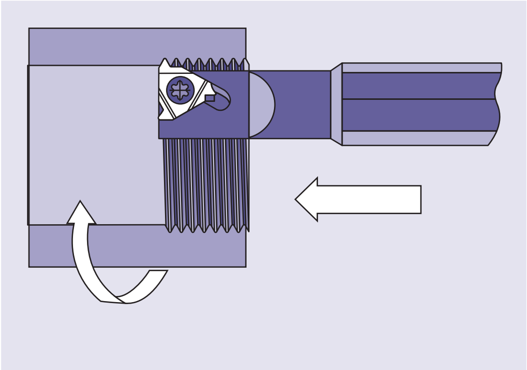
7. Left-hand internal thread

Right-hand version (NR)



8. Left-hand internal thread

Left-hand version (NL)

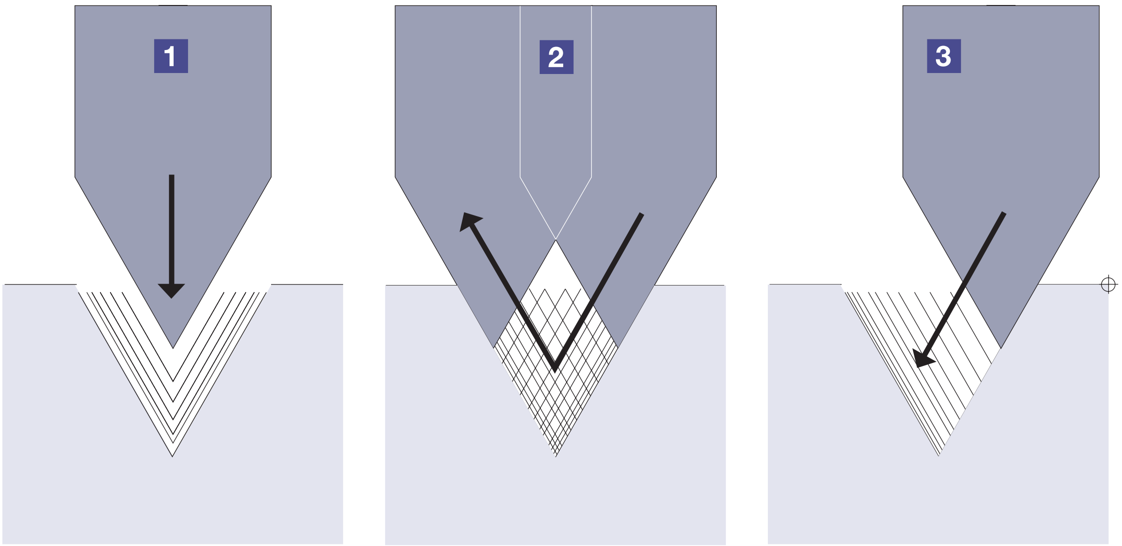


(4.5) Programming principles

Thread turning

Thread turning is executed in several successive passes, each deeper than the last. There are 3 execution options, with the first one the most commonly used.

The 3 thread turning options



Programming example

M20 x 1 thread

ISO machining cycle (SCHAUBLIN)

T01	**Tool no.**
G0 **G97** M3 S1000 X22 Z5	**Auto cutting speed in relation to Ø**
G197 X20 Z-20 H0.64 S0.03 Q0.3 M0.03 V3 A60 F1	**Multiple pass thread cutting cycle**
M30	

Large pitch machining tools

For pitches of between 10 and 25 mm, Xactform produces a range of specially adapted TS inserts which offer high strength thanks to their special seating.

All shapes are available on request, including for applications other than thread cutting, such as tools for O-ring grooves, grooving, etc.

Europe

Champs-Montants 16A
CH - 2074 Marin-Epagnier
Switzerland

Tel +41 (0) 32 729 10 00
Fax +41 (0) 32 729 10 01

info@xactform.ch
www.xactform.com

USA

Xactform USA, Inc

11 Miracle Street Loop
PO Box 20307
Panama City Beach, FL 32407

Tel: 850-230-9848
Toll Free: 888-783-8122
Fax: 850-230-9851

Information: info_usa@xactform.com
Orders: sales_usa@xactform.com
Website: www.xactform.com