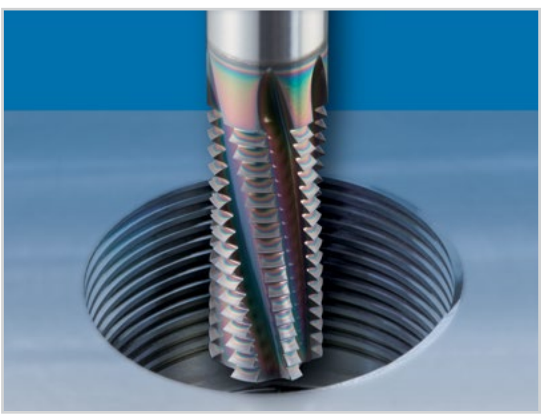
Vol 1

TECHNICAL GUIDE
THREAD MILLS

For more information
scan the QR code to visit:
osgtool.com/thread-milling

Technical Data *Thread Mills*

CONTENTS



Today's machining environment emphasizes rapid turnaround of limited product quantities, demanding more just-in-time production from all sources. While manual machining operations still widely exist, most modern operations are conducted using advanced machining centers. To make the best use of these machining centers, advanced tooling is required, ensuring continued operation even while the machine is not under direct supervision.

However, internal threading processes are sometimes undesirable to leave unattended due to the stresses put on tooling, especially in tapping operations. Unexpected problems can be costly, resulting in a difficult trade – manpower or possible added costs.

Thread mills exist to combat this, making use of the modern computer numerical control (CNC) that ensures machine accuracy even when moving in the X-, Y-, and Z-axes. This allows for simultaneous movement to create a thread without the full engagement of a tap, providing an avenue for unmanned threading operations.

OSG is a forerunner in thread mill development and, through the experience developing and testing the suite of thread milling tools, has created this technical guide to assist the reader in the proper application and usage of thread milling tools.

1 Thread mill series

1.1 Features

(1) One tool can create threads of multiple diameters if they have the same pitch, and can also create screw thread insert (STI) threads. One tool can also thread both right and left handed threads.

(2) Since CNC programming allows for direct adjustment of machining dimensions, one tool can thread for multiple different thread limits. This eliminates the need for carrying multiple taps with various thread limits.

(3) Since the mechanism of cutting is similar to side milling, thread mills can machine large diameter threads at lower machine power requirements than conventional tapping. (Requires Machine Rigidity)

(4) Thread milling produces small, broken chips, similar to those seen in solid end milling operations. Since there are no long, stringy chips produced, there are rarely any problems associated with chips, allowing for stable, stress-free machining.

(5) Water soluble coolant is ideal for thread milling, eliminating the need for cutting oil.

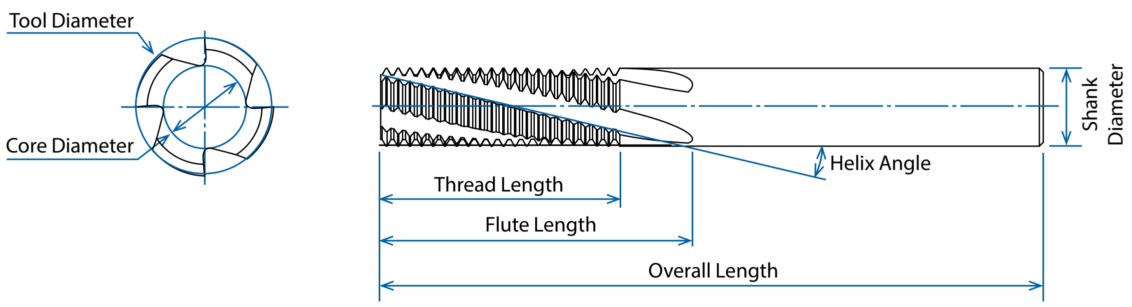
(6) Unlike taps, since there is no chamfer portion, thread mills can machine closer to the bottom of blind holes.

(7) In tapered pipe taps, thread mills are able to overcome the previously unavoidable "stop marks" while also producing holes with better circularity.

1.2 Structures

Figure 1 below shows the geometry of a basic thread mill. Although they are used for machining internal threads, there is no chamfer portion that is found in regular taps. Thread mills have threads oriented in a circular pattern around the tool, contrasting with a tap's helical thread which wraps around the tool. Since thread mills are held in regular milling holders (like milling chucks and collets), they do not have a square on the end. The outer diameter of the thread portion has an eccentric relief applied to minimize contact of anything but the cutting edge of each tooth.

Figure 1: Thread Mill Geometry

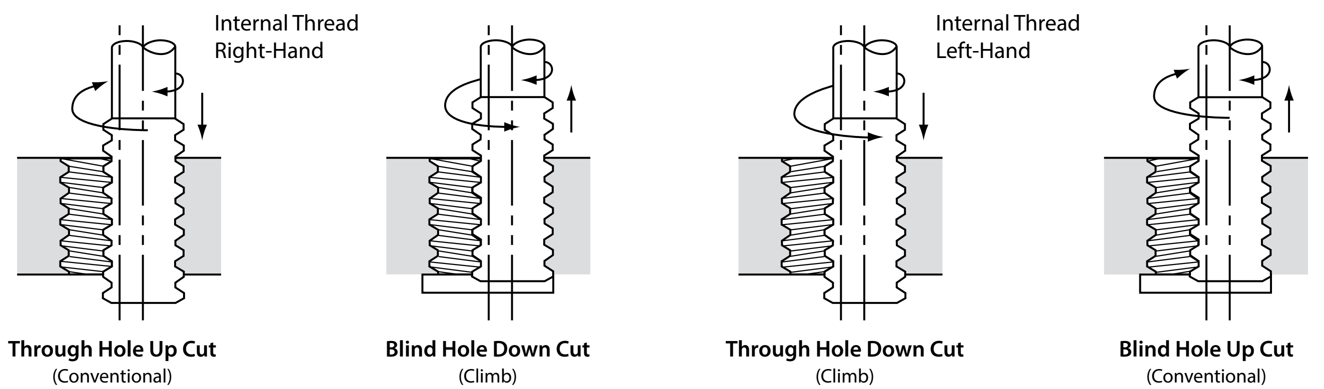


2 Threading process

2.1 WX-PNC, OT-SFT-PNGT, OT-PNGT, PNGT, HY-PRO P, WX-ST-PNC, WXO-ST-PNC

Unlike taps, thread mills do not have a helical lead on the threads. Positioning the tool parallel to the hole's axis, the tool is both rotated (spindle rotation), and revolved (helical feed, usually in the X- and Y-axes) while moving 1 pitch length per revolution (usually in the Z-axis). This is how the helical thread is created. Since the tool moves in the X, Y, and Z axes at the same time, the process requires CNC machinery capable of simultaneous 3-axis movement. Right hand or left hand thread, as well as climb or conventional milling approaches are all determined by both the direction of revolution and the axial feed (fig. 2). Generally speaking, climb milling is preferred for preserving tool life, while conventional milling produces better surface finish and accuracy. When thread milling blind holes with a maximum effective thread length, it is recommended to feed the tool in the upward direction (starting at the maximum depth) to avoid interference with the chips created from cutting the material.

Figure 2: Internal Thread Processing Methods



2.2 DR-PNAC, DR-O-PNAC (Super Thread mill)

①② The tool drills holes with the drill point, continuing until the chamfer portion engages and cuts the hole chamfer.

Since the thread cutting diameter is smaller than the drill point diameter, the threading teeth do not contact the drilled hole in these steps.

③ The tool retracts axially by 1 or 2 pitches of distance - this is to ensure the chamfer portion is no longer contacting with the workpiece.

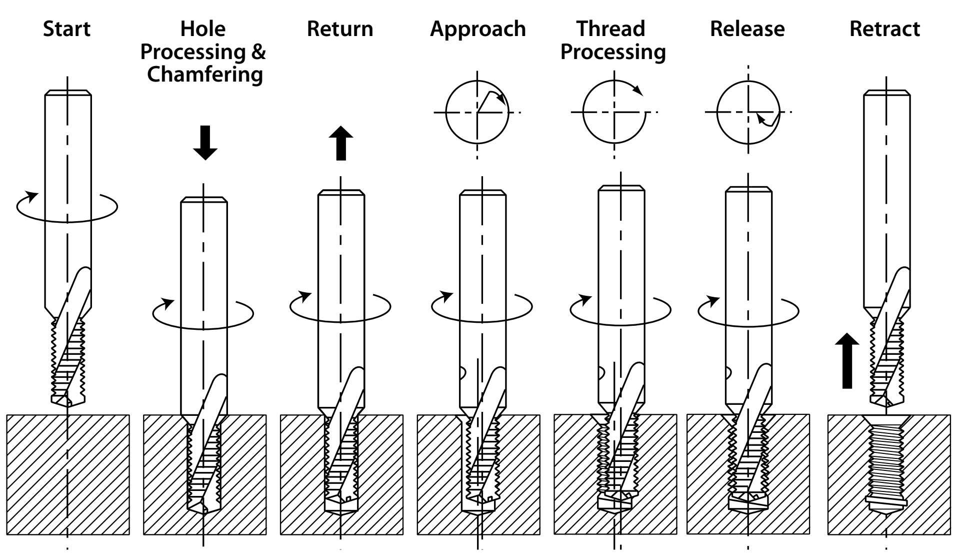
The amount of retraction is determined by the tool feed direction and thread length.

④⑤⑥ The threads are cut by helical interpolation (the tool is rotated while being revolved into the hole at one revolution for pitch length).

The tool approaches the interior wall in an arc to ensure the best surface finish and productivity. Since the drill point will still be cutting, some threads will have a small relief cut into them.

⑦ The tool returns to the center of the hole and retracts straight upward, completing the threaded hole.

Figure 3: Thread Mill Processing



2.3 WH-VM-PNC (General Work Material),

Since the WH-VM-PNC tool has a length of cut of three pitches, it machines a maximum of 3 pitches per revolution. In common materials such as carbon steels, aluminum alloys, and stainless steels, threading should be completed starting at the bottom and continued three pitches at a time, each time returning to center and moving up to the next set of three pitches. With a thread mill of insufficient cutting length, a thread can be produced with multiple steps moving axially out of the hole.

2.4 WH-VM-PNC (HRSA)

For difficult materials, such as heat-resistant alloys, the WH-VM-PNC cutter is recommended.
Since machining these materials typically results in high cutting forces, the machining method must change to accommodate. The first cut remains the same as in 2.3, but to ensure the best quality instead of a three-pitch movement each time the tool should perform helical interpolation for the entire length of thread, creating the thread one pitch at a time. In this case there is no retraction back to the center until the complete length of thread is machined.

Figure 4:
Machining Method with WH-VM-PNC
(General Work Material)

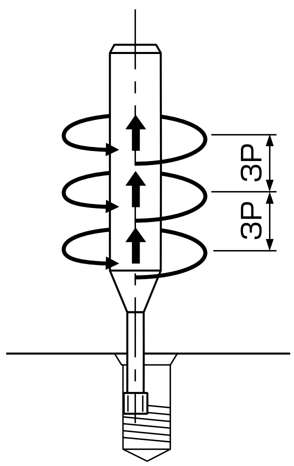
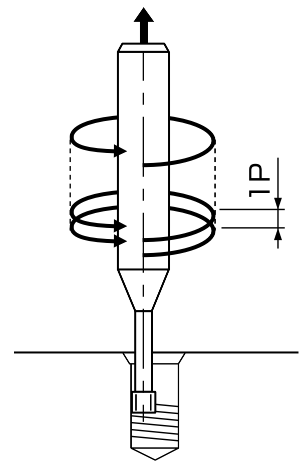


Figure 5:
Machining Method with WH-VM-PNC
(HRSA)



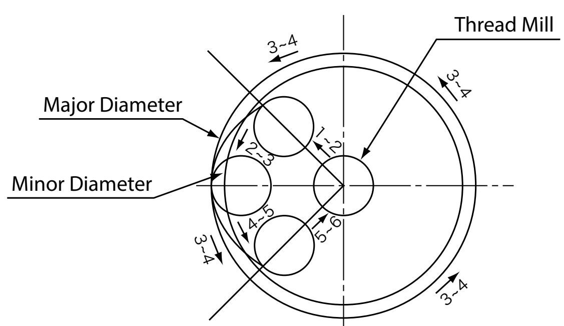
3 Threading process

3.1 Thread milling process

Process OrderProcess Description
1-2 Rapid movement to near material, leaving clearance for entry
2-3 Entering material in an arc
3-4 Thread milling (360° helical interpolation)
4-5 Exiting material in an arc
5-6 Rapid movement to center

The optimal path of a thread mill during entry into and exit from the material is an arc. This will shorten the thread length machined vs. the tool's thread length - the tool must be moving along the thread lead during entry and exit. For optimal quality and productivity, thread milling should follow the process in Figure 6.

Figure 6: Cutting Operation



Top View of Internal Threading

3.2 Internal thread cross section and tool path

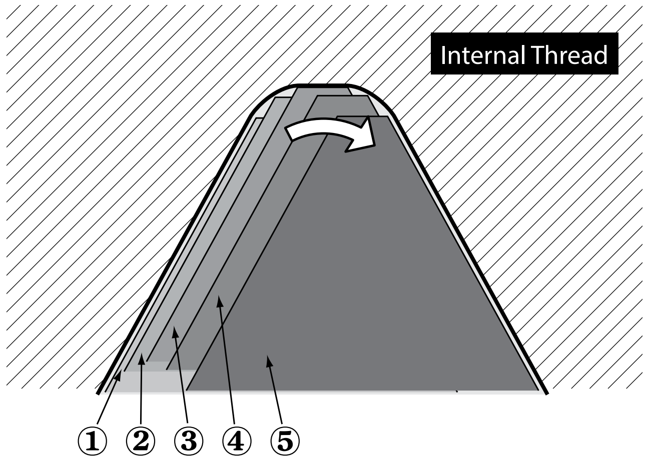
Thread mills are threaded cutters with no helical lead, and are used as a rotary tool to cut threads.
Figure 7 shows a cross section of the movement of a thread mill when creating internal threads.
A cutter with a smaller diameter than the thread is used to create the internal thread profile. The amount of tool engagement is determined by the hole diameter, tool diameter, internal thread lead, and the internal thread flank angle. As the tool diameter gets closer to the pre drill hole diameter, the total tool engagement increases. In order to create an effective thread, truncation of crest on tap must be designed to be smaller.

The cross section processed in the order of ① to ⑤ to finally create a full internal thread.

3.3 Cutting torque

When compared to taps, thread mills require significantly less torque to machine threads. Figure 8 shows the difference in tapping torque between a conventional tap and a thread mill. When machining large diameter holes, the large amount of torque required may cause less powerful machines to stall, while thread mills mitigate the torque required and allow for large diameter tapping with relative ease.

Figure 7: Cross Section of Internal Thread and Movement of Tool Thread

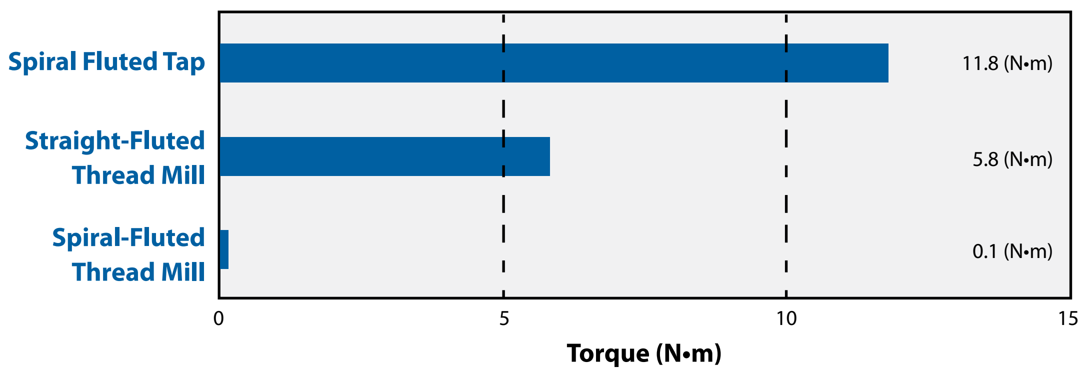


The cross section processed ① to ⑤ creates a full internal thread.

**Thrust force may be higher than tap.*

Figure 8: Difference in Threading Torque

Internal Thread: M16 x 1.5 • **Work Material:** 1045 Steel • **Thread Depth:** 30mm



3.4 Chip shape

Regardless of the work material, chips from thread milling resemble those found in figure 9.
With spiral fluted taps the chips in carbon steels, mild steels, and stainless steels resemble figure 10.
The difference in chip shape indicates the ease of evacuation – thread mills have very minor concern about chip packing and resulting damage or breakage, whereas spiral fluted taps may have chip packing or birdnesting due to the extended chip length.

Figure 9: Thread Mill Chips

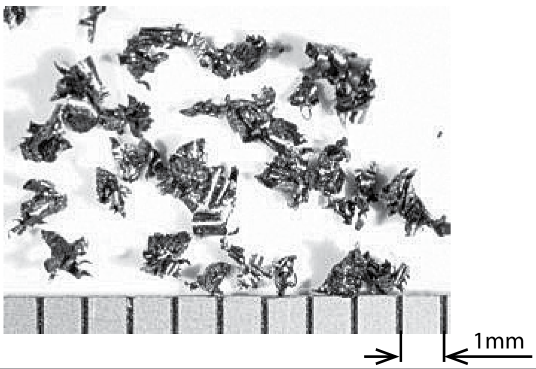
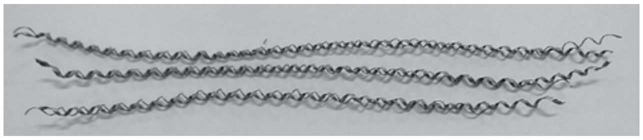


Figure 10: Spiral Flute Tap Chips



3.5 Surface Finish

When machining threads with taps, we are often left with "stop marks" (fig. 11) at the point where the tap stops cutting and begins to reverse. This stop mark is a remnant of material left uncut after chips break.
Generally the stop marks exist on a small portion of the finished thread, but tapered pipe threads will have them along the entire thread length, potentially interfering with a tight seal following assembly.
When the male and female end of a pipe fitting are connected, this stop mark can create enough of a gap to allow the leakage of fluid. Thread mills do not leave a stop mark (fig. 12), thus are the optimal solution for leak-free pipe threading.

Figure 11: Tapped Tapered Pipe Threads

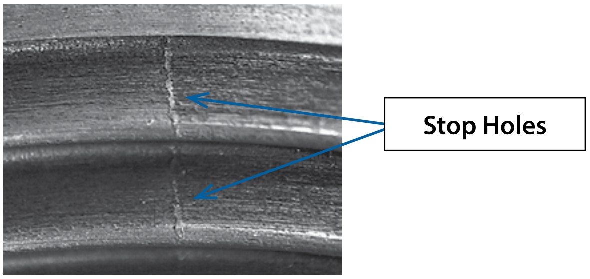
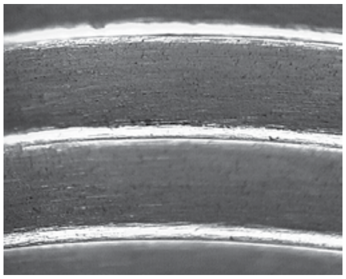


Figure 12: Thread Milled Tapered Pipe Threads



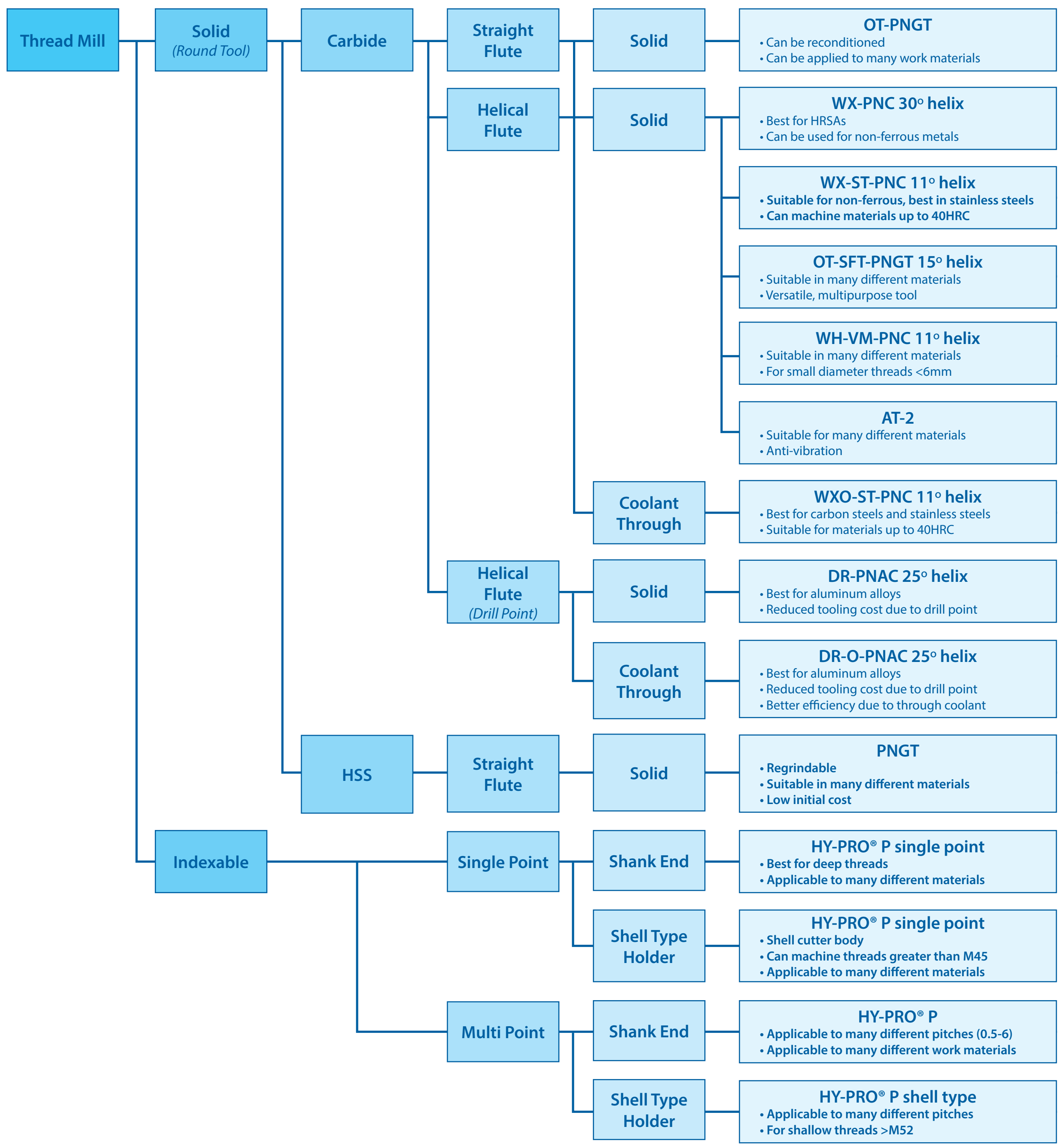
4 Tool Selection

In order to create effective threads, choosing the correct tool is a very important step. It is important to understand the function and purpose of a tool when making a selection.

4.1 Types of Thread Mills and Points for Selection

Thread mills are roughly classified as shown in Fig. 13. This chart outlines the features of each tool in order to aid tool selection.

Figure 13: Types and Features of Thread Mill Series



4.1.1 Carbide Thread Mill (OT-PNGT, OT-SFT-PNGT, WX-PNC, WX-ST-PNC, WXO-ST-PNC, WH-VM-PNC)

Thread mills typically have been used in hardened materials; however, recent advancements in thread mill geometry have allowed for an increase in application range for thread milling cutters. Straight fluted thread mills, such as OT-PNGT, are advantageous due to the ease of regrinding, but for general purpose machining the spiral fluted thread mills, like OT-SFT-PNGT, allow for increased application range.

It is still important to consider the material type when choosing a thread mill. For HRSAs and non-ferrous metals the WX-PNC cutter is best. For carbon steels, stainless steels, and alloy steels under 45HRC, the WX-ST-PNC and WXO-ST-PNC are recommended. The WXO-ST-PNC has coolant-through capabilities allowing for improved chip evacuation, resulting in trouble-free machining.
For smaller diameter threads (under M6), the WH-VM-PNC thread mill is recommended. This thread mill is also capable of machining titanium- and nickel-based HRSAs.

4.1.2 Super Thread Mill (DR-PNAC, DR-O-PNAC)

The Super Thread Mill is capable of machining the pilot hole, then threading and chamfering with a single tool. Due to its multi-purpose nature the DR-PNAC is best suited for machines with limited locations in their tool magazine. Although the DR-PNAC can combine multiple processes into one, it is only recommended for aluminum alloys and cast iron. The DR-O-PNAC cutter has through coolant capabilities allowing for greater productivity compared to its solid tool counterpart.

4.1.3 High Speed Steel Thread Mill (PNGT)

When machining larger threads, a high speed steel thread mill is recommended to avoid substantial up front cost. Furthermore, high speed steel thread mills are available in larger diameters than their carbide counterparts, allowing machining of threads size M34 or greater. Since HSS thread mills are straight fluted, they can easily be reconditioned.

4.1.4 Indexable Thread Mill (HY-PRO P, HY-PRO P single point)

Indexable thread mills are best suited for large threads (larger than M30) and lower production volumes due to the limited number of flutes available on the tool. A single tool body can be used for multiple different pitches, making indexable thread mills a good option for job shops or small production runs. Additionally, though initial cost is typically somewhat high, the long-term running cost is typically lower than solid carbide options. For thread lengths greater than 80mm, the HY-PRO P single point thread mill is recommended – the tool engages just one pitch of the thread at any time, limiting the amount of force applied to the tool and all but eliminating tool deflection.

4.2 Tools for Internal and External Threads

In general, thread mills can be used for both internal and external threads. However, due to the differences in thread crest truncation, there are thread mills for both internal and external threads exclusively, as well as some that are suitable for both. When choosing a thread mill it is important to refer to the catalog to determine if the tool is appropriate for internal or external threads.

4.2.1 Thread Geometry of Metric Thread Tool

Figure 14 shows the difference in the thread crest truncation requirements for internal and external threads. For this example, the thread profile of the OT-PNGT is shown, but these geometries apply to all thread mills. There is a difference in height between the crest and root of the thread for external and internal threads. Since thread standards require a radius in external thread roots, the crest of the thread mill cutting edge has a radius.

4.2.2 Pipe Thread Geometry

Thread geometry for tapered pipe thread mills are shown in figure 15 below. Most pipe thread mills are capable of milling internal and external threads as the thread form is typically identical for internal and external threads.

Figure 14: Geometry of External and Internal Thread Tools (Metric Thread)

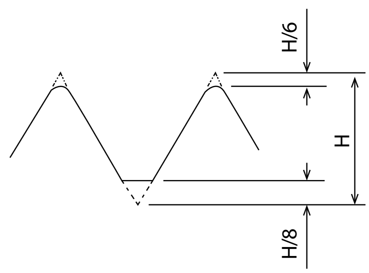
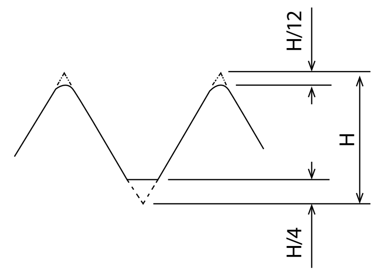
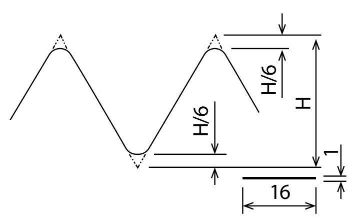
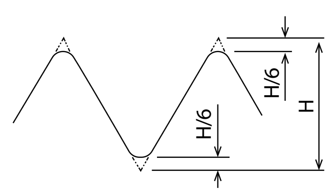
Thread Type	External	Internal
Geometry H="sharp-V" thread height		
Application	EXT	INT
Crest Profile	R	Flat
Root Profile	Flat	Flat
Manufacturing	Made to Order	Special

Figure 15: Angle Shape of External and Internal Thread Tool (Pipe Thread)

Thread Type	R (PT) external and Rc (PT) internal	Rp (PS) internal /G(PF) external and internal
Geometry H=theoretical crest height		
Application	Rc	Rp
Crest Profile	R	R
Root Profile	R	R
Manufacturing	Special	Special

4.3 Selection of Tool Diameter

4.3.1 Conventional Thread Mills (WX-PNC, OT-SFT-PNGT, OT-PNGT, PNGT, HY-PRO P single point, HY-PRO P)

When machining threads using a thread mill, it may seem logical to choose the shortest length and largest diameter possible to increase rigidity. However, as the tool diameter gets larger, it increases the engagement amount and duration, resulting in higher force and larger deflection than a smaller tool would cause. Typically, a cutter diameter that is 80% of the thread diameter or smaller is recommended. Please refer to the catalog for minimum size threads pertaining to different thread mill diameters.

4.3.2 Drill-Thread Mill Combinations (DR-PNAC, DR-O-PNAC)

When using a super thread mill, the size and pitch of the thread is predetermined by the tool diameter. Please refer to the catalog when choosing these tools.

4.3.3 Thread Mills for Pipe Threads (WX-PNC, OT-PNGT, PNGT, HY- PRO P)

When thread milling pipe threads, tools are chosen based on diameter and pitch of the thread. Please refer to the catalog when choosing pipe thread mills.

5 Cutting Conditions

5.1 Cutting Speed and Feed Rate

Cutting speed and feed rate are most often determined by the work material to be machined. Cutting speed and feed rate are directly related to the process time, tool life, and thread accuracy. It is important to consider all of these factors when choosing machining parameters.

5.1.1 Cutting Speed

In general, softer materials and free machining materials are cut at a higher surface speed, while harder and more difficult to machine materials are cut at a lower speed. Cutting speed needs careful control when attempting to meet a surface finish requirement – higher speeds may cause chatter, adversely impacting finish.

5.1.2 Feed Rate

Feed rate is also determined based on the work material and usually is provided in a feed per tooth measure, e.g. inches per tooth. In circumstances where there is a high surface finish requirement, or in harder, stronger materials, the feed rate is typically slower. Softer, free-machining materials are typically machined with a higher feed rate. However, it is important to consider the increased cutting forces associated with a larger feed rate which can cause greater tool deflection and a thread that tapers deeper into the hole. Consequently, smaller diameter tools with a relatively large pitch should be fed at a slower rate. It is important to consider other factors other than the work material such as setup rigidity, machine specifications, tool overhang, and orientation as these are all factors that influence setting a proper feed rate.

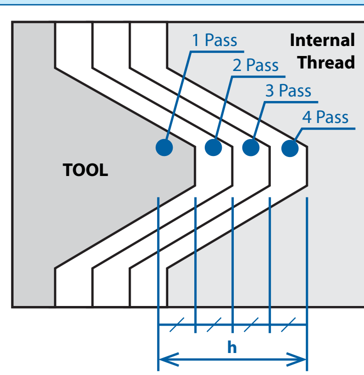
5.2 Depth of Cut

When machining free cutting materials such as carbon steels or aluminum, single pass thread milling is generally recommended. Figure 16 shows an example of multiple cutting passes, recommended for the following situations:

- Even in free machining materials, threads with a pitch greater than 3mm (8 TPI)
- Materials harder than 25HRC
- When the finished threads are not perpendicular, i.e. become tapered due to deflection, and the thread does not gauge
- HRSA

Spring or zero pass machining – a final pass identical to the pass just before it – is useful for ensuring thread perpendicularity. When machining hardened steels or HRSAs, lowering the depth of cut (adding additional passes) will help to increase tool life. Figure 17 shows a recommendation for the number of passes based on work material.

Figure 16: Multi-Pass Threading Operation



Using reduced depth of cut to create thread in multiple passes.

Figure 17: Work Material & Number of Passes

Material	Number of Passes
General Steels	1
Tempered Steels Ti Alloys General Steels (3mm Pitch or Greater)	2
Inconel	4

5.3 Cutting Direction

Similar to end mills, thread mills can be used in either conventional or climb milling, but in order to increase tool life climb milling is typically recommended. Typically the thread mill moves from the bottom of the thread upward toward the opening of the hole – in right-hand threads this ensures usage of climb milling. This method is preferred for blind holes as it leaves more room for chip clearance. When using a climb milling direction results in deflection, conventional milling helps reduce radial cutting forces to create a more perpendicular thread, but will result in shorter tool life and may require care to avoid chip packing.

5.4 Pilot Hole

Thread mills require the same drill size as used in conventional tapping. Pilot hole depth can be as small as 1 pitch length longer than the desired thread length.

5.5 Processing Time

Processing time is calculable by using a combination of the thread, the tool, and cutting parameters. In reality, factors such as machine capabilities (spindle acceleration, table acceleration) will cause actual cycle times to be slightly longer than the estimate. The following equation calculates the cycle time based on one pass thread milling. When the approach angle (Alpha) and release angle (Beta) cannot be determined, assume Alpha = 45deg, Beta = 45deg.

$$T = \pi (D-Dc) (1 + (\alpha + \beta) / 360) / Vf \text{ minutes}$$
(internal thread processing)
$$= \pi (D + Dc) (1 + (\alpha + \beta) / 360) / Vf \text{ minutes}$$
(male thread processing)
$$= \pi D (1 + (\alpha + \beta) / 360) / (f \times z \times n) \text{ minutes}$$
(male thread, internal thread processing)

Where:

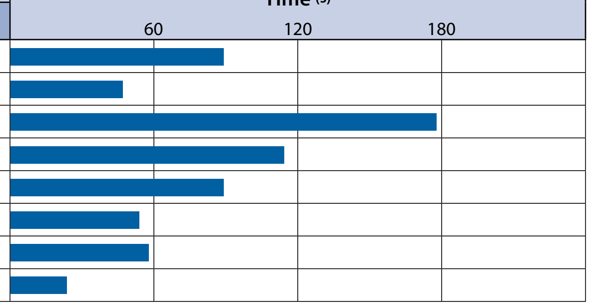
D: Thread diameter (mm)
Dc: Tool diameter (mm)
α: Approach angle (°)
β: Release angle (°)
Vf: Table Feed (mm/min)
f: Feed rate (mm/t)
z: Number of Flutes
n: Spindle speed (min -1)

5.5.1 Thread Milling Cycle Time vs. Conventional Tapping

When comparing the cycle times of cut taps and thread mills, the cutting parameters, thread length, and tool chosen all influence the amount of time. Figure 18 below shows the difference in cycle times between cut tapping and thread milling carbon steel. In order to thread a hole faster than a tap at 16 SFM, a thread mill must run at 130 SFM at a feed rate of 0.002 IPT, which are relatively high parameters for surface speed and feed rate. When synchronous tapping threads under M30, a cut tap will typically process threads in a shorter amount of time than a thread mill. This is a common trend among carbon steels, aluminum alloys, and cast iron.

Figure 18:

Tool	Size	Parameters		Time (s)
		Speed (SFM)	Feed (IPT)	
Cutting Tap	M36x2	16		
		32		
Thread Mill	20mm Dia. 5FL	66	0.0012	
			0.0020	
		132	0.0012	
			0.0020	
		198	0.0012	
			0.0020	



5.5.2 HY-PRO P, HY-PRO P Single Point

When thread milling using the HY-PRO P or HY-PRO P Single point, the thread length will almost always be longer than the tool's length of cut and requires a helical tool path that is broken into multiple Z-axis levels. As such, the cycle time equals one helical pass multiplied of how many Z-level step up/downs are required. By using the formula to estimate one pass of threading time and multiplying it by the number of Z-level steps, the cutting time can be estimated.

5.5.3 Super Thread Mill

The cycle time for the super thread mill is determined by summing the threading time and the hole processing time. Although the super thread mill also performs a chamfering operation, it does so at the same time as drilling, so the time for chamfering does not need to be taken into consideration. Additionally, due to the removal of tool changes, the overall cycle time will be shorter. However, when tapping at 5,000 RPM or greater, the super thread mill may still have a longer overall cycle time.

5.6 Selection of Cutting Fluid

Cutting fluids are mainly chosen based on lubricity, cooling capacity, and anti-welding properties to prolong tool life. Coolant is also an effective method of clearing chips from the work area. When thread milling, oil-based lubricants as well as water-soluble emulsion coolant are applicable, as thread mills do not require high lubricity coolant used in tapping. For example, when tapping nickel alloys, there are some situations where oil-based coolant must be used, but thread milling can use water-soluble emulsion coolant.

6 NC Program Creation

6.1 Concept of NC Program

When creating an NC program for thread milling, there are two primary methods. OSG recommends the second of the two.

1.Programming the tool using the center of tool and adjusting the tool diameter offset in the machine until an accurate thread is created.
2.Programming the tool using the root diameter of the cutting edge, and setting the machine offset to accommodate the pitch diameter of the thread.

CAM software is also useful to create a program for thread milling, but this requires a higher level of operator skill as well as access to the software itself.
Since tapered pipe threads have a different diameter at the shank end and end cut of the tool, OSG recommends using the tip diameter of the tool to create a program.

6.2 Setting Values for Machining Diameter and Tool Diameter

When preparing a program for thread milling, it is important to program the tool to create as accurate of a thread as possible during the first thread milling attempt. Adjusting the tool diameter offset from here is very important. Figure 19 details the relationship between tool diameter and programmed offset. During actual machining, the amount of tool engagement, the sharpness of the tool, setup rigidity, and other factors can cause actual machined pitch diameter to stray from the programmed values. Even following the process outlined in figure 19, reprogramming of the part after adjusting the programmed tool diameter and tool offset is common.

Figure 19:
Relationship Between Tool Diameter and Machining Diameter
(Parallel Internal Thread)

Tool Diameter Input	Tool Diameter when Programming NC
Actual tool diameter - target pitch diameter tolerance*	The root diameter of thread cutting edge
Tempered Steels Ti Alloys General Steels *(3mm Pitch or Greater)*	Internal thread root diameter + pitch diameter

** target pitch diameter tolerance: the difference between target internal thread pitch diameter and the actual pitch diameter*

6.3 Calculating Spindle Speed and Table Feed

When choosing thread milling parameters, surface speed and feed rate are typically provided in the catalog. However, to create an NC program, the spindle speed and table feed must be determined.

6.3.1 Spindle Speed

Since the tool rotation speed and the spindle rotation speed are the same, we can calculate the spindle speed using the equation below. For holders that increase spindle speed, adjust the spindle speed based on the multiplication factor of the holder.

$$n=(3.82 \times Vc)/Dc$$

n: Spindle Speed
Vc: Surface Speed
Dc: Diameter of Cutter (inch)

6.3.2 Tool Feed (Thread Processing)

When programming an arc or circular tool path, feed rate differs from that of a straight line which must be considered during calculation. This factor differs between internal and external threads. The equation below shows the feed calculation as well as how the internal or external thread factors into the calculation.

$$Vf=(f \times z \times n \times (D \pm Dc))/D$$

Vf: Table feed (IPM)
D: Thread major diameter (inch)
Dc: Tool diameter (inch)
z: Number of flutes
f: Feed rate (IPT)
n: Spindle speed (RPM)
For internal thread: -
For external thread: +

6.3.3 Tool Feed (Drilling)

The super thread mill has a drill point for creating a pilot hole. In order to correctly drill a hole, a drill feed rate is calculated as shown in the equation below:

$$Vf=Fz \times n$$

Fz: Feed per revolution (IPR)
N: Spindle speed (RPM)

6.4 NC Program Example

The following is an example of how OSG recommends creating a program for thread milling. When creating a program, adjustments to the following program are required based on work material, thread dimensions, etc.
OSG also has a complimentary program called ThreadPro, available for download at osgtool.com, which automatically generates NC programs for thread mills.

Compatible thread types:

- Metric screw (external and internal)
- Unified screw (external and internal)
- Tapered pipe thread (external and internal)
- Parallel pipe thread (external and internal)

https://www.osgtool.com/thread-milling-threadpro

6.4.1 Internal Thread (Right-Handed)

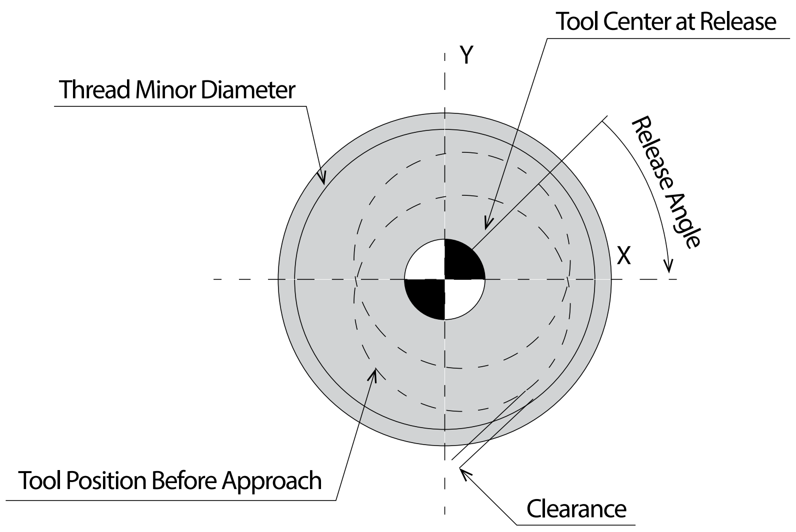
WX-PNC, OT-SFT-PNGT, PNGT, OT-PNGT, HYPRO P

Variables	
Thread Diameter	D(mm)
Internal Thread Inner Diameter	D1(mm)
Thread Length	Aa(mm)
Pitch	P(mm)
Tool Diameter	Dc(mm)
Number of Flutes	z(flutes)
Clearance	C(mm)
Release Angle	β(º)
Cutting Speed	V(m/min)
Feed Rate	fz(mm/t)
Tool Length Correction	No. 10
Tool Diameter Correction No. 20	No. 20
Recommended Release Angle	45º

Equations

#101 = 1000×Vc / π / Dc
1 = 180 / π × 2 × sin-1{(D1 - 2C - Dc) / (D - Dc)}
#106 = (D + Dc) / 4
#103 = (D - Dc) / 4 - # 106 × cos (# 1 × [pi / 180)
#104 = -#106 x sin (#1 x π / 180)
2 = 180 / π × cos-1{(D1 - 2C - Dc) / (D - Dc)}
#102 = Aa + P x (1 + #2 / 360)
#105 = D / 2
#107 = Aa + P
#108 = fz × z × #101 × (2 × #106 – Dc) / 2 / #106
#109 = D / 2
#110 = Aa
#111 = fz x z x #101 x (D - Dc) / D
#3 = 180/π × sin-1{(D – Dc)/(D + Dc)×sin (β × π/180)}
#112 = (D - Dc) / 4 + #106 x cos{(#3 + β) x π / 180}
#113 = #106 x sin {(#3 + β) x π / 180}
#114 = #106
#115 = Aa – P × β / 360

Figure 20: Internal Thread Processing



**In the main program example, the thread is created at the coordinates of X0 and Y0.*
**The sub program controls the spindle speed and moves the tool to the x, y coordinates and stops 5mm above the top of the workpiece. (Z0)*
**Before executing the program, check thoroughly with a dry run.*

NC Program

Main program example

```
O1000
G92 X0 Y0 Z100.0
G90 G00 X0 Y0 S#101 M03
G43 Z5.0 H10 M08 Z - #102
G17 G01 G41 X#103 Y#104 D20 F500
G03 X#105 Y0 R#106 Z - #107 F#108
I - #109 J0 Z - #110 F#111
X#112 Y#113 R#114 Z - #115
G01 G40 X0 Y0 F1000 M09
G00 Z100.0 G49 M05
M30
```

Subprogram example

```
O2000
G90 G00 Z - #102
G91 G17 G01 G41 X#103 Y#104 D20 F500
G03 X (#105 - #103) Y - #104 R#106 Z (#102 - #107) F#108
I - #105 J0 Z (#107 - #110) F#111
X (#112 - #105) Y#113 R#114 Z (#110 - #115)
G01 G40 X - #112 Y - #113 F1000
G90 G00 Z5.0
M99
```

6.4.2 Internal Thread (Right-Handed)

With drilling operation DR-PNAC, DR-O-PNAC

Variables	
Thread Diameter	D(mm)
Internal Thread Inner Diameter	D1(mm)
Thread Depth	Hd(mm)
Pitch	P(mm)
Tool Diameter	Dc(mm)
Number of Flutes	z(flutes)
Clearance	C(mm)
Release Angle	β(º)
Cutting Speed	V(m/min)
Feed Rate	fz(mm/t)
Drill Feed	f(mm/rev)
Tool Length Correction	No. 10
Tool Diameter Correction No. 20	No. 20
Recommended Release Angle	45º
C=(D-Dc)/2	

Equations

#101 = 1000 x V/π/D1
1 = 2 x sin-1 {(D1-2 x C-Dc)/(D-Dc)} x 180/π
#106 = (D + Dc)/4
#103 = (D-Dc)/4-#106 x cos (#1 x π/180)
#104 = #106 x sin (#1 x π/180)
#2 = cos-1{(D1-2 x C-Dc)/(D-Dc)} x 180/π
#102 = Hd
#105 = D/2
#107 = Hd-1.75 x P
#108 = fz x Z x #101 x (2 x #106-Dc)/2/#106
#109 = D/2
#110 = #107 + P
#111 = fz x Z x #101 x (D-Dc)/D
3 = sin-1{(D-Dc)/(D+Dc)×sin(β×π/180)}×180/π
#112 = (D-Dc)/4 + #106 x cos {(#3 + β) x π/180}
#113 = - #106 x sin {(#3 + β) x π/180}
#114 = (D + Dc)/4
#115 = #110 + P x β/360

NC Program

Main program example

```
O1000
G92 X0 Y0 Z100.0
G90 G00 X0 Y0 S#101 M03
G43 Z5.0 H12 M08
G01 Z-#102 F(#101•f)
G00 Z[{2+(#2-90)/360}•P-#102]
G17 G01 G42 X#103 Y#104 D20 F#108
G02 X#105 Y0 R#106 Z-#107
  I-#109 J0 Z-#110 F#111
  X#112 Y#113 R#114 Z-#115
G01 G40 X0 Y0 F1000 M09
G00 Z100.0 G49 M05
M30
```

Subprogram example

```
O2000
G90 G01 Z-#102 F(#101•f)
G00 Z[{2+(#2-90)/360}•P-#102]
G91 G17 G01 G42 X#103 Y#104 D20 F#108
G02 X (#105-#103) Y-#104 R#106 Z-•/360•P
I-#105 J0 Z (#107-#110) F#111
X(#112-#105) Y#113 R#114 Z(#110-#115)
G01 G40 X-#112 Y-#113 F1000
G90 G00 Z5.0
M99
```

**When using internal coolant change M08 to the desired M-code value for internal coolant.*
**In the main program example, the thread is created at the coordinates of X0 and Y0.*
**The sub program controls the spindle speed and moves the tool to the x, y coordinates and stops 5mm above the top of the workpiece. (Z0)*
**Before executing the program, check thoroughly with a dry run.*

6.4.3 Male Thread (Right-Handed) WX-PNC, OT-SFT-PNGT, PNGT, OT-PNGT, HY-PRO P

Variables	
Thread Diameter	D(mm)
Male Thread Root Diameter	D1(mm)
Thread Length	aa(mm)
Pitch	P(mm)
Tool Diameter	Dc(mm)
Number of Flutes	z(flutes)
Clearance	C(mm)
Approach Angle	α(º)
Release Angle	β(º)
Cutting Speed	V(m/min)
Feed Rate	fz(mm/t)
Tool Length Correction	No. 10
Tool Diameter Correction No. 20	No. 20
Recommended Approach Angle/Release Angle	45º
D1=D-1.2269xP (UN or ISO Thread)	

Equations

#101 = D/2 + 50
#102 = 1000 x V/π/Dc
#103 = aa-P x α/360
#106 = (D1)/2
#1 = (D/2 + C) x cos (α x π/180)
#2 = (D/2 + C) x sin (α x π/180)
#3 = tan-1 {#2/(#106-#1)} x 180/π
#107 = 0.5 x (((D1)/2-#1)2+#22)0.5/cos (#3×π/180)
#4 = {(#107+Dc/2)2-(D/2+Dc/2+C)2-(#107-(D1)/2)2}/{2×(#107-(D1)/2)}
#5 = ((D/2 + Dc/2 + C)2 - # 42)0.5
#6 = tan-1{#5/(#107-(D1)/2+#4)} x 180/π
#104 = #4-0.5 x Dc x cos (#6 x π/180)
#105 = #5-0.5 x Dc x sin (#6 x π/180)
#108 = aa
#109 = fz x z x #102 x (#107 + d/2)/#107
#110 = L + P
#111 = fz x z x #102 x (D + Dc)/D
#112 = (D/2 + C) x cos (β x π/180)
#113 = (D/2 + C) x sin (β x π/180)
#7 = tan-1{#113/(D1/2-#112)} x 180/π
#114 = (((D1)/2-#112)2+ #1132))0.5/2/cos (#7×π/180)
#115 = aa + P x (1 + β/360)

NC Program

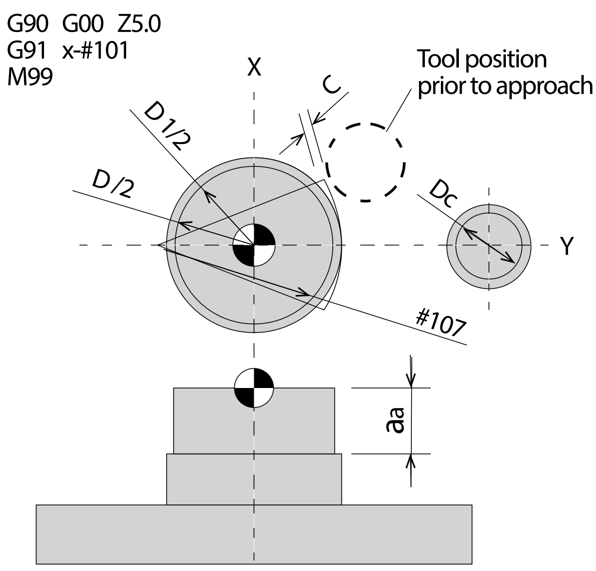
Main program example

```
O1000
G92 X0 Y0 Z100.0
G90 G00 X#101 Y0 S#102 M03
G43 Z5.0 H10 M08
 Z-#103
G17 G01 G41 X#104 Y#105 D20 F500
G02 X#106 Y0 R#107 Z-#108 F#109
 L-#106 J0 Z-#110 F#111
 X#112 Y-#113 R#114 Z-#115
G01 G40 X#101 Y0 F1000 M09
G00 Z100.0 G49 M05
X0 Y0
M30
```

Subprogram example

```
O2000
G91 G00 X#101
 Z-(#103+5)
G17 G01 G41 X(#104-#101) Y#105 D20 F500
G02 X(#106-#104) Y-#105 R#107 Z-(#108-#103) F#109
 L-#106 J0 Z-(#110-#108) F#111
 X(#112-#106) Y-#113 R#114 Z-(#115-#110)
G01 G40 X(#101-#112) Y#113 F1000
G90 G00 Z5.0
G91 X-#101
M99
```

Figure 21: External Thread Processing



** In the main program example, the thread is created at the coordinates of X0 and Y0.*
** The sub program controls the spindle speed and moves the tool to the x, y coordinates and stops 5mm above the top of the workpiece. (Z0)*
** Before executing the program, check thoroughly with a dry run.*

6.4.4 Tapered Internal Pipe Thread (Right-Handed)

WX-PNC, PNGT, OT-PNGT, HY-PRO P

Since pipe threads are tapered, the tool path for pipe threads follows a spiral curve pattern.
This program is broken into 4 blocks to create a spiral arc through helical interpolation.

Parameters	
Root Diameter	D(mm)
Internal Thread Pilot Hole Diameter	D1(mm)
Pitch	P(mm)
Thread Length	Hd(mm)
Tool Nominal Diameter	Dc (mm)
Tool LOC	ℓ(mm)
Number of Flutes	z(flutes)
Clearance	C(mm)
Release Angle	β(º)
Cutting Speed	V(m/min)
Feed Rate	fz(mm/t)
Tool Small End Root Diameter	D1s=D-Hd/16
Tool Outer Diameter on the Small End Side	TD=Dc-ℓ/16
Tool Length Correction	No. 10
Tool Diameter Correction No. 20	No. 20
Recommended Release Angle	45º

2 Calculation Formula

#101 = 1000 x V/π/TD
1 = 360/π x sin-1[[(D1)/4-{C+(Hd+P)/32}/2-TD/4]/(#107-TD/2)]
#103 = (D1s-P/16)/2-#107×{1+cos (#1×π/180)}
#104 = -#107 x sin(#1 x π/180)
#2 = 180/π×cos-1 [[(D1)-2×{C+(Hd+P)/32}-TD]/{(D1s-P/16)-TD}]
#102 = Hd + P x (1 + #2/360)
#105 = D1s/2-P/32
#107 = (D1s-P/16+TD)/4
#108 = Hd + P
#109 = fz x z x #101 x (#107-TD/2)/#107
#111 = #105 + P/128
#112 = #105 + P/256
#113 = Hd + 3 x P/4
#114 = fz x z x #101 x (#112-TD/2)/#112
#115 = - (#105+P/64)
#117 = #105 + 3 x P/256
#118 = Hd + P/2
#120 = - (#105 + 3 x P/128)
#121 = #105 + 5 x P/256
#122 = Hd + P/4
#123 = D1s/2
#125 = #105 + 7 x P/256
#126 = Hd
#3 = 180/π×sin-1{(D1s-TD)×sin(β×π/180)/(D1s+TD)}
#127 = (D1s-TD)/4+(D1s+TD)/4×cos{(#3+β)×π/180}
#128 = (D1s + TD)/4 x sin{(#3 + β) x π/180}
#129 = (D1s + TD)/4
#130 = Hd-P x β/360

NC program *(NC program based on major diameter of small end of tool)*

Main program example

```
O1000
G92 X0 Y0 Z100.0
G90 G00 X0 Y0 S#101 M03
G43 Z5.0 H10 M08
Z-#102
G17 G01 G41 X#103 Y#104 D20 F500
G03 X#105 Y0 R#107 Z-#108 F#109
X0 Y#111 R#112 Z-#113 F#114
X#115 Y0 R#117 Z-#118
X0 Y#120 R#121 Z-#122
X#123 Y0 R#125 Z-#126
X#127 Y#128 R#129 Z-#130
G01 G40 X0 Y0 F1000 M09
G00 Z100.0 G49 M05
M30
```

Subprogram example

```
O2000
G90 G00 Z-#102
G91 G17 G01 G41 X#103 Y#104 D20 F500
G03 X(#105-#103) Y-#104 R#107 Z(#102-#108)
F#109
X-#105 Y#111 R#112 Z(#108-#113) F#114
X#115 Y-#111 R#117 Z(#113-#118)
X-#115 Y#120 R#121 Z(#118-#122)
X#123 Y-#120 R#125 Z(#122-#126)
X(#127-#123) Y#128 R#129 Z(#126-#130)
G01 G40 X-#127 Y-#128 F1000
G90 G00 Z5.0
M99
```

When using a left-handed holder with HY-PRO P, reverse the spindle (Change M03 to M04) before use.

**In the main program example, the thread is created at the coordinates of X0 and Y0.*
**The sub program controls the spindle speed and moves the tool to the x, y coordinates and stops 5mm above the top of the workpiece. (Z0)*
**Before executing the program, check thoroughly with a dry run.*

Figure 22: Pipe Thread Reference Values

Size	Threads per inch	Basic Dia.	Pitch Diameter Length	Pilot Hole Size
		Major Dia.		
R 3/8	19	16.662	9.7	14.4
R 1/2	14	20.995	12.7	18
R 3/4	14	26.441	14.1	23
R 1	11	33.249	16.2	29
R 1-1/4	11	41.910	18.5	38
R 1-1/2	11	47.803	18.5	44
R 2	11	59.614	22.8	55

The pilot hole diameter is a recommendation for a non-tapered hole. This is a recommendation based on a single incomplete thread allowed at the hole bottom.

Figure 23: NPT Thread Reference Values

Size	Threads per inch	Basic Dia.	Pitch Diameter Length
		Major Dia.	
1/16	27	7.770	0.242 (6.15)
1/8	27	10.117	0.332 (8.43)
1/4	18	13.426	0.438 (11.13)
3/8	18	16.866	0.562 (14.27)
1/2	14	20.980	0.703 (17.86)
3/4	14	26.325	0.906 (23.01)
1	11-1/2	32.943	1.141 (28.98)
1-1/4	11-1/2	41.689	1.484 (37.69)
1-1/2	11-1/2	47.760	1.719 (43.66)
2	11-1/2	59.797	2.188 (55.58)

6.4.5 Internal Thread (Right-Handed)

WH-VM-PNC (hard-to-cut materials), HY-PRO P single point

Variables	
Thread Diameter	D(mm)
Internal Thread Inner Diameter	D1(mm)
Pitch	P(mm)
Thread Length	aa(mm)
Tool Diameter	Dc(mm)
Number of Flutes	z(flutes)
Clearance	C(mm)
Release Angle	β(º)
Cutting Speed	V(m/min)
Feed Rate	fz(mm/t)
Tool Length Correction	No. 10
Tool Diameter Correction No. 20	No. 20
Recommended Release Angle	45º

Equations

#101 = 1000 x V/π/Dc
1 = 180/π x 2 x sin-1{(D1-2C-Dc)/(D-Dc)}
#106 = (D + Dc)/4
#103 = (D-Dc)/4-#106 x cos(#1 x π/180)
#104 = - #106 x sin(#1 x π/180)
#2 = 180/π x cos-1{(D1-2C-Dc)/(D-Dc)}
#102 = aa + P x (1 + #2/360)
#105 = D/2
#107 = aa + P
#108 = fz x z x #101 x (2 x #106-Dc)/2/#106
#109 = D/2
#110 = aa
#111 = fz x z x #101 x (D-Dc)/D
#3 = 180/π x sin-1{(D-Dc)/(D+Dc) x sin(β x π/180)}
#112 = (D-Dc)/4 + #106 x cos{(#3 + β) x π/180}
#113 = #106 x sin{(#3 + β) x π/180}
#114 = #106
#115 = aa-P x β/360-P x (N-1)
#116 = P
#117 = FUP (0.3 + aa)/P

NC program

Main program example

```
O1000
G92 X0 Y0 Z100.0
G90 G00 X0 Y0 S#101 M03
G43 Z5.0 H10 M08
Z-#102
G17 G01 G41 X#103 Y#104 D20 F500
G03 X#105 Y0 R#106 Z-#107 F#108
I-#109 J0 Z-#110 F#111
I-#109 J0 Z (#116-#110)
I-#109 J0 Z (#116*2-#110)

.
.
.

I-#109 J0 Z(#116*(#117-2)-#110)
I-#109 J0 Z(#116*(#117-1)-#110)
X#112 Y#113 R#114 Z-#115
G01 G40 X0 Y0 F100 M09
G00 Z100.0 G49 M05
M30
```

(Repeat N Times)

Subprogram example

```
G90 G00 Z-#102
G91 G17 G01 G41 X#103 Y#104 D20 F500
G03 X(#105-#103) Y-#104 R#106 Z(#102-#107) F#108
I-#105 J0 Z#116 F#111
I-#105 J0 Z#116

.
.
.

I-#105 J0 Z#116
I-#105 J0 Z#116
X(#112-#105) Y#113 R#114 Z(#110-#115)
G01 G40 X-#112 Y-#113 F1000
G90 G00 Z5.0
M99
```

(Repeat N Times)

**In the main program example, the thread is created at the coordinates of X0 and Y0.*
**The sub program controls the spindle speed and moves the tool to the x, y coordinates and stops 5mm above the top of the workpiece. (Z0)*
**Before executing the program, check thoroughly with a dry run.*

6.4.6 Internal Thread (Right Hand) ③

WH-VM-PNC (general work material)

Variables	
Thread	D(mm)
Internal Thread Inner Diameter	D1(mm)
Pitch	P(mm)
Thread Length	aa(mm)
Tool Diameter	Dc(mm)
Number of Flutes	z(flutes)
Clearance	C(mm)
Release Angle	β(º)
Cutting Speed	V(m/min)
Feed Rate	fz(mm/t)
Tool Length Correction	No. 10
Tool Diameter Correction No. 20	No. 20
Recommended Release Angle	45º

Equations

#101 = 1000 x V/π/Dc
1 = 180/π x 2 x sin-1{(D1-2C-Dc)/(D-Dc)}
#106 = (D + Dc)/4
#103 = (D-Dc)/4-#106 x cos(#1 x π/180)
#104 = - #106 x sin(#1 x π/180)
#2 = 180/π x cos-1{(D1-2C-Dc)/(D-Dc)}
#102 = aa + P x (1 + #2/360)
#105 = D/2
#107 = aa + P
#108 = fz x z x #101 x (2 x #106-Dc)/2/#106
#109 = D/2
#110 = aa
#111 = fz x z x #101 x (D-Dc)/D
#3 = 180/π x sin-1{(D-Dc)/(D+Dc) x sin (β x π/180)}
#112 = (D-Dc)/4 + #106 x cos{(#3 + β) x π/180}
#113 = #106 x sin{(#3 + β) x π/180}
#114 = #106
#115 = aa-P x β/360
#116 = P x 3
#117 = FUP (0.3 + aa) / #116

NC program

Subprogram example

Repeat N times by changing z height (First Time)

```
G90 G00 Z-#102 • • • ①
G91 G17 G01 G41 X#103 Y#104 D20 F500
G03 X(#105-#103) Y-#104 R#106 Z(#102-#107) F#108
I-#105 J0 Z#116 F#111
X(#112-#105) Y#113 R#114 Z(#110-#115)
G01 G40 X-#112 Y-#113 F1000
```

(Second Time)

```
G90 G00 Z (#116-#102)
G91 G17 G01 G41 X#103 Y#104 D20 F500
G03 X(#105-#103) Y-#104 R#106 Z(#102-#107) F#108
I-#105 J0 Z#116 F#111
X(#112-#105) Y#113 R#114 Z(#110-#115)
G01 G40 X-#112 Y-#113 F1000
```

.
.
.

(N-1)th Time

```
G90 G00 Z(#116*(#117-2)-#102)
G91 G17 G01 G41 X#103 Y#104 D20 F500
G03 X(#105-#103) Y-#104 R#106 Z(#102-#107) F#108
I-#105 J0 Z#116 F#111
X(#112-#105) Y#113 R#114 Z(#110-#115)
G01 G40 X-#112 Y-#113 F1000
```

(Final Time)

```
G90 G00 Z(#116*(#117-1)-#102)
G91 G17 G01 G41 X#103 Y#104 D20 F500
G03 X(#105-#103) Y-#104 R#106 Z(#102-#107) F#108
I-#105 J0 Z#116 F#111
X(#112-#105) Y#113 R#114 Z(#110-#115)
G01 G40 X-#112 Y-#113 F1000
```

```
G90 G00 Z5.0
M99
```

**In the main program example, the thread is created at the coordinates of X0 and Y0.*
**The sub program controls the spindle speed and moves the tool to the x, y coordinates and stops 5mm above the top of the workpiece. (Z0)*
**Before executing the program, check thoroughly with a dry run.*

shaping your dreams

Safe use of cutting tools

- Use safety cover, safety glasses and safety shoes during operation.
- Do not touch cutting edges with bare hands.
- Do not touch cutting chips with bare hands. Chips will be hot after cutting.
- Stop cutting when the tool becomes dull.
- Stop cutting operation immediately if you hear any abnormal cutting sounds.
- Do not modify tools.
- Please use appropriate tools for the operation. Check dimensions to ensure proper selection.

FOR MORE INFORMATION CONTACT US

800-837-2223 • osgtool.com

© Copyright 2023 OSG USA, INC. All rights reserved. PDF Only. EDP#800390CA