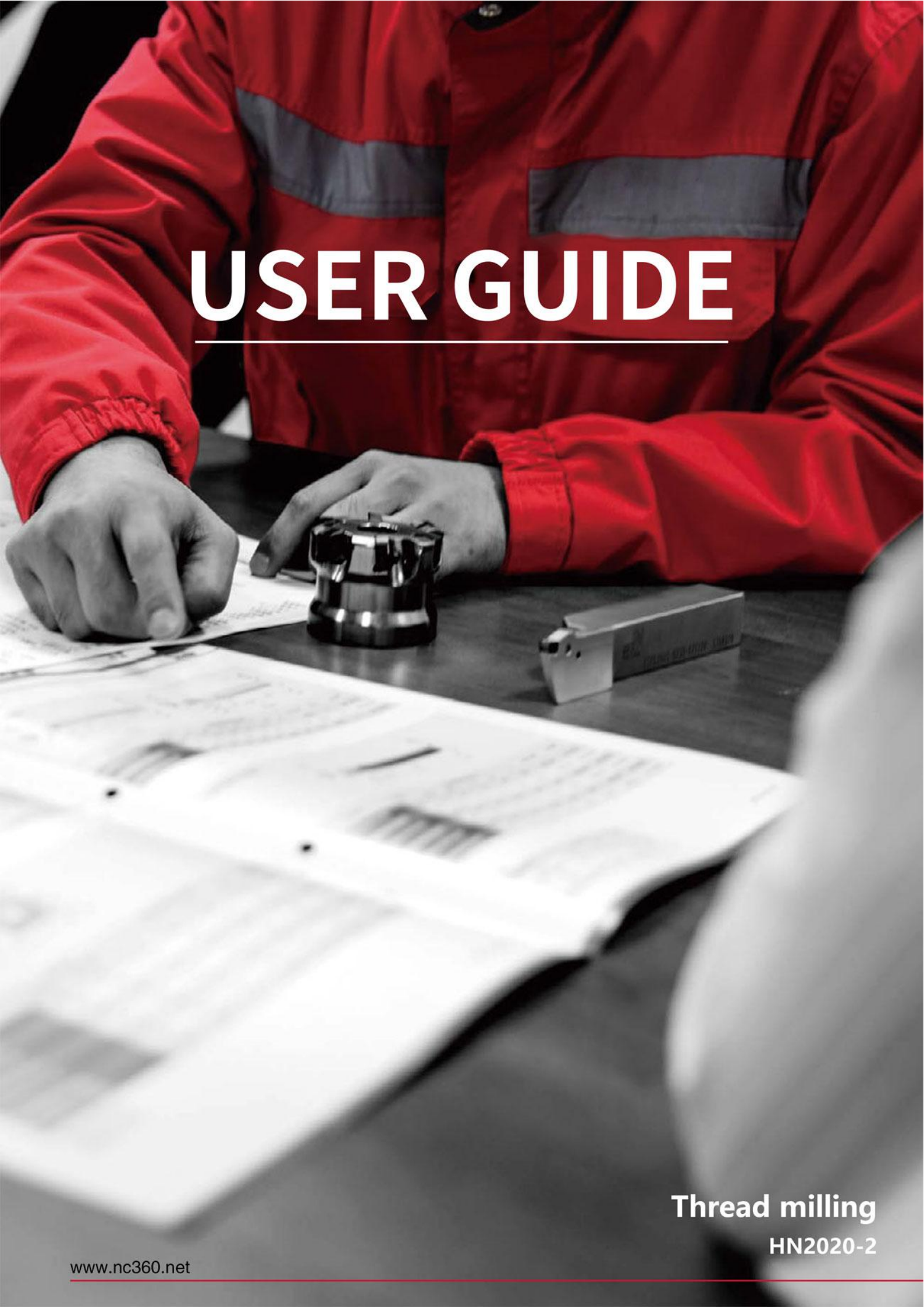


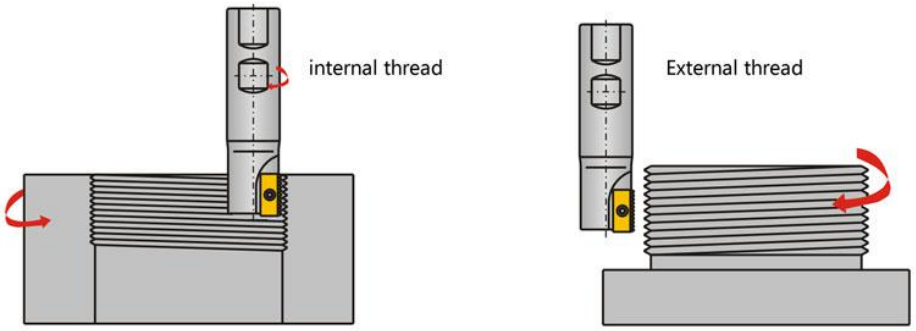
USER GUIDE

Thread milling

HN2020-2

www.nc360.net

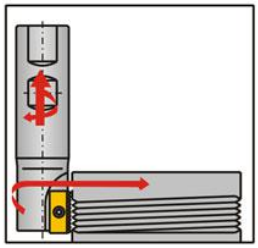
Application data of indexable thread milling cutter



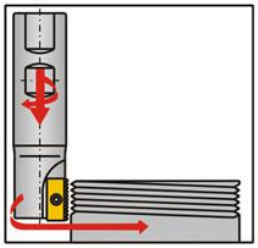
For 40-shank machine tools, when the thread is larger than M24 and 50-shank machine tools, when the thread is larger than M42, tapping is no longer applicable, and thread milling is an inevitable choice.

● Thread milling method

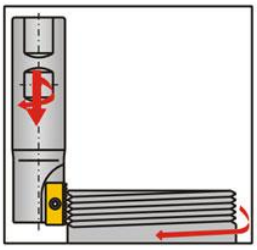
Right hand thread
Counter clockwise feed and reverse milling



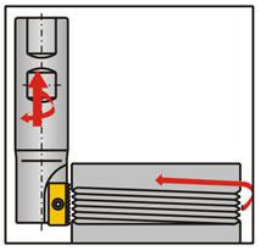
Left hand thread
Counter clockwise feed and reverse milling



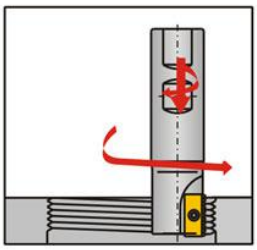
Right hand thread
Clockwise feed down milling



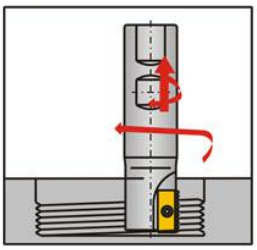
Left hand thread
Clockwise feed down milling



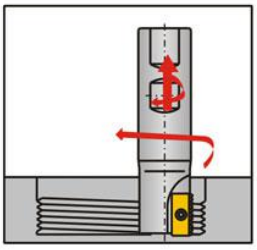
Right hand thread
Clockwise feed up milling



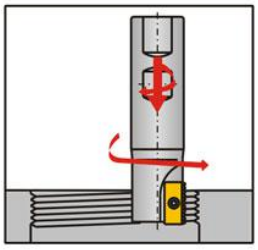
Left hand thread
Clockwise feed up milling



Right hand thread
Counter clockwise feed clockwise milling



Left hand thread
Counter clockwise feed clockwise milling

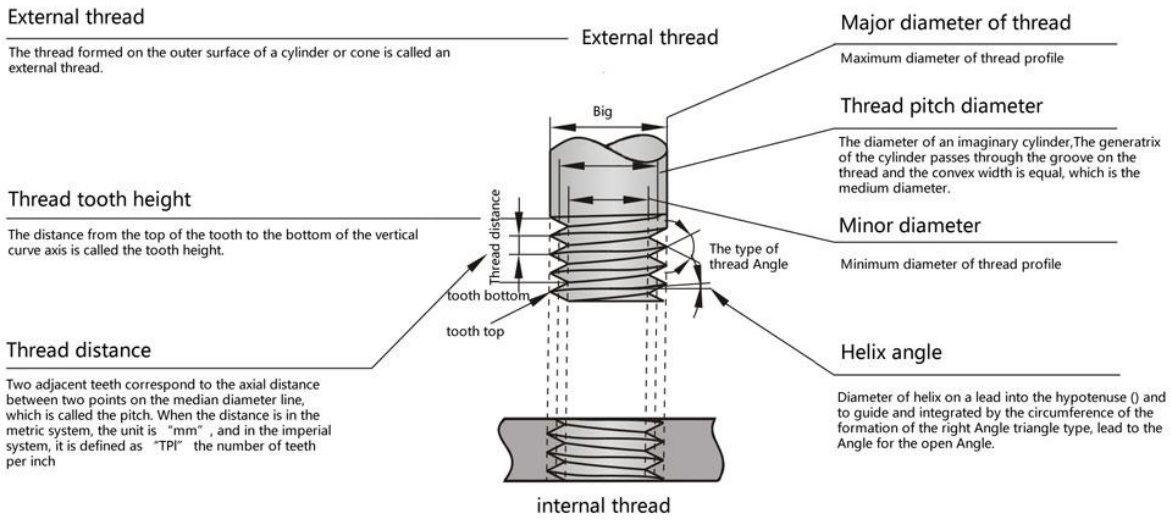


● Recommended cutting parameters

ISO	Workpiece material	Cutting speed (m/min)	
		CP5200	CP5300
P	Low carbon steel and medium carbon steel	100-250	115-280
	High-carbon steel	110-180	130-200
	alloy steel	90-160	105-180
M	Martensitic stainless steel	110-170	130-190
	Austenitic stainless steel	130-170	150-190
K	cast iron	70-150	80-170
N	Non-ferrous metals	160-300	180-340
	Non-metallic materials	100-400	115-460
S	Nickel-based alloy, titanium alloy	20-80	25-90

Application data of indexable thread milling cutter

● Terminology of thread elements



Nominal size

According to thread deviation and grade
The limit size of the thread diameter can be calculated.

The thread formed in the hole of the cylinder or cone is called the internal thread

Straight thread

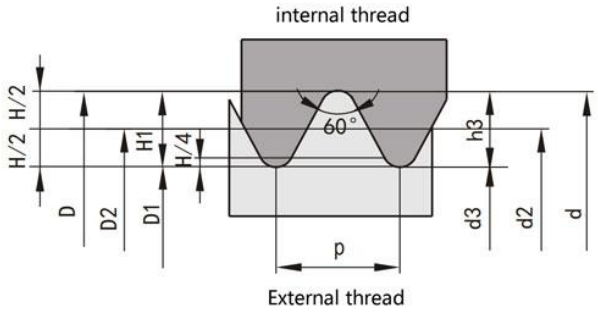
A thread formed on a cylinder.

Taper thread

The thread formed on the cone.

● Thread size calculation

Standard metric thread
It is defined as follows:



$H=\frac{P/2}{\tan(\varphi/2)}=0.866p \quad \varphi=60°$

$D2=D+\frac{H}{8}\times 2-H=D\frac{3}{4}-H=D-0.75H$

$D2=D-0.64953p$

$d3=D-1.41667h=D-1.22687p$

$D1=D-H1=D-1.083p$

$h3=0.61343p$

$H1=0.54127p$

其中:

d =D =公称直径（mm）

d2=D2=螺纹中径（mm）

d3= 外螺纹小径（mm）

D1= 内螺纹小径（mm）

h3= 外螺纹牙高（mm）

H1= 内螺纹牙高（mm）

r= 齿根圆角半径（mm）

Application data of indexable thread milling cutter

In order to better apply thread milling, the machine tool must have three-axis linkage performance. The spiral interpolation function is realized, and the machine tool controls the tool to realize the spiral trajectory. The spiral
interpolation is formed by the planar arc interpolation and the linear motion linkage perpendicular to the plane.
For example: the spiral trajectory from point A to point B (figure A) is formed by the linkage of the X-Y plane circular interpolation movement and the linear linear movement of the Z axis.
For most CNC systems, this function can be achieved through the following two different commands.

G02: Clockwise arc interpolation command

G03: Anti clockwise arc interpolation command

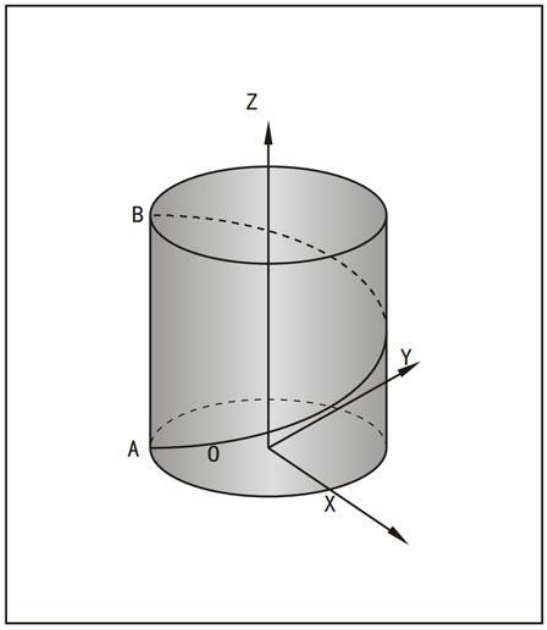


Illustration A

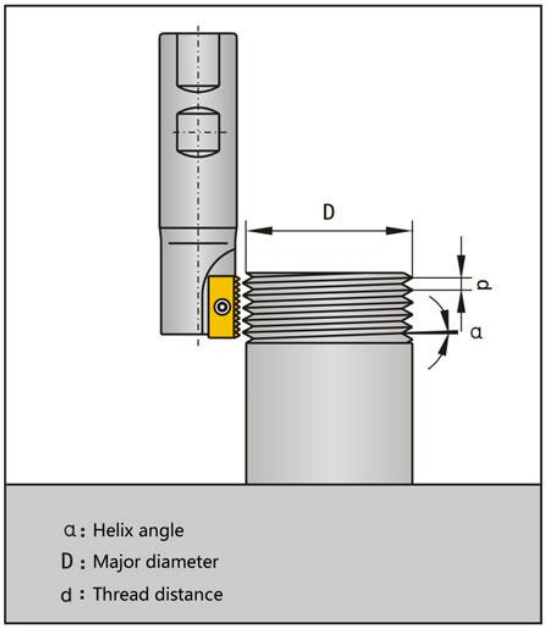


Illustration B

Thread milling (Figure B) is formed by the rotation of the tool and the spiral interpolation of the machine tool. In a circular interpolation process, the geometry of the tool is used, combined with the movement of the tool to move a pitch in the axial direction, to process the required thread.

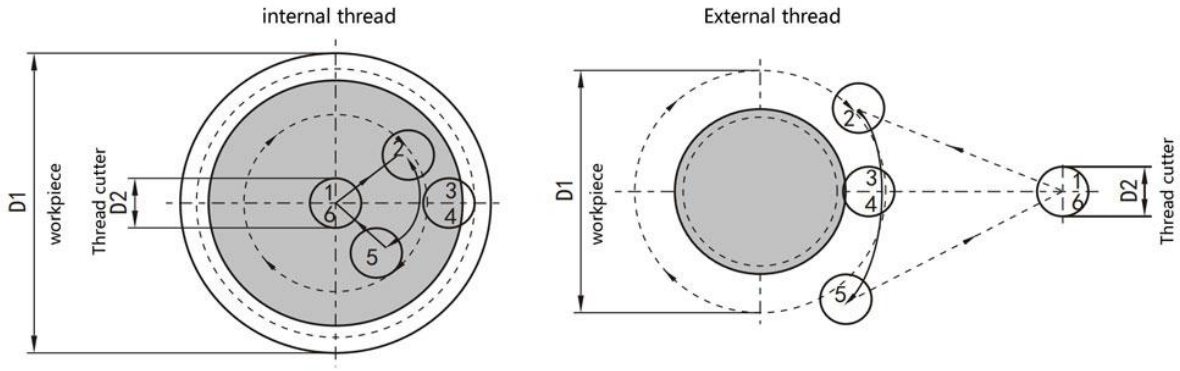
- Thread milling can use the following three cutting methods

  - Arc cut method
  - Radial cut method
  - Tangential approach

Application data of indexable thread milling cutter

● Arc cut method

With this method, the tool cuts in and cuts out in balance, leaving no traces, and no vibration, even when processing hard materials.
The preparation of this method is more complicated than the radial cutting method. It is recommended to use this method when machining precision threads.



1–2: Rapid positioning

2–3: The tool cuts in along the arc feed, while interpolating along the Z axis

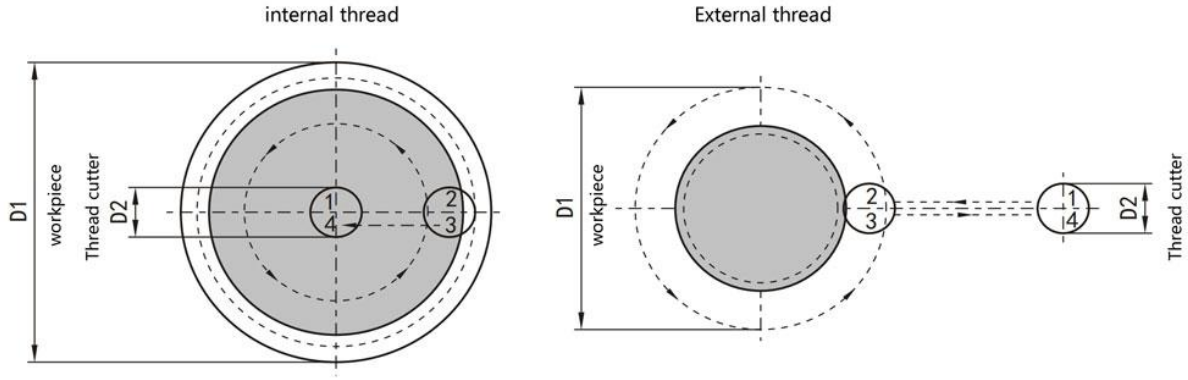
3–4: 360° full circle cutting and interpolation one week, one lead of axial movement

4–5: The tool cuts along the arc feed and at the same time interpolates along the Z axis

5–6: Quick return

● Radial cut method

This method is the simplest, but sometimes there are two situations:
A. Small vertical marks will be left at the cut-in and cut-out points, but the thread quality will not be significantly affected.
B. When processing very hard materials, when the cutting is close to the full tooth profile, due to the large contact area between the tool and the workpiece, vibration may occur.
Note: In order to avoid the vibration when the cutting is close to the full tooth, the feed amount should be reduced to 1/3 of the spiral interpolation feed as much as possible.



1–2: Radial fast cutting

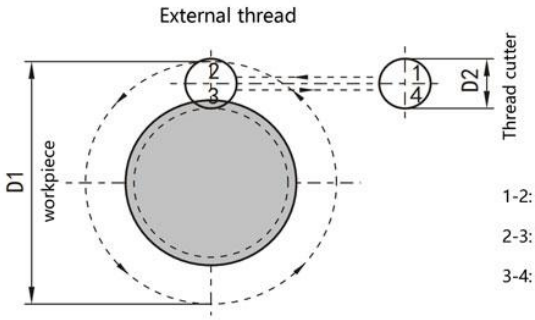
2–3: 360° full circle cutting and interpolation one week, one lead of axial movement

3–4: Quick exit

Application data of indexable thread milling cutter

● Tangential approach

This method is very simple and has the advantage of arc cutting, but it is only suitable for milling of external threads.



1-2: tangential quick cut

2-3: 360 ° full circle cutting and interpolation for one cycle and axial movement for one lead

3-4: quick exit

● Calculate tool feed rate

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

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

$$F1=NZf$$

N-tool speed [r.p.m]

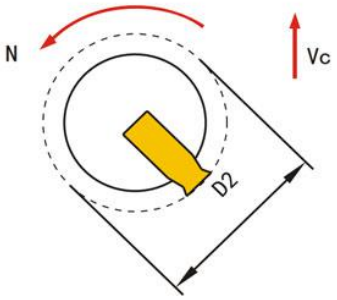
V-cutting linear speed [M / min]

D_2 - tool diameter (including blade) [mm]

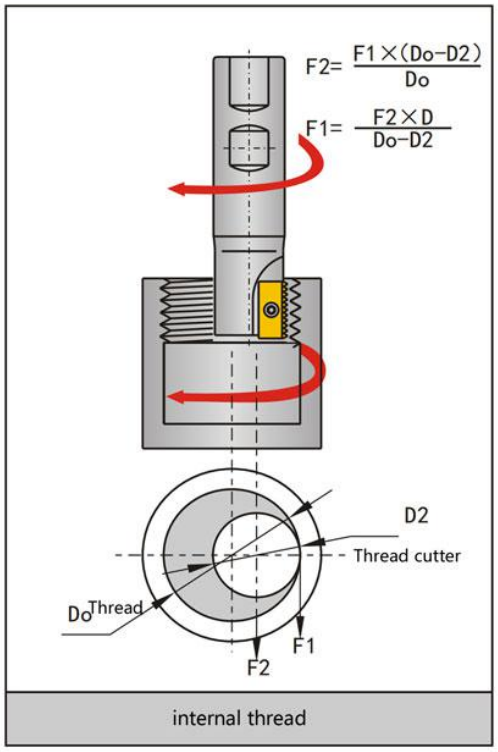
F1 - radial feed rate of tool [min]

z-Number of cutting edges

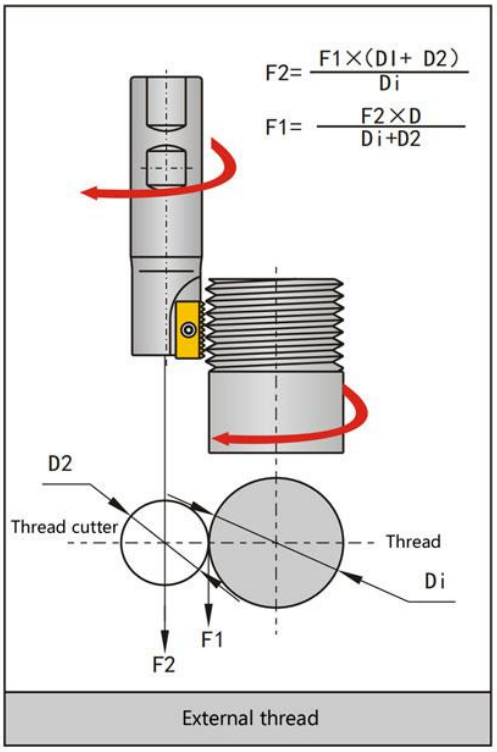
F - feed rate per blade per revolution [mm / rev]



● Calculation formula



internal thread



External thread

In most CNC machine tools, tool center feed programming is required. The feed rate of the tool is determined by the feed rate of the tool center, but the feed rate of the tool center is not directly given, but can be obtained by the relationship equation between the tool feed rate and the tool

Application data of indexable thread milling cutter

● Tool selection steps

（1）Select the blade according to the pitch to be machined.
（2）Select DC less than the size to be processed.
（3）According to the table, the tools that meet the conditions 1 and 2 are selected according to the maximum tool diameter.

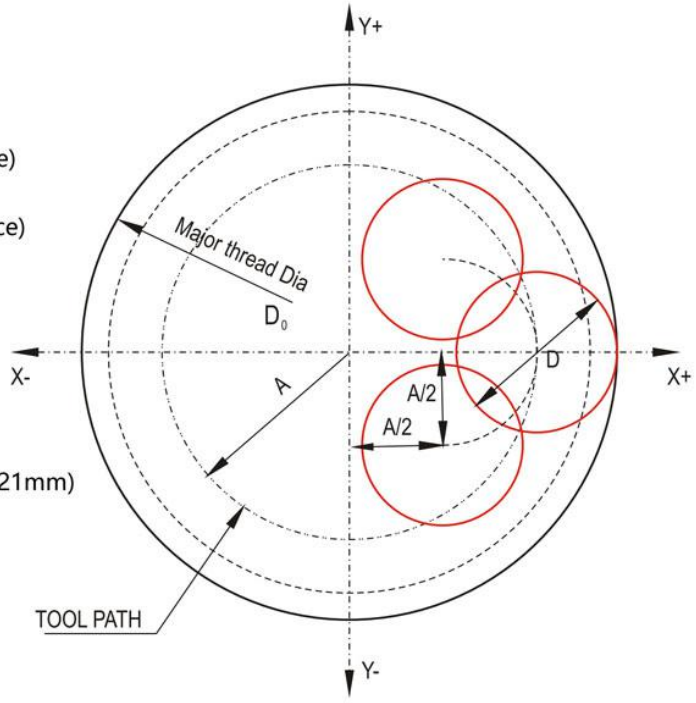
Programming of internal thread milling on CNC machine tool

$$A = \frac{D_0 - D}{2}$$

D_0 : Nominal diameter of thread
D : Tool diameter
A : Tool interpolation path radius

● General program

G90 G00 G54 G43 H1X0 Y0 Z10 S-
G00 Z- (thread depth)
G01 G91 G41 D1 X(A/2) Y-(A/2) Z0 F-
G03 X(A/2) Y(A/2) R(A/2) Z(1/8 thread distance)
G03 X0 Y0 I-(A) J0 Z(Thread distance)
G03 X-(A/2) Y(A/2) R(A/2) Z(1/8 thread distance)
G01 G40 X-(A/2) Y-(A/2) Z0
G90 X0 Y0 Z0



● Internal thread machining program example

Thread size: M32X2.0(Thread depth 18mm)
Tool model: ST90-21R1T21-B20(Tool diameter 21mm)
blade: 21 I2.0 ISO
A=(32-21)/2=5.5

G90 G00 G54 G43 H1X0 Y0 Z10 S2800
G00 Z-18
G01 G91 G41 X2.75 Y-2.75 Z0 F85 D1
G03 X2.75 Y2.75 R2.75 Z0.25
G03 X0 Y0 I-5.5 J0 Z2
G03 X-2.75 Y2.75 R2.75 Z0.25
G01 G40 X-2.75 Y-2.75 Z0
G90 X0 Y0 Z0

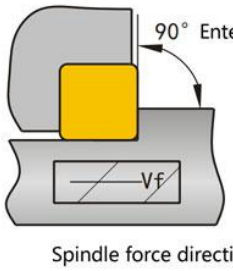
● Advantages of internal thread milling cutter

A thread milling cutter can be used to machine left and right threads
A thread milling cutter can process internal and external threads of different diameters
The precise tool holder design allows the blade to be accurately and firmly fixed on the tool holder
Most blades are double-edged
When thread milling, it can be formed at one time
Can process taper threads
The increase in cutting speed and the multi-tooth blade structure can greatly improve the processing efficiency
When machining blind hole threads, the thread depth can reach the bottom of the hole
Special composite coating can greatly extend tool life
Tool cost is much lower than tap and die
Due to the small cutting force, large threads can be formed and processed at one time on low-power equipment, and the idle time of the equipment and the number of tool changes are reduced.

Milling technology application data

● Choice of entering angle

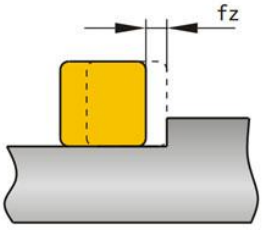
Form of entering angle

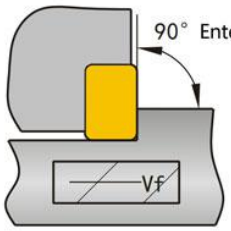


Spindle force direction

Applied to thin-walled parts
Requires correct 90° formation
Low-strength structure and poor clamping

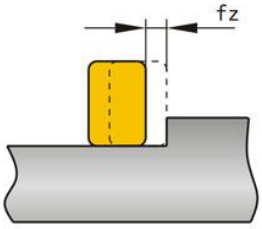
Influence of entering angle on chip thickness

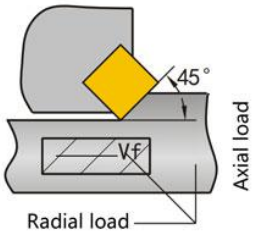




Spindle force direction

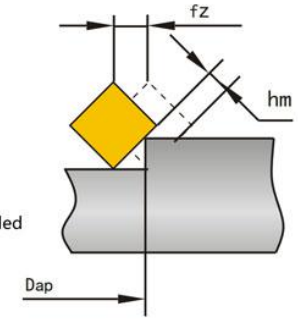
Applied to thin-walled parts
Requires correct 90° formation
Low-strength structure and poor clamping

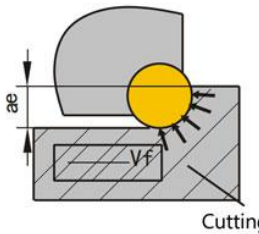




Spindle force direction

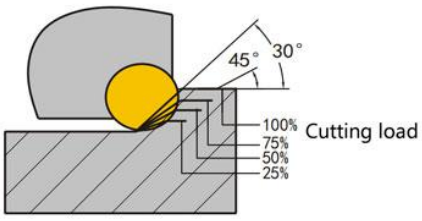
The entering angle is 45°, which provides excellent cutting edge strength, especially for overhang milling, and the axial cutting force is nearly equal to the radial cutting force.
Easy to chip when milling cast iron, 45° entering angle is recommended





Cutting force

The strongest cutting edge that can be rotated many times
Thin cutting, most suitable for heat-resistant alloy processing
The most used roughing tool



If the shape of the workpiece makes it difficult to locate the cutting position of the tool, a smaller entering angle is better.

Haina Precision Cutting Technology Co., Ltd.

Haina Precision Machinery Co., Ltd.

Address: No. 3, Daqing Road, Lian District, Xi’ an, China

Phone: +86 029-81333283 14729292929

Email: nc360@vip.163.com

Website: www.nc360.net