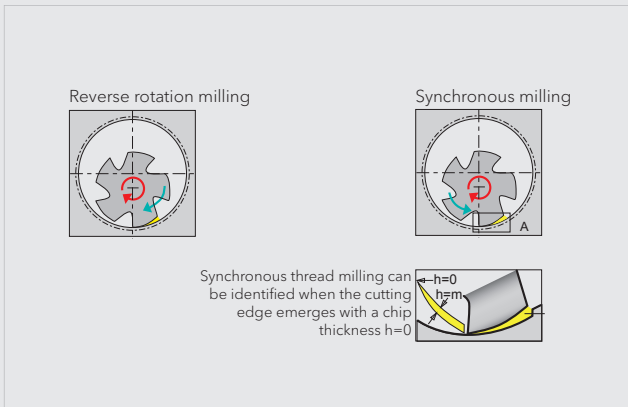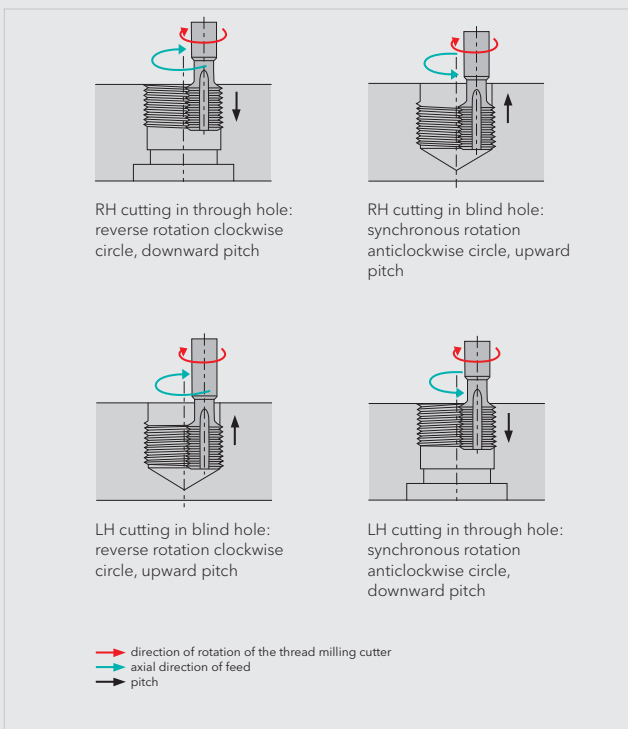# PROCESS AND TECHNOLOGY OF THREAD MILLING

MACHINING COMBINATIONS (REVERSE ROTATION/ SYNCHRONOUS MILLING)

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

Reverse rotation milling

Synchronous milling

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

h=0
h=m

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

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

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

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

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

→ direction of rotation of the thread milling cutter
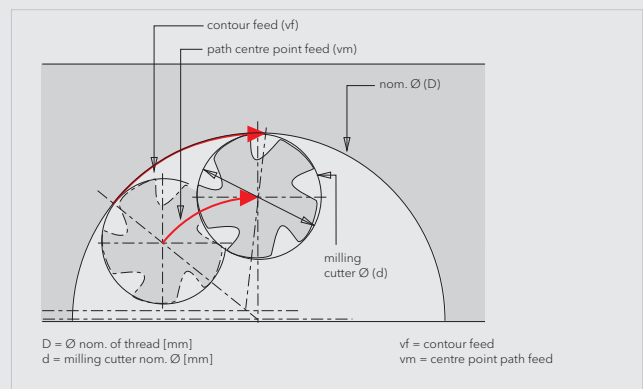→ axial direction of feed
→ pitch

PRESSURE ANGLE AND FEED RATIO

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

pressure angle of thread milling cutter

nom. Ø of thread milling cutter

core Ø of internal thread

nom. thread diameter
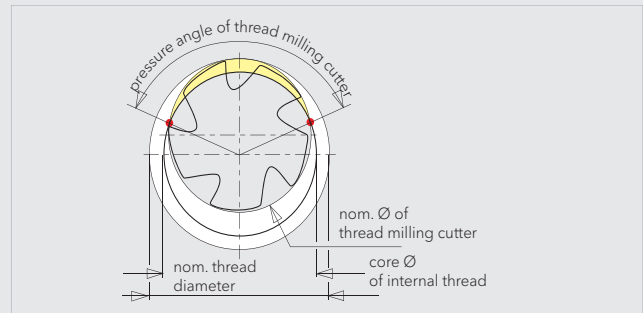
The diameter of the thread milling cutter and the profile depth determine the pressure angle to the thread diameter.

The feed at the cutting edge of the thread milling cutter is calculated by the cutting speed (revolutions) and the feed rate per tooth. With linear movement, the feed rate at the cutting edge is identical to that at the tool's centre. However, the helical interpolation follows the path of a circle in the plane. As the machine tool always calculates to the tool's centre, a command must be programmed for converting the cutting speed (contour related program). If such a command does not exist or the program is centre-related, the feed rate must be converted first.


contour feed (vf)

path centre point feed (vm)

nom. Ø (D)

milling cutter Ø (d)

D = Ø nom. of thread [mm]   vf = contour feed
d = milling cutter nom. Ø [mm]   vm = centre point path feed

ℹ You can find further materials and cutting values in the cutting data calculator.

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

## THREAD MILLING CUTTER ENTRY CYCLES

90° Quarter circle entry cycle (WZG 17223)

With an entry cycle of 90° and a small difference in diameter between the tool and the thread, a large part of the total chip volume is removed during the linear section of the entry cycle. This method is therefore only recommended for relatively large differences in diameter between hole size and thread milling cutter. The advantages presented by this entry method are simple programming and a relatively short entry path.
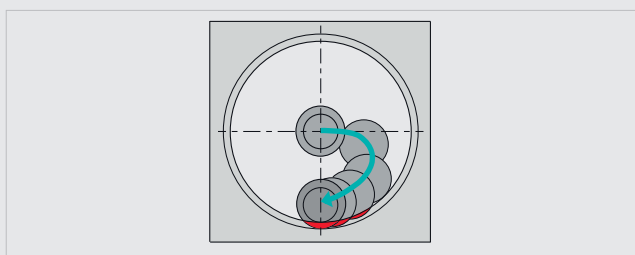

180° Semicircle entry cycle (WZG 17123)

With a 180° entry cycle, the loading on the tool is the lowest when plunging, as the angle of contact is relatively small during the complete entry cycle.
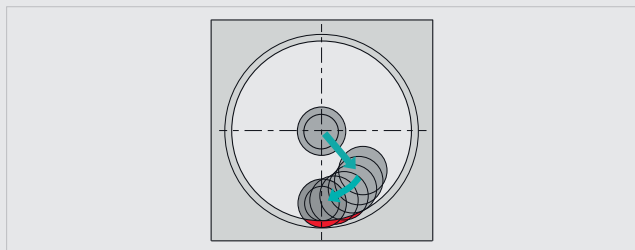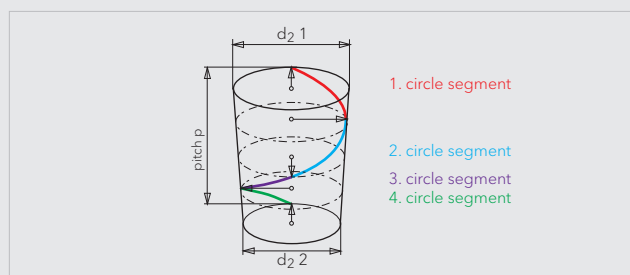

Screw interpolation (cyl. thread)

The screw interpolation is a superposition of a circular and a linear motion. Various threads can be produced by the type of superimposition of pitch direction and direction of rotation of circular motion.
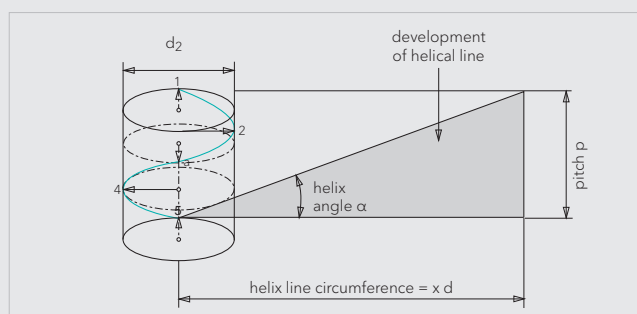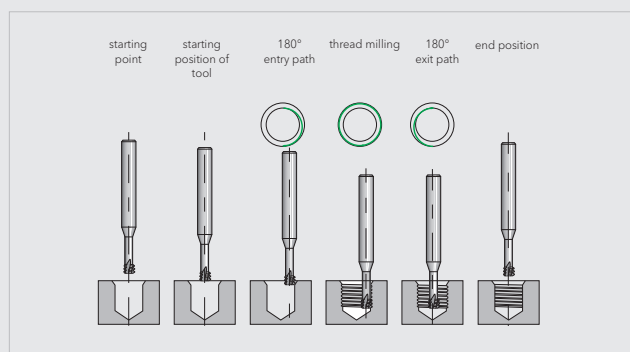

Helical interpolation (conical thread)

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

1. circle segment
2. circle segment
3. circle segment
4. circle segment

Programming sequence for thread milling (Conventional, up-cut right-hand thread milling)

starting point — starting position of tool — 180° entry path — thread milling — 180° exit path — end position

Radial cutting index

1st cut — 2nd cut

1st cut (conventional up-cut milling)
2nd cut (conventional up-cut milling)