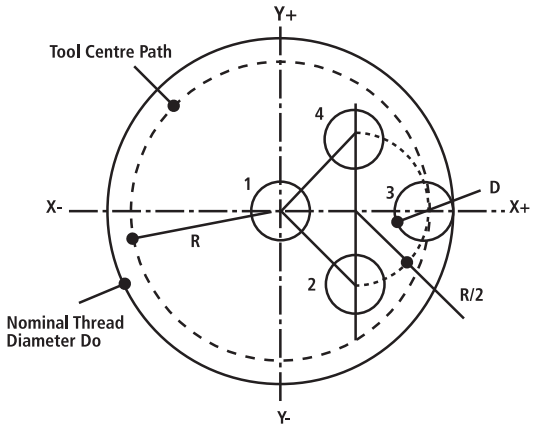
Programming examples

The following programming examples are based on the thread milling tool centre path, which requires no tool radius compensation other than an offset for tool wear. These examples are purely for guidance only and may need to be altered to suit the CNC control you are using. Sample programmes can be supplied to suit your individual requirements.

Internal Parallel Thread Milling Example



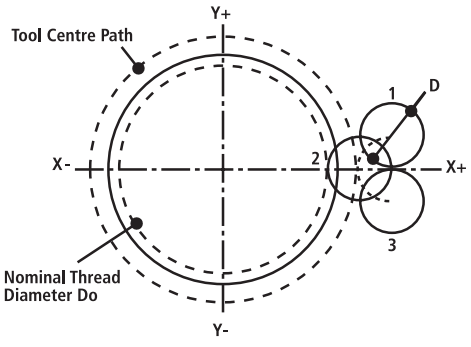
Radius of tool centre path R = (Do-D)/2
Where Do = Nominal Thread Diameter
D = Cutting diameter

Assuming X0 Y0 is centre of component and thread is climb milled.
Cutting data obtained from chart on Grades, Speeds & Feeds page AA25

M24 x 3.0p ISO x 32min full thread internal RH thread in austenitic stainless steel using HN16 B 159 3.0ISO RS21
Cutting speed =60m/min (1201rpm)
Cutting diameter =15.9mm
Feed per tooth =0.025mm/tooth
Cutting edge feed =120mm/min
Tool centre feed =41mm/min

S1201 M03
(Set cutting speed at 1201rpm)
G00 G90 X0 Y0 Z3
(Rapid feed to position 1 at 3mm above component)
G01 Z-32.375 F1000
(Feed to 32mm thread depth plus 1/8 of thread pitch)
G01 X2.025 Y-2.025
(Feed to position 2)
G03 X4.05 Y0 I0 J2.025 Z-32 F41
(Feed to nominal thread diameter at position 3 moving up 1/8 pitch at feed rate of 41mm/min)
G03 X4.05 Y0 I-4.05 J0 Z-29
(Interpolate counter clockwise one revolution to position 3 moving up one pitch of the thread)
G03 X2.025 Y2.025 I-2.025 J0 Z-28.625
(Feed out of thread to position 4 moving up 1/8 pitch)
G01 X0 Y0 F1000
(Feed out to position 1)
G00 X0 Y0 Z3
(Rapid feed to position 1 at 3mm above component)

External Parallel Thread Milling Example



Radius of tool centre path = (Do+D)/2
Where Do = Nominal Thread Diameter
D = Cutting diameter

Assuming X0 Y0 is centre of component and thread is climb milled.
Cutting data obtained from chart on Grades, Speeds & Feeds page AA25

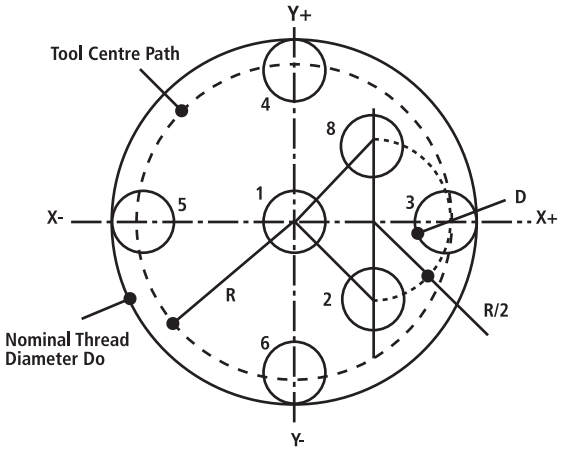
M24 x 3.0p ISO x 32min full thread external RH thread in austenitic stainless steel using HN16 B 159 3.0ISO RS21
Cutting speed =60m/min (1201rpm)
Cutting diameter =15.9mm
Feed per tooth =0.025mm/tooth
Cutting edge feed =120mm/min
Tool centre feed =214mm/min
Pitch of arc-in / out movement = 0.5869mm

S1201 M03
(Set cutting speed at 1201rpm)
G00 G90 X26.0597 Y7.95 Z3
(Rapid feed to position 1 at 3mm above component)
G01 X26.0597 Y7.95 Z-28.4131 F1000
(Feed to 32mm thread depth minus a thread pitch minus the pitch of arc-in / out movement)
G03 X18.1097 Y0 I0 J-7.95 Z-29 F214
(Feed to nominal thread diameter at position 2 moving down the pitch of arc-in / out movement at feed rate of 214mm/min)
G02 X18.1097 Y0 I-18.1097 J0 Z-32
(Interpolate clockwise one revolution to position 2 moving down one pitch of the thread)
G03 X26.0597 Y-7.95 I7.95 J0 Z-32.5869
(Feed out of thread to position 3 moving down the pitch of arc-in / out movement)
G00 Z3
(Feed out to position 3 at 3mm above component)

Programming examples

The following programming examples are based on the thread milling tool centre path, which requires no tool radius compensation other than an offset for tool wear. These examples are purely for guidance only and may need to be altered to suit the CNC control you are using. Sample programmes can be supplied to suit your individual requirements.

Internal Taper Thread Milling Example



Assuming X0 Y0 is centre of component and thread is climb milled. Cutting data obtained from chart on Grades, Speeds & Feeds page AA25
With tapered threads, the radius of the part diameter increases as the tool rises in cut.

R at start position 3 = (nominal thread diameter - effective cutting diameter) / 2
R at position 4 = R + (0.25 x pitch/32)
R at position 5 = R + (0.5 x pitch/32)
R at position 6 = R + (0.75 x pitch/32)
R at end position 3 = R + (pitch/32)
R at position 8 = (R + pitch/32)) / 2

1/2" x 14NPT internal thread in austenitic stainless steel using HX16 B 145 14NPT RS21
Effective cutting diameter =15.5mm at 16.907mm from front of tool
Cutting speed =60m/min (1232rpm)
Feed per tooth =0.025mm/tooth
Cutting edge feed =154mm/min
Tool centre feed =42mm/min

S1232 M03
(Set cutting speed at 1232rpm)
G00 G90 X0 Y0 Z3
(Rapid feed to position 1 at 3mm above component)
G01 Z-17.2839 F1000
(Feed to start depth position)
G01 X1.4305 Y-1.4305
(Feed to position 2)
G03 X2.861 Y0 I0 J1.4305 Z-17.0571 F42
(Feed to nominal thread diameter at position 3
moving up 1/8 pitch of the thread at feed rate of 49mm/min)
G03 X0 Y2.8752 I-2.8752 J0 Z-16.6036
(Interpolate counter clockwise to position 4 moving up
one 1/4 pitch of the thread and with increasing radius)
G03 X-2.8893 Y0 I0 J-2.8893 Z-16.15
(Interpolate counter clockwise to position 5 moving up one 1/4 pitch
of the thread and with increasing radius)
G03 X0 Y-2.9035 I2.9035 J0 Z-15.6964
(Interpolate counter clockwise to position 6 moving up one 1/4 pitch
of the thread and with increasing radius)
G03 X2.9177 Y0 I0 J2.9177 Z-15.2429
(Interpolate counter clockwise to position 3 moving up one 1/4 pitch
of the thread and with increasing radius)
G03 X1.4588 Y1.4588 I-1.4588 J0 Z-15.0161
(Feed to position 8 moving up 1/8 pitch of the thread with increasing radius)
G01 X0 Y0 F1000
(Feed out to position 1)
G00 X0 Y0 Z3
(Rapid feed to position 1 at 3mm above component)

Speed & Feed Calculations

To calculate rpm of cutter	N = rpm V = Recommended cutting speed in m/min d = Cutting diameter of tool in mm	$N = \frac{1000 \times V}{d \times \pi}$ (rpm)
To calculate feed at cutting edge	F1 = Feed at cutting edge in mm/min fz = Recommended feed per tooth mm/tooth Z = No of teeth / flutes N = rpm	F1 = fz x z x N (mm/min)
To calculate feed at tool centre (for internal threads)	F2 = Feed at centre of tool in mm/min F1 = Feed at cutting edge in mm/min Do = Nominal diameter of thread d = Cutting diameter of tool in mm	$F2 = \frac{F1 \times (Do - d)}{Do}$ (mm/min)
To calculate feed at tool centre (for external threads)	F2 = Feed at centre of tool in mm/min F1 = Feed at cutting edge in mm/min Do = Nominal diameter of thread D = Cutting diameter of tool in mm	$F2 = \frac{F1 \times (Do + d)}{Do}$ (mm/min)