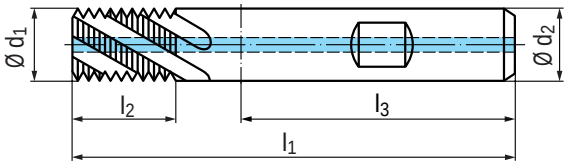
Tool Description Of Thread Mills Type GF

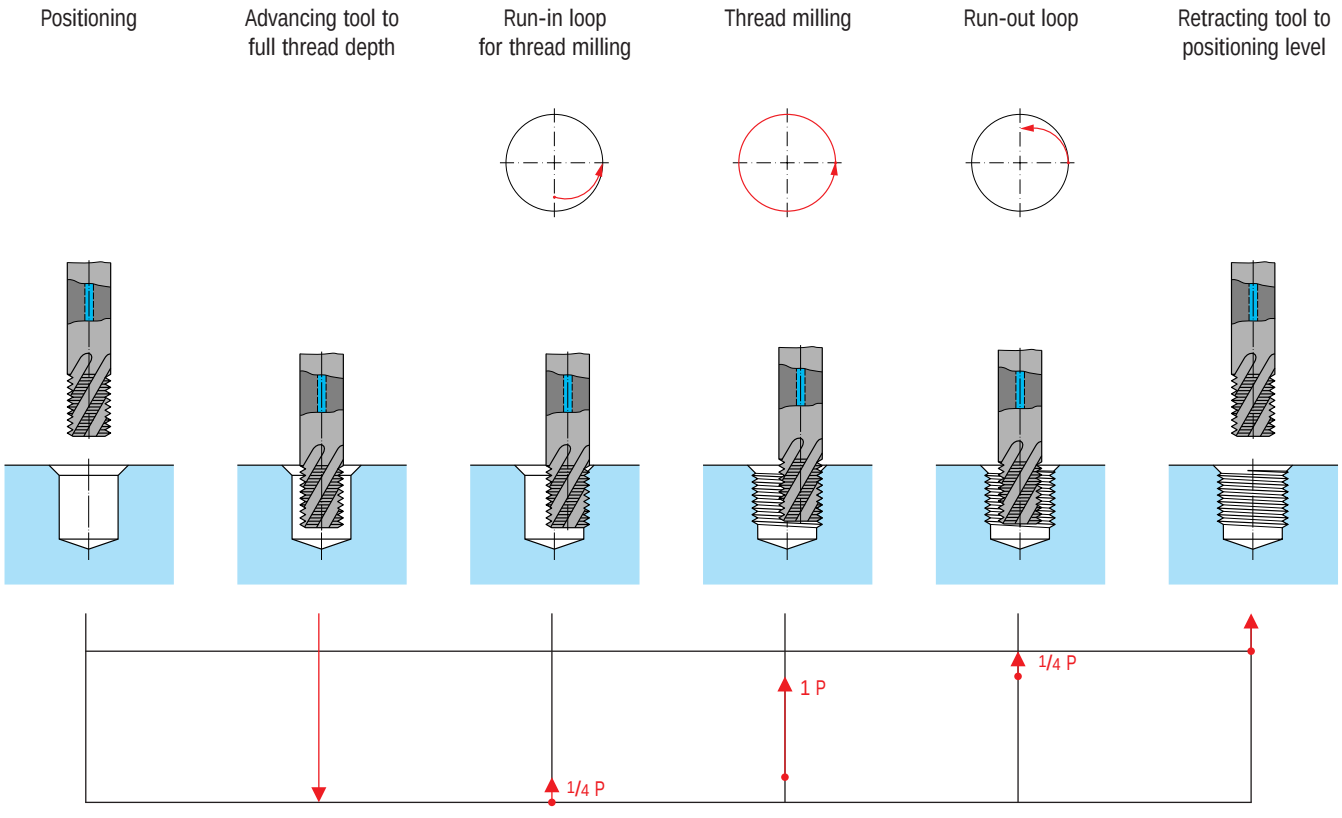


Description:

- Thread mills with shank for the production of larger threads
- In order to avoid profile distortions, it is important to make sure that the cutter diameter does not exceed approx. 2/3 (for fine threads 3/4) of the nut thread diameter
- Also availabe with a cutting face for cutting or chamfering the thread hole upon request
- More economic than milling cutters with inserts

Application areas: All materials

Preparatory work: Drilling of thread hole
(in aluminium, casting the hole with an allowance is also possible)



Programming Instruction
CNC Thread Milling With Thread Mills Type GF

EMUGE

Dimension: _____ Company: _____

Nominal thread diameter D: _____ Inch Workpiece: _____

Thread pitch P: _____ Inch Material: _____

Drilled hole diameter D_1: _____ Inch Date: _____

Tool: _____ Cutting material: _____

Article no.: _____ Coating: _____

Flutes z: _____

Cutter diameter d_1: _____ Inch (measured on the cutting part)

Cutter radius compensation k [1]: _____ Inch 0.05 x P

Cutter radius to be programmed [2]: _____ Inch ($1/2\ d_1 - k$)

Thread depth b: _____ Inch

Cutting speed v_c: _____ SFM ($n = \frac{v_c \times 12}{d_1 \times \pi}$) S = _____

Feed (milling) f_z: _____ Inch ($v_f = f_z \times z \times n$) F = _____ (contour)

Please note: For milling feed related to the path of the tool axis: $v_f = \frac{v_{f\ contour} \times (D - d_1)}{D}$. F = _____ (axis)

Then, in block N40 1/2 F (tool axis) and in block N60 F (tool axis).

Required parameters:

E1	=	1/2 nominal thread diameter	1/2 D	=	_____	Inch
E2	=	thread pitch [3]	P	=	_____	Inch
E3	=	thread depth	b	=	_____	Inch
E4	=	1/4 P (for climb milling and right-hand threads)	1/4 P	=	_____	Inch
E5	=	beginning of contour in y	0.5 x P	=	_____	Inch
E6	=	run-in loop	(E1 – E5)	=	_____	Inch

CNC internal thread milling (climb milling, on the contour, incremental, acc. DIN 66025)

N 10	G 54	G 90	G 00	X	Y	Z 2	S	T 01 [2] M 03
N 20	G 91	G 00	Z– . . . . (**E3** + 2)					
N 30	Y . . . . . (**E5**)							
N 40	G 41	G 01	X . . . . . (**E6**)	F . . . . . **(milling, contour)**				
N 50	G 03	X– . . . . . (**E6**)	Y . . . . . (**E6**)	Z . . . . . (**E4**)	I– . . . . . (**E6**)	J 0		
N 60	G 03	X 0	Y 0	Z . . . . . (**E2**)	I 0	J– . . . . . (**E1**)		
N 70	G 03	X– . . . . . (**E6**)	Y– . . . . (**E6**)	Z . . . . . (**E4**)	I 0	J– . . . . . (**E6**)		
N 80	G 00	G 40	X . . . . . (**E6**)	Y– . . . . (**E5**)				
N 90	G 90	Z 2						

[1] The cutter radius measured over the tooth crests of the threaded part must be reduced by the amount of the cutter radius compensation. This is necessary to achieve a depth of cut to the middle of the 6H/ISO2 nut tolerance. Please note, however, that this also depends on the radial deflection of the tool (tensile strength of the material, projecting length of the tool).

[2] The cutter radius to be programmed is normally included in the tool memory.

[3] For right-hand threads + (screwing out); for left-hand threads – (screwing in).