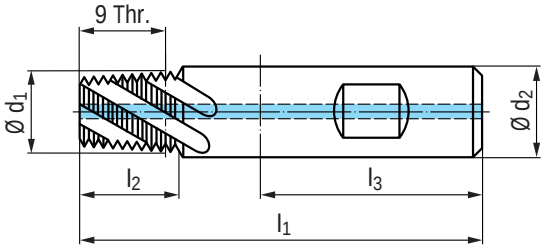
Tool Description Of Thread Mills Type GF Tapered

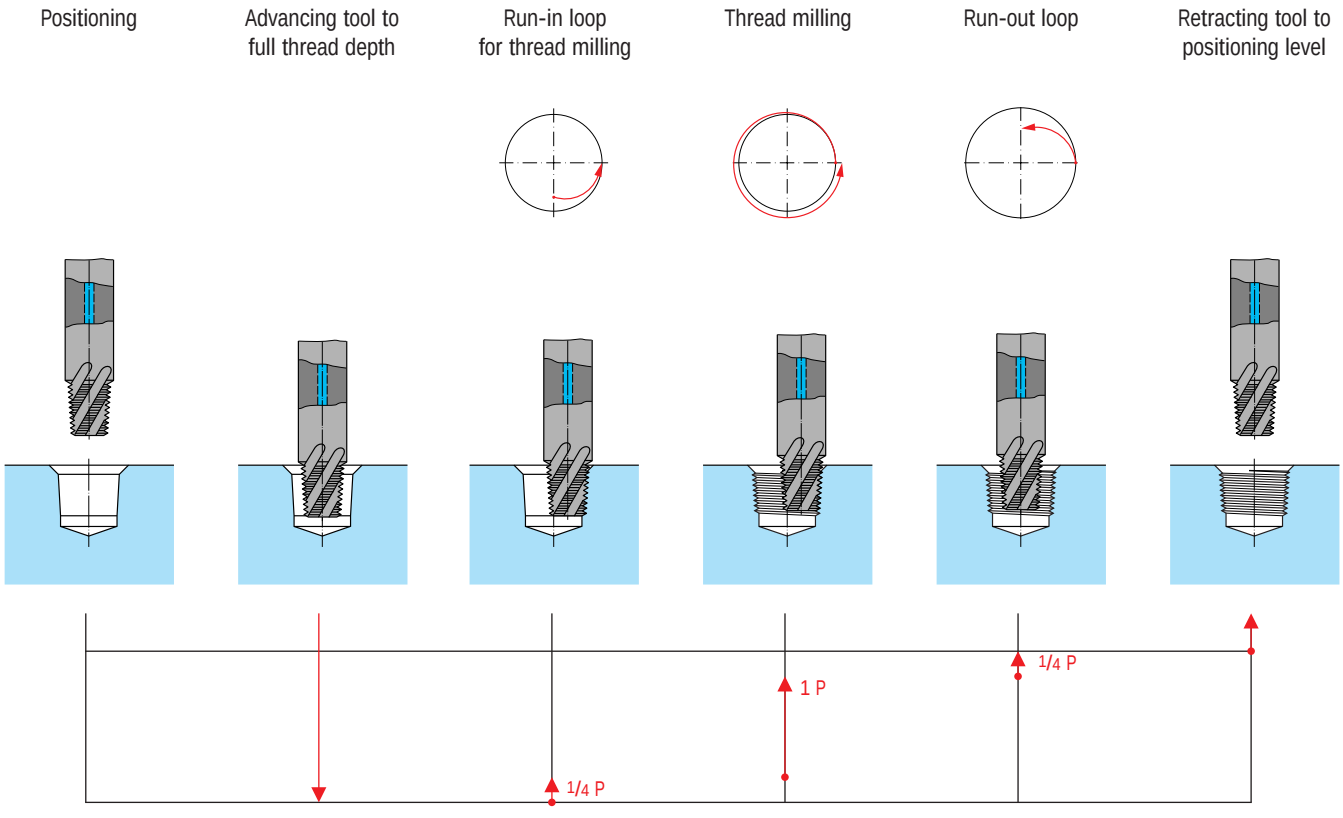


Description:

- Thread mills with shank for the production of tapered pipe threads
- Corrected thread profile, for producing a true-to-gauge thread even with a very small difference between thread diameter and tool diameter
- Also availabe with a cutting face for chamfering the thread hole upon request

Application areas: All materials

Preparatory work: Drilling of a straight thread hole
(in difficult materials, drilling of a tapered thread hole is advisable)



Programming Instruction
CNC Thread Milling With Thread Mills Type GF Tapered

EMUGE

Programming example for a tapered thread 1:16 NPT 1/2 - 14

Dimension:	**NPT 1/2 - 14**		Company:	
Outside diameter of nut D:	0.830	Inch	Workpiece:	
Taper ratio:	1 : 16		Material:	
Thread pitch P:	0.071	Inch	Date:	
Usable depth:	0.606	Inch		

Tool:	Thread milling cutter - NPT tapered		Cutting material:	Solid carbide
Article no.:			Coating:	
Flutes z:	4			

Cutter diameter d_1:	0.561	Inch	(measured on the cutting part, measuring point 9 threads from the tool face)		
Cutter radius to be programmed:	0.281	Inch	(1/2 d_1)		
Cutting speed v_c:	328	SFM	($n = \frac{v_c \times 12}{d_1 \times \pi}$)	S =	2235
Feed (milling) f_z:	0.002	Inch	($v_f = f_z \times z \times n$)	F =	18 (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 = 6 (axis)

Then, in block N50 1/2 F (tool axis) and in block N60 F (tool axis).

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

N 10	G 54	G 90	G 00	X	Y	Z 0.1	S 2235	T 01	M 03
N 20	G 91	G 01	Z– 0.684		F 18 **(milling, contour)**				
N 30	Y 0.081								
N 40	G 41	X 0.335							
N 50	G 03	X– 0.335	Y 0.335		R 0.335	[F 6] 1)			
N 60	G 03	X– 0.416	Y– 0.415		Z 0.0179	I– 0.0003	J– 0.415		
N 70	G 03	X 0.416	Y– 0.416		Z 0.0179	I 0.416	J– 0.007		
N 80	G 03	X 0.417	Y 0.416		Z 0.0179	I 0.0003	J 0.417		
N 90	G 03	X– 0.417	Y 0.417		Z 0.0179	I– 0.417	J 0.0003		
N 100	G 03	X– 0.335	Y– 0.335		R 0.335				
N 110	G 01	G 40	X 0.335						
N 120	G 90	X	Y						
N 130	Z 0.1								

Please note that it is essential to gauge the first finished thread! This will make it possible to introduce a tool radius or depth compensation which may be necessary. Compensation is made by adjusting the distance of the measuring steps on the plane side of the plug gauge form the workpiece.

Variables for influencing the thread diameter on the workpiece:

1. The cutter radius to be programmed in the tool memory T01
2. The plunging depth (thread depth) –Z in block N20

Radius compensation = lacking screw-in depth x taper ratio (1:16) x 0,5

Please note: A smaller tool radius will create an increased screw-in depth!

1) If your control does not calculate the centre point feed automatically (command M110 and M111) please use the feed values printed in brackets.

All information without obligation!