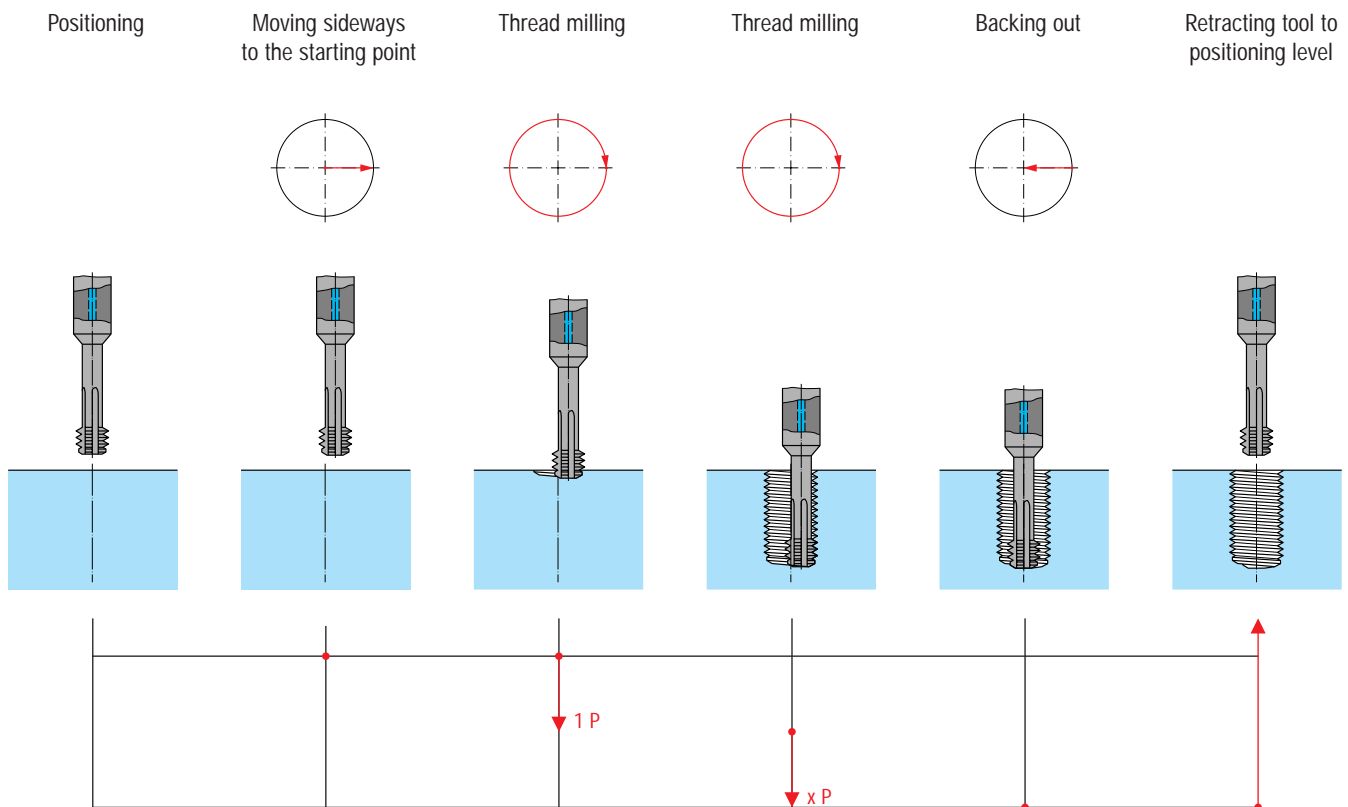


**Description:**

- Thread mills for work in hard materials
- Corrected profile, to be used for different thread sizes
- Circular movement produces thread hole, thread and chamfer at the same time
- Produces a short, easy-to-remove chip (even in long-chipping materials)
- Variable thread depth up to  $2 \times D$

**Application areas:** Hardened and difficult-to-machine materials

**Preparatory work:** None



# Programming Instruction

## CNC Thread Milling With Circular Thrillers Type ZBGF



**Dimension:** \_\_\_\_\_

Nominal thread diameter D: \_\_\_\_\_ Inch

Thread pitch P: \_\_\_\_\_ Inch

Drilled hole diameter D<sub>1</sub>: \_\_\_\_\_ Inch

Company: \_\_\_\_\_

Workpiece: \_\_\_\_\_

Material: \_\_\_\_\_

Date: \_\_\_\_\_

**Tool:** \_\_\_\_\_

Cutting material: \_\_\_\_\_

Article no.: \_\_\_\_\_

Coating: \_\_\_\_\_

Flutes z: \_\_\_\_\_

Cutter diameter d<sub>1</sub>: \_\_\_\_\_ Inch (measured on the cutting part)

Cutter radius compensation k<sup>1)</sup>: \_\_\_\_\_ Inch (cutter radius compensation k acc. workcase)

Cutter radius to be programmed 2): \_\_\_\_\_ Inch (1/2 d<sub>1</sub> – k)

Thread depth b: \_\_\_\_\_ Inch

Cutting speed v<sub>c</sub>: \_\_\_\_\_ SFM ( $n = \frac{v_c \times 12}{d_1 \times \pi}$ ) S = \_\_\_\_\_

Feed (milling) f<sub>z</sub>: \_\_\_\_\_ Inch ( $v_f = f_z \times z \times n$ ) F = \_\_\_\_\_ (contour)

**Please note:** For milling feed related to the path of the tool axis:  $v_f = \frac{v_{f \text{ contour}} \times (D - d_1)}{D}$  . F = \_\_\_\_\_ (axis)  
Then, in block N30 F (tool axis).

### Required parameters:

E1 = 1/2 nominal thread diameter	1/2 D = _____ Inch
E2 = thread pitch	P = _____ Inch
E3 = thread depth 3)	b = _____ Inch
E4 = chamfer length 4)	l <sub>A</sub> = _____ Inch
E5 = safety distance = pitch P	P = _____ Inch
E6 = number of repetitions	(E3+E4+E5)/E2-1 = _____ 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 ..... (E5)  S .....      T 01 2)      M 04
N 20      G 91
N 30      G 42      G 01      X 0      Y- ..... (E1)  F ..... (milling, contour)
N 40      G 02      X 0      Y 0      Z- ..... (E2)  I 0      J ..... (E1)
.....5)
N 50      G 40      G 01      X 0      Y ..... (E1)
N 60      G 90      G 00      Z ..... (E5)
    
```

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) The thread depth b as entered must be divisible by the pitch P.  
 4) The chamfer length l<sub>A</sub> usually amounts to one thread.  
**5) Block N 40 must be repeated with the number of threads (E6).**