MACRO SHOP

Pulley groove

Fanuc 0/16/18/21/31

Rev. 5

FIELD OF APPLICATION

The macro-instruction carries out the groove machining for both straight and inclined sides. It is particularly suitable for grooving pulleys with trapezoidal slot. In fact, in the parameters for defining the geometry, the dimensions of the slot refer to the effective diameter of the pulley. In particular, the macro carries out roughing with radial cuts and finishing by profiling along the two sides of the groove. It is possible to choose whether to make a chamfer or a corner round on the top edge. If several grooves are to be machined with a constant pitch, simply set the number of grooves and the pitch to repeat machining on all the grooves. For better chip management, the macro has a chip breakage function; by setting the increment value to a value other than zero, it will perform increments with chip breaks in which the macro will execute a small retraction to break and release the chip. Macro designed for all Fanuc controls from series 0 to series 31. Read the manual carefully for a correct use, paying attention to the "Warnings" paragraph.

PARAMETER DESCRIPTION

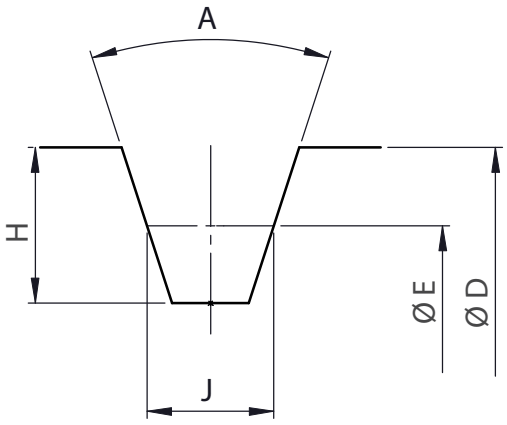
Below is a combination of the letter and its meaning:

D= EXTERNAL DIAMETER
E= EFFECTIVE DIAMETER
H= GROOVE DEPTH
J= EFFECTIVE WIDTH
I= RADIAL DEPTH OF CUT
A= FULL GROOVE ANGLE
Z= POSITION IN Z OF THE CENTRE OF THE FIRST GROOVE
S= TOOL THICKNESS
R= INSERT RADIUS
Q= RADIAL SAFETY DISTANCE
W= STEP (WITH SIGN)
K= NUMBER OF GROOVES
C= ROUND/CHAMFER SELECTION 0=NONE 1=ROUND 2=CHAMFER
M= TOOL OVERLAPPING FACTOR
F= FINISHING FEEDRATE
V= SIDE ALLOWANCE FOR FINISHING
U= RADIAL FLOOR ALLOWANCE FOR FINISHING
B= CHAMFER OR ROUND SIZE
T=CHIP BREAKAGE STEP
X= RETRACTION X COORDINATE
Y=0/NULL roughing+finishing; 1=roughing only; 2=finishing only

FOCUS PARAMETERS

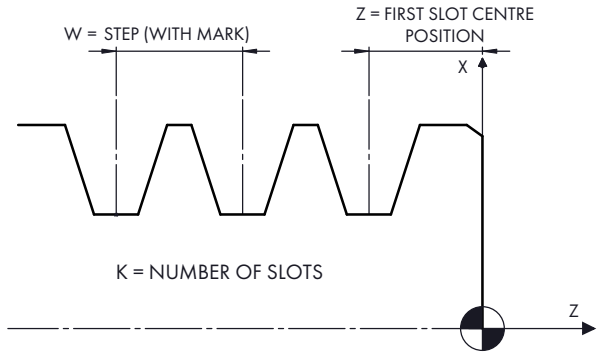
PAR. E D H J A

Using the parameters E, D, H, J and A, the groove shape is defined. Where D is the external diameter, A the groove internal angle and H the groove depth. For the definition of the groove width, the effective diameter is used, which is mainly used for trapezoidal profile grooves. The groove width is defined with the parameter J on the effective diameter. If you do not have the effective diameter but only the width on the external diameter, simply set the diameter E to the same value as the external diameter D and the width J will be the groove width on the external diameter. Setting A equal to 0 will result in a groove with straight sides.

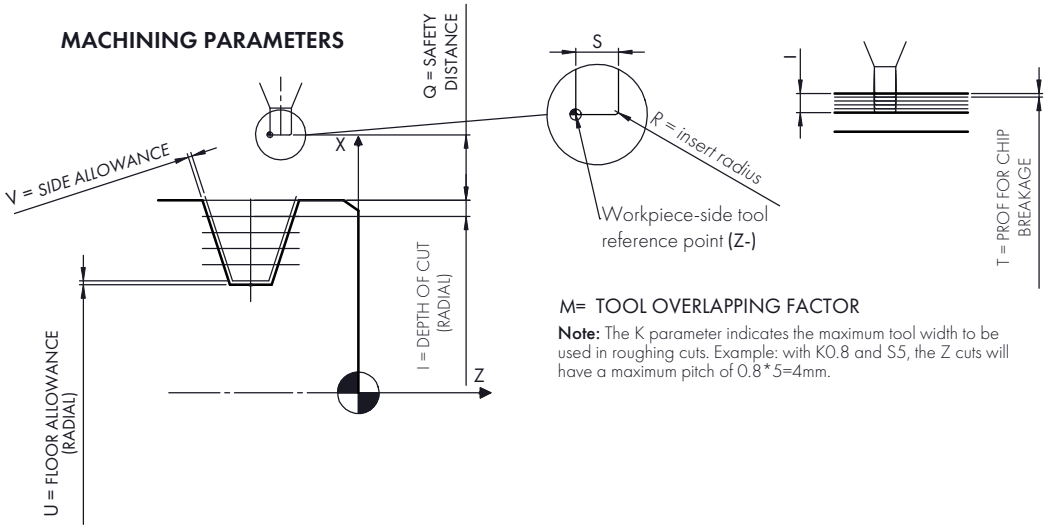


PAR. Z,K,W,

MULTIPLE GROOVES



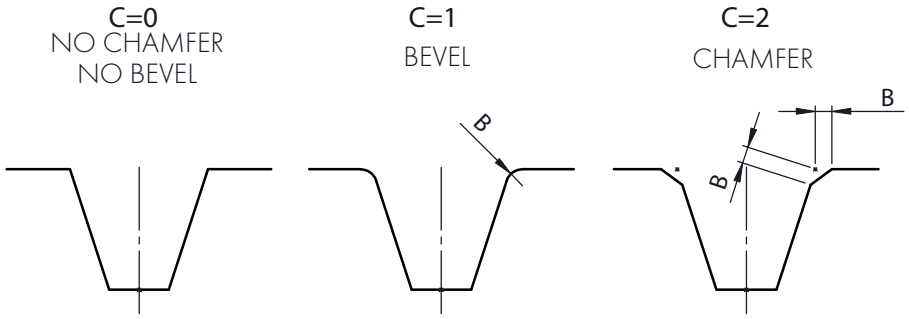
The macro allows one or more grooves to be executed. The position of the first groove is defined with the Z parameter that indicates the co-ordinate along the Z axis of the centre of the first groove. If you want to execute more than one groove, just set in the K parameter the number of grooves to be executed and with W the pitch between the grooves. The parameter W requires a sign, because it also defines the direction in which you move from the first groove to the following ones. For example, if the first groove I want to execute is the one that is in Z+, I will have to indicate W with a negative sign and vice versa.



PAR. V,U

To define the machining parameters, we will use V and U to set the side and bottom groove allowances respectively.

The I parameter is used to set the depth of cut with which the roughing cuts are to be made. When reaching the depth of cut, the macro will retract to allow the chip to be broken and released. The increment before the chip releasing is defined with the T parameter. Setting T equal to zero will not perform chip breakage. For tool definition, the insert widtht and any radius on the edges must be set. The most important thing is the correct position of the tool corrector, in fact the macro-instruction sets the program with the tool measuring point on the side closest to the workpiece.
Once the tool is brought to depth on the basis of the I parameter, it is moved with plunge cuts until the entire width of the groove is roughened; the tool overlapping factor (M) is used for this movement. This factor multiplied by the tool width will define the maximum tool load. With M=1 it will execute maximum cuts equal to the insert width, therefore a value between 0.6 and 0.8 is recommended.



In the finishing cut, chamfers or rounds can be executed on the two external edges. With C=0 the edge remains sharp; with C=1 it will make corner rounds; with C=2 it will make chamfers.

CYCLE DESCRIPTION

Machining will begin with roughing the first groove by performing plunge cuts for a depth of cut equal to the I parameter. Once the depth of cut I has been reached, it will move with plunge cuts until the entire groove width has been roughened. Once the depth of cut I has been reached, it moves on with plunge cuts until the entire groove width has been roughened and continues with a further increase along the X axis of the value I until the groove has been completely cleared. When carrying out the radial grooving cuts equal to the depth of cut I, it is possible to carry out chip breaking defined with the parameter T (set a value other than 0). Once the groove has been roughened, it will execute the finishing by moving down the two sides and finishing the bottom. The movement on the bottom is calculated automatically to always ensure that it does not collide with the opposite edge but guarantees the overlap of the two cuts.

Once the first groove has been made, it will return to the height defined by the X parameter, and then move on to the next grooves if several grooves have been programmed with the K parameter different from 1, or it will end the cycle by remaining at the safety X height and in Z equal to the last machining position.

PROGRAMMING

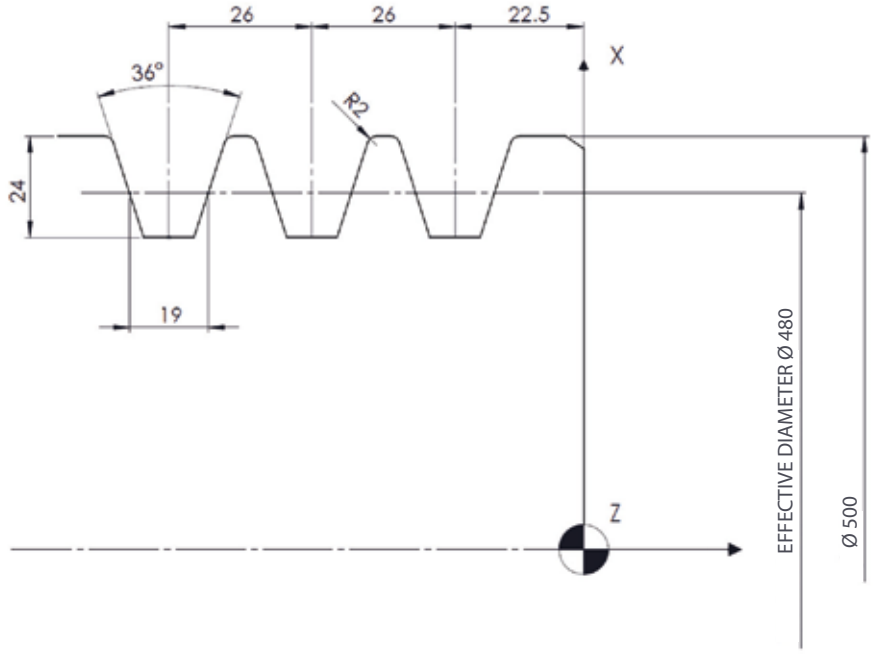
The correct tool for machining with this macro is a groove tool, the width of which must be less than the groove bottom, also taking into account the oversize on the sides (V). If the width (S) of the tool is greater than the width at the bottom of the groove, the macro will check it and if necessary display an error message.
Groove tools do not usually have very large radii on the cutting edge, but it is recommended to set the radius with the R parameter anyway, since in finishing it carries out the profile by compensating the nose radius.
Compensation is done analytically without using the G41/G42 functions, so it is not necessary to set the tool tip orientation T and the radius R in the tool list.
It is not possible to use circular inserts, only inserts with a flat face.

The cycle is to be used as a subprogram to be called with the function G65 and indicating on the same line the parameters respecting the letters indicated in the "Parameters description" section.
The subprogram is provided with numbering O8015, so the subprogram will be called with G65P8015 followed by the parameters. If the subprogram needs to be renumbered, the letter P must be followed by the new program number.
Before calling the cycle, start the spindle and define the technological parameters. In particular, the feed rate F for roughing must be programmed before calling the cycle, while the feed rate F programmed in the call line will be the feed rate used in finishing.
Please note that the parameters I, J and K must be written exactly in alphabetical order, while the order in which the others are written is irrelevant.

FOCUS PARAMETERS

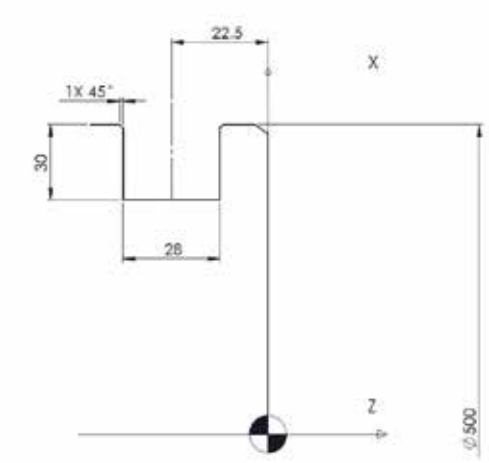
NOTE TO PAY ATTENTION TO

EXAMPLE 1: TRAPEZOIDAL PULLEY GROOVE




```
(PARAMETRIC TRAPEZOIDAL GROOVE PROGRAM)
T0404 (GROOVE T. SQ.5 R0.2)
G92S600
G96G95S200F0.35M4
G65 P8015 D500 E480 H24 I2.5 J19 K3 A36 Z-22.5 S5 R0.2 Q5 W-26 M0.8 F0.15
V0.15 U0.05 C1 B2 X650
G0 Z300
G0 X300
M30
%
```

EXAMPLE 2: STRAIGHT GROOVE



O0001

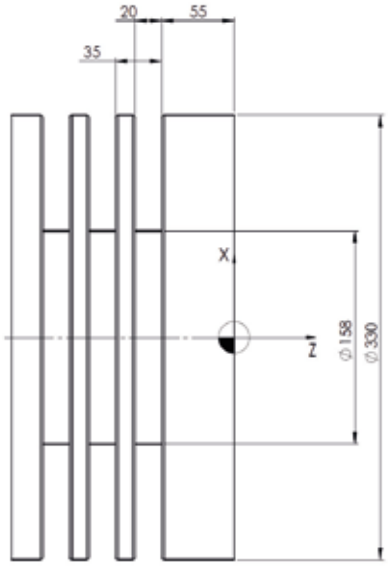
(STRAIGHT SIDE GROOVE PROGRAM)
T0404 (SQ5 R0.2)
G92S600
G96G95S200M4F.3

(EXAMPLE OF A SINGLE GROOVE LAUNCH)
G65 P8015 D500 E500 H30 I2.5 J28 K1 A0 Z-22.5 S5 R0.2 Q5 M0.8 F0.15 V0.15 U0.05 C2 B1 X600

G0 Z300
G0 X300
M30

EXAMPLE 3: MULTIPLE STRAIGHT GROOVES

Machining with sq. tool 4



Solution no. 1: first the roughing of all the grooves and then the finishing of all the grooves.

```
T0101(GROOVE T. SQ4)
M42
G96S240F0.5M3
G0X600Z20
(ROUGHING)
G65P8015D330E330H86I2J20A0Z-65S4R1Q5W-35K3C1M0.
8V0.1U0.05B2T0.3X500Y1
M0 (INSERT CHECK)
(FINISHING)
G65P8015D330E330H86I2J20A0Z-65S4R1Q5W-35K3C1M0.
8V0.1U0.05B2T0.3X500Y2F0.1
G0X1200Z300
M5
```

Solution No. 2: since the grooves are very deep, it is recommended to carry out a multi-step roughing of the three grooves at the same time and then finish all the three grooves.

```
T0101(GROOVE T. SQ4)
M42
G96S240F0.5M3
G0X600Z20
G65P8015D330E330H30I2J20A0Z-65S4R1Q5W-35K3C0M0.8V0.1U0X400Y1T0.3
G65P8015D270E270H30I2J20A0Z-65S4R1Q5W-35K3C0M0.8V0.1U0X400Y1T0.3
G65P8015D210E210H26I2J20A0Z-65S4R1Q5W-35K3C0M0.
8V0.1U0.05X400Y1T0.3
M0 (FINISHING)
G65P8015D330E330H86I2J20A0Z-65S4R1Q5W-35K3C1M0.
8V0.1U0.05B2T0.3X400Y2F0.1
G0X1200Z300
M5P11
M30
```

AVVERTENZE

1. Inside the macro, in the first few lines of the program, you will find parameters that will remain constant for all the programs in which the macro is called. They are located just below the words "INTERNAL PARAMETER". Parameters #140, #141 and #146 are the default values that are used by the macro. Parameter #140 defines whether to perform increments in Z at the same coordinate as X by going in the Z+ or Z- direction. The macro is delivered with the parameter set to 1, which corresponds to executing the cuts by moving in the Z+ direction.

Parameter #141 is the safety distance in X to increase from one Z coordinate to another. Finally, parameter #146 defines how much the tool retracts during chip breakage.

```
(***INTERNAL PARAMETER***)
#140=1(1=STEP Z+ DIRECTION  -1=STEP Z- DIRECTION)

IF[#4006EQ20]GOTO7878
(MM)
#141=0.5(SAFETY DISTANCE BETWEEN DIFFERET Z STEP AT THE SAME DIAM)
#146=0.3 (RETRACTION GOR CHIP-BROKEN)

GOTO7877
N7878
(INC)
#141=0.019(SAFETY DISTANCE BETWEEN DIFFERET Z STEP AT THE SAME DIAM)
#146=0.011 (RETRACTION GOR CHIP-BROKEN)
(***END INTERNAL PARAMETER***)
```

2. The macro uses parameters #100 to #149, so it is necessary to check that these parameters can be used, if necessary contacting the machine builder. If it is necessary to use parameters with a different numbering, request the modification of the macro.

3. The macro is delivered already tested, but it is advisable for the first few times to do the necessary tests at no-load or away from the workpiece.

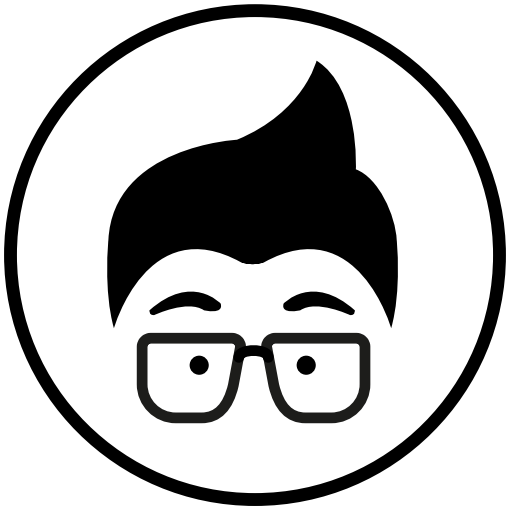
4. The cycle automatically sets the absolute coordinates by setting function G90. If you need to set the incremental coordinates after the macro-instruction, set function G91.

5. The macro only works with the G65 call and not with the G66 modal call function. Follow the instructions in the "Programming" paragraph.

6. To improve tool life and reduce stress and vibration, it is recommended that if the sides are square-shouldered in relation to the bottom, to always use the machining strategy with the sides recovery as described in example 1.

7. When calling the macro, the parameters I, J and K must be written in alphabetical order, not necessarily one after the other, but the order between them must be as follows: I, J, K.

8. To be able to use the macro on your machine, make sure that the parametric programming option is enabled. Although most machines have B macro programming enabled, check that your machine does. To do this, simply go to the MDI window and enter #100=1, press start and if no alarm message is displayed it means that B macro programming is enabled. On some lathes in the 0 series, it may be that the # key is missing; to test this, simply load a program with only line #100=1 and have it executed automatically.



www.cncofcourse.com