MACRO
SHOP

TROCHOIDAL MILLING SLOT

FANUC 0/16/18/21/31

✎ Rev. 1

FIELD OF APPLICATION

The macro-instruction carries out the roughing of an open slot using the trochoidal strategy, whereby production times can be significantly reduced, especially on highly alloyed materials such as duplex, heat-resistant alloys and titanium alloys. The trochoidal path developed by the macro does not have any linear line, but is obtained by chaining several circular interpolations. Since it is not a point path as could be generated by a CAM, but is a circular interpolation, there is no need for machines with a high-speed block reading function. Furthermore, by developing the path completely in circular interpolation, the approach to the workpiece is very smooth and ensures greater process reliability. With the macro it is possible to choose whether to carry out only the roughing or possibly also the contour finishing of the sides. The trochoidal path is fully parameterised using the parameters defined in the call, starting from the direction of orientation of the groove, up to the pitch of the trochoidal milling. It is also possible to define the direction of machining, whether concordant or discordant, and a possible depth of cut in Z, if you do not want to carry out the groove in a single cut. With a single block, the operator can generate the roughing and finishing path in less than a minute, the time to fill in the necessary parameters.
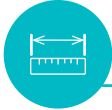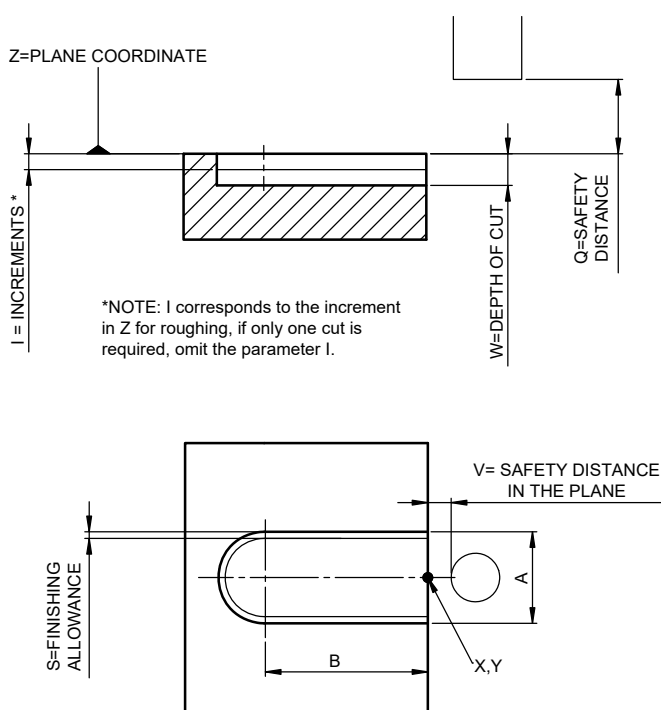
PARAMETER DESCRIPTION

Below is a combination of the letter and its meaning:

```
X=POSITION IN X
Y=POSITION  IN Y
A=SLOT WIDTH
B=SLOT LENGTH
Z=Z-DIMENSION OF THE PLANE
W=DEPTH
I=INCREASE IN Z, ROUGHING
D=INCREASE IN Z, FINISHING
S=LATERAL ALLOWANCE, FINISHING
U=THREAD PITCH
Q=SAFETY DISTANCE IN Z
V=LATERAL SAFETY DISTANCE
K=SLOT DIRECTION
M=MILLING DIRECTION      0/NONE=CONCORDANT; 1=DISCORDANT
F=ROUGHING FEED RATE
E=FINISHING FEED RATE
T=MACHINING TYPE    T=0 rough+finishing; T=1 rough. only;
T=2 finishing only
```
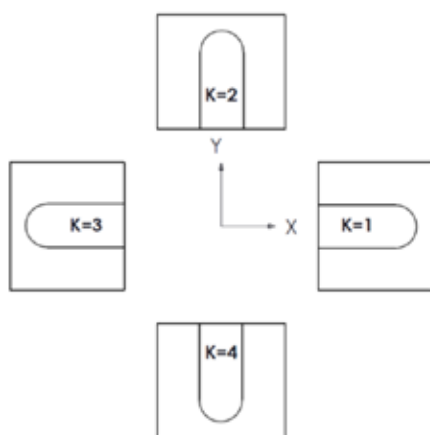
The key can be executed in all orthogonal directions by setting the parameter K as shown in the drawing above.

Z=PLANE COORDINATE

I = INCREMENTS *

*NOTE: I corresponds to the increment in Z for roughing, if only one cut is required, omit the parameter I.

W=DEPTH OF CUT

Q=SAFETY DISTANCE

S=FINISHING ALLOWANCE

V= SAFETY DISTANCE IN THE PLANE

A

B

X,Y

The X and Y coordinates are used to define the position of the most external point of the slot, that is the open edge of the slot. Parameters A and B are used to define the width and length of the slot. Regarding the position in the working axis, the Z parameter is used to define the top co-ordinate of the workpiece and the W parameter to define the depth of the slot. The W parameter is always expressed as positive, the macro will still perform the slot by descending in the Z-direction in relation to the workpiece.

Mediante il parametro Q si definisce la distanza di sicurezza espressa in modo incrementale rispetto al piano Z; se si imposta Q pari a 50, la macro eseguirà gli svincoli a 50mm dal piano Z. Nel caso in cui la cava fosse molto profonda rispetto al diametro fresa e si volessero eseguire più passate di sgrossatura trocoidale, basterà impostare il parametro I per definire la profondità di passata in Z (ap).

If you wish to carry out several cuts in Z in the finishing phase as well, it is possible to insert the parameter D equal to the depth of cut in Z. If parameter I is omitted, the macro will carry out only one cut in Z during roughing; if parameter D is omitted, the macro will carry out only one cut in Z during finishing.

The macro automatically reads the tool radius (by setting the internal parameter correctly, see the "Warnings" paragraph) and then automatically performs the calculations in relation to the cutter diameter, as well as the starting positions where it adds the tool radius to the V parameter (lateral safety distance).

If you want to carry out both slot roughing and flank finishing, simply omit the T parameter or set it to 0. If you only want to perform roughing, set T equal to 1 and then T equal to 2 to perform only finishing. In cases where finishing is present, the feed rate must be set using parameter E. In cases where roughing has been defined, the sides allowance should be set with parameter S and the roughing feed with parameter F.

In the macro, there are two internal parameters defined in the "Warnings" section that are used for settings that will remain constant each time the macro is used.

PROGRAMMAZIONE

The cycle is to be used as a subprogram to be called with the G65 function and indicating the parameters on the same line, respecting the letters indicated in the "Parameter description" section.

The subprogram is delivered with the numbering O8018, so the subprogram will be called with G65P8018 followed by the parameters. If the subprogram has to be renumbered, the letter P must be followed by the new program number.

EXAMPLE 1
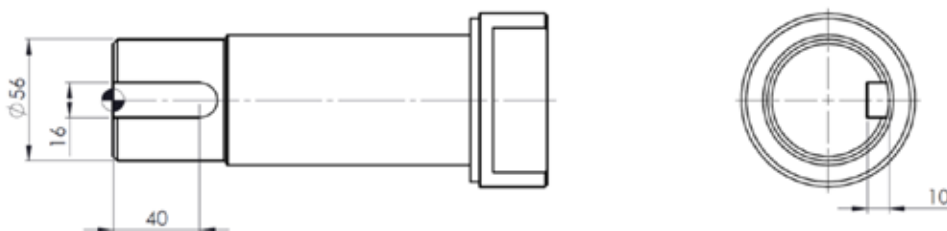
Roughing and finishing machining with the same tool. The work offset in Z in this case is positioned on diameter 56 so the starting plane is Z0, whereas if the work offset in Z were in the centre I would set Z28 W10.

```
%
O0009
G17G90
T3M6

G43H3D3Z300M7
S6000F2000M3
G65P8018X0Y0Z0A16B40W10I12S0.3U0.8Q50V3K1M0F3000E500T0
G0Z300M5
G53Y0
M30
%
```

If roughing and finishing is to be carried out with two separate tools, the program will be as follows:

```
%
O0009
G17G90
T3M6 (ROUGHING)
G43H3D3Z300M8
S5000M3
G65P8018X0Y0Z0A16B40W10S0.3U0.8Q50V3K1M0F3000T1
G0Z300M5

T4M6 (FINISHING)
G43H4D4Z300M8
S5000M3
G65P8018X0Y0Z0A16B40W10Q50V3K1M0F300T2
G0Z300M5
M30
%
```

In the programs just shown, only one roughing cut was performed at depth Z because in the first case the I value was set equal to depth W, while in the second case it was omitted, but in both cases only one roughing cut was performed. To perform more than one cut, it was sufficient to enter the parameter I equal to the desired depth of cut (Ap).

! WARNINGS

1. The macro automatically reads the tool radius stored in the corrector table, so before calling the cycle it is necessary to activate the radius corrector with the D address followed by the corresponding corrector number. In addition, it is necessary to set internal parameter #145, which can be found in the first blocks of the macro file that is sent, to define what type of corrector table is present on the machine. Opening the O8018 file, you will find the following blocks immediately after the program number:

%
O8018(TROCHOIDAL SLOT)

(***INTERNAL PARAMETERS***)
#145=3(TOOL LIST DEFINITION A=1 B=2 C=3)
#146=0.05(DETACHMENT FROM BOTTOM IN FINISHING)
 (***END INTERNAL PARAMETERS***)

Parameter #145 must be set according to the following indications:

#145=3 C-TYPE TOOL MEMORY (most present version set by default)
Case where in the corrector table (OFFSET/SETTING) you have a column for the length corrector (H) and a column also for the radius corrector (D) with the respective wear.

NO. CORRECTOR	CORRECTOR LENGTH (H)		RADIUS CORRECTOR (D)	
	GEOMETRY	WEAR	GEOMETRY	WEAR
1				
2				

#145=2 B-TYPE TOOL MEMORY
Case where in the corrector table (OFFSET/SETTING) you have only one column for correctors, so a corrector can correspond to both length and radius and in a program there can never be H1D1 because they would read the same value. In addition to the corrector there is also the wear column.

NO. CORRECTOR	CORRECTOR LENGTH	
	GEOMETRY	WEAR
1		
2		

#145=1 A-TYPE TOOL MEMORY
Case where in the corrector table (OFFSET/SETTING) you have only one column for correctors, so a corrector can correspond to both length and radius and in a program there can never be H1D1 because they would read the same value. There is no wear column.

NO. CORRECTOR	CORRECTOR	
	GEOMETRY	
1		
2		

The macro is delivered with parameter #145=3 which is the most common case on recent Fanuc controlled machines. If your machine has a different setting, an alarm message will be emitted and in any case, to confirm a correct reading of the tool radius, simply start the macro and, keeping the feed potentiometer at 0, consult the macro variable #110, which must have a value equal to the radius of the cutter. To display the values of the macro variables, go to OFFSET/SETTING and select the MACRO menu.

2. There is a further internal parameter in the macro, parameter #146, which allows you to manage the detachment in Z from the bottom of the slot during the finishing phase. In other words, if I set Z0 and the slot depth W12, the roughing cut will be performed at Z-12. In order to avoid touching the bottom of the slot but have only the sides of the cutter working against it, the internal parameter defines how much it should be raised when finishing. By default the macro is given as 0.05, so it would make the finish at -11.95.

3. To be able to use the macro on your machine, make sure that the parametric programming option is enabled. Although most machines have macro B programming enabled, check that your machine does too. To do so, simply go to the MDI window and enter #100=1, press START and if no alarm message is emitted it means that programming with macro B is enabled. Some controls in the 0 series may not have the # key, so to test this, load a program with only line #100=1 and have it executed automatically. Again, no alarm message should appear.

4. The macro is provided already tested, but it is advisable for the first few times to carry out the necessary checks in a no-load condition or away from the workpiece.

5. The macro only works in work plan G17. If you activate the macro in a different work plan by mistake, it will stop with error No. 28.

6. 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.

7. The macro only works with the non-modal call G65 and not with the modal call G66.

8. Warning, the macro uses parameters #100 to #149; check that these parameters can be used.

Made in Italy

www.cncofcourse.com