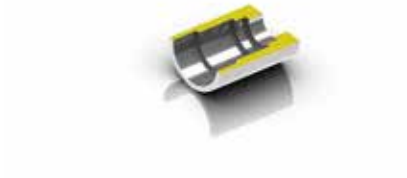
MACRO SHOP

Helical interpolation milling

Fanuc serie 0/16/18/21/31



Rev. 1

FIELD OF APPLICATION

The macro-instruction performs the helical interpolation contouring of internal/external diameters or for pace milling circular flanges. Thanks to the macro-instruction, it will be very simple and immediate to program the machining of holes in a helical interpolation. All you have to do is set the diameter to be machined, the start and end Z dimension and the pitch of the helical interpolation. The advantage is that the parameters can be modulated and varied very quickly.

PARAMETER DESCRIPTION

Below is a combination of the letters and their meaning:

X= CENTRE POSITION IN X

Y= CENTRE POSITION IN Y

D= DIAMETER

Z= END QUOTE IN Z (ABSOLUTE)

W= STARTING QUOTE IN Z (ABSOLUTE)

I= HELICAL INTERPOLATION PITCH

S= LATERAL SAFETY DISTANCE S=0 AND U=1 STARTS FROM THE CENTRE HOLE

Q= SAFETY DISTANCE IN Z (INCREMENTAL TO W)

K= LEAD ANGLE

U= 1= INTERNAL MACHINING 2= EXTERNAL 3= FACE MILL. WITH CENTRE CUTTER

B= INTERP. DIRECTION 2= CLOCKWISE 3= COUNTERCLOCKWISE

T= CLEARANCE SELECTION

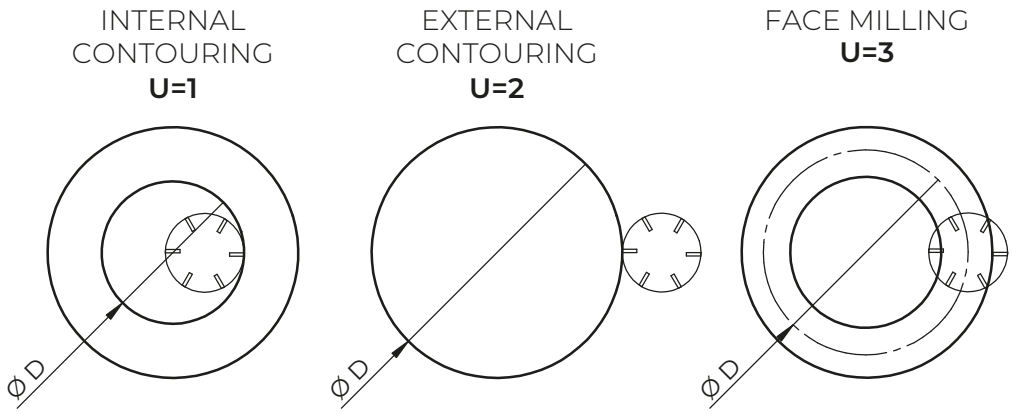
T=0 or NULL, PERFORMS ENTRY AND EXIT CLEARANCE

T=1, CLEARANCE AT Q ONLY ON ENTRY

T=2, NEVER PERFORMS THE CLEARANCE

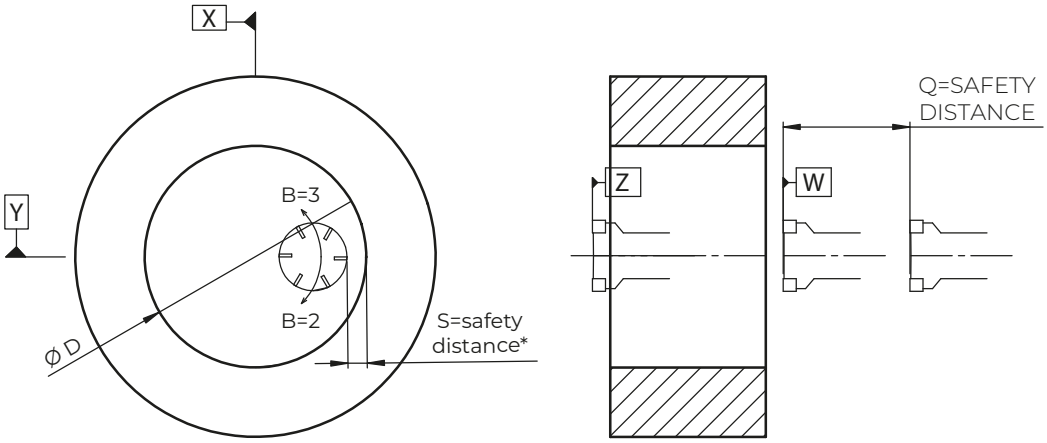
T=3, CLEARANCE ONLY ON EXIT

FOCUS PARAMETERS



In the case of U=3, the diameter D corresponds to the diameter covered by the cutter centre.

The choice of machining type is made with the parameter according to the scheme above. In the case of internal and external contouring, diameter D corresponds to the diameter that will be created, automatically taking into account the cutter diameter. In the case of U=3, diameter D corresponds to the diameter created exactly by the cutter centre.



* By setting S=0 the positioning before and after helical interpolation will be in the centre of the hole (only for internal contours U=1).

The X and Y parameters are used to set the coordinates of the centre of the diameter that will be contoured. Whereas the Z position of the helical interpolation will be done with the parameters W and Z, where W is the absolute coordinate of the start of the milling process and Z is the coordinate of the end of the helical interpolation. W must therefore always be greater than Z. Parameter Q is used to set the safety distance referred to value W at which the cycle will exit at the start and end of machining. Parameter Q is incremental, so if I set W10 and Q50, the clearance will be at Z60.

The parameters related to the approach are the lead angle K and the lateral safety distance S. K defines the angle in relation to the X+ axis at which the entry and exit trajectory of the interpolation will be executed.

Setting for example K90 defines the approach in the Y+ direction. Parameter K can be set either positive or negative. Parameter S is used to set the lateral safety distance, the distance from the finished diameter to which the cutter will be brought with the cutter edge before engaging the circular path. In the case of internal contours, setting S=0 means that the approach and the exit will be from the centre of the hole.

The macro automatically calculates the number of revolutions required to reach the final Z dimension with a specific pitch represented by the I parameter.

Finally, to choose the rotation direction, it is sufficient to set parameter B equal to 2 to perform a clockwise direction or equal to 3 to perform a helical trajectory in an anti-clockwise direction.

Using the T parameter, it is possible to define whether the macro will perform the clearance at the Q quote only on entry, only on exit, never or always according to the following criterion:

0 or null = PERFORMS ENTRY AND EXIT CLEARANCE

1 = CLEARANCE AT Q ONLY ON ENTRY

2 = NEVER PERFORMS THE CLEARANCE

3 = CLEARANCE ONLY ON EXIT

The reason why the operator can define the exit is to offer the possibility of executing several consecutive machining operations without any exit between them (see programming example 4).

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 provided with the numbering O8008, so the subprogram will be called with G65P8008 followed by the parameters. If it is necessary to renumber the subprogram, the letter P must be followed by the new program number.

Before calling the macro-instruction with its parameters, the tool must be called with its length and diameter correctors, and the speed and feed rate F must be set.

EXAMPLE 1

Internal contouring of a d.100 hole starting from Z5 to Z-20 by performing a helical interpolation with a 2 mm pitch per revolution. The start is from the centre of the hole and the approach is along the X+ axis by setting the angle of approach K0. The helical interpolation will be performed counterclockwise by setting B equal to 3.

T1M6

S1500F2500M3

G0G43H1D1Z150

G65P8008X0Y0D100B3W5Z-20I2S0K0Q10U1

G0Z300M5

M30

EXAMPLE 2

Example of external contouring in helical interpolation with a 45° inclined lead in relation to the X axis. Finished diameter of 200 mm starting from Z2 up to Z-30 and a pitch of 3 mm per revolution. As there is a side allowance of 5mm, a lateral safety of 10 is set with the S parameter.

T1M6

S1500F2500M3

G0G43H1D1Z150

G65P8008X0Y0D200B2W2Z-30I3S10K45Q50U2

G0Z300M5

M30

EXAMPLE 3

Face milling of a flat flange with an external diameter of 80mm. Using a 50 mm cutter, the diameter of the helical interpolation is set to D40 so that the cutter overhangs the edge by 5 mm. The finished Z dimension is at Z0 with 10mm of allowance.

T1M6

S1500F2500M3

G0G43H1D1Z150

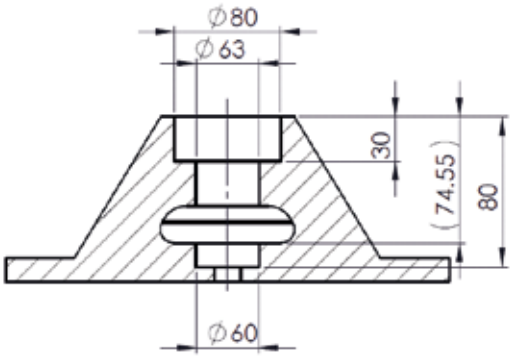
G65P8008X0Y0D40B2W10Z0I2S0K0Q50U3

G0Z300M5

M30

EXAMPLE 4

Helical interpolation milling of three holes machined in succession (see drawing below).



%

O0001

T1M6

S1500F250M3

G0G43H1D1Z150

G65P8008X-100Y200D100W2Z-30I2S0K0Q50U1B3T1

G65P8008X-100Y200D80W-29Z-55I2S0K0U1B3T2

G65P8008X-100Y200D60W-72Z-80I2S0K0Q100U1B3T3

G0Z300

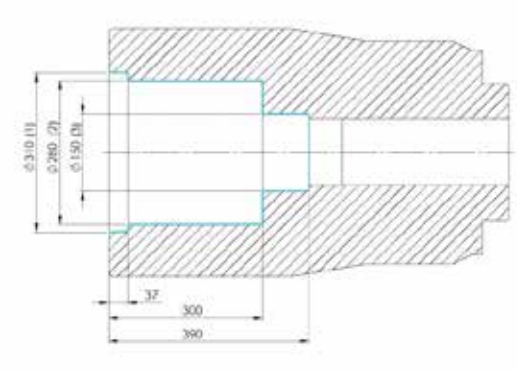
M30

EXAMPLE 5

Milling three consecutive holes.

Programming example

Milling three consecutive holes in helical interpolation.



T2M6(A.A. MILL D125)
G17G90G55
S3000F2500M3
G0X0Y0
G43H2D102Z300

(ROUGHING D.310)
G65P8008X0Y0D310Z-37W5I2S0U1Q10K0B3T1

(ROUGHING D.280)
G65P8008X0Y0D280Z-300W-34I2S0K0U1B3T2

(ROUGHING D.150)
G65P8008X0Y0D150Z-390W-295Q400I1.5S0K0U1B3T3

G0Z300M5M9

T2M6(A.A. MILL D125)

G17G90G55

S3000F2500M3

G0X0Y0

G43H2D102Z300

(ROUGHING D.310)

G65P8008X0Y0D310Z-37W5I2S0U1Q10K0B3T1

(ROUGHING D.280)

G65P8008X0Y0D280Z-300W-34I2S0K0U1B3T2

(ROUGHING D.150)

G65P8008X0Y0D150Z-390W-295Q400I1.5S0K0U1B3T3

G0Z300M5M9

M30

WARNINGS

1. The macro automatically reads the tool radius stored in the corrector list, so before calling the cycle it is necessary to activate the radius corrector with the address D followed by the corresponding corrector number. In addition, it is necessary to set the internal

parameter #145, which is found in the first blocks of the macro file that is sent, to define what type of corrector list is present on the machine. Opening the O8008 file that is sent you will find, immediately after the programme number, the following blocks:

%
O8008(HELICAL INTERPOLATION)

(***INTERNAL PARAMETERS***)
#145=3(TOOL LIST DEFINITION A=1 B=2 C=3)
(***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. The Q quote is the end clearance, calculated incrementally in relation to the W quote, so if, as in example no.5, the w quote was W-295, to get clearance off the workpiece, Q must be at least greater than 295mm, for example Q300.

3. 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.

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. If you enable the optional block M1, the macro will stop at each cut in order to allow the eventual chip cleaning or machining control. If you want to proceed continuously, disable the optional block M1.

www.cncofcourse.com