MACRO SHOP

Cylinder face milling

FANUC 0/16/18/21/31

Rev. 3

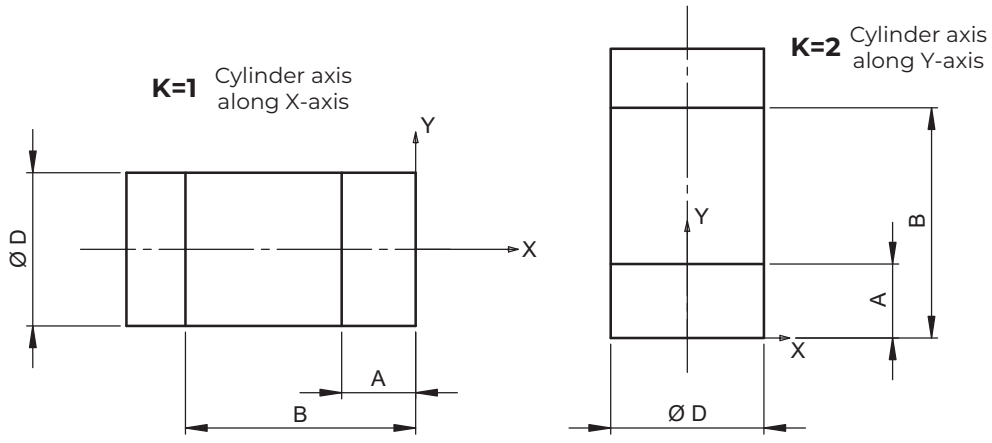
FOCUS PARAMETERS

FIELD OF APPLICATION

The cycle performs the milling of planes created on cylindrical surfaces, automatically calculating an optimised path that follows the cylindrical surface, reducing production times and with fast and safe programming. It is suitable for machining supply planes on valves or hydraulic cylinders, operating keys on rods and planes for lubricant fittings. The macro automatically calculates the depth and width cuts on the basis of the cutter. In fact, in addition to following the cylindrical surface, it automatically calculates the cuts to achieve the required surface.

DESCRIZIONE DEI PARAMETRI

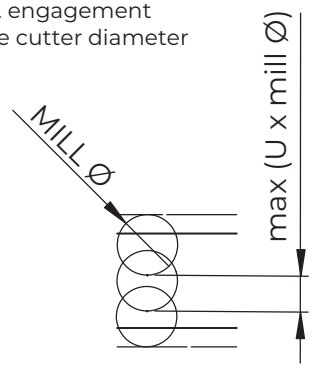
A=FIRST SIDE EDGE COORDINATE
B=SECOND SIDE EDGE COORDINATE
D=CYLINDER DIAMETER
V=LATERAL SAFETY DISTANCE
S=LATERAL EXIT CUTTER CENTRELINE
Z=END DIMENSION IN Z
W=DIMENSION IN Z EXTERNAL DIAMETER
I=INCREMENT IN Z
U=NUMBER OF CUTS ON THE PLANE / % CUTTER DIAM.
Q=SAFETY DISTANCE IN Z
K=TOOL TIP ORIENT. STARTING CUT 1=INCREMENTS IN X 2=INCREM. IN Y
T=STRATEGY SELECTION



The macro allows you to choose the axis direction of the cylindrical part. If it is oriented in the X direction, parameter K is set to 1. If it is oriented in the Y direction, parameter K is set to 2. The sides within which the surface is to be created are indicated by parameters A and B. Parameters A and B are the finished dimensions of the sides within which the face milling is carried out. The macro automatically finds the cutter diameter by reading the cutter radius from the tool offset list using the D corrector, and calculates the cuts to be created in the plane according to the U parameter setting.

if U<1

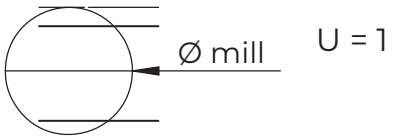
U = max. engagement factor of the cutter diameter



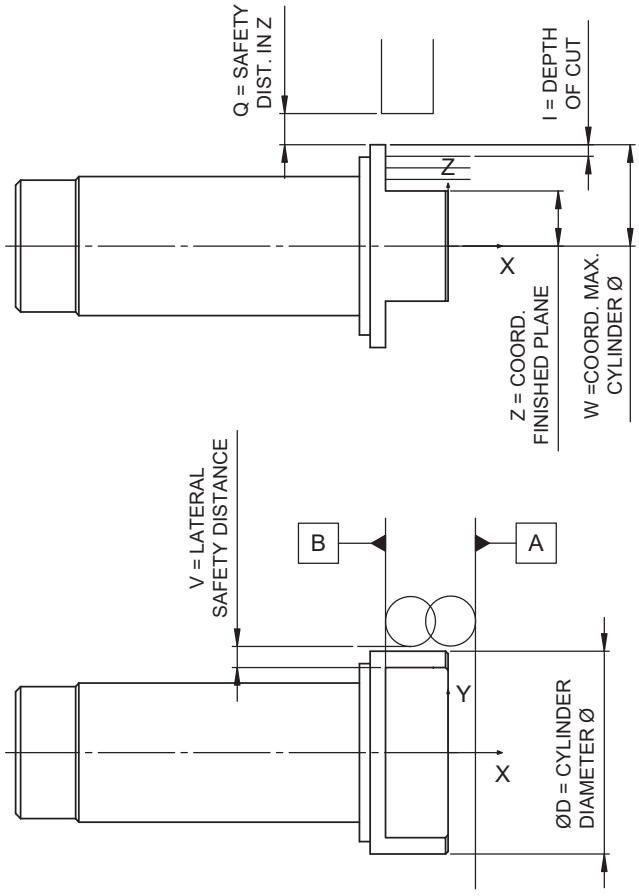
if U≥1

U = number of cuts

By setting U equal to the number of cuts, the macro divides the section to be milled into U cuts.



If I set U smaller than 1, the macro reads it as the load factor of the cutter diameter when calculating the number of cuts. For example, if I set U=0.75 with a cutter diameter of 50, it will create cuts with a maximum load of 37.5. On the other hand, if I set U greater than or equal to 1, the macro reads it as the direct number of cuts to be made to cover the area between A and B. Therefore, if I set U=1 it will perform a single cut in tangency with side A, if I set U=2 it will perform one cut along side A and one cut along side B, if I set U=5 the macro will perform a total of five cuts starting from side A up to side B, dividing the cutter load section in a constant way.



To manage the cuts in Z, the macro starts with the first cut from the highest point of the cylinder represented by the W parameter, until it reaches the height defined by the Z parameter. If I were to create a surface positioned 180 mm from the centre, on a 400 diameter, and the origin in Z was positioned in the centre of the diameter, I would set parameters D, Z and W as follows: **D**400**Z**180**W**200.

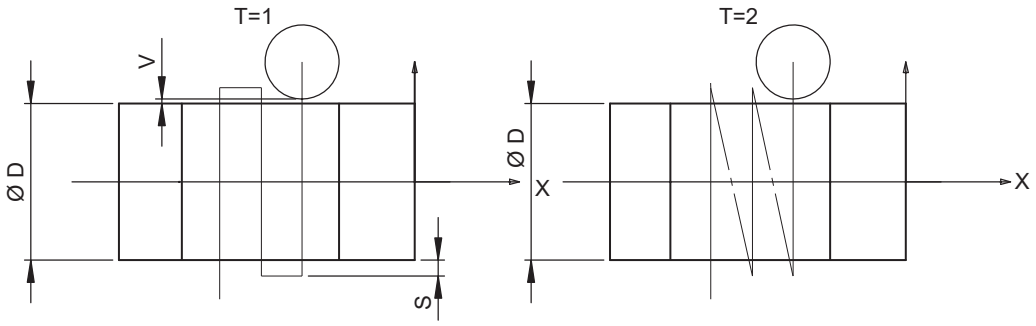
The same machining, but with the origin in Z on the external surface of the cylinder, will be carried out by setting the parameters D, Z, W as follows: **D**400**Z**-20**W**0.

The definition of the depth of cut will be followed with the I parameter, then the macro will automatically calculate the number of cuts in Z.

Finally, there are two parameters for defining safety distances: the Q parameter for safety in Z, to which it goes in the first and last movement. The Q parameter is incremental to the external diameter. So it is sufficient to leave Q equal to 50mm and it will always move 50mm from the external diameter, either with the offset in the centre or with the offset on the external diameter.

For the safety distance on the plane to which the milling cutter is brought to increase in depth and to start machining, the parameter V will be used. Setting V=5, for example, will bring the cutter with the edge 5mm away from the diameter tangency point.

CUTS STRATEGY



Parameter T is used to select the type of path to be followed in the lateral cuts: with T1 it will carry out continuous cuts with a return, while with T2 it will carry out cuts only in one direction, raising each time to the safety level and then carrying out the following cuts always in the same direction.

The S parameter defines how far the cutter exits from the edge with its centre, so if you set S0 the cutter will only exit half way.

CYCLE DESCRIPTION

The macro will carry out the face milling contained between the coordinates set by parameters A and B in several depth increments. The macro will reach the point where it performs the increment along the Z axis and then follow the face milling equal to the dimensions defined by parameters A and B. The cuts on the plane will be carried out with a lateral cutter load equal to the V parameter. The macro will make the first cut at the tangent of the side A until it reaches the side B. The path taken to carry out the face milling will be round trip or one way depending on the T parameter. The second dimension of the face milled surface will be automatically calculated on the basis of the section generated on the cylindrical surface.

PROGRAMMING

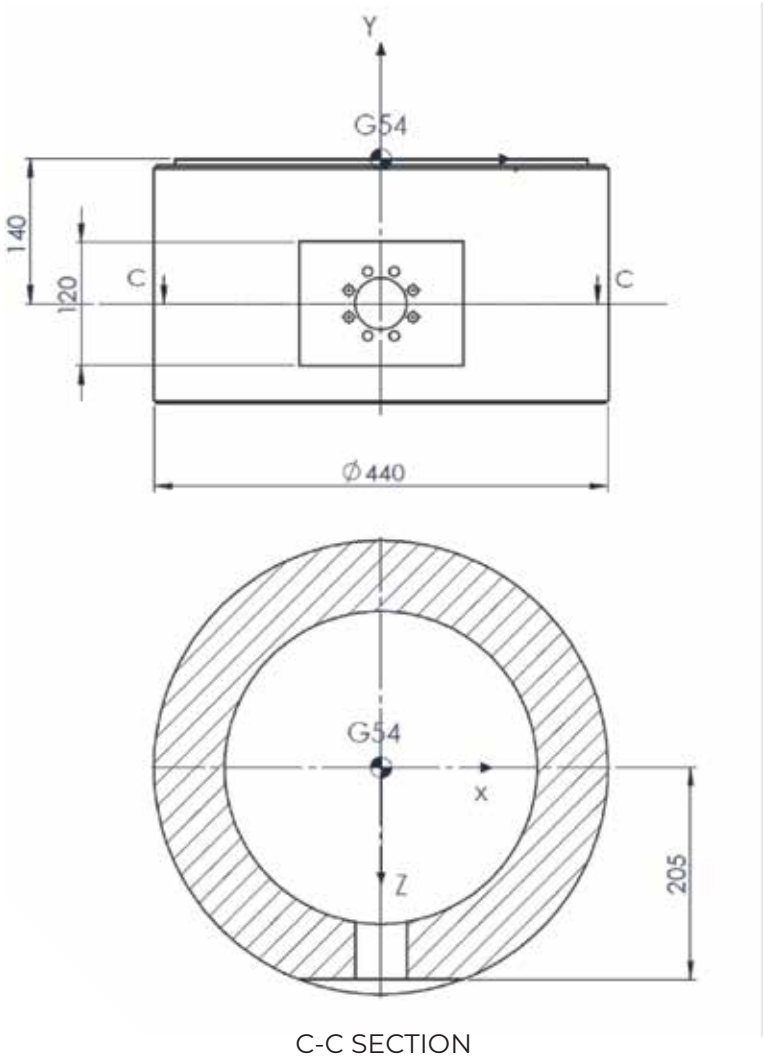
The cycle is to be used as a subprogram to be called with the function G65 and indicating on the same line the parameters, respecting the letters indicated in the "Parameter description" section.
The subprogram is provided with the numbering O8007, so the subprogram will be called with G65P8007 followed by the parameters.
If the subprogramme needs to be renumbered, the letter P must be followed by the new programme number.

Call example:

G65P8003A-20B50D400Z-20W0I2S5U1Q5V5K2T2

EXAMPLE N°1

Milling the supply table on a 440 mm valve body on which a 120 mm wide supply table must be created with the centre positioned 140 mm from the workpiece zero offset. A milling cutter with a diameter of 40 mm will be used with a maximum load of 75% (U0.75). As it is made on a boring machine and the cylinder axis is oriented in the Y direction, I will set K2.

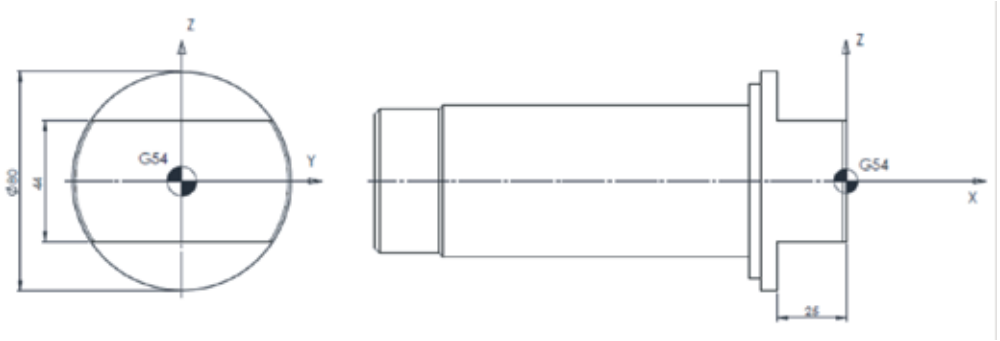


C-C SECTION

```
%
O0001
G90G40G17
T1M6
S1500F250M3

G0G43H1D1Z300
G65P8003A-80B-200D440Z205W220I2S5U0.75Q50V5K2T2
G0Z300
M30
%
```

ESEMPIO N°2
Milling of a spanner, first side at 22 mm from the centre made with a 32 mm cutter in a single cut on the plane and with 2 mm cuts in Z.




```
%
O0001

G90G40G17
T1M6
S1500F250M3
G0G43H1D1Z300
G65P8007A-25B20D80Z22W40I2S5U1Q50V5K1T2
G0Z300
M30
%
```

If the same plane were to be made with a d.20 milling cutter, then two cuts would be required in the plane, dimension A would be just outside the workpiece so that the first cut would not be made on it, while the letter B would indicate the end stop (B-25) and force the cuts to two with the U parameter equal to 2.

The program will become as follows:

```
%
O0001
G90G40G17
T1M6
S1500F250M3
G0G43H1D1Z300
G65P8007A8B-25D80Z22W40I2S5U2Q50V5K1T2
G0Z300
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 corector with the address D followed by the corresponding corrector number. In addition, it is necessary to set an internal parameter #145, which can be 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 O800 file that is sent you will find, immediately after the program number, the following blocks:

%
O800

(***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. To be able to use the macro on your machine, make sure that the parametric programming option is enabled. Although most machines have the programming of B macros enabled, check that your machine is also enabled. 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, the # key may be missing, so to test this, simply load a program with only line #100=1 and have it executed automatically; no alarm message should appear in this case either. Furthermore, the macro uses the parameters from #100 to #149, it is responsibility of the user of the macro to ensure that these parameters are usable, if necessary contact the builder of the machine tool.

3. Whenever using the macro for the first time, test it with no load or away from the workpiece.

4. The macro only works in working plane G17. If you activate the macro in a different working plane by mistake, the macro will stop with error No. 28.

5. 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.

6. The macro only works with the non-modal G65 call-up and not with the modal G66 call-up function.

www.cncofcourse.com