MACRO SHOP

Conical hole milling

Fanuc

Rev. 0

FOCUS PARAMETERS

FIELD OF APPLICATION

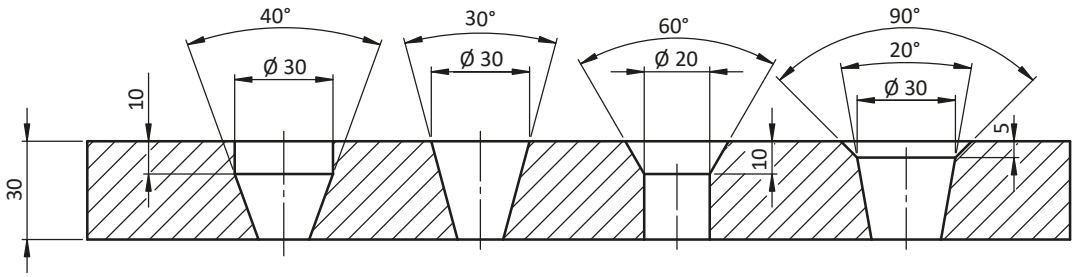
The conical hole milling macro uses two different strategies: constant Z cuts and helical cuts. In the case of constant Z cuts, it performs an increment in Z for each cut, starting from the inside of the pre-drill and performs circular interpolation with a circular approach. In the case where the helical strategy is chosen, the interpolation is continuous without ever leaving the face of the hole. In this second case, the helical conical interpolation is performed by dividing the entire helical path into angular sectors; the Fanuc option of helical conical interpolation is not necessary. The hole geometry can be of various configurations, for example the initial chamfer and the tapered hole, a single taper, a cylindrical initial section and a second tapered section or vice versa. It is also possible to define with a special parameter whether the hole is blind or through. The macro will calculate the start and end points correctly, automatically compensating for the cutter's toric radius. The macro can be called up modally at several points or only for a single position. The macro is compatible with all FANUC versions, please read the section on warnings.

DESCRIZIONE DEI PARAMETRI

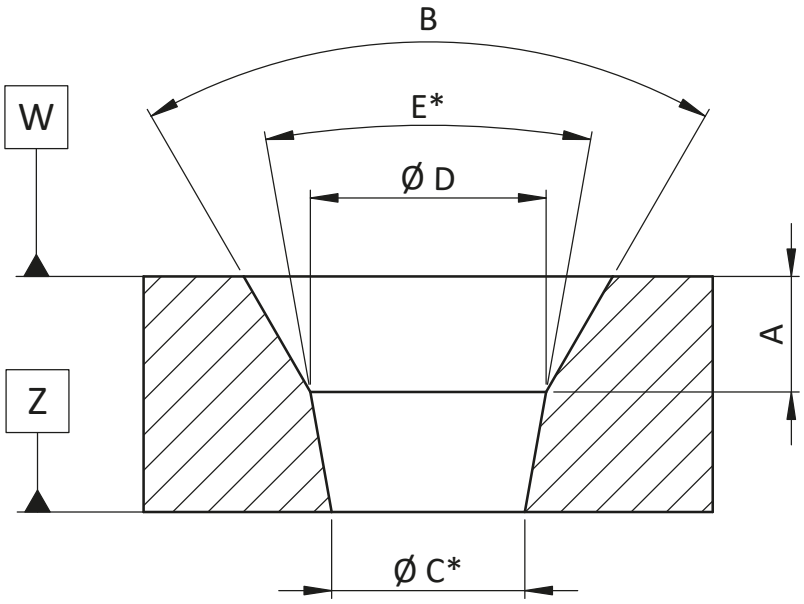
The list of parameters is given below:

X=CENTRE POSITION IN X
Y=CENTRE POSITION IN Y
D= VERTEX DIAMETER TAPER
C=END DRILL DIAMETER
A=FIRST VERTEX HEIGHT
B=FIRST VERTEX ANGLE
E=SECOND VERTEX ANGLE
W=SURFACE Z COORDINATE
Z= END DRILL COORDINATE
I= DEPTH PITCH
R =RHOTIC MILL RADIUS
S=MILL EXTERNAL DIAMETER
M=WORKPIECE HOLE DIAMETER
Q=SAFETY DISTANCE IN Z
T=INTERPOLATION DIRECTION
2=CLOCKWISE 3=COUNTERCLOCKWISE
V= HOLE BOTTOM TYPE
0 or NULL= THRU-HOLE
1=BLIND-HOLE
U= STRATEGY
0=Z CUTS
1=HELICAL

The macro allows the realisation of different geometric types of hole, some examples of which are given below:

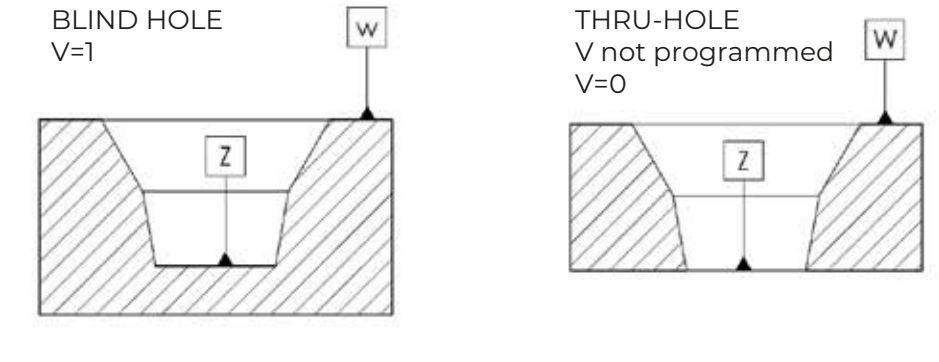


The individual parameters are analysed below:

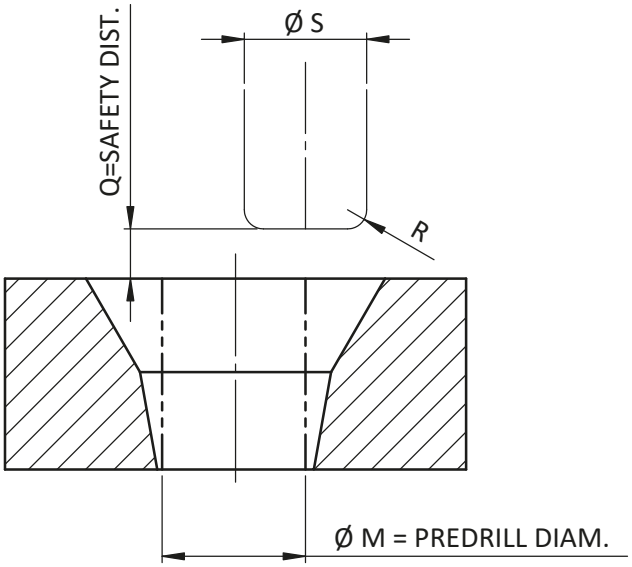


The B and A parameters are used to define the geometry of the first conical section, which may correspond to the chamfer at the opening of the hole. Specifically, B corresponds to the angle and A to the height of the initial conical section. Parameter A is defined as the distance between the surface and the starting edge of the second hole section. Parameter D is used to define the diameter at the point of intersection of the first conical section and the second conical section. The second taper has two parameters: the taper angle (E) and the diameter at the bottom of the hole (C). The user may program only one of these parameters, E or C. If both are programmed, the macro takes the diameter at the bottom of the hole as the reference value and the angle is only checked. If the E angle set when the macro is called up differs from the calculated angle, the macro will display an error message. It is advisable to program both parameters E and C when the angle is quoted in the drawing between the parenthesis in order to have a second check on the accuracy of the data entered.

The start-of-hole and end-of-hole coordinates are to be defined with the W and Z parameters. The W parameter corresponds to the surface coordinate at the start of the hole, while Z corresponds to the coordinate at the end of the hole, both expressed in absolute values.



In the case of a blind hole, set the V parameter to one; in the case of a through hole, do not set the V parameter or set it to zero.



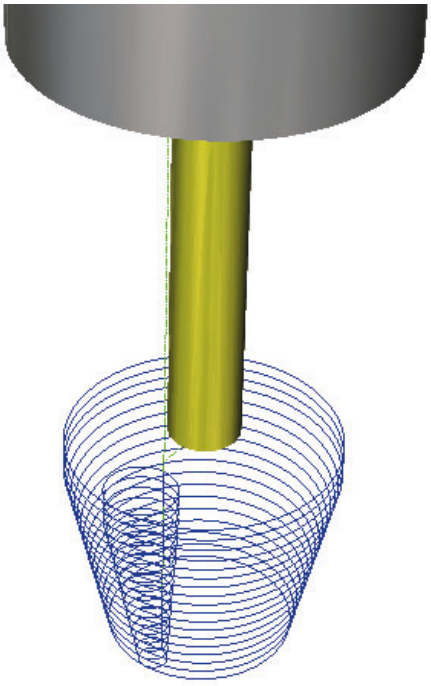
The macro allows to carry out a machining operation from the workpiece or a machining operation performed with a predrill. Use the M parameter to define the diameter of the predrill. If there is no predrill, do not program the M parameter or set it equal to 0.

The Q parameter is used to define the safety distance from the Z plane at the start of the hole, from which the macro will start the milling cuts. The macro automatically calculates radius compensation and toric radius compensation, the parameters of which are defined by the parameters R for the toric radius and S for the diameter of the milling cutter respectively.

The macro will perform a constant Z milling if the U parameter is set to 0, or a helical interpolation if the U parameter is set to 1.

CYCLE DESCRIPTION

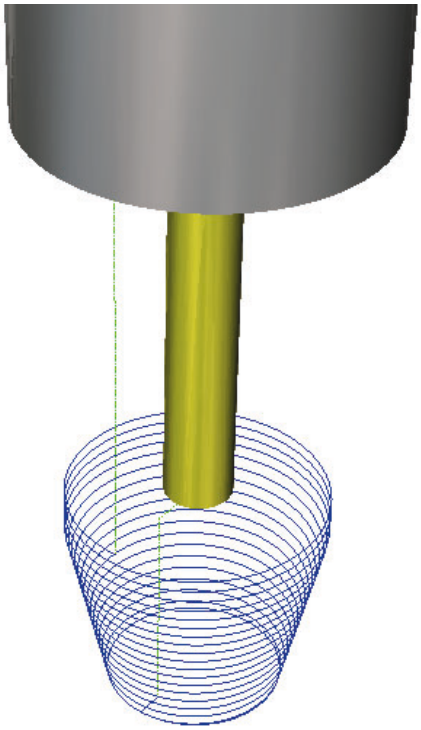
CONSTANT Z CUTS, U=0



In case of constant Z cuts, obtained by setting the U parameter equal to 0, the macro firstly moves to the X and Y co-ordinate of the hole, and then moves down to the initial Z dimension obtained as W+Q. In the case where there is no predrill, the increment position in Z is in the centre, while in the case of a predrill, the X and Y co-ordinate is calculated in such a way as to remain inside the predrill and at a distance from the predrill equal to the value indicated in the macro's internal parameter, set by default to 5mm. From this point the macro performs a circular approach to reach the diameter, it performs circular interpolation at the end of which it retracts in a circular way, returning inside the predrill. It then proceeds to make several cuts in Z, with the depth of the cut defined by the I parameter.
Parameter I corresponds to the maximum depth of cut in Z, since the macro will recalculate the depth of cut on the basis of the whole number of cuts required to drill the entire hole.

NOTE: It is better to not program U0 without the predrill (M=0), in which case make sure the milling tool is drilling in the centre.

HELICAL STRATEGY, U=1



Setting parameter U equal to 1 sets a helical milling. From the first starting point, the milling tool will always remain in contact with the profile, executing a helical interpolation until the end of the hole. The transition from one conical section to the next will be continuous, always remaining in contact with the profile. The helical interpolation is generated by a point-to-point path segmentation. The angular pitch with which the helical path is divided is defined with an internal parameter of macro #148.

```
(***INTERNAL PARAMETER***)
#148=1.5 (ANGULAR PITCH FOR HELICAL STRATEGY)
```

When you open the macro program, you will find internal parameter #148 in the first few program blocks. Parameter #148 is used to set the angular step by which the helical trajectory is divided. For example, if we set #148=1, each degree of the circular sector will correspond to a point on the trajectory. A linear path is taken from one point to the next, so for a more accurate path reduce parameter #148.

PROGRAMMING

The macro is delivered in the form of a parametric program numbered O8032. To execute the machining operation, the macro call must be inserted in the program with the respective parameters, using function G65 or G66. The G65 function is to be used in the case of a single hole. Function G66 is used for multiple holes. To close function G66, program G67 at the end of the hole coordinates (see example below).

SINGLE HOLE

```
G65P8032X0Y0D50A2.5B0E25W0Z-30I2R4S12M40Q1T3U1
```

NON-MODAL MACRO CALL

HOLE COORDINATE

MULTIPLE HOLES

MODAL MACRO CALL

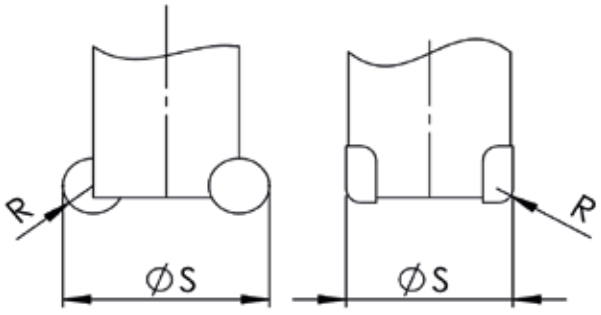
```
G66P8032D50A2.5B0E25W0Z-30I2R4S12M40Q1T3U1
X150Y50
X200
X250Y100
G67
```

HOLE COORDINATE

MACRO MODAL CALL DEACTIVATION

If it is necessary to renumber the macro program, also change the call G65 and G66.

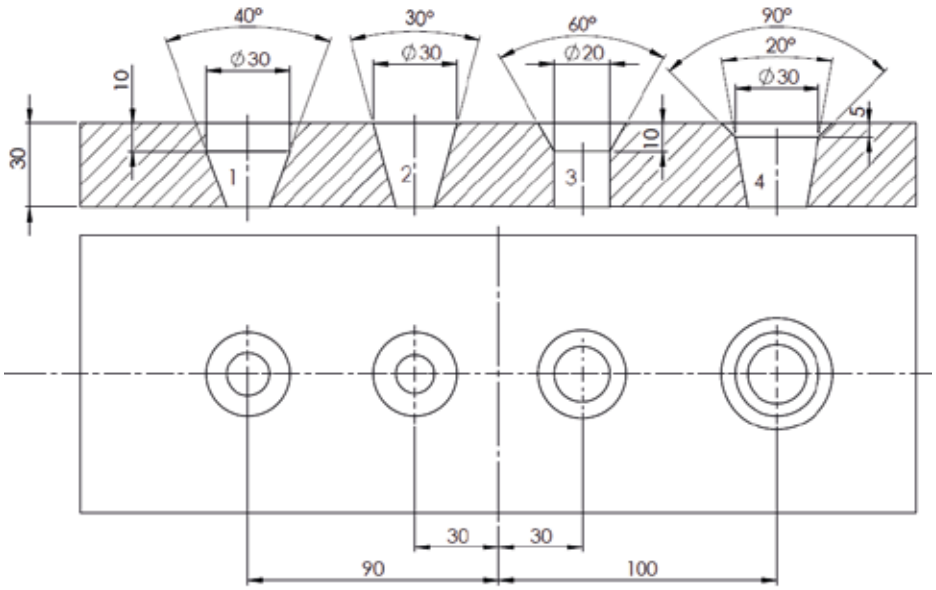
EXAMPLE: renumbered macro program P7000
G65P7000X0Y0D50A2.5B0E25W0Z-30I2R4S12M40Q1T3U1



Regarding the tool data, the macro does not use the radius compensation functions G41-G42 internally, so it is not necessary to define the tool radius in the OFFSET list. The geometric data of the tool are defined by the parameters R and S. R corresponds to the toric radius, while S corresponds to the tool diameter.

FOCUS PARAMETERS

PROGRAMMING EXAMPLES



%
O0600

T1M6
G43D1H1S3000F1000M3

(FORO1)
G65P8032X-90Y0D30A10B0E40W0Z-30I1R2S12M15Q3T3U1
(FORO2)
G65P8032X-30Y0D30E30W0Z-30I1R2S12M15Q3T3U1
(FORO3)
G65P8032X30Y0D20E0B60A10W0Z-30I1R2S12M15Q3T3U1
(FORO4)
G65P8032X100Y0D30A5B90E20W0Z-30I1R2S12M22Q3T3U1
G0Z300
M5
M30

In the example, if the U parameter is set to 1, machining is carried out with a helical trajectory. By setting U equal to 0 or omitting it, constant Z milling is performed. If the holes were blind holes, it would be enough to set parameter V equal to 1; in this case V has been omitted as they are thru-holes.

WARNINGS

Within the macro, in the first few lines of the program, are entered some parameters that will remain constant for all the programs in which the macro is called.

(***INTERNAL PARAMETER***)
#148=10 (ANGULAR PITCH FOR HELICAL STARTEGY)
IF[#4006EQ20]GOTO7878
(MM)
#149=5.0 (SIDE SAFE DISTANCE)
GOTO7877
N7878
(INC)
#149=0.19 (SIDE SAFE DISTANCE)
N7877
(***END INTERNAL PARAMETER***)

Parameter #148 defines the angular pitch at which the helical path is divided into points. Setting a low number reduces the chordal error, but may affect the speed of the execution. The speed of the execution will depend on the axes parameterisation. It is recommended to work with the G64 function active.

The second parameter within the macro is parameter #149, which defines the lateral safety distance, the distance from the predrill from which the milling tool is laterally detached in order to perform the Z increments.

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 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.

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.

www.cncofcourse.com