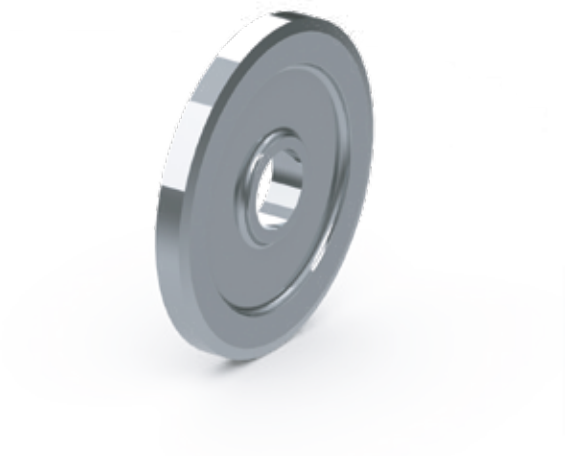
MACRO SHOP

Face grooving

Fanuc 0/16/18/21/31



Rev. 0

FIELD OF APPLICATION

The macro carries out the face grooving using a high performance turning strategy. This macro is particularly suitable for removing material between two straight or inclined sides with a high chip volume. The macro uses the principle of high feed rates in turning. A typical application of the macro is the machining of face grooves on flanges, pulleys and gear wheels. The macro allows a choice of various geometric shapes for the groove, which are perfectly suited to the most common geometries of face grooves. It also allows the programmer to set two working strategies: one strategy involves roughing with transverse cuts with rounded entry and rounded exit. The second strategy, which can only be applied when the two sides are at 0 degrees, performs roughing with transverse cuts, generating a temporary profile with two inclined sides, and then returns to the inclined sides with longitudinal cuts. This second option is recommended for particularly resistant materials and especially when the depths are very high, as it greatly reduces the occurrence of vibrations. Finally, for materials that are difficult to chip, the macro allows transverse cuts to be made with chip breakage. The geometric parameters include every possible configuration of the slot, even for cases where the flanges have different dimensions. The macro can be executed on all lathes with Fanuc control from series zero to series 31. For details, please refer to the manual with particular attention to the " Warnings " paragraph.

PARAMETER DESCRIPTION

D=MAX. DIAMETER SIDE 2
U=MIN. DIAMETER SIDE 1
Z=Z-COORDINATE OF GROOVE START ON THE X SIDE MIN.
W=Z-COORDINATE OF GROOVE STAR ON THE X SIDE MAX. (omit if same as Z)
E=Z-COORDINATE OF GROOVE BOTTOM
X=Z-DIMENSION OF ANGULAR START (optional parameter)
A=ANGLE X SIDE MAX.
B=ANGLE X SIDE MIN.
M=RADIUS SIZE OF THE GROOVE BOTTOM
V=SIDE ALLOWANCE
R=INSERT RADIUS
Q=RADIAL SAFETY DISTANCE
I =DEPTH OF CUT
T = 0 or null S/R T=1 ONLY START IN X-
S= 1=ROUGH. 2=ROUGH.+FINISHING 3=FINISHING
4=RADIUS RECOVERY 5=ROUGH.+RADIUS RECOVERY
6=ROUGH.+RADIUS RECOVERY+FINISHING
C=RADIUS/CHAMFER SIDE 1
H=RADIUS/CHAMFER SIDE 2

FOCUS PARAMETERS

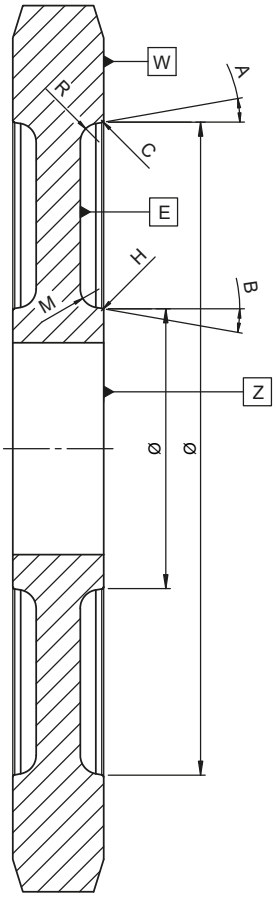
Y= BOTTOM RADIUS OF THE RADIUS RECOVERY (to be inserted only if the finished profile has 0° sides and roughing is to be carried out with modified profile)
J=VALUE FOR CHIP BREAKAGE NULL OR ZERO WITHOUT BREAKAGE
F=STARTING FEEDRATE
K=FINISHING FEEDRATE

NOTE 1: THE ROUGHING FEEDRATE IS TAKEN FROM THE PROGRAM

NOTE 2: PARAMETERS I,J,K MUST BE ENTERED IN ALPHABETICAL ORDER EVEN IF THEY ARE NOT NECESSARILY CONSECUTIVE.

PAR. D, U

A series of parameters allow the profile of the groove to be freely configured. In particular, it will be possible to define the maximum groove diameter (D) and the minimum groove diameter (U). The two diameter parameters refer to the diameter at the edge formed by the front face with the groove side. Parameters A and B are used to define the sides inclination angle. Parameter A refers to the angle in relation to the Z axis of the corner with the largest diameter, while angle B refers to the wall with the smallest diameter.



PAR. A,B

PAR. E

Parameter E is used to define the Z-coordinate of the groove bottom, while two different coordinates can be configured for the front faces. The co-ordinates of the front faces are defined using the parameters Z and W. Z is the co-ordinate of the face on the bottom edge and W is the parameter for defining the co-ordinate of the face on the top edge.

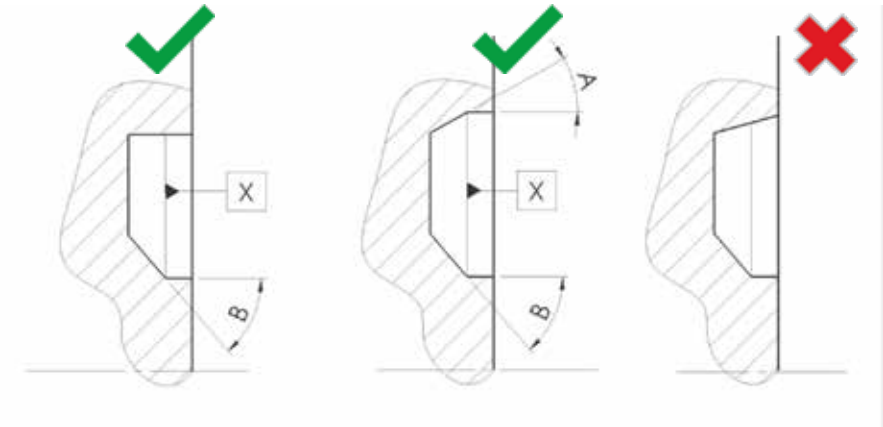
PAR.Z,W

If the pocket has two edges with the same co-ordinate along the Z axis, it will be sufficient to define the Z parameter for the co-ordinate of the face, omitting the W parameter.

If you define two faces with different co-ordinates, the macro will carry out the roughing operation starting from the co-ordinate that creates the smaller depth of the pocket. Example: if I set W-15 Z0 E-25, the macro will rough a pocket starting from a depth along the Z axis of -15 to reach the bottom of the groove at coordinate -25. It is very important to always define both the Z and W parameters in the case of pockets with edges in different planes for the correct profile calculation and to correctly execute the finishing cut and any external radius.

PAR. C,H

The parameters C and H are used to define the size of the external corner. It is possible to make only rounds (NO CHAMFER) or, if necessary, by setting C and H equal to zero, the finishing cut will continue along the side without making any round on the external edge.



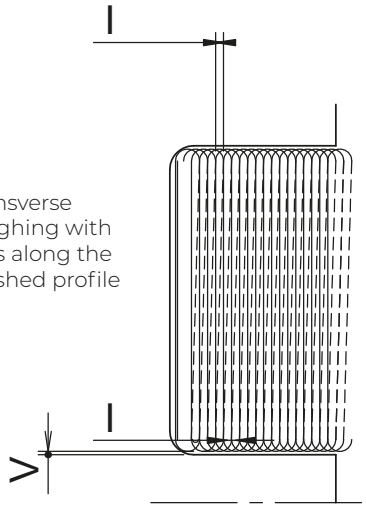
PAR. X

The X parameter can be used to define the Z-coordinate at the start of the inclined stroke if the inclined stroke does not start from the external edge. It is possible to define only one inclined side or both inclined sides, but the start of the two inclined sides must be at the same coordinate defined with the X parameter.

PAR. Z,W,E

The parameters Z, W, E and X, which define the various coordinates along the Z axis, are all expressed in absolute value, so they all refer to the active workpiece offset.

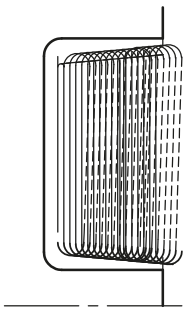
PAR. S



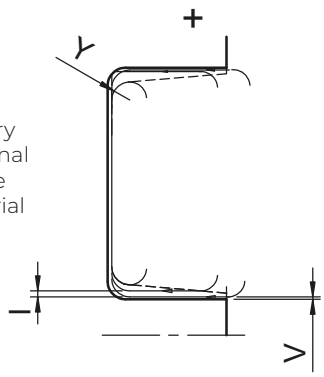
The S parameter is used to define which machining operation is to be carried out. In the most general case, so for any pocket shape, the following coding applies:

S=1 only carries out roughing.
S=2 performs roughing and at the end, with the same call of the macro and therefore with the same tool, also finishing.
S=3 only carries out finishing.

Transverse roughing with cuts along an inclined profile



Radius recovery with longitudinal cuts to remove residual material



For only pockets with finished sides at 0°, it is possible to carry out roughing with a modified profile. By setting the parameter Y equal to the bottom radius for recovering inclined faces, the macro understands that the inclination values A and B are only for generating a modified profile, to facilitate roughing on a more suitable inclined profile to reduce vibration and cutting forces. It will therefore be necessary to add a roughing operation to recover the inclined faces with longitudinal cuts.

The S parameter is used to define which operation is to be performed when the macro is called:

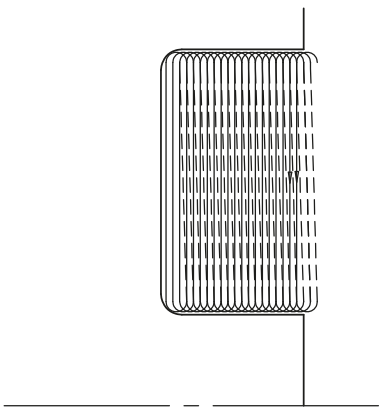
S=1 ROUGHING
S=3 FINISHING
S=4 RADIUS RECOVERY
S=5 ROUGH.+RADIUS RECOVERY
S=6 ROUGH.+RADIUS RECOVERY+FINISHING

If the operations are carried out with different calls because different tools are to be used, the important thing is that all the geometric parameters are always the same by always rewriting the Y parameter, which serves to make the macro understand in all operations what the finished profile will look like.

TECHNOLOGICAL PARAMETERS

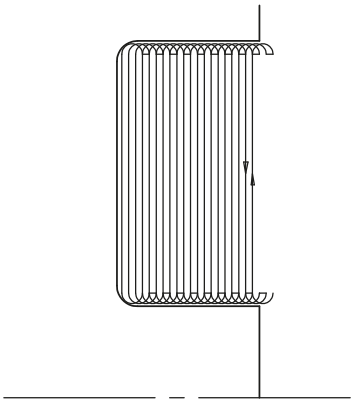
Only cuts in X- direction

T=1



Round trip cuts

T=0 or null



PAR.T

PAR. F,K

The T parameter is used to select the direction of the transverse cuts. Setting the T parameter equal to 1 allows cuts to be made only in the negative X direction, so cuts directed towards the axis of the workpiece. If the T parameter is set to zero or null, cuts are made in both directions X- and X+, optimising passive times.

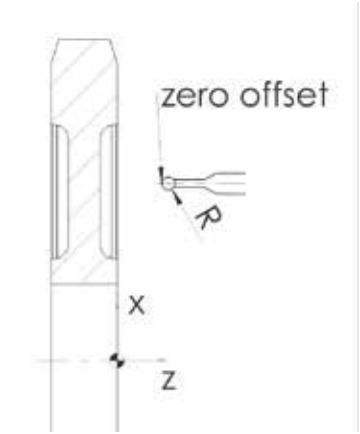
To set the cut feed rate, write the value of this feed rate in the blocks preceding the macro call always with the letter F. While the feed rate F indicated on the line of the macro call corresponds to the feed rate in the radius entry, the finishing feed rate will be indicated with the K parameter.

The depth of cut is defined by the I parameter. Parameter I corresponds to the pitch between cuts, both in transverse roughing and in longitudinal cuts, for the recovery of inclined sides.

Example:
T0101
G97S1500F0.5
G0X150Z20
G65P8029D_U_Z_E_.....F0.35K0.15

In this case, the roughing cuts in the cross section will be carried out at F0.5, while the radius entry at the beginning of each cut and the radius exit at each cut will be carried out with feed rate F0.35. In the finishing cut, a feed rate of F0.15 is used throughout the profile.

CYCLE DESCRIPTION



The macro is set up to carry out roughing and finishing using a radial tool. The tool must be measured along the X axis at the centreline, and along the Z axis at the external point. The R parameter is used to set the radius R of the tool, so there is no need to fill in the tool radius in the table, nor the T tool tip orientation, as the radius compensation is carried out automatically within the macro.

PAR. R

Roughing is carried out in several cuts in depth by making increments in Z equal to the I parameter. Each entry and exit of the profile is carried out in a radial approach. If the finished groove has 90° sides, it is advisable to carry out transversal roughing by setting 5-10° on the sides (e.g. A=5 B=5), a large bottom radius M which will then be recovered with the longitudinal cuts and Y equal to the bottom radius of the groove required in the drawing.

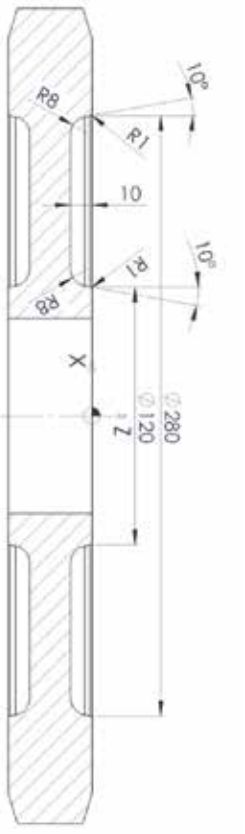
If there are problems with chip breakability, despite the high feed rates that should facilitate chip breakage, the macro offers the possibility of performing transverse cuts with a chip breakage that can be set with the J parameter. For example, if J5 is set, during the X cut, the tool performs a retraction every 5mm in X to break the chip.

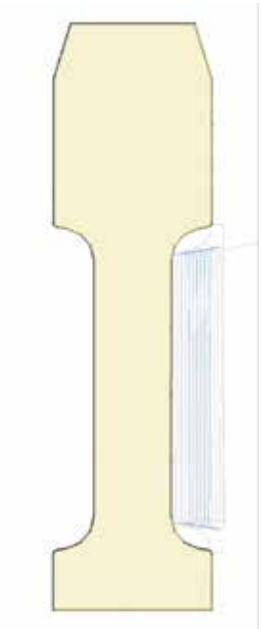
Finishing is done in a single cut along the profile starting from the most positive side.

PROGRAMMING

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 supplied with the numbering O8029, so the subprogram will be called with G65P8029 followed by the parameters. If it is necessary to renumber the subprogram, the letter P must be followed by the new program number.
If you want to carry out roughing and finishing with two different tools (both radiused), the geometrical parameters of the groove must be the same, changing only the R parameters relating to the tool radius and any technological parameters.

EXAMPLE 1: Face groove machining on a gear wheel.



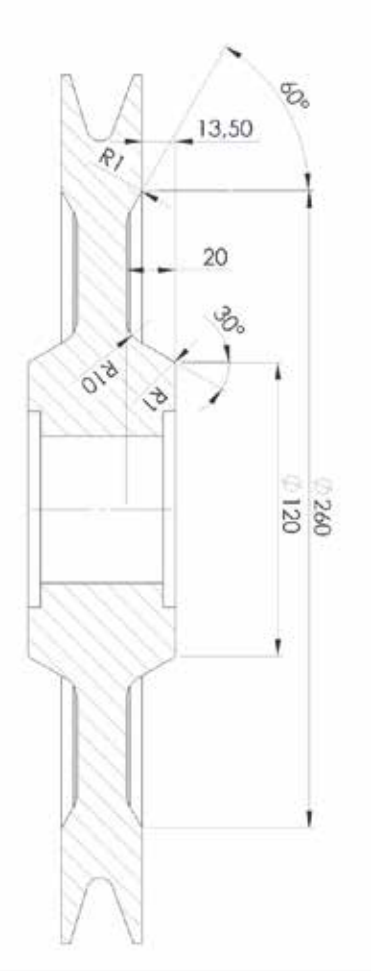


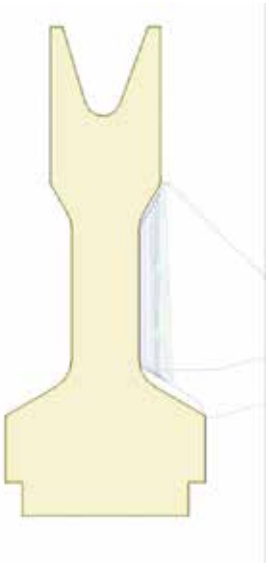
Both roughing and finishing are carried out with the same 6mm tool radius.

%
O0095(LAUNCHING FACE GROOVE)

T0101(T. RADIUS6)
G50S800
G96S240F3M3
G0X200Z20M8
G65P8029D280U120Z0E-10A10B10V.4R6I1M8Q3S2F0.5K0.2C1H1T1
G0Z300
G0X300M5
M30
%

EXAMPLE 2: Roughing and finishing a pulley relief.





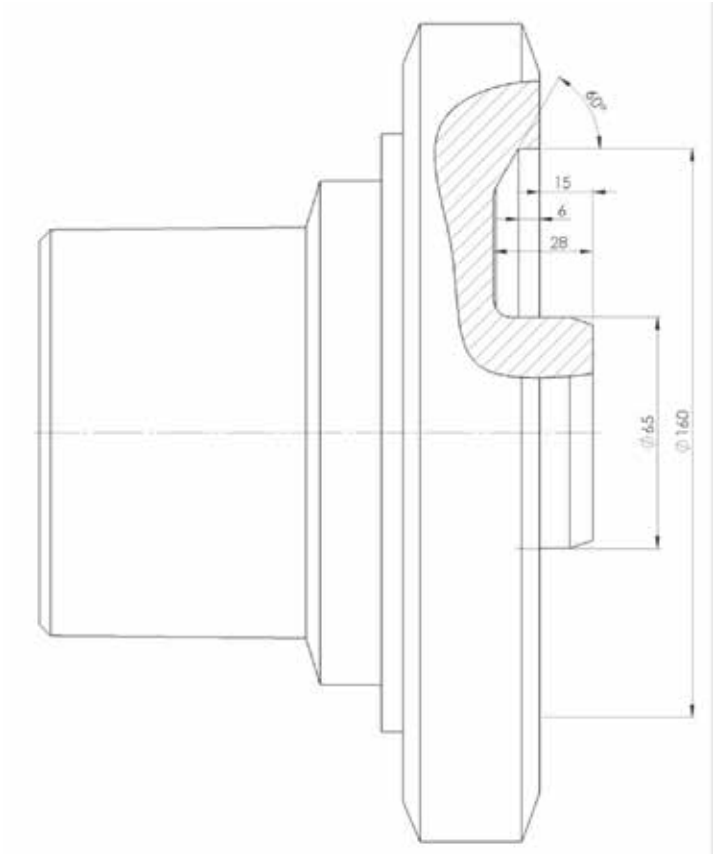
Roughing performed with ra adius 6 tool and finishing performed with a radius 4 tool.

%
O0094(LAUNCHING FACE GROOVE)

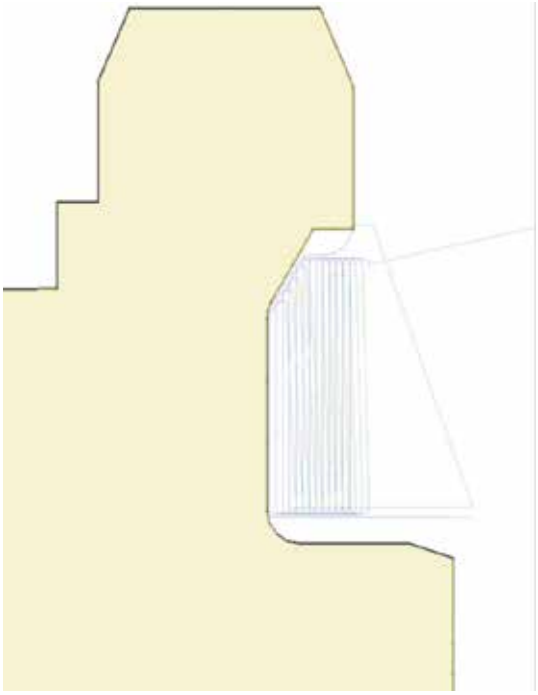
T0101(T. RADIUS6 ROUGHING)
G50S800
G96S240F3M3
G0X200Z20
G65P8029Z0W-13.5D260U120E-20A60B30V.4R6I1M10Q3S1F0.5C1H1T1
G0X300Z300

T0909(T. RADIUS4 FINISHING)
G50S800
G96S220F0.15M3
G0X200Z20
G65P8029Z0W-13.5D260U120E-20A60B30R4M10Q3S3K0.4C1H1
G0X300Z300
M5
M30
%

EXAMPLE 3: Roughing and finishing a face pocket on a spigot.



Roughing will start from Z-15 then perform external roughing during turning of d.65 up to Z-15.
In finishing, the corner radius is set to 0.5mm on the top edge only, while on the bottom edge it is set to zero so that the tool will have a straight exit.



%

O0096(EXAMPLE 3)

T0101(T. RADIUS4)
G50S1500
G96S240F0.5M3
G65P8029Z0W-15E-28X-21D160U65A60B0V.4R4I1M5Q3 S2F0.25K0.15C0.5H0T1
M5
M30
%

FOCUS PARAMETERS

WARNINGS

1. Within the macro in the first few lines of the program, two parameters are inserted that will remain constant for the programs in which the macro is called. One of the two parameters is the entry radius when the macro approaches the cut (#149). The second concerns the value of how much you want to retract at each chip break when the macro programs the chip break with the J parameter (#122).

```
(REV.2)
(***INTERNAL PARAMETER***)
IF[#4006EQ20]GOTO7878

(MM)
#149=1.0 (APPROACH RADIUS)
#122=0.3 (RETRACT DIST FOR CHIP BREAK)

GOTO7877
N7878

(INC)
#149=0.04 (APPROACH RADIUS)
#122=0.012 (RETRACT DIST FOR CHIP BREAK)

(***END INTERNAL PARAMETER***)
```

The two parameters are different if the machine is set in millimetres or inches. In your case, only change the values in the unit of measurement you are using.

2. The macro uses parameters #100 to #180, so it is necessary to check that these parameters can be used, if necessary contacting the machine builder. If you need to use parameters with a different numbering, request the modification of the macro.

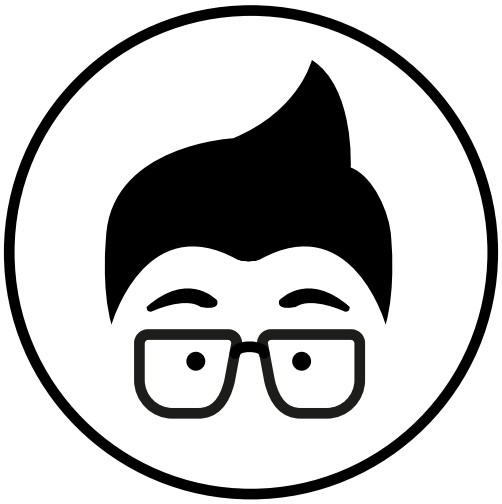
3. The macro is provided already tested, but it is advisable for the first few times to carry out the necessary checks in a no-load condition or away from the workpiece.

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.

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

6. In order to improve tool life and reduce stress and vibration, we recommend that if the sides are at square shoulders compared to the bottom, to always use the machining strategy with side recovery.

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.



www.cncofcourse.com