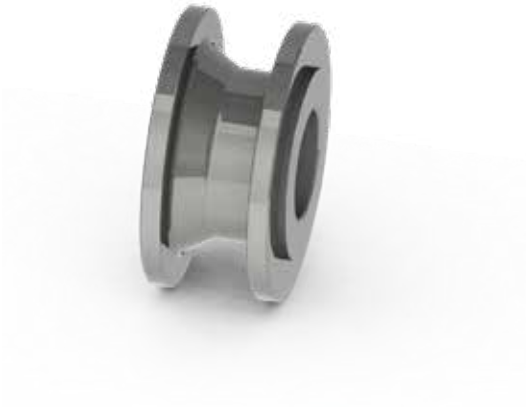
MACRO SHOP

HIGH PERFORMANCE GROOVE

Fanuc 0/16/18/21/31



Rev. 3

FIELD OF APPLICATION

The macro performs the grooving using a high-performance turning strategy. The macro is particularly suitable for removing material between two straight or inclined walls 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 the clearance between the flanges of valve closures. The macro makes it possible to choose various geometric shapes for the groove, which are perfectly suited to valve closures. It also allows the programmer to set two working strategies. One strategy involves roughing with longitudinal cuts with a radial entry and radial exit. The second strategy, which can only be applied when the two walls are at 90°, carries out roughing with longitudinal cuts generating a temporary profile with two inclined walls, and then takes up the inclined walls with transverse cuts. This second option is especially suitable for particularly resistant materials and especially when depths are very high, as it greatly reduces the occurrence of vibrations. Finally, for materials that are difficult to chip, the macro allows longitudinal cuts with chip breakage. The geometric parameters provide for every possible configuration of the groove, even for cases where the flanges have different dimensions. The macro can be run on all Fanuc-controlled lathes from series 0 to series 31.

PARAMETER DESCRIPTION

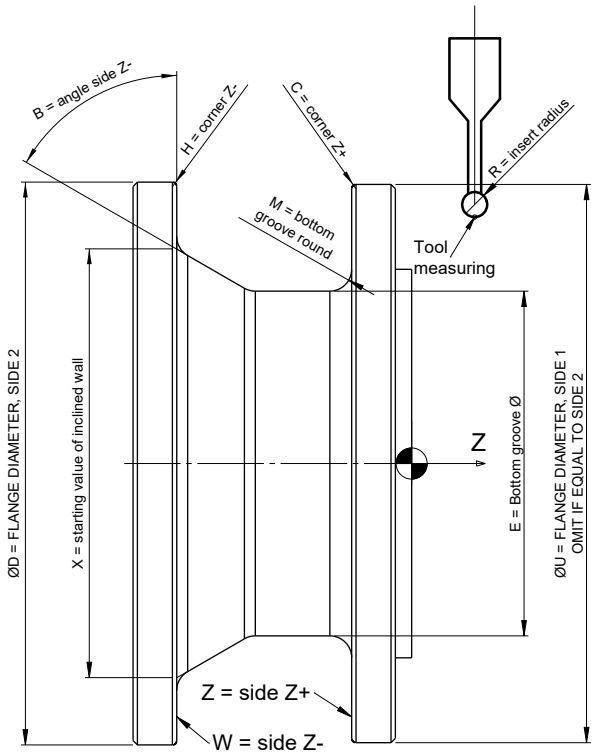
D=EXTERNAL DIAMETER OF SIDE2 (DEFAULT VALUE FOR BOTH FLANGES IF PAR. U IS USED)
U= EXTERNAL DIAMETER SIDE1 (TO BE ENTERED ONLY IF THE TWO STARTING FLANGES ARE DIFFERENT)
E=INTERNAL DIAMETER
Z=POSITION IN Z GROOVE SIDE Z+
A=ANGLE SIDE Z
W=POSITION IN Z-GROOVE SIDE Z-
B=GROOVE ANGLE ON SIDE Z-
M=RADIUS ENTITY (BE CAREFUL THAT (M-V) MUST BE GREATER THAN R)
V=ALLOWANCE ON PROFILE
R=INSERT RADIUS
Q=RADIAL SAFETY DISTANCE
I =DEPTH OF RADIAL CUT
T = 0 or none S/R T=1 ONLY START Z-
S 1=ROUGH 2=ROUGH+FINISHING 3=FINISHING
4=RADIUS RECOVERY 5=ROUGH+RADIUS RECOVERY
6=ROUGH+RADIUS RECOVERY+FINISHING
C CHAMFER RADIUS SIDE 1
H CHAMFER RADIUS SIDE 2
X ANGLE START DIMENSION
Y BOTTOM RADIUS FOR RADIUS RECOVERY

FOCUS PARAMETERS

J VALUE FOR CHIP BREAKAGE ZERO OR ZERO WITHOUT BREAKAGE
F ENTRY FEEDRATE
K FINISHING FEEDRATE

NOTE 1: THE ROUGHING FEEDRATE IS TAKEN FROM THE PROGRAM

NOTE 2: PARAMETERS I, J AND K MUST BE ENTERED IN ALPHABETICAL ORDER, ALTHOUGH NOT NECESSARILY CONSECUTIVELY.



A series of parameters allow you to freely configure the profile of the groove to be cleared. In particular, it is possible to define the external diameter (D) and the groove end diameter (E). The Z and W parameters are used to define the position of the two sides of the groove. If the two sides are not square shouldered, it is possible to set an inclination angle for the two sides. The X parameter can also be used to start the inclined part from a diameter other than the external one. The macro, when it sees that at least one of the two parameters A and B relating to the inclination angle of the sides is different from zero, and when parameter X is programmed, will execute a groove section with the sides straight until it reaches the diameter defined with parameter X, from which the side inclination will start. Parameter M is used to define the corner radius at the bottom of the groove, which also corresponds to the radius that each cut will make once the diameter of the longitudinal turning cut has been reached.

FOCUS PARAMETERS

PAR. D, E

PAR.Z, W

PAR. X

PAR. A,B

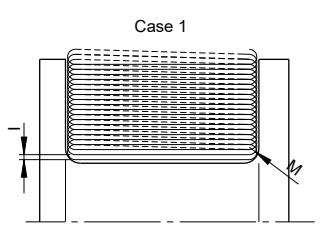
PAR. X

If you have chosen to carry out roughing with recovery of the radii and the sides, the Y parameter is used to define the radius of the end of the groove at the end of the transverse cuts.

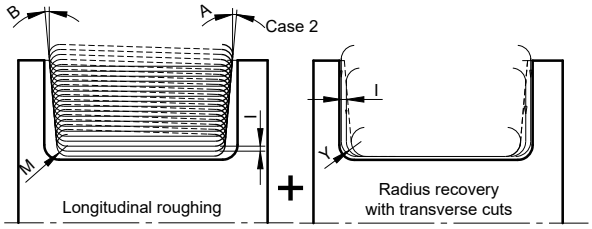
The depth of cut I is given radially and for longitudinal cuts it corresponds to the radial increment between the turning cuts. In the case of side milling with transverse cuts, the parameter I corresponds to the pitch in Z between each cut.

The V parameter is used to set the allowance on the profile to be left for finishing. The allowance is left constant over the entire groove profile, including the sides, the radius and the bottom of the groove.

The S parameter can be used to define the type of machining. In particular, this choice must be differentiated according to the shape of the groove.

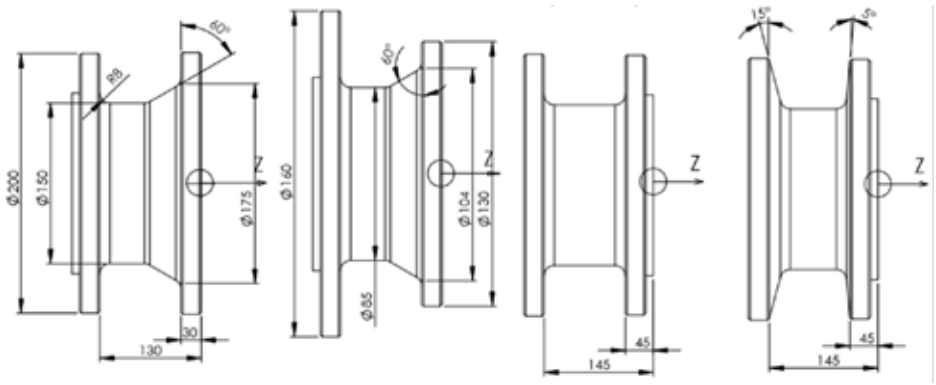


D200E150X175 M8 Z-30W-130A0B0I1.5...
S=1 ROUGHING
S=2 ROUGHING+FINISHING
S=3 FINISHING



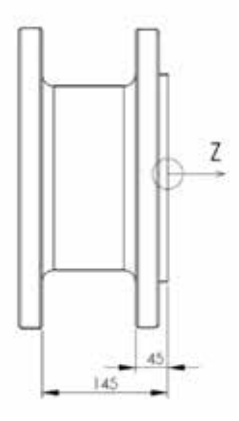
D200E150X175 M8Y10 Z-30W-130A15B15I1.5...
S=1 ROUGHING
S=3 FINISHING
S=4 RADIUS RECOVERY
S=5 ROUGH. + RADIUS RECOVERY
S=6 ROUGH. + RADIUS RECOVERY + FINISHING

CASE1: VALID FOR ANY SHAPE THAT CAN BE REALISED WITH THE MACRO (Y ZERO)



Using the S parameter, it is possible to choose whether to carry out only rough machining (S=1), only finishing (S=3) or both the machining operations (S=2).

CASE2: ONLY VALID FOR GROOVES WITH FINISHED SQUARE-SHOULDER SIDES (Y NON-ZERO)



For grooves with straight sides, so at 90° to the bottom, a further strategy is possible. In fact, it is possible to set the inclination degrees A and B in order to carry out an initial roughing with longitudinal cuts and then rework the radii and inclined sides with transversal cuts. As soon as the Y parameter is set to non-zero in the macro, the macro recognises the intention to execute the strategy by taking the radius and the inclined sides. Therefore by setting the Y non-zero parameter, it will be possible to choose between 5 different operations:
S=1 carries out roughing with longitudinal cuts;
S=3 performs only the finishing that will be carried out on the finished profile, so with square shoulder walls;
S=4 carries out only the radius recovery and the angled sides, leaving any allowance on the profile and leaving the corner radius equal to the Y parameter;
S=5 performs roughing with longitudinal cuts as well as radius and side machining with transverse cuts;
S=6 carries out the roughing in longitudinal cuts, radius and side machining in transverse cuts and the finishing of the entire groove.

The T parameter is used to select the direction of the longitudinal cuts. Setting the parameter equal to 1 allows the cuts to be made only in the negative Z direction, so against the spindle. If parameter T is set to zero or zero, cuts are made in both Z- and Z+ directions, thereby improving passive times.

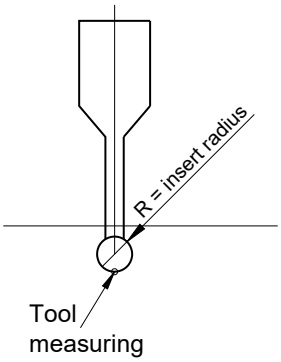
To set the cut feed rate, write the value of this feed in the blocks preceding the macro callout always with the letter F. While the feed rate F indicated on the line of the macro callout corresponds to the feed rate in the radial entry section. The finishing feed will be indicated with the parameter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

PAR. T

FEED RATE

CYCLE DESCRIPTION

The macro is set up to carry out roughing and finishing using a radial tool. Tool measuring must be carried out along the X axis at the maximum point of the tool, while along the Z axis it must be done in the centre of the tool. (see figure below)



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 quadrant, as radius compensation is performed automatically within the macro.

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

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

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

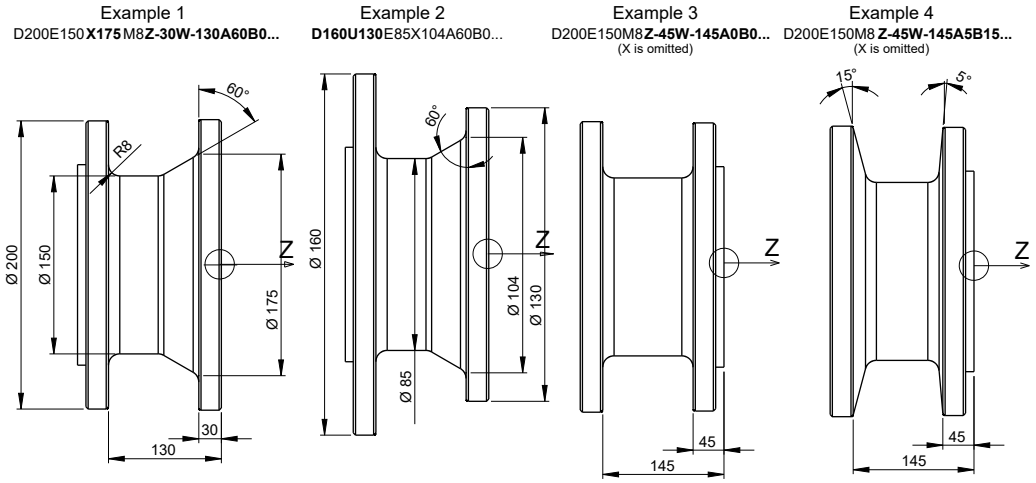
PROGRAMMING

The cycle is to be used as a subprogram to be called up 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 O8026, so the subprogram will be called up with G65P8026 followed by the parameters. If the subprogram needs to be renumbered, the letter P must be followed by the new program number.

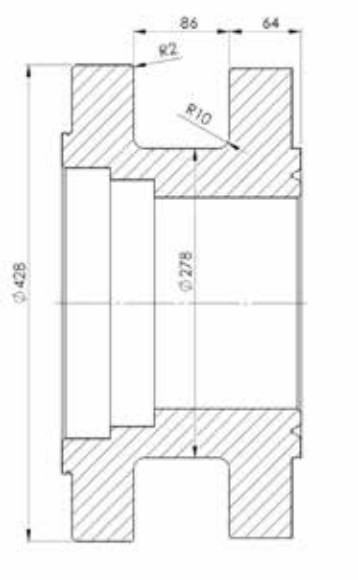
If roughing and finishing is to be carried out with two different tools (both rounded), the geometrical parameters of the groove must be the same.

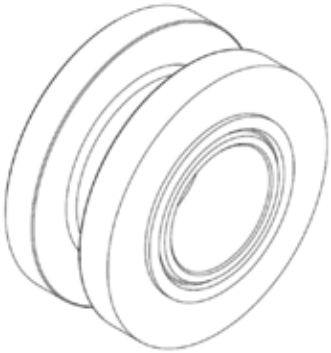
Various examples of groove shapes are given with the respective geometric parameters of the groove; the machining parameters must then be added to these parameters.

Example of setting the geometric parameters



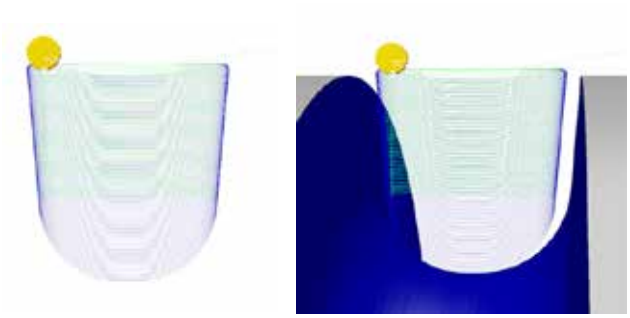
EXAMPLE1: Closure body with square-shouldered sides. Roughing with flanks at a modified inclination, reworking with transverse cuts and finishing



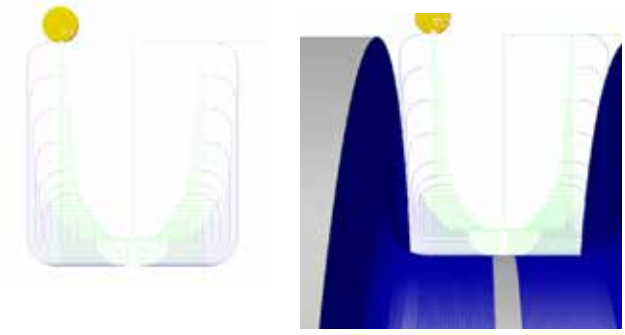


T0101
G96S180F2M4
G65P8026Z-64W-150D428E278A5B5V1R6I1M35Q2S6F0.1K0.1C2H2T1Y15
M5
M30
%
The program will sequentially perform the three operations of longitudinal roughing with modified flanks, radius and flank recovery and finishing.

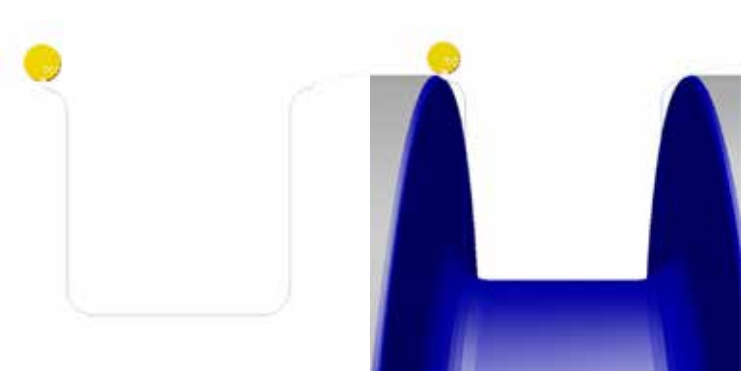
Transverse roughing cuts



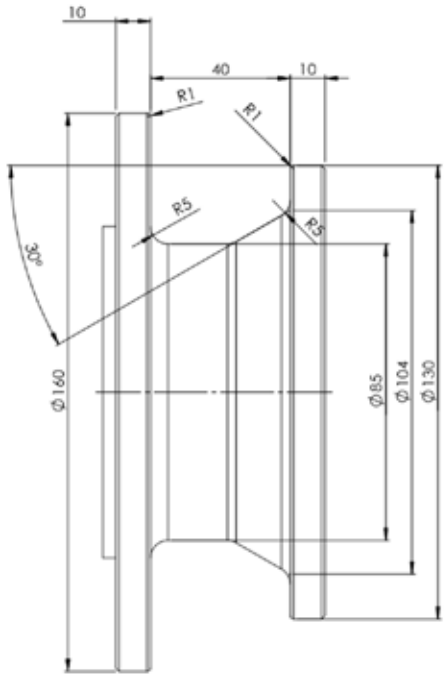
Recovery of the radius and the sides with transverse cuts



FOCUS PARAMETERS



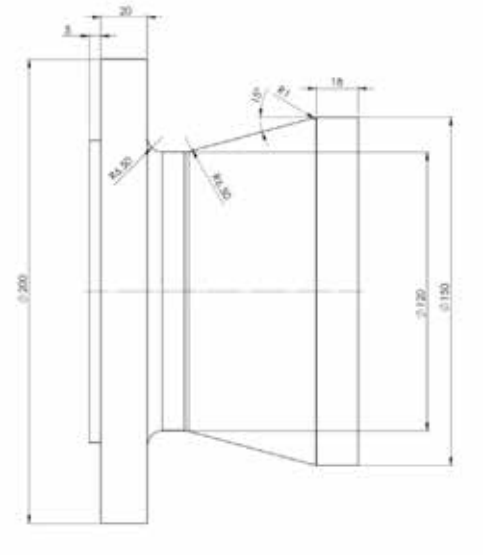
EXAMPLE 2: TURNING THE CLOSURE WITH DIFFERENT FLANGE DIAMETERS AND AN INCLINATION ON ONE SIDE ONLY



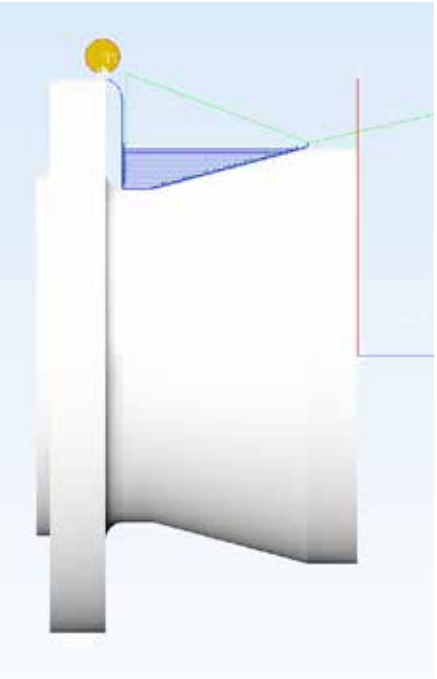
Roughing and finishing the closure with a front flange of a smaller diameter than the rear flange. Roughing the groove starts from the diameter of the lower flange, so it is a good idea to include turning cuts up to the diameter of the lower flange.

T020
G92S1500
G96S500F3M3
G65P8026Z-10W-50D160U130E85A60B0V.2R4I1Q2S2F0.4K.3C1H1T0M5X104
M5
M30
%

EXAMPLE 3: TURNING THE CLOSURE WITH DIFFERENT FLANGE DIAMETERS



T0101
G92S1500
G96S250F3M3
G65P8026Z-18W-91D200U150E120A75B0V.4R6I1M6.5Q2S2F0.5K0.4C1H1T0



FOCUS PARAMETERS

WARNINGS

Within the macro, parameters are inserted in the first few lines of the program that will remain constant for the programs in which the macro is called. One of the two parameters is the radius of entry 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 in inches. In your case, only change the values in the unit of measurement you are using.

2. The macro uses parameters #100 to #199, 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 delivered already tested, but it is recommended for the first few times to do the necessary tests at no-load or away from the workpiece.

4. The cycle automatically sets the absolute coordinates by setting the G90 function. If you need to set the incremental coordinates after the macro-instruction, set the G91 function.

5. The macro only works with the G65 call-up and not with the G66 modal call-up function. Follow the instructions in the "Programming" section.

6. If the flanks are square-shouldered compared to the bottom, it is advisable to always use the flanking machining strategy as described in example no. 1, for better tool life and a reduction in stress and vibration.

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