

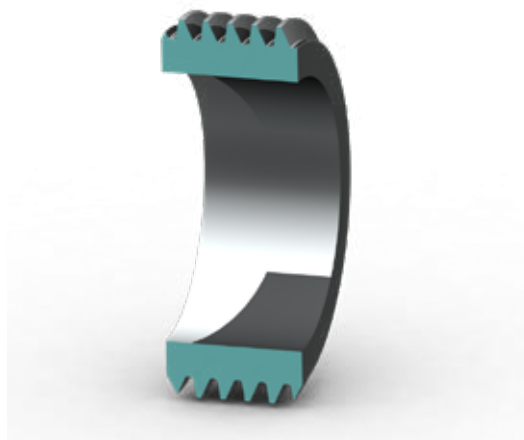


MACRO SHOP

Large diameter thread



Fanuc 0/16/18/21/31



 Rev. 5



FIELD OF APPLICATION

The macro for Fanuc controls carries out the roughing and the finishing of large or standard threads, internal or external, using roughing and finishing strategies that allow threads to be made with very high pitches while maintaining low cutting forces and good surface finishing. In the case of both the screw and the nut, roughing and finishing can be carried out with a standard tool with a rounded shape, such as a groove tool or a tool with an R shape insert. Better still, when angles allow it, with standard turning tools with a V (e.g. VNMG) or D (e.g. DNMG) shape. It is also possible to use grooving tools. The macro allows you to freely set the inclination angles of the two sides of the thread, and allows you to set the corner radius on both external and internal edges. For example, it can be used to machine buttress with standard and non-standard angles, trapezoidal threads, metric threads. By setting the angle of the two flanks equal to zero, it is possible to carry out oil grooving for bronze bushing. In roughing, the macro will perform for each X step a series of cuts to open the thread groove, using the tool radius equal to a maximum value set with a parameter. In finishing the macro allows you to set the value of the maximum crest along the profile, in order to control the roughness on the whole profile, both on the sides and on the floor and also on the radius. In addition, for trapezoidal threads where the radiuses at the floor are almost zero, it is possible to set a tool such as parting tools or special tools. The macro can be executed on all lathes with Fanuc control from series zero to series 31. Please refer to the manual for details, with particular attention to the warnings section.



PARAMETERS DESCRIPTION

D= CREST DIAMETER
I= MINOR DIAMETER
R= TOOL NOSE RADIUS
U= DIAMETRAL CUTTING DEPTH
W= THREAD PITCH
Z= Z COORDINATE STARTING POINT
J= TAPERED
K= Z COORDINATE FINISHING POINT
X= X CLEARANCE COORDINATE
A= FLANK ANGLE SIDE 1
B= FLANK ANGLE SIDE 2
C= LENGTH THREAD GROOVE AT CREST DIAMETER
E= FLAT STROKE TOOL WIDTH ONLY FOR FINISHING WHERE A TOOL WITH A FLAT STROKE IS USED





F= EXTERNAL CORNER RADIUS SIDE 1 AND SIDE 2

S= FLOOR RADIUS SIDE 1

T= FLOOR RADIUS SIDE 2

Q= -1=INTERNAL THREAD 1= EXTERNAL THREAD

H= MACHINING OPERATION 1=ROUGHING 2=FINISHING

M= % TOOL NOSE ENGAGEMENT/PROFILE CREST HEIGHT

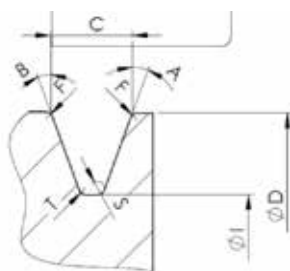
Y= SIDE ALLOWANCE (ONLY FOR ROUGHING)

V= FLOOR ALLOWANCE (ONLY FOR ROUGHING)

NOTE: PARAMETERS I,J AND K MUST BE ENTERED IN THIS ORDER, NOT NECESSARILY ONE AFTER THE OTHER.

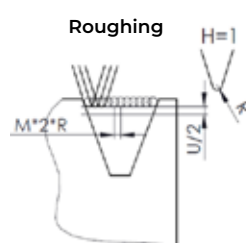
The D parameter is used to set the diameter at the start of the thread, while the I parameter is used to set the diameter at the end of the thread.

The Z parameter is used to set the co-ordinate of the start of the thread, it is advisable to set the pitch outside the workpiece at least twice. The K parameter is used to set the end threading co-ordinate. The X parameter defines the clearance coordinate X at the end of each cut.



Regarding the parameters of the thread profile, it is possible to set the two inclination angles of the thread sides. In particular, parameter A sets the angle that will be created on the flank in the Z+ direction, while parameter B sets the angle of the flank in the Z- direction. To define the

size of the thread, the width will be set to the diameter D using parameter C. Finally, parameter F is used to define the size of the corner radius on the external edge and parameters S and T are used to define the value of the fillet radii on the floor of the thread.



To carry out the roughing operation, the H parameter must be set equal to 1. Only when roughing the U parameter is used to define the depth of cut expressed in diametral mode. If, for example, U0.3 is set, the macro will perform pass by increasing the diameter every 0.3 mm. At each roughing pass, there may be several roughing pass

as the macro calculates the maximum engagement equal to parameter M. If, for example, M0.8 is set and the insert radius is R2, the macro will execute pass with an offset of a maximum of $0.8 \cdot 4 = 3.2\text{mm}$, so if the first cut has a groove of 8mm, the macro will execute 3 cuts at the same X co-ordinate.

PAR. D

PAR. Z, K, X

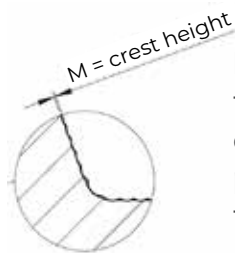
PAR. A,B

PAR. C,S,T,

PAR. H, U

PAR. M

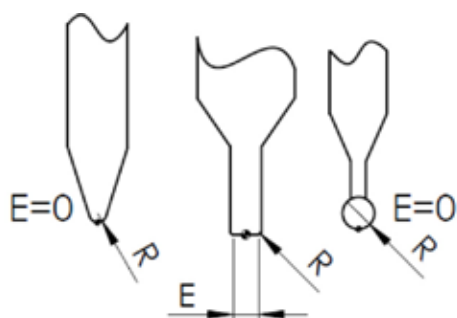




For finishing, the H parameter is set to 2. In finishing, the step will be calculated while keeping the roughness constant; the crest height generated by the cuts on the profile will be set with the M parameter it corresponds to the Rp roughness.

To define whether it is an external or internal thread, the parameter Q will be used. Q of -1 corresponds to internal threading, while Q +1 corresponds to external threading.

For threads where there must be little or no radius at the floor, it is possible to use tools with a flat stroke. In such cases, it is still necessary to indicate the tip radius of the tool but also the width of the flat stroke using the E parameter. When using a tool with a flat stroke, the tool must be set in the centre of the tool, therefore in the middle of the flat stroke (see drawing). For example, if you are using a 3 mm wide cutting tool with a radius of 0.2 at the corners for finishing, give the following parameters: R0.2 E=2.6, since the tool flat stroke width will be the width of the cutter minus twice the nose radius.



The J parameter is used to create a tapered thread. Parameter J is the difference between the final and initial diameters. If J is set to a positive value, both the internal and external thread at the end point will have a greater X value.

The parameters Y and V are used to define the finishing allowances, Y being the allowance on the sides and V being the radial allowance to be left on the bottom.

PAR.Q

TOOL
PARAMETER

PAR.J

PAR.Y,V

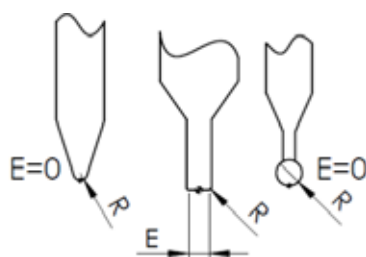




CYCLE DESCRIPTION

The macro is set to perform roughing and finishing operation with round nose tools (e.g. form R or radiused grooves), general turning tools of D or V shape when the threads have angles greater than the cutting angle of the tool. Finally, when there is a flat stroke and a large radius is not allowed, it is also possible to use tools with a flat stroke, for example grooving tools. In the latter case, it is required to indicate the radius on the edge, although small, which will be used to calculate the cuts starting from the crest height in the finishing operation. Use parameter E to indicate the width of the flat stroke of the tool. Please note that the parameter E indicates exactly the width of the flat stroke of the tool and not the total width of the insert.

If roughing and finishing are carried out with the same tool, the tool measuring can be performed at any point. If two different tools are used, it would be necessary to set the tool along the Z-axis at the centreline and at the most overhanging point of the tool along the X-axis (see figure below).



Roughing is carried out with several pass in depth by carrying out diametral increments equal to the U parameter. For each diametral increment the macro will carry out several lateralpass to open up the entire width of the slot. The lateral pass will be calculated automatically to maintain the maximum tool engagement value defined by the M parameter.

In finishing, the cuts will only be made along the sides until they reach the bottom of the thread. The pitch between each cut in finishing will be calculated automatically on the basis of the crest height required with the M parameter. Be careful in finishing if you use a flat stroke tool; also indicate the radius on the edges as this is used to calculate the cuts on the basis of crest height.

ROUGHING AND
FINISHING TOOLS





PROGRAMMING

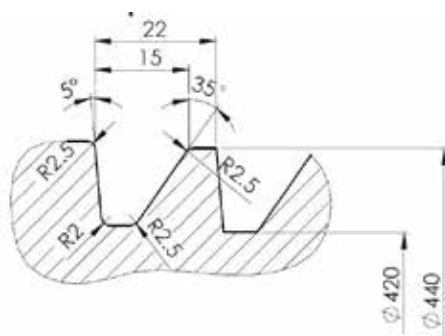
The cycle is to be used as a subprogram to be called up with the G65 function and indicating the parameters on the same line, respecting the letters indicated in the "Parameters description" section.

The subprogram is supplied with the numbering O8027, so the subprogram will be called up with G65P8027 followed by the parameters. If the subprogram needs to be renumbered, the letter P must be followed by the new program number.

The macro must be called up for both the roughing tool and the finishing tool. The geometric parameters of the finishing, as well as the start and end of thread parameters Z and K, must be the same.

EXAMPLE 1

Special EXTERNAL thread execution similar to a buttress.



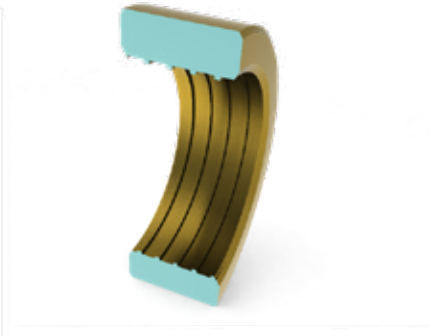
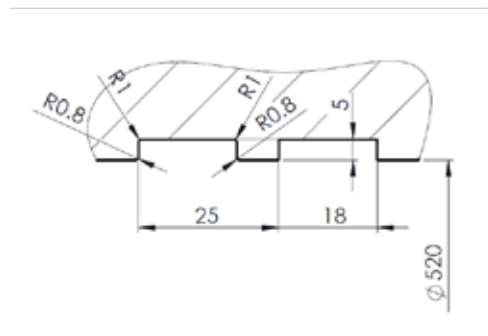
```
%  
O0093  
T0404  
G97S350M3  
  
G0X500Z100  
(ROUGHING)  
G65P8027D440I420A35B5C15R1.2U0.3W22Z35K-100Q1H1M0.7X460S2.5T2F2.5Y0.3V0.1  
(FINISHING)  
G65P8027D440I420A35B5C15R1.2W22Z35K-100Q1H2M0.02X460S2.5T2F2.5  
M30
```





EXAMPLE 2

Oil grooves for Bronze Bushing



%

O0093

T0404

G97S250M3

G65P8027D520I530X480A0B0C18R0.8U0.2W25Z40K-250Q-1H1M0.7S1T1F0.8Y0.2V0.1

G65P8027D520I530X480A0B0C18R0.8W25Z40K-250Q-1H2M0.05S1T1F0.8

G0Z400

M5

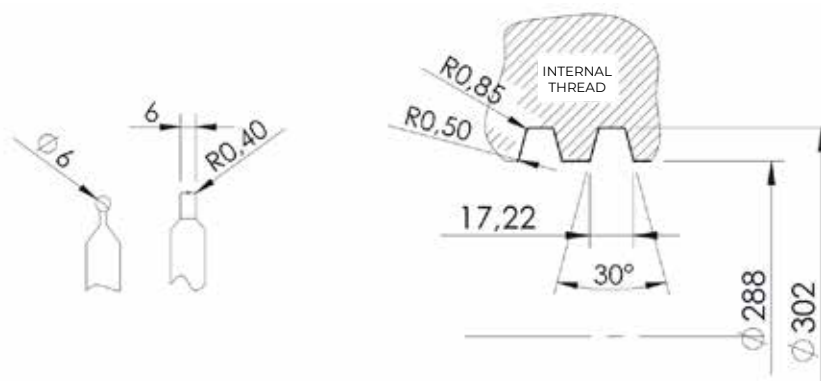
M30

EXAMPLE 3

Internal turning of a trapezoidal thread TR300 pitch 28. Roughing is carried out with a round nose tool of radius 3mm and finishing is carried out with a 6mm thick parting tool with radius 0.4 on the edges, having a flat stroke equal to $6\text{mm} - (0.4 \times 2) = 5.2\text{mm}$.

The floor thread allows a radius of up to 1mm, which is equal to the floor clearance.

In this case, a certain safety clearance is maintained and a floor radius of 0.85 is executed. A taper of 0.1mm (parameter J) is also inserted to compensate for geometric errors on the lathe.





```
%  
O0093  
T0404  
G97S250M3  
G65P8027D276I302X250A15B15C17.22R3U0.2W28Z40K-250Q-1H1M0.7S0.85T0.85F0.5Y0.  
3V0.05J0.1  
G65P8027D276I302X250A15B15C17.22R0.4W28Z40K-250Q-  
1H2M0.05S0.85T0.85F0.5E5.2J0.1  
G0Z400  
M5  
M30
```



WARNINGS

1. The Z and K parameters represent the coordinates of the start and end of the thread in relation to the centreline of the thread. From that height will be developed the descent along the sides of the cuts, it is therefore advisable to keep an adequate safety distance outside the workpiece equal to at least two times the pitch.

2. Machining can be restarted with the same precautions as for standard threading operations.

3. In the initial part of the macro part-program, set parameter #140 equal to 1 for machines with type A, G codes (e.g. for lathes where the limitation is written with G50S...) or set #140=2 for machines with type B, G codes (e.g. for lathes where the limitation is written with G92S).

(***INTERNAL PARAMETERS***)

#140=1(1=TYPE A CODES 2=TYPE B CODES)

4. 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 needed to use parameters with different numeration, request the macro modification.

5. The macro is supplied already tested, but it is advisable for the first few times to carry out the necessary tests at no-load or away from the workpiece to make the necessary checks.

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

7. The macro only works with the non-modal G65 call up and not with the modal G66 call up function, as indicated in the "Programming" section.

8. If the thread needs to be wider, finishing can be restarted by changing the C parameter.

FOCUS PARAMETERS





9. If a tool with a flat stroke is used for finishing, indicate the radius on the tool edges, even if minimal, for the calculation of the cuts with the crest height.

FOCUS PARAMETERS





www.cncofcourse.com