



MACRO
SHOP

Fanuc cable reel groove



Fanuc 0/16/18/21/31



 Rev. 7



FIELD OF APPLICATION

The macro carries out the roughing and finishing of a circular groove in helical interpolation in the Z axis direction, similar to a thread but with a circular profile. The macro can be used to machine grooves on cable reels, radial grooves for conveying cooling fluid or any application where a radial profile with a constant pitch must be executed. The same macro can be used whether machining is carried out in the negative Z direction or in the positive Z direction.

The entry must always take place in an unloaded area or external to the workpiece, but the exit can also take place in the workpiece, as the macro automatically calculates a smooth exit from the workpiece, always following the helix angle. It is possible to choose the type of exit from the workpiece using a parameter. In roughing, the calculation of the cuts for opening the cable reel groove is calculated automatically on the basis of the tool's percentage application.

In fact, the CNC macro will automatically calculate the number of cuts required to rough the profile for each diametral co-ordinate. While in finishing, it will be possible to set the maximum crest height that determines the roughness of the profile, and the macro will automatically calculate the cuts both on the round radius with the external diameter and on the whole rounded profile.



PARAMETERS DESCRIPTION

I = BOTTOM GROOVE DIAMETER

A = DIAMETRAL PITCH DEPTH (for roughing only)

E = CREST DIAMETER

R = GROOVE RADIUS

U = INSERT RADIUS

W = THREAD PITCH

Z = Z VALUE FOR START GROOVE

K = Z VALUE FOR END GROOVE

X = RETRACTION VALUE

S = SAFETY DISTANCE

V = EXIT TYPE

V=0 STRAIGHT EXIT

V=1 INTERP. EXIT FOLLOWING THE THREAD

M = TOOL DIAMETER ENGAGEMENT FACTOR (EXAMPLE M0.7) (for roughing only)

H = MACHINING TYPE =1 ROUGHING =2 FINISHING =3=ROUGH.+FINISH.

Y = FINISHING ALLOWANCE (for roughing only)

Q = CREST HEIGHT (for finishing only)

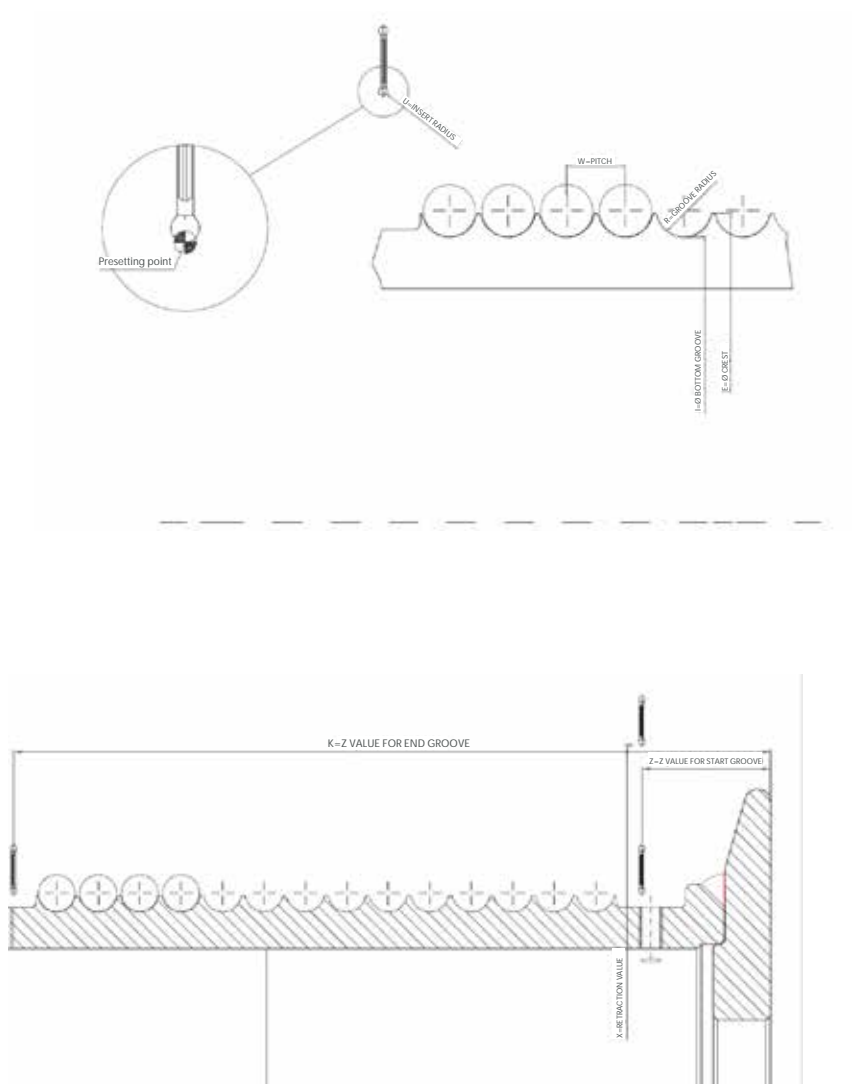
F = ROUNDING RADIUS (for finishing only)





FOCUS PARAMETERS

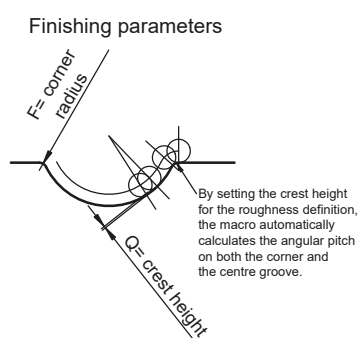
PAR.M





CYCLE DESCRIPTION

The cycle consists of a multiple-pass roughing of the circular groove, moving to the safety distance X and then moving in Z to the start of the groove. It quickly approaches the workpiece at the crest diameter plus the S parameter (SAFETY DISTANCE). At this point it performs for each value in X N cuts to open the groove starting from the centre, using the tool diameter equal to the factor defined with the M parameter. The cycle calculates the cuts automatically, keeping the depth of cut A constant and expressing it diametrically, starting from the crest diameter (E) and ending with the diameter at the bottom of the groove (I). At the end of the cycle the tool returns to the clearance height X. For a first section, the increment will be along a tangent line until the insert radius has reached tangency with the circular profile. The exit from the workpiece occurs with two types of path depending on the V parameter. If the V parameter is equal to 0, the exit will be without helical interpolation, therefore to be used when the exit takes place in an empty space, for example in a relief groove or outside the workpiece. If you set parameter V equal to 1, the output will also follow the thread; use it in the case of grooves with an ending in the workpiece, so that the profile will always end up outside the workpiece following the helical trajectory. The distance covered in Z when exiting will depend on the X path to reach the exit point S outside the workpiece. If a radial distance equal to half the pitch is travelled to reach the retraction point X from the bottom of the groove, the exit will take place in half a turn. If you also want to perform finishing cuts along the profile, you must enter the allowance to be left for the finishing in the parameters.



profile that you wish to leave for finishing. Parameter Q identifies the crest height that corresponds to the roughness Rt.



PROGRAMMING

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

The subprogram is provided with the numbering O8013, so the subprogram is called up with G65P8013 followed by the parameters. If the subprogram needs to be renumbered, the letter P must be followed by the new program number.

Call-up example:

G65P8013I344A0.5E354R7.5U4W15Z-428K-2458X370S5V0M0.5H3Q0.01F1Y0.3

If you wish to carry out the finishing operation with the same tool as soon as you have finished roughing, you can program a single macro call-up with all the parameters entered. If you want to carry out roughing and finishing in two separate operations and perhaps even with two different tools, you will need to program the roughing callout and then the finishing callout. It is possible to leave out the parameters that are not needed for an operation, but it is also possible to leave all the parameters between roughing and finishing and change only the letter H.

Example:

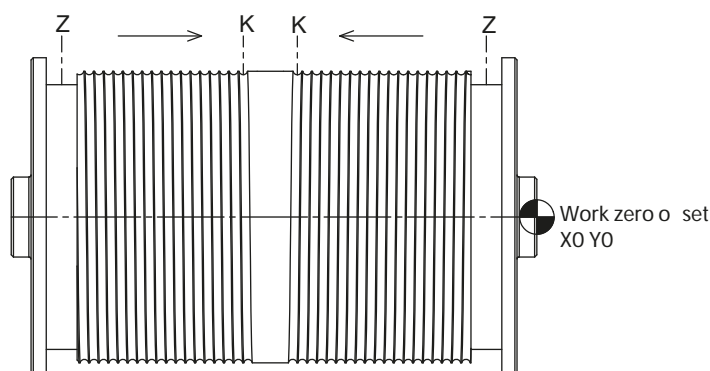
Roughing

G65P8013I309.1A0.5E318R25U4W16Z-736K-32X350S1V0M0.5H1Y0.3

Finishing

G65P8013I309.1E318R25U4W16Z-736K-32X350S1V0H2Q0.01F2

Example of right and left groove



The groove to be machined has a radius of 10.5 (parameter R) and is made with a rounded parting tool R6 (parameter U). The grooves have a pitch of 22 mm (parameter W) and have to be machined on an external diameter of 796 mm (parameter E).





The Z and K values are as shown in the figure, so they are both symmetrical in relation to the workpiece centreline so that the two helixes end symmetrically in relation to the centre of the workpiece.
The depth of cut is 0.2 mm diametral (parameter A).

```
T808(PART.TOOL R.6)
G97S50M4
(LEFT HELIX THREAD)
G0X867
Z-150
(S.P.8013 CALL)
G65P8013I780A0.2E796R10.5U6W22Z-144K-657X805S5V1M0.6H-
3Q0.01F1Y0.15
G0X867M9
M0(POSITION START Z)
Z-1275
(S.P.8013 CALL)
G65P8013I780A0.2E796R10.5U6W22Z-1275K-762X805S5V1M0.6H-
3Q0.01F1Y0.15
G0X867M9
M5
M30
```

FOCUS PARAMETERS

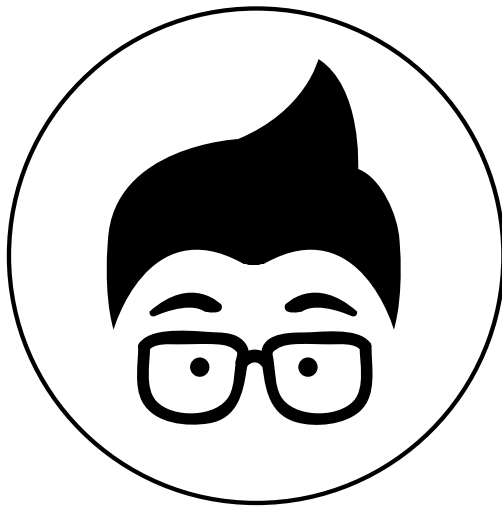




WARNINGS

1. The finishing of the groove will depend on the depth of cut (A) and the insert radius. It is recommended to use an insert radius of 40% to 80% of the groove radius.
2. The entry into the workpiece is in rapid traverse. It is therefore advisable to use the Z value to indicate a position outside the workpiece or inside a groove where rapid feed is possible.
3. Regarding the restart of a machining operation or replacement of the insert, the same rules apply as for a threading operation. It is therefore important not to change the Z value of the groove start, otherwise synchronism will be lost. The same applies to the angular position at which the workpiece is clamped in the spindle; with the workpiece position unchanged and the tool position unchanged, the cycle can be resumed even after a reset.
4. In the 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).
5. In order to use the macro on your machine, make sure that the parametric programming option is enabled. Although most machines have B macro programming enabled, check that your machine does too. To do this, simply go to the MDI window and enter #100=1, press start and if no alarm message appears, this means that programming with macro B 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.
6. The macro is supplied already tested, but it is advisable for the first few times to carry out the necessary tests without the workpiece or away from it.
7. In case of vibration, it is recommended to reduce the tool radius.





www.cncofcourse.com