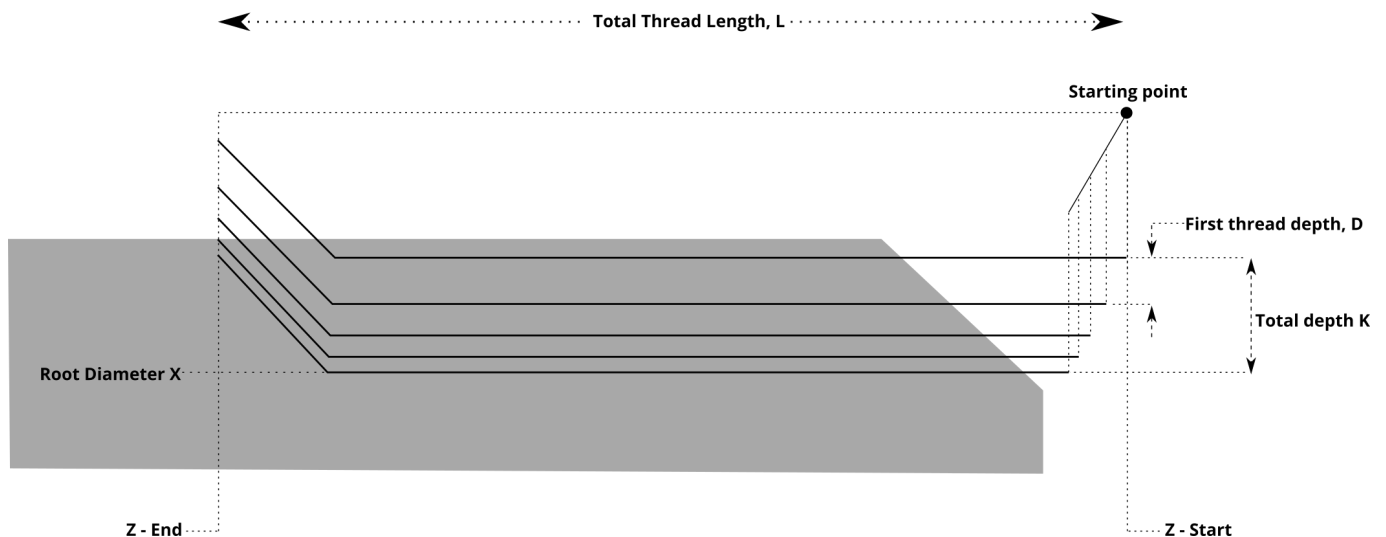


G76 Thread Cycle

The G76 cycle is a threading cycle that allows the program to go through any single thread in either a single or a double block of program code. myCNC uses the 10/11/15T and the 0T/16T/18T style parameters.

G76



Single line thread cutting

G76 X.. Z.. I.. K.. D.. F.. A.. P..

- X = Diameter of the last threading pass
- Z = Position of the thread end
- I = Taper over the total length
- K = Single depth of thread - positive
- D = Depth of first pass - positive
- A = Angle of thread - positive
- F = Pitch of thread
- P = Infeed method (one of 4) - positive

G-code example of a single line thread cycle:

```
G76 X2.89 Z-1.6 I0 K0.051 D0140 A60 P4 F0.0833
```

Two-line thread cycle

The two-line thread cycle consists of a command in the following style:

```
G76 P(11)(22)(33), Q, R G76 X, Z, R, P, Q, F
```

First line:

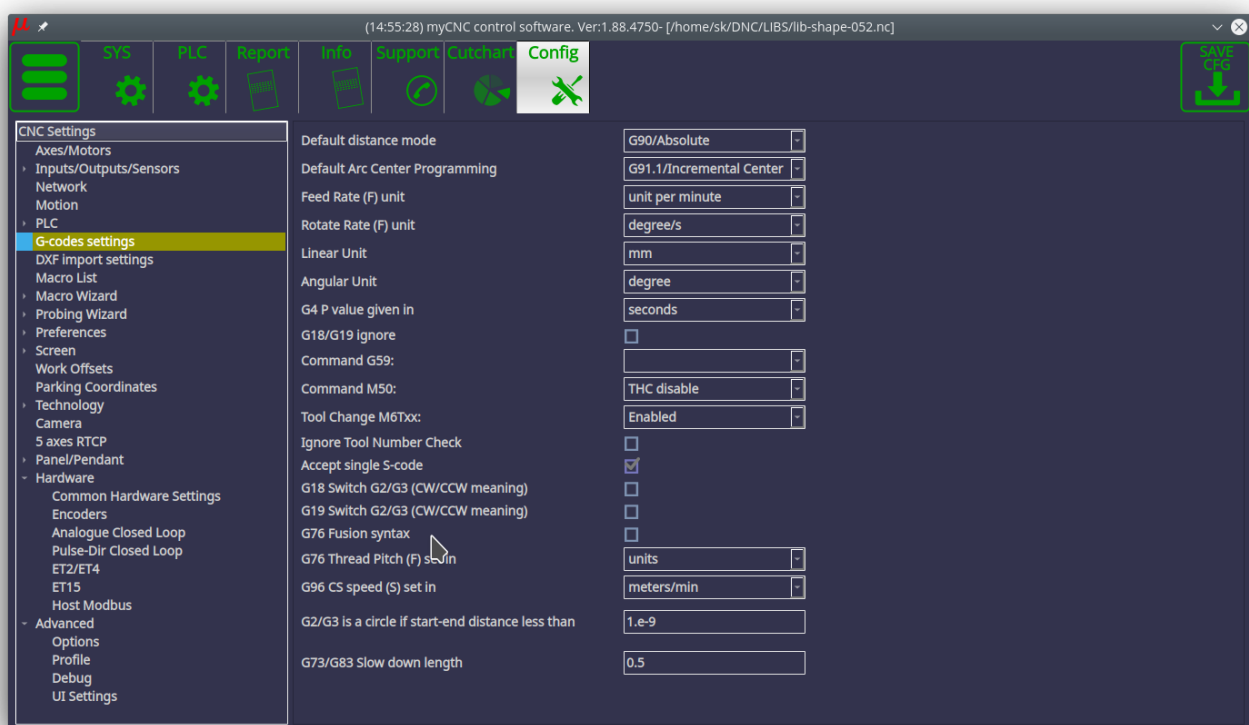
- P(11) = Number of finishing cuts (01-99)
- P(22) = Number of leads for gradual pull-out (0.0-9.9 times lead). There is no decimal point used in this block (00-99)
- P(33) = Angle of thread
- Q = Minimum cut depth
- R = Finish allowance

Second line:

- X = Last diameter of the thread
- Z = Endpoint of thread
- R = Radial difference between start and end positions at final pass - omit for straight threads
- P = Depth of thread
- Q = Depth of first cut pass
- F = Feedrate of the thread (same as thread lead)

NOTE: The R/P/Q addresses of the first and the second line are not the same and have their own meaning within each block only.

Due to different CAM software packages using different units (for example, mm in Fusion 360 versus 1/1000th of a mm in SolidCAM), a toggle in Settings > Config > G-codes settings > G76 Fusion Syntax is necessary:



The above toggle allows the user to switch between the two units systems.

G-code examples of a double line G76 cutting command:

```
G76 P009900 Q40 R100
```

G76 X16.18 Z-6.3 P560 Q40 F1

N27 G76 P001000 Q0.147 R0

N28 G76 X11.39 Z-10. P0.88 Q0.147 F1.337

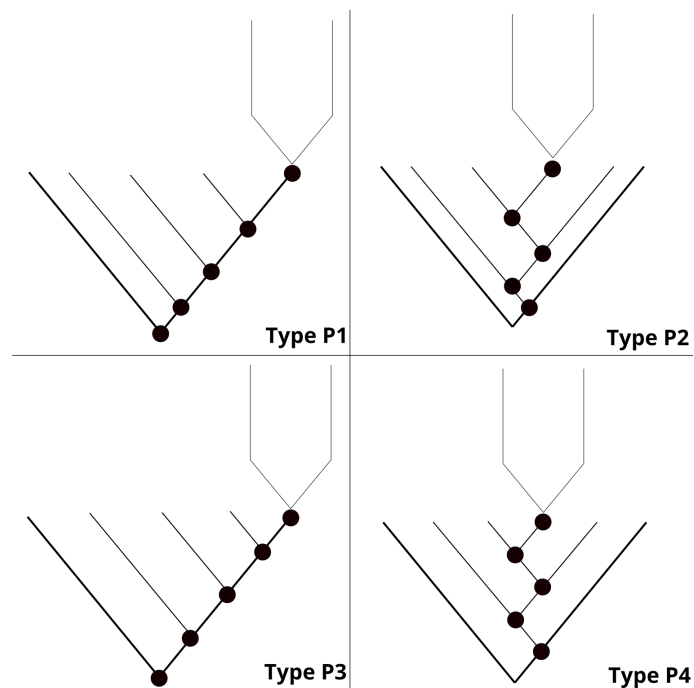
Thread cutting depth infeed

The P- and A-parameters allow to control for the thread cutting type and thread insert angle respectively. The A parameter table is available below:

A0	Straight infeed
A29	ACME thread type
A30	Metric trapezoidal thread type
A55	Whitworth 55 degree thread type
A60	Standard 60 degree V thread type
A80	German PG thread type

The radial infeed is programmed with the A0 parameter, while non-zero A parameter represents compound infeed. Additionally, the P parameter can also be selected for a one-edge or a two-edge cutting mode which either has a constant amount or a constant depth for the infeed:

G76 infeed



- P1 - one-edge cutting, constant cutting amount
- P2 - two-edges cutting, constant cutting amount
- P3 - one-edge cutting, constant cutting depth
- P4 - two-edges cutting, constant cutting depth

If the P-parameter is omitted in the G76 command, then the P1 mode is used by default.

From:
<http://cnc42.com/> - **myCNC Online Documentation**

Permanent link:
http://cnc42.com/mycnc/gcodes_list/g76-thread-cycle

Last update: **2021/06/18 17:09**

