

CNC

Programming
Tutorials
Examples
G & M Codes



Thanh Tran

CNC Programming Tutorials Examples G & M Codes

G & M Programming Tutorial Example Code for Beginner to Advance Level CNC Machinist.

TABLE OF CONTENTS:

- [1. Advanced Level](#)
- [2. Beginner Level](#)
- [3. Bolt Hole Circle](#)
- [4. Boring CNC Lathe](#)
- [5. Chamfer Radius](#)
- [6. CNC Lathe Machine](#)
- [7. CNC Milling Machine](#)
- [8. Drilling](#)
- [9. G02 G03 I J K](#)
- [10. G02 G03 R](#)
- [11. G40 G41 G42](#)
- [12. G81 Drilling Cycle](#)
- [13. G91 Incremental Programming](#)
- [14. Grooving](#)
- [15. Intermediate Level](#)
- [16. Pattern Drilling](#)
- [17. Peck Drilling Lathe](#)
- [18. Peck Drilling-Mill](#)
- [19. Peck Milling](#)
- [20. Ramping Milling](#)
- [21. Slot Milling](#)
- [22. Step Turning CNC Lathe](#)
- [23. Subprogram](#)
- [24. Taper Threading](#)
- [25. Tapping](#)
- [26. Threading](#)

CNC Program Examples *

Advanced Level

Fanuc G72.1 G72.2 Figure Copy Program Example (Bolt Hole Circle)

This CNC program example shows how both G72.1 and G72.2 figure copy functions can call one-another in one part program, read Programming Notes below carefully.

Programming

Fanuc G72.1 Rotational Copy

- Using G72.1 Rotational Copy G-code a figure specified by a subprogram can be repeatedly produced with **Rotational movement**.
- Read G72.1 definition with program example Fanuc G72.1 Rotational Copy (Figure Copy Function CNC Mill)

Fanuc G72.2 Linear Copy

- Using G72.2 Linear Copy G-code a figure specified by a subprogram can be repeatedly produced with **Linear movement**.
- Read G72.2 definition with program example Fanuc G72.2 Linear Copy (Figure Copy Function CNC Mill)

Fanuc G81 Drilling Cycle

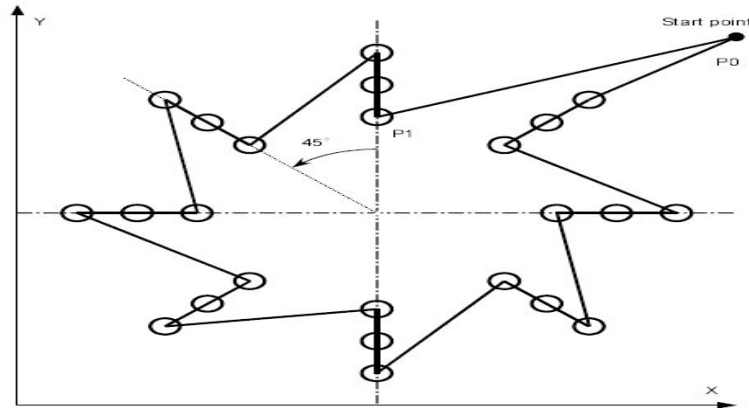
- G81 drilling cycle is used for simple drilling/spot drilling operations.
- Read G81 drilling cycle definition with program examples Fanuc G81 Drilling Cycle

Programming Notes

Specifying two or more commands to copy a figure

- G72.1 cannot be specified more than once in a subprogram for making a rotational copy (If this is attempted, alarm PS0900 will occur).
- G72.2 cannot be specified more than once in a subprogram for making a linear copy (If this is attempted, alarm PS0901 will occur).
- In a subprogram that specifies rotational copy, however, linear copy can be specified. Similarly, in a subprogram that specifies linear copy, rotational copy can be specified.

Fanuc G71.2 G72.2 Program Example



Main program

```
O4000 ;
N10 G90 G00 G17 X240. Y230. Z100. ; (P0)
N20 G72.1 P4100 X120. Y120. L8 R45. ;
N30 G80 G00 X240. Y230. ; (P0)
N40 M30 ;
```

Sub program_ Rotation copy_ (G72.1)

```
O4100 N100 G72.2 P4200 I0 J20. L3 ;
N200 M99 ;
```

Sub program_ Linear copy (G72.2_)

```
O4200 N110 G90 G81 X120. Y180. R60. Z10. F200. ; (P1)
N210 M99 ;
```

Fanuc G72.2 Linear Copy (Figure Copy Function CNC Mill)

Fanuc G72.2 Linear Copy

Using G72.2 Linear Copy G-code a figure specified by a subprogram can be repeatedly produced with **Linear** movement.

Programming

G72.2 P... L... I... J...

Parameters

Parameter	Description
P	Subprogram number
L	Number of times the operation is repeated
I	Shift along X-axis
J	Shift along Y-axis

G-Code Data

Modal/Non-Modal	G-Code Group
Non-Modal	00

Programming Notes

Notes

- In the G72.2 block, addresses other than P, L, I and J are ignored.
- P, I and J must always be specified.
- If L is not specified, the figure is copied once.
- For shifts (I, J), specify increments. The n-th geometric shift is equal to the specified shift times (n – 1).

First block of the subprogram

Always specify a move command in the first block of a subprogram that performs a linear copy. If the first block contains only the program number such as

O00001234; and does not have a move command, movement may stop at the start point of the figure made by the n-th (n = 1,2, 3, ...) copying.

Example of an incorrect program

```
O00001234 ;  
G00 G90 X100.0 Y200.0 ;  
_____  
_____  
M99 ;
```

Example of a correct program

```
O00001000 G00 G90 X100.0 Y200.0 ;  
_____  
_____  
M99 ;
```

Limitation

Specifying two or more commands to copy a figure

- G72.2 cannot be specified more than once in a subprogram for making a linear copy (If this is attempted, alarm PS0901 will occur).
- In a subprogram that specifies linear copy, however, rotational copy (G72.1) can be specified. Similarly, in a subprogram that specifies rotational copy, linear copy can be specified.

Commands that must not be specified

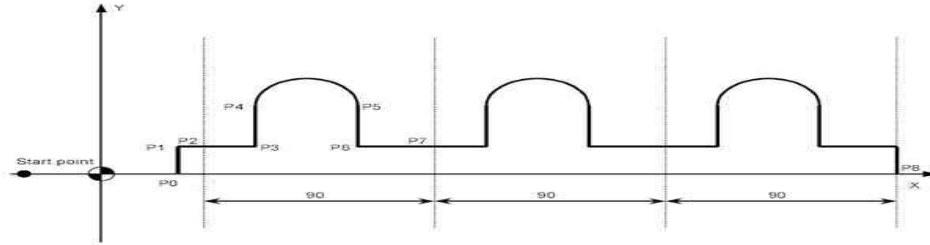
Within a program that performs a linear copy, the following must not be specified:

- Command for changing the selected plane (G17 to G19)
- Command for specifying polar coordinates (G16)
- Reference position return command(G28)
- Axis switching
- Coordinate system rotation (G68)
- scaling (G51)
- programmable mirror image (G51.1)

Single block

Single-block stops are not performed in a block with G72.1 or G72.2.

G72.2 Programming Example



Main program

```
O3000 ;
N10 G90 G00 X-30. Y0 ;
N20 X0 ;
N30 G01 G17 G41 X30. D01 F100 ; (P0)
N40 Y20. ; (P1)
N50 X40. ; (P2)
N60 G72.2 P3100 L3 I90.0 J0 ;
N70 G90 X310. Y0 ; (P8)
N80 X0 ;
N90 G40 G00 X-30.0 ;
N100 M30 ;
```

Sub program

```
O3100 G91 G01 X20. ; (P3)
N100 Y30. ; (P4)
N200 G02 X40. I20. ; (P5)
N300 G01 Y-30. ; (P6)
N400 X30. ; (P7)
N500 M99 ;
```

Fanuc G72.1 Rotational Copy Program Example

Fanuc G72.1 Rotational copy programming example, G72.1 G-code is used to repeatedly produce a figure with rotational movement.

Fanuc G72.1 Rotational Copy

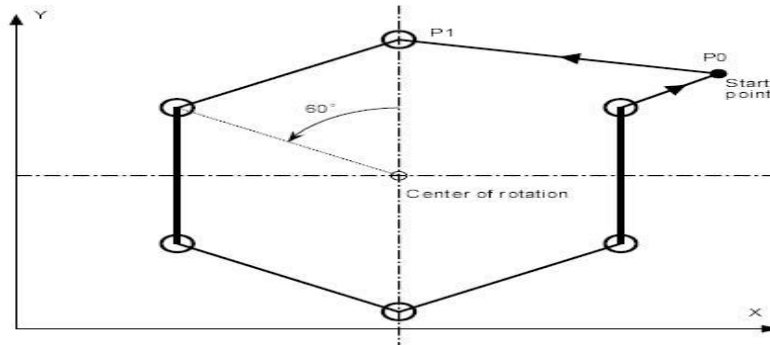
- Using G72.1 Rotational Copy G-code a figure specified by a subprogram can be repeatedly produced with Rotational movement.

- Read more Fanuc G72.1 Rotational Copy (Figure Copy Function CNC Mill)

Fanuc G81 Drilling Cycle

- G81 drilling cycle is used for simple drilling/spot drilling operations.
- Read complete article with program examples Fanuc G81 Drilling Cycle

Fanuc G72.1 Program Example



Main program

```
O2000 ;
N10 G90 G00 G17 X250. Y100. Z100. ; (P0)
N20 G72.1 P2100 L6 X100. Y50. R60. ;
N30 G80 G00 X250. Y100. ; (P0)
N40 M30 ;
```

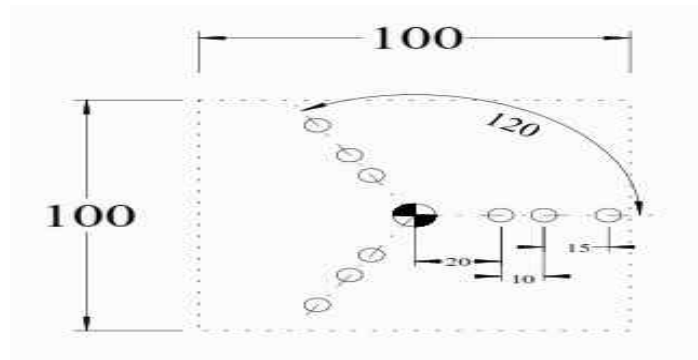
Sub program

```
O2100 N100 G90 G81 X100. Y150. R60. Z10. F200. ; (P1)
N200 M99 ;
```

Fanuc G68 Coordinate Rotation Program Example

- Fanuc G68 Coordinate Rotation G-Code makes it easy for cnc machinist to run a pattern of operations in a rotated angle.
- Here is a basic cnc programming Example which helps to understand the actual working of G68 coordinate rotation.

Fanuc G68 Program Example



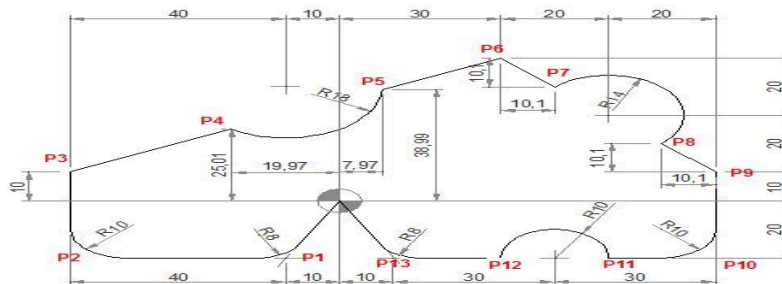
```
T1 M6
G0 G90 G40 G21 G17 G94 G80
G54 X20 Y0 S1500 M3
G43 Z100 H1
Z5
G81 R3 Z-20 F? M8
X30
X45
G68 X0 Y0 R120
X20 Y0
X30
X45
G68 X0 Y0 R240
X20 Y0
X30
X45
G69 G80
G0 G90 Z100 M30
```


Beginner Level

CNC Mill Example Program G01 G02 G03 G90 G91

A cnc mill program for cnc machinists programmers, who have started to learning basic cnc programming techniques.

CNC Mill Example Program



CNC Program

```
N40 G90 G00 X0 Y0  
N50 G01 X-10 Y-20 R8 (P1)  
N60 G01 X-50 R10 (P2)  
N70 Y10 (P3)  
N80 X-19.97 Y25.01 (P4)  
N90 G03 X7.97 Y38.99 R18 (P5)  
N100 G01 X30 Y50 (P6)  
N110 G91 X10.1 Y-10.1(P7)  
N120 G90 G02 X59.9 Y20.1 R14 (P8)  
N130 G01 X70 Y10 (P9)  
N140 Y-20 R10(P10)  
N150 X50 (P11)  
N160 G03 X30 R10 (P12)  
N170 G01 X10 R8 (P13)  
N180 X0 Y0
```

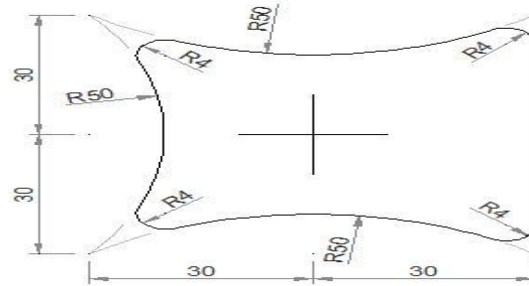
G M S T Codes Explanation

Code	Description
G00	Rapid traverse

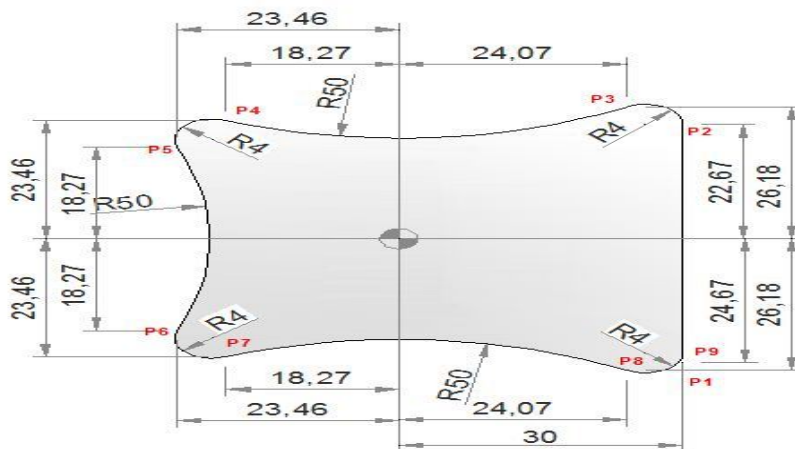
G01	Linear interpolation
G02	Circular interpolation CW
G03	Circular interpolation CCW
G90	Absolute command
G91	Increment command

G02 G03 Example CNC Mill

G02 G03 Circular interpolation CNC mill example program.



G02 G03 Example CNC Mill



CNC Part Program

```
G0 X30 Y-30 (P1)
G1 Y22.67 (P2)
G3 X24.07 Y26.18 R4 (P3)
G2 X-18.27 Y23.46 R50 (P4)
```

G3 X-23.46 Y18.27 R4(P5)
 G2 X-23.46 Y-18.27 R50 (P6)
 G3 X-18.27 Y-23.46 R4 (P7)
 G2 X24.07 Y-26.18 R50 (P8)
 G3 X30 Y-24.67 R4 (P9)
 G1 X33

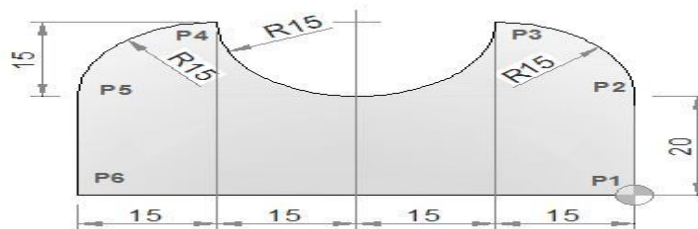
G M S T Codes Explanation

Code	Description
G0	Rapid traverse
G1	Linear interpolation
G2	Circular interpolation CW
G3	Circular interpolation CCW
M30	End of program (Reset)

Multiple Arc CNC Mill Program G2 G3 I J

CNC milling machine program which combines/joins multiple arcs.

Multiple Arc CNC Mill Program G2 G3 I J



CNC Part Program

```

N10 M6 T1 G43 H1 M3
N15 S500 F120
N20 G0 X0 Y0 (P1)
N25 G1 Y20(P2)
N30 G3 X-15 Y35 I-15 J0 (P3)
N35 G2 X-45 Y35 I-15 J0 (P4)
N40 G3 X-60 Y20 I0 J-15 (P5)
N45 G1 Y0 (P6)
N50 G1 X0 (P1)

```

G M S T Codes Explanation

Code	Description
G0	Rapid traverse
G1	Linear interpolation
G2	Circular interpolation CW
G3	Circular interpolation CCW
G43	Tool length compensation + direction
M3	Spindle start forward CW
M6	Tool change
M30	End of program (Reset)
T	Tool
S	Speed
F	Feed

Haas Corner Rounding and Chamfering Example G01 C R

Haas Corner Rounding and Chamfering

Haas CNC program example to show how Chamfer and Corner Radius can be programmed.

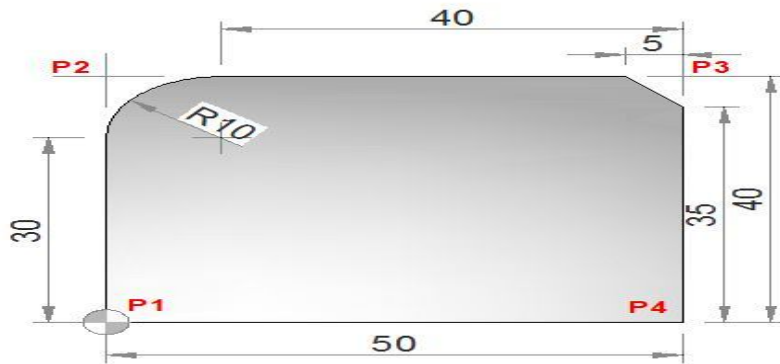
Haas Chamfering

To program Chamfer
N10 G01 X20 Y30 ,C3

Haas Corner Rounding

To program Radius
N10 G01 X20 Y30 ,R3

Haas Corner Rounding and Chamfering Example



Haas CNC Program

```

O1234 (Corner Rounding and Chamfering Example);
T1 M6;
G00 G90 G54 X0. Y0. S3000 M3; (P1)
G43 H01 Z0.1 M08;
G01 Z-0.5 F20.;
Y40. ,R10.;(P2)
X50. ,C5.;(P3)
Y0. ; (P4)
G00 Z0.1 M09;
G53 G49 Z0.;
G53 Y0.;
M30;

```

Haas G M S T Codes

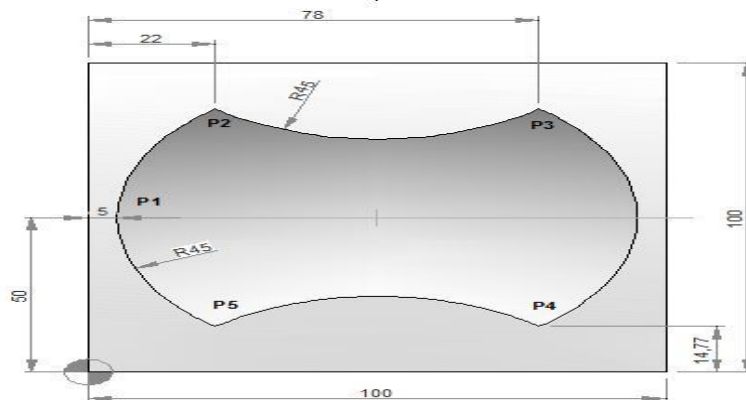
Code	Description
G00	Rapid Motion
G01	Linear Interpolation Motion
G43	Tool Length Compensation +
G49	G43/G44 Cancel
G53	Non-Modal Machine Coordinate Selection
G54	Select Work Coordinate System 1
G90	Incremental Programming
M3	Spindle On, Clockwise (S)
M6	Tool Change (T)
M08	Coolant On
M09	Coolant Off
M30	Program End and Reset
S	Spindle speed
T	Tool

CNC Mill Subprogram Example

Joining Multiple Arcs G02 G03 G41

CNC milling program to describe how two or more radii can be joint together in a cnc mill program.

CNC Mill Subprogram Example



CNC Part Program

```
N10 T1 H1 M6 G43 M3
N20 F150 S250
N30 G0 X-21 Y50 Z0.5
N40 G0 Z0
N50 M98 P040050
N60 G49
N70 G0 Z50
N80 M30
```

Subprogram

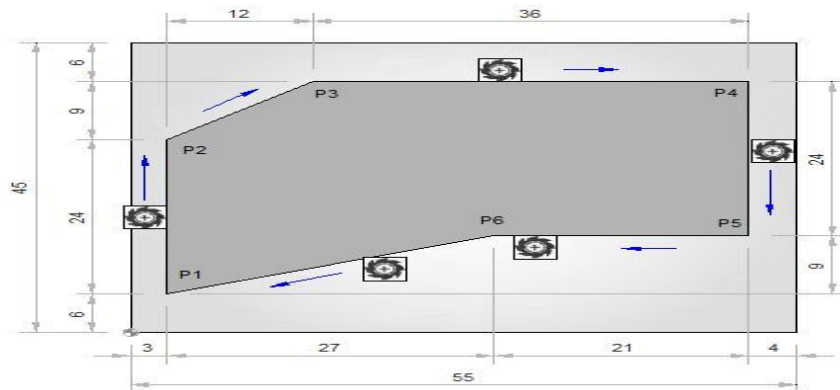
```
O0050
N10 F160 S400
N20 G0 Z-2.5 G91
N30 G1 G90 X5 Y50 G41 (P1)
N40 G2 X22 Y85.23 I45 J0 (P2)
N50 G3 X78 Y85.23 R45 (P3)
N60 G2 X78 Y14.77 R45 (P4)
N70 G3 X22 Y14.77 R45 (P5)
N80 G2 X5 Y50 R45 (P1)
N90 G0 G40 X-21
N100 M99
```

G M S T Codes Explanation

Code	Description
G00	Rapid traverse
G01	Linear interpolation
G02	Circular interpolation CW
G03	Circular interpolation CCW
G40	Cutter compensation cancel
G41	Tool nose radius compensation left
G43	Tool length compensation + direction
G49	Tool length compensation cancel
G90	Absolute command
G91	Increment command
M03	Spindle start forward CW
M06	Tool change
M30	End of program (Reset)
M98	Subprogram call
M99	End of subprogram
T	Tool
S	Speed
F	Feed

CNC Mill Program G91 G41 G43

CNC Mill Program G91 G41 G43



CNC Part Program

```
N05 G54  
N10 M6 T1 G43 H1 M3  
N15 S500 F120  
N20 G0 X-22 Y-22  
N25 Z-3  
N30 G1 X3 Y6 G41 H2 (P1)  
N35 G91 X0 Y24(P2)  
N40 X12 Y9(P3)  
N45 X36 (P4)  
N50 Y-24 (P5)  
N55 X-21 (P6)  
N60 G90 X3 Y6 (P1)  
N65 G0 X-21 G40
```

G M S T Codes Explanation

Code	Description
G00	Rapid traverse
G01	Linear interpolation
G40	Cutter compensation cancel
G41	Tool nose radius compensation left
G43	Tool length compensation + direction
G54	Workpiece coordinate system 1 selection
G90	Absolute command
G91	Incremental command
M06	Tool change
T	Tool
S	Speed
F	Feed

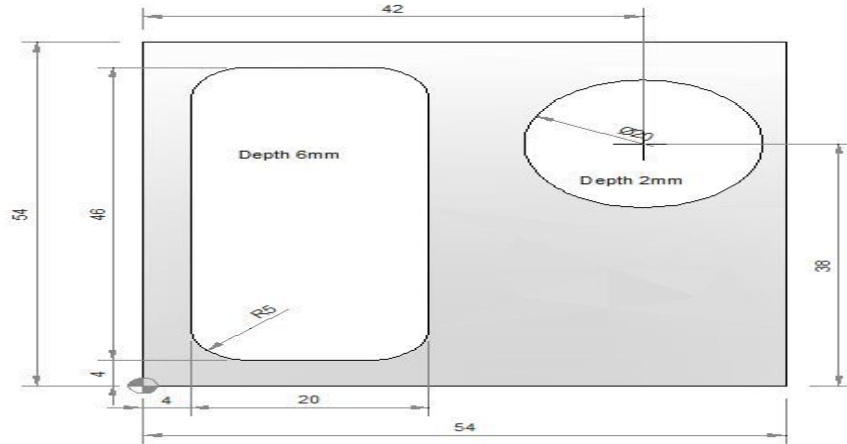
CNC Pocket Milling Program

Example – Peck Milling

CNC milling program example which shows how a cnc program can be made to machine Pockets on a cnc mill.

This program example uses Peck milling to cut material to machine a rectangular and one round pocket.

CNC Pocket Milling Program Example



Main Program

Milling cutter diameter: 10mm

```
N05 G55  
N10 M6 T2 H3 G43 M3  
N15 S1000 F60  
N20 G0 X9 Y9 Z1  
N25 G1 Z0  
N30 M98 P030035  
N35 G0 Z1 G90  
N40 X42 Y38  
N45 G1 Z-2 F30  
N50 X47 F300  
N55 G3 X47 Y38 I-5 J0  
N60 G0 Z100  
N65 G49  
N70 M30
```

Subprogram

```
O0035  
N05 G1 Z-2 G91 F30  
N10 X10 F100  
N15 Y36  
N20 X-10  
N25 Y-36  
N30 M99
```

Explanation

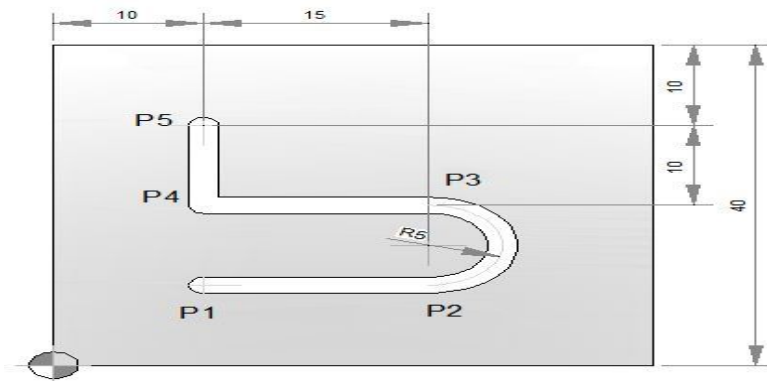
Although this cnc mill program is self explanatory

```
M98 P030035
```

CNC Milling Program Example G03 G90 G91

CNC milling program example, which illustrates how a cnc program can be easily converted from G90 Absolute program mode to G91 Incremental program mode.

CNC Milling Program Example



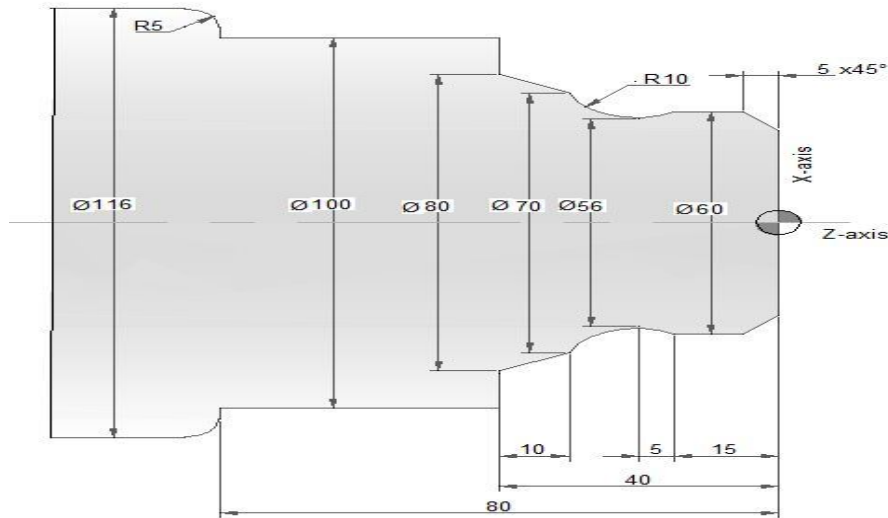
G90 Absolute Program	G91 Incremental Program
N10 T1 M6 M3 G43	N10 T1 M6 M3 G43
N15 S2000 F80	N15 S2000 F80
N20 G0 X10 Y10 Z1 (P1)	N20 G0 X10 Y10 Z1
N25 F30	N25 F30
N30 G1 Z-2	N30 G1 Z-2
N35 X25 F80 (P2)	N35 G91 X15 F80
N40 G3 X25 Y20 I0 J5 (P3)	N40 G3 X0 Y10 I0 J5
N45 G1 X10 (P4)	N45 G1 X-15
N50 Y30 (P5)	N50 Y10
N55 G0 Z100 G49	N55 G90 G0 Z100 G49
N60 M30	N60 M30

CNC Turning Center Programming Example

Easy to understand CNC turning center/cnc lathe programming example for cnc machinists who work on cnc turning centers/cnc lathe machines.

This cnc programming example can be used as a cnc learning programming exercise for beginners level cnc programmers/machinists.

CNC Turning Center Programming Example



```
N10 T03 D03 M06
N20 G97 S900 M04
N30 G00 G42 X40 Z5
N40 G92 S3500
N50 G96 S250
N60 G01 X60 Z-5 F0.1
N70 Z-15
N80 X56 Z-20
N90 G02 X70 Z-30 R10
N100 G01 X80 Z-40
N110 X100
N120 Z-80
N130 X106
N140 G03 X116 Z-85 R5
N150 G01 X120
N160 G00 X150 Z100 G40 G97 S900 M05
N170 M30
```

Wire EDM Programming Example

Complete cnc programming example for machinists who work on a Wire EDM machine (Electrical Discharge Machining)

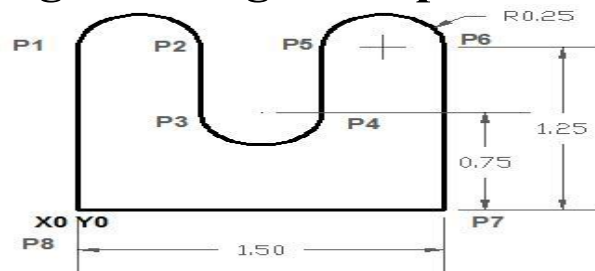
CNC Wire Cut Introduction

Wire EDM cutting, also known as electrical discharge machining, is a process that uses an electrically energized thin wire to slice through metal. Wire EDM cutting uses rapid, controlled, repetitive spark discharges from the wire to the workpiece, thereby eroding the metal away. The workpiece must be electrically conductive.



Wire EDM cutting can provide high dimensional accuracy for close fitting parts. The process can make sharp inside corners.

Wire EDM Programming Example



Wire EDM Programming Example

```
G92 X0. Y0.  
G01 Y1.25 (P1)  
G02 X.5 I.25 J0.(P2)  
G01 Y.75(P3)  
G03 X1. I.25 J0.(P4)  
G01 Y1.25 (P5)  
G02 X1.5 I.25 J0. (P6)  
G01 Y0. (P7)  
X0. (P8)  
M02
```

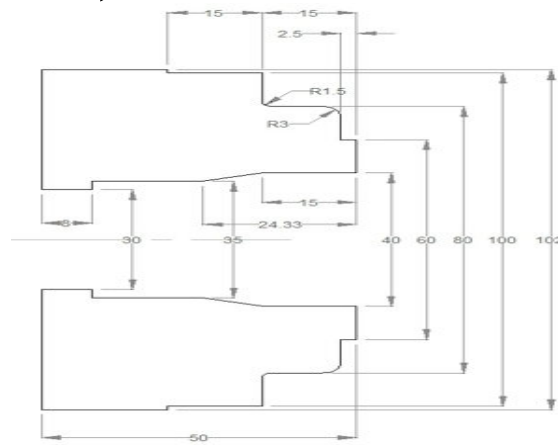
G92 command is used to assign program zero. You will include the distances and directions from the program zero point to the wire start point in your *G92* command.

CNC Lathe Basic Programming Example ID/OD Turning/Boring Operations (No Canned Cycle Used)

A full CNC programming example with ID/OD (Turning/Boring operations) for cnc machinists who work on a cnc lathe machine. A must to learn/practice for those who are learning cnc programming.

The added benefit of this programming example is that no cnc lathe canned cycle is used in this programming example.

CNC Lathe Basic Programming Example (Turning Boring Operations)



CNC Lathe Example Turning Boring

```
O0000  
N10 (ø30 DRILL)  
G50 T0200  
G97 S250 M03  
G00 X0 Z5.0 T0202 M08  
G01 Z-5.0 F0.07  
W1.0  
Z-40.0 F0.25  
G00 Z5.0  
Z-39.0  
G01 Z-60.0  
G00 Z10.0  
X200.0 Z200.0 T0200  
M01  
N20 (Outside diameter stock removal)  
G50 S1500 T0100
```

G96 S180 M03
 G00 X94.0 Z5.0 T0101 M08
 G01 Z-14.8 F0.27
 G00 U2.0 Z0.5
 G01 X28.0 F0.23
 G00 X87.0 W1.0
 G01 Z-14.8 F0.27
 G00 U2.0 Z1.0
 X80.5
 G01 Z-14.1
 G02 X81.9 Z-14.8 R0.7
 G00 X100.5 W1.0
 G01 Z-29.8
 G00 U2.0 Z-1.0
 G01 X60.5 F0.23
 G00 X82.0 W1.0
 Z-2.4
 G01 X60.5
 X72.9
 G03 X80.5 Z-6.2 R3.8
 G00 U2.0 Z5.0
 X200.0 Z200.0 T0100
 M01
 N30 (Inside diameter stock removal)
 G50 S1500 T0400
 G96 S180 M03
 G00 X34.5 Z3.0 T0404 M08
 G01 Z-41.8 F0.27
 G00 U-0.5 Z1.0
 X39.5
 G01 Z-15.0
 X34.5 Z-24.3
 G00 Z10.0
 X200.0 Z200.0 T0400
 M01
 N40 (Out diameter finishing)
 G50 S1800 T0500
 G96 S200 M03
 G00 X63.0 Z5.0 T0505 M08
 Z0
 G01 X38.0 F0.2
 G00 X60.0 Z3.0
 G42 Z1.0
 G01 Z-2.5 F0.2
 X74.0
 G03 X80.0 Z-5.5 R3.0
 G01 Z-13.5
 G02 X83.0 Z-15.0 R1.5
 G01 X100.0
 Z-30.0
 X102.0
 G40 G00 U2.0 W1.0
 G00 Z10.0
 X200.0 Z200.0 T0500

```

M01
N50 (Inside diameter finishing)
G50 S1800 T0600
G96 S200 M03
G00 X40.0 Z5.0 T0606 M08
G41 Z1.0
G01 Z-15.0 F0.2
X35.0 Z-24.33
Z-42.0
X29.0
G40 G00 Z10.0
X200.0 Z200.0 T0600 M09
M30

```

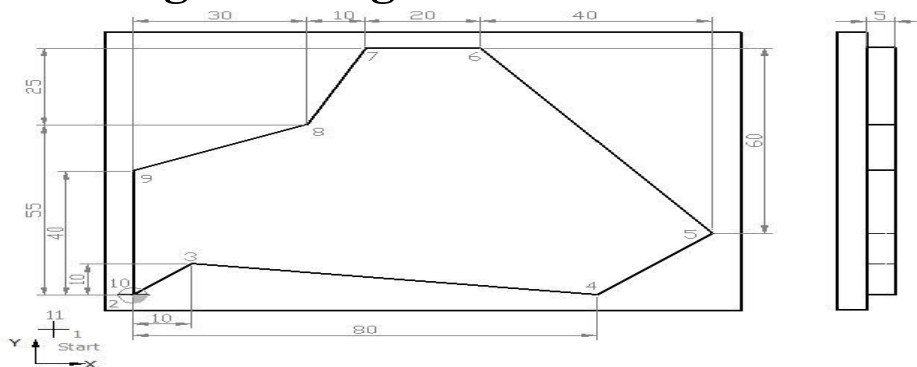
CNC Mill Programming Exercise using G91 Incremental Programming

This CNC mill programming exercise will give you more idea to understand drawing and how to find exact coordinate of each drawing point.

Completely based on G91 Incremental Dimensioning System, so cnc programmers/machinists can learn and understand how G91 Incremental Programming works.

This program is written for 4-axis CNC mill, where C-axis is used for rotary table. 3-axis CNC mill can also run this program, so for this just remove program block no. N15.

CNC Mill Programming Exercise

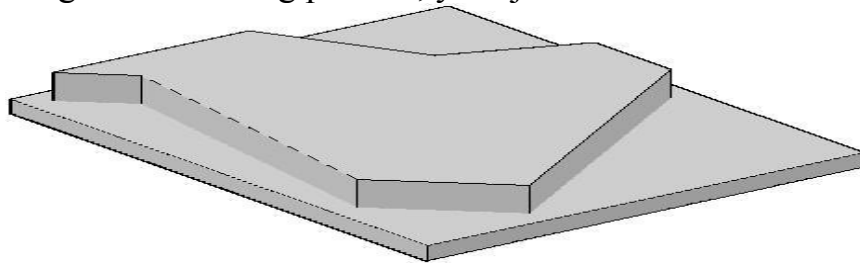


G91 Incremental Programming CNC Mill Exercise

```
N5 G00 G54 G64 G90 G17 X-20 Y-20 Z50 ;Point 1
N10 S450 M03 F250 D01 (12.5 MM DIA)
N15 C0
N20 Z5
N25 G01 Z0
N30 Z-5
N35 G42 G91 X20 Y20 ;Point 2
N40 X10 Y10 ;Point 3
N45 X70 Y-10 ;Point 4
N50 X20 Y20 ;Point 5
N55 X-40 Y60 ;Point 6
N60 X-20 ;Point 7
N65 X-10 Y-25;Point 8
N70 X-30 Y-15;Point 9
N80 X0 Y-40 ;Point 10
N85 G40 G90 X-20 Y-20;Point 11
N90 G00 Z50
N95 Y100
N100 M30
```

Finished Part

After completing the machining process, your job will look like this in 3D view,



Complete Machined Part in 3D

Explanation of CNC G-Code

G00 : Rapid traverse.

G54 : Zero Offset no. 1.

G64 : Continuous-path mode.

G90 : Absolute dimensioning system.

G91 : Incremental dimensioning system.

G17 : X-Y plan selection.

G42 : Cutter radius compensation activation (right hand side movement)

G40 : Cutter radius compensation de-active

S : Spindle speed

F : Axes motion feed

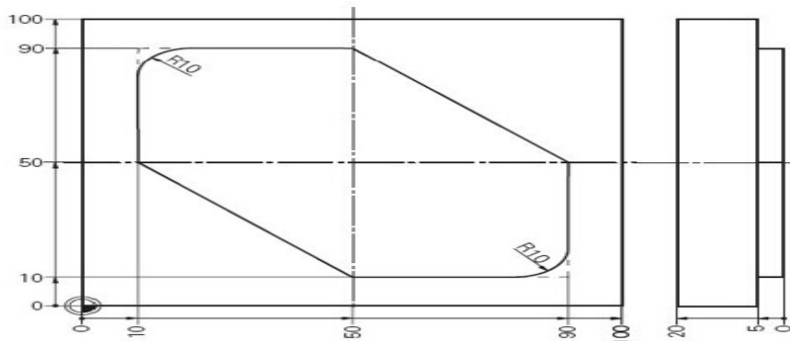
M : Cutter motion (3=clockwise, 4=anti-clockwise)

D : Tool no

Vertical Machining Center Programming Example CNC

Vertical machining centers are a vital part of a cnc machine workshop, So here is a Vertical machining center programming example for cnc machinists who have just started learning cnc programming.

Vertical Machining Center Programming Example



Vertical Machining Center Programming Example

```
N10 T1
N20 S1000 M3
N30 G00 G40 G90 G17 X-30 Y60
N40 G43 Z50 H1
N50 Z-5
N60 G01 G41 X10 Y60 D1 F200
N70 Y80
N80 G02 X20 Y90 R10
N90 G01 X50
N100 X90 Y50
N110 Y20
N120 G02 X80 Y10 R10
N130 G01 X50
N140 X10 Y50
N150 Y65
N160 G40 X-30 Y60
N170 G00 Z200 M02
```

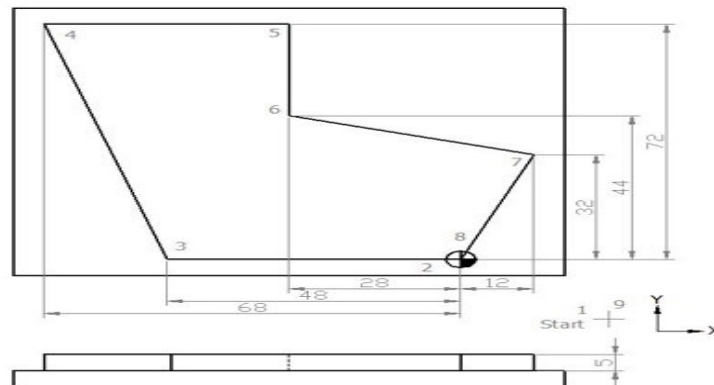
G41 G40 Cutter Radius Compensation Example CNC Mill Program

Cutter Radius Compensation Example program shows how G41, G40 can be used in a cnc mill program.

Cutter Compensation code used in this program are,

- G41 Cutter Radius Compensation Left
- G40 Cutter Radius Compensation Cancel

Cutter Radius Compensation Example



G41 G40 Cutter Radius Compensation Example

```
N5 G00 G54 G64 G90 G17 X20 Y20 Z50
```

```
N10 S450 M03 F250 D01 (12.5 MM DIA)
```

```
N15 C0
```

```
N20 Z5
```

```
N25 G01 Z0
```

```
N30 Z-5
```

```
N35 G41 X0 Y0
```

```
N40 X-48
```

```
N45 X-68 Y72
```

```
N50 X-28
```

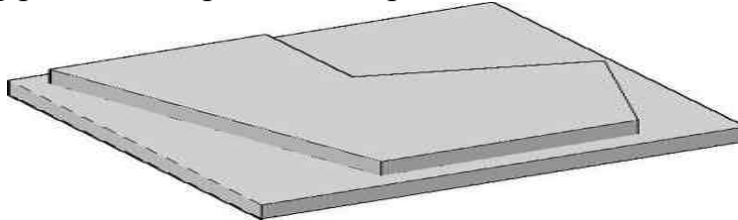
```
N55 Y44
```

```
N60 X12 Y32
```

```
N65 X0 Y0
```

```
N70 G40 X20 Y-20
N75 G00 Z50
N80 Y100
N85 M30
Finished Part
```

After machining process completion, component will look like



Cutter Radius Compensation Example Finished Part

Explanation of CNC G-Code

- G00 : Rapid traverse.
- G54 : Zero Offset no. 1.
- G64 : Continuous-path mode.
- G90 : Absolute dimensioning system.
- G17 : X-Y plan selection.
- G41 : Cutter radius compensation activation (left hand side movement)
- G40 : Cutter radius compensation de-active
- S : Spindle speed
- F : Axis motion feed
- M : Cutter rotation (3=clockwise, 4=anti-clockwise)
- D : Tool offset no

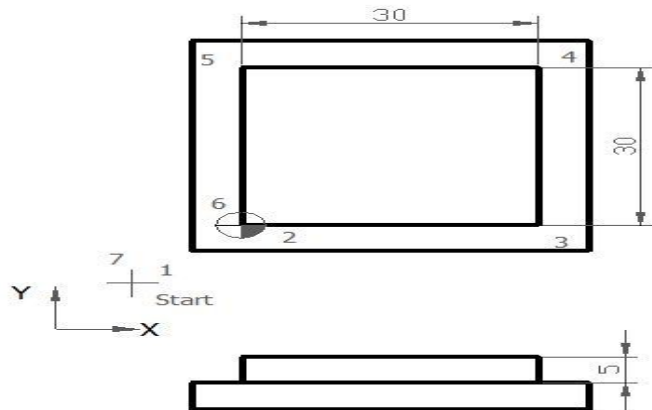
Siemens Sinumerik Milling Programming Example

A very simple cnc milling program example which will show cnc machinists the use of Siemens Sinumerik milling programming concepts.

This program is written for 4-axis cnc mill, where C is used for rotary table.

But a simple cnc mill can also run this program just remove program block N15

Sinumerik Milling Program

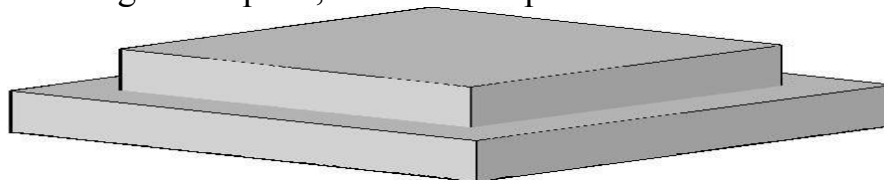


Sinumerik Mill Programming Example

```
N5 G00 G54 G64 G90 G17 X-20 Y-20 Z50  
N10 S450 M03 F250 D01 (12.5 MM DIA)  
N15 C0  
N20 Z5  
N25 G01 Z0  
N30 Z-5  
N35 G42 X0 Y0  
N40 X30  
N45 Y30  
N50 X0  
N55 Y0  
N60 G40 X-20 Y-20  
N65 G00 Z50  
N70 Y100  
N75 M30
```

Finished Part

After the machining is complete, this finished part will look like this



Finished Part

Explanation of G-Code

- G00 – Rapid traverse.
- G54 – Zero Offset no 1.
- G64 – Continuous-path mode.
- G90 – Absolute dimensioning system.
- G17 – X-Y plan selection.
- G42 – Cutter radius compensation activation

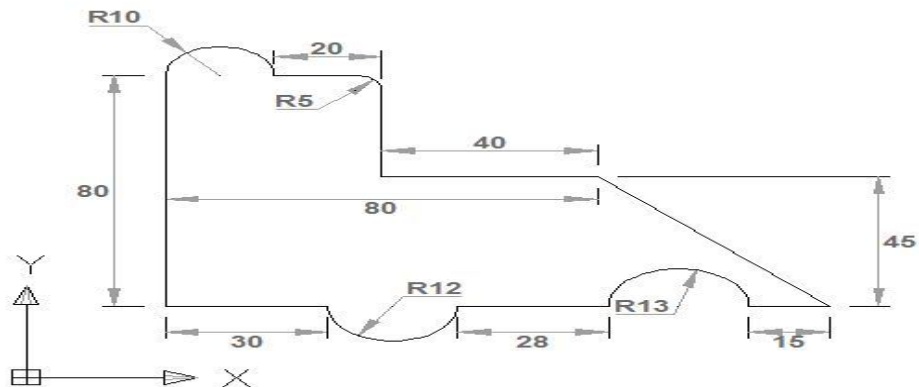
G40 – Cutter radius compensation cancel
M03 – Cutter rotation clockwise
S – Spindle speed
F – Axis motion feed
D – Tool no

CNC Mill G02 G03 Circular Interpolation Programming Example

CNC mill program to show the use of

- G02 Circular Interpolation CW
- G03 Circular Interpolation CCW

CNC Mill Program



CNC Mill G02 G03 Circular Interpolation Programming Example

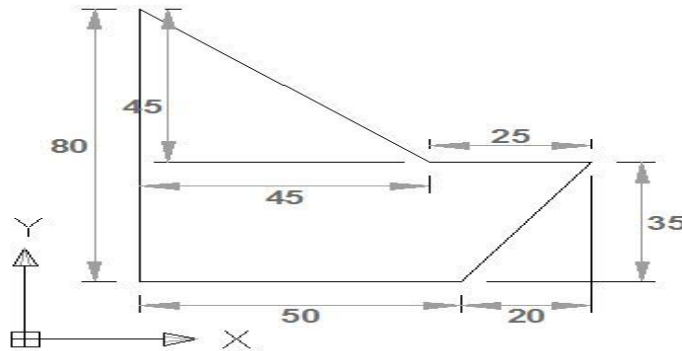
```
G90 G01 X0 Y0
X30
G03 X54 R12
G01 X82
G02 X108 R13
G01 X123
X80 Y45
X40
Y75
G03 X35 Y80 R5
G01 X20
G03 X0 Y80 R10
G01 Y0
M30
```

CNC Mill Programming Exercise using G90 Absolute Programming G91 Incremental Programming

A very basic cnc mill programming exercise for cnc machinists.
This cnc example code shows the use of

- G90 Absolute Programming
- G91 Incremental Programming

CNC Mill Programming Exercise



Using G90 Absolute Programming G91 Incremental Programming

G90 Absolute Programming

```
G90 G01 X0 Y0  
X50.0  
X70.0 Y35.0  
X45.0  
X0 Y80.0  
Y0  
M30
```

G91 Incremental Programming

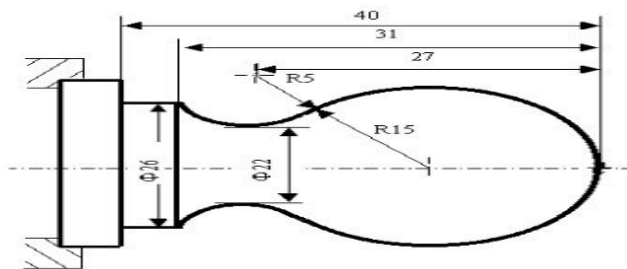
```
G91 G01 X0 Y0  
X50.0  
X20.0 Y35.0
```

X-25.0
X-45.0 Y45.0
Y-80.0
M30

CNC Arc Programming G02 G03 Example

CNC arc programming example this cnc program shows how two arcs G03 G02 can be joint together.

CNC Arc Programming G02 G03 Example



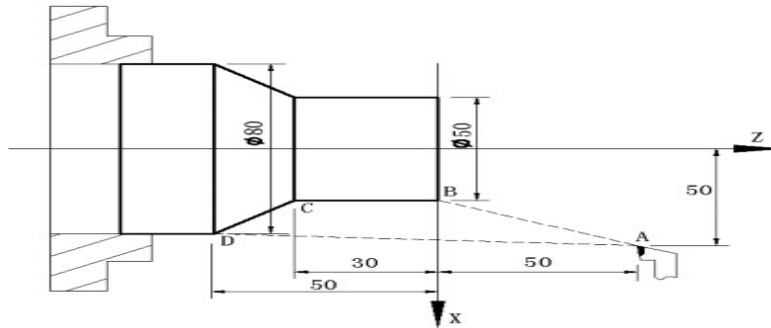
CNC Arc Programming G02 G03 Example

```
O0001
N001 G0 X40 Z5 i¼ˆ Rapid position i¼%
N002 M03 S200 i¼ˆ Start spindle i¼%
N003 G01 X0 Z0 F900 i¼ˆ (Approach workpiece)
N005 G03 U24 W-24 R15 i¼ˆ i¼ˆ Cut R15 arc i¼%
N006 G02 X26 Z-31 R5 i¼ˆ i¼ˆ Cut R5 arc i¼%
N007 G01 Z-40 i¼ˆ (Cut i¼, 26)
N008 X40 Z5 i¼ˆ (Return to starting point)
N009 M30 i¼ˆ i¼ˆ End of program i¼%
```

CNC Lathe Simple G Code Example – G code Programming for Beginners

CNC lathe simple g code example for CNC machinists/CNC programmers who just started their journey to learn CNC programming.

Simple G Code Example CNC Lathe



CNC Lathe Simple G Code Example

```
O0001
N5 M12
N10 T0101
N15 G0 X100 Z50
N20 M3 S600
N25 M8
N30 G1 X50 Z0 F600
N40 W-30 F200
N50 X80 W-20 F150
N60 G0 X100 Z50
N70 T0100
N80 M5
N90 M9
N100 M13
N110 M30
N120 %
```

Code Explanation

N5 Clamping workpiece
N10 Changing No.1 tool and executing its offset
N15 Rapidly positioning to A point
N20 Starting the spindle with 600 r/min
N25 Cooling ON
N30 Approaching B point with 600mm/min
N40 Cutting from B point to C point

N50 Cutting from C point to D point
N60 Rapidly retracting to A point
N70 Canceling the tool offset
N80 Stopping the spindle
N90 Cooling OFF
N100 Releasing workpiece
N110 End of program, spindle stopping and Cooling OFF

Fanuc Circular Interpolation G02 G Code Example

Fanuc Circular Interpolation

Fanuc Circular Interpolation commands (G02, G03 G Codes) are used to move a tool along a circular arc.

G02 – Circular Interpolation Clockwise.

G03 – Circular Interpolation Anti-Clockwise.

How to Program Circular Interpolation?

On CNC lathe machines with Fanuc control Clockwise Circular Interpolation (Arc) can be programmed in following ways.

N10 G02 X Z R

or

N10 G02 X Z I K

Explanation

N10 G02 X Z R

X – Arc end-point in X-axis

Z – Arc end-point in Z-axis

R – Arc Radius.

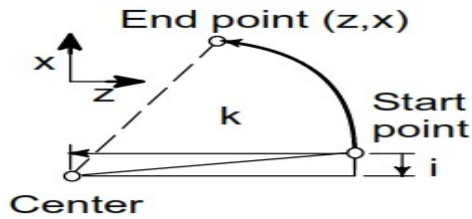
N10 G02 X Z I K

X- Arc end-point in X-axis.

Z – Arc end-point in Z-axis.

I – Distance from arc-start-point to arc-center-point in X-axis.

K – Distance from arc-start-point to arc-center-point in Z-axis.

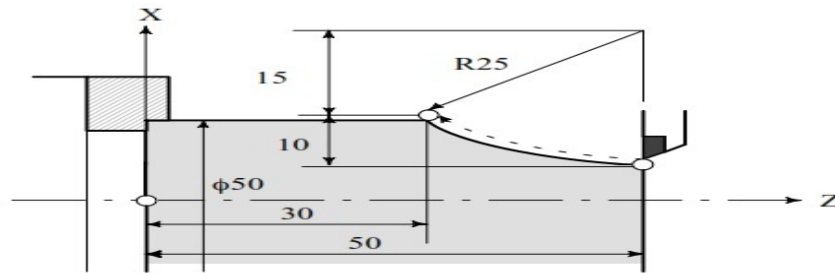


Fanuc Circular Interpolation G02

Fanuc Circular Interpolation G02 G Code Example

The following Fanuc circular interpolation program example is programmed in multiple different ways.

CNC programmers/machinists can use G02/G03 X Z R method or G02/G03 X Z I K method if control allows.



Fanuc Circular Interpolation G02 G Code Example

(Diameter programming)

```
G02 X50 Z30 I25 F0.3  
G02 U20 W-20 I25 F0.3  
G02 X50 Z30 R25 F0.3  
G02 U20 W-20 R25 F0.3
```

G Code Example Mill – Sample G Code Program for Beginners

Sample G Code Program for Beginners

Sample G code program example for cnc programmers / cnc machinists who work or want to learn cnc mill programming.

A similar G code example can be found here

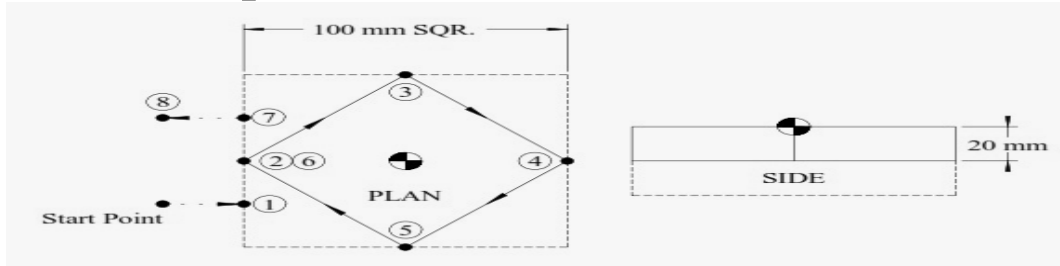
Simple G Code Example Mill – G code Programming for Beginners

If you are interested in G code subroutine example (sub program) read

CNC Mill Contour Pecking – FANUC Subprogram Repeat Example

This is the same component but this time we are machining it in taper.

G Code Example



G Code Example Mill

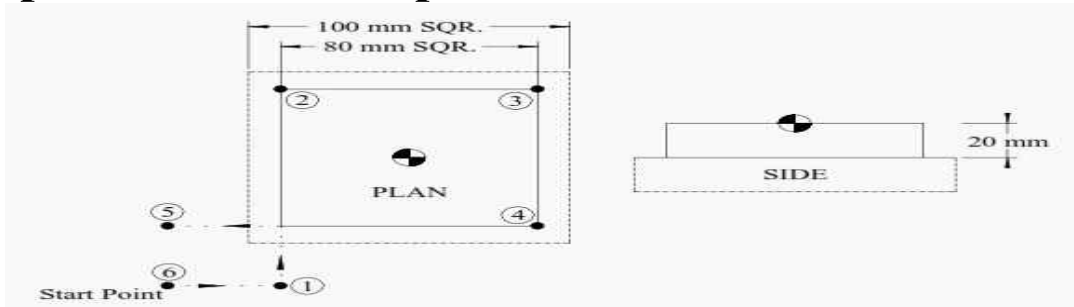
```
O1000
T1 M6
G0 G90 G40 G21 G17 G94 G80
G54 X-75 Y-25 S500 M3 (Start Point)
G43 Z100 H1
Z5
G1 Z-20 F100
X-50 M8 (Position 1)
Y0(Position 2)
X0 Y50(Position 3)
X50 Y0(Position 4)
X0 Y-50 (Position 5)
X-50 Y0 (Position 6)
Y25 (Position 7)
X-75 (Position 8)
G0 Z100
M30
```

Simple G Code Example Mill – G code Programming for Beginners

G code Programming for Beginners

- Learning G code programming is not that difficult.
- Read and learn about a G code.
- Understand how that G code work.
- Then make a simple G code example drawing.
- Make program for that example.
- Run the program on your cnc machine (Safety first, keep a professional around).
- Or you can simply run a cnc program Simulation (but this always doesn't substitute the actual cnc machining process.)
- The last thing is the most crucial which is **Just practice practice and practice.**
- This is a very simple G code example for beginner level cnc programmers / cnc machinists.
- This G code program example don't use Tool radius compensation G40/G41/G42 G code.

Simple G Code Example Mill



Simple G Code Example

```
O1000
T1 M6
(Linear / Feed - Absolute)
G0 G90 G40 G21 G17 G94 G80
G54 X-75 Y-75 S500 M3 (Position 6)
G43 Z100 H1
Z5
G1 Z-20 F100
X-40 (Position 1)
Y40 M8 (Position 2)
X40(Position 3)
Y-40 (Position 4)
X-75 (Position 5)
Y-75 (Position 6)
G0 Z100
M30
```


Bolt Hole Circle

Fanuc G72.1 G72.2 Figure Copy Program Example (Bolt Hole Circle)

This CNC program example shows how both G72.1 and G72.2 figure copy functions can call one-another in one part program, read Programming Notes below carefully.

Programming

Fanuc G72.1 Rotational Copy

- Using G72.1 Rotational Copy G-code a figure specified by a subprogram can be repeatedly produced with **Rotational movement**.
- Read G72.1 definition with program example Fanuc G72.1 Rotational Copy (Figure Copy Function CNC Mill)

Fanuc G72.2 Linear Copy

- Using G72.2 Linear Copy G-code a figure specified by a subprogram can be repeatedly produced with **Linear movement**.
- Read G72.2 definition with program example Fanuc G72.2 Linear Copy (Figure Copy Function CNC Mill)

Fanuc G81 Drilling Cycle

- G81 drilling cycle is used for simple drilling/spot drilling operations.
- Read G81 drilling cycle definition with program examples Fanuc G81 Drilling Cycle

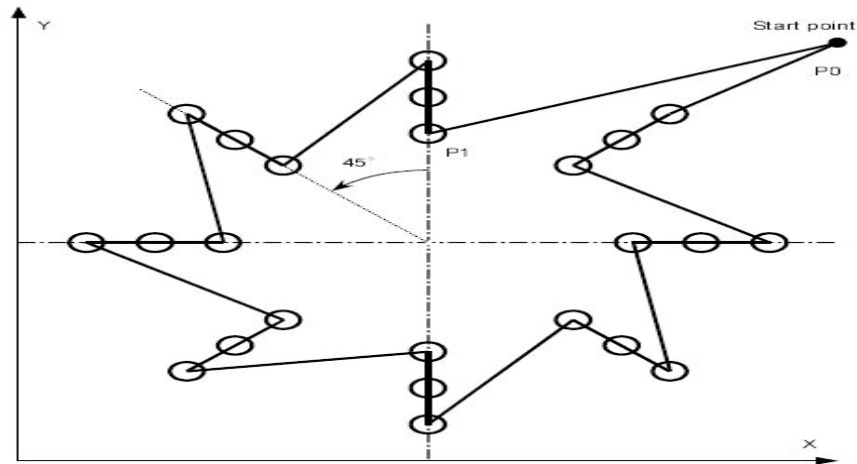
Programming Notes

Specifying two or more commands to copy a figure

- G72.1 cannot be specified more than once in a subprogram for making a rotational copy (If this is attempted, alarm PS0900 will occur).

- G72.2 cannot be specified more than once in a subprogram for making a linear copy (If this is attempted, alarm PS0901 will occur).
- In a subprogram that specifies rotational copy, however, linear copy can be specified. Similarly, in a subprogram that specifies linear copy, rotational copy can be specified.

Fanuc G71.2 G72.2 Program Example



Main program

```
O4000 ;
N10 G90 G00 G17 X240. Y230. Z100. ; (P0)
N20 G72.1 P4100 X120. Y120. L8 R45. ;
N30 G80 G00 X240. Y230. ; (P0)
N40 M30 ;
```

Sub program_Rotation copy_(G72.1)

```
O4100 N100 G72.2 P4200 I0 J20. L3 ;
N200 M99 ;
```

Sub program_Linear copy (G72.2_)

```
O4200 N110 G90 G81 X120. Y180. R60. Z10. F200. ; (P1)
N210 M99 ;
```


Boring CNC Lathe

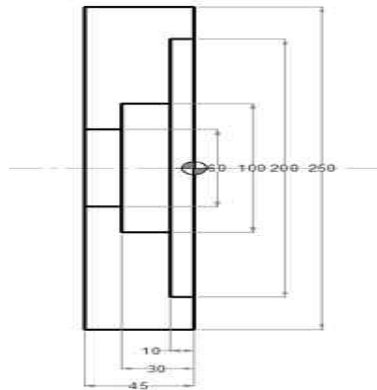
Fanuc G71 G72 G70 Canned Cycle CNC Lathe Internal Machining Example (Boring & Facing)

Fanuc programming example which shows the use of multiple fanuc canned cycle in cnc programming, Following canned cycle are used in this cnc lathe programming example

Fanuc G71 for Boring Opertaion

This programming example also illustrate how G71 turning cycle can be used for internal machining (boring operation). So if you want to remove extra stock from inside of a component, you can use Fanuc G71 turning cycle for internal stock removal as well.

Fanuc Programming Example



Fanuc Canned Cycle Example

```
G28 U0 W0  
G50 S1500  
N1 T0101 M8 (OD FACING)  
G96 S180 M3  
G0 X255.0 Z5.0  
G72 W2.0 R0.5
```

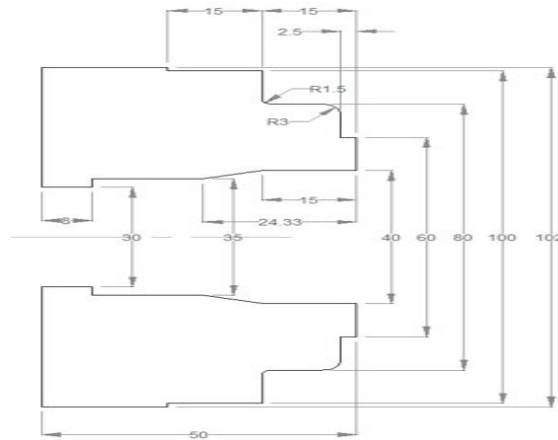
```
G72 P100 Q200 F0.25
N100 G0 Z0 G41
G1 X-2.0 F0.18
N200 G0 Z5.0
G40
X255.0
G28 U0 W0
N2 T0404 M8 (ID ROUGH)
G96 S180 M3
G0 X50.0 Z5.0
G71 U2.0 R0.5
G71 P500 Q600 U-0.5 W0.1 F0.25
N500 G0 X202.0 G41
G1 Z0 F0.12
X200.0
Z-10.0
X100.0
Z-30.0
X60.0
Z-45.0
N600 X50.0
G40
G0 Z5.0
G28 U0 W0
N3 T0505 M8 (ID FINISH)
G96 S220 M3
G0 X50.0 Z5.0
G70 P500 Q600
G0 X50.0 Z5.0
G28 U0 W0
M5 M9
M30
```

CNC Lathe Basic Programming Example ID/OD Turning/Boring Operations (No Canned Cycle Used)

A full CNC programming example with ID/OD (Turning/Boring operations) for cnc machinists who work on a cnc lathe machine. A must to learn/practice for those who are learning cnc programming.

The added benefit of this programming example is that no cnc lathe canned cycle is used in this programming example.

CNC Lathe Basic Programming Example (Turning Boring Operations)



CNC Lathe Example Turning Boring

O0000

```

N10 (ø30 DRILL)
G50 T0200
G97 S250 M03
G00 X0 Z5.0 T0202 M08
G01 Z-5.0 F0.07
W1.0
Z-40.0 F0.25
G00 Z5.0
Z-39.0
G01 Z-60.0
G00 Z10.0
X200.0 Z200.0 T0200
M01
N20 (Outside diameter stock removal)
G50 S1500 T0100
G96 S180 M03
G00 X94.0 Z5.0 T0101 M08
G01 Z-14.8 F0.27
G00 U2.0 Z0.5
G01 X28.0 F0.23
G00 X87.0 W1.0
G01 Z-14.8 F0.27
G00 U2.0 Z1.0
X80.5
G01 Z-14.1
G02 X81.9 Z-14.8 R0.7
G00 X100.5 W1.0
G01 Z-29.8
G00 U2.0 Z-1.0
G01 X60.5 F0.23

```

G00 X82.0 W1.0
Z-2.4
G01 X60.5
X72.9
G03 X80.5 Z-6.2 R3.8
G00 U2.0 Z5.0
X200.0 Z200.0 T0100
M01
N30 (Inside diameter stock removal)
G50 S1500 T0400
G96 S180 M03
G00 X34.5 Z3.0 T0404 M08
G01 Z-41.8 F0.27
G00 U-0.5 Z1.0
X39.5
G01 Z-15.0
X34.5 Z-24.3
G00 Z10.0
X200.0 Z200.0 T0400
M01
N40 (Out diameter finishing)
G50 S1800 T0500
G96 S200 M03
G00 X63.0 Z5.0 T0505 M08
Z0
G01 X38.0 F0.2
G00 X60.0 Z3.0
G42 Z1.0
G01 Z-2.5 F0.2
X74.0
G03 X80.0 Z-5.5 R3.0
G01 Z-13.5
G02 X83.0 Z-15.0 R1.5
G01 X100.0
Z-30.0
X102.0
G40 G00 U2.0 W1.0
G00 Z10.0
X200.0 Z200.0 T0500
M01
N50 (Inside diameter finishing)
G50 S1800 T0600
G96 S200 M03
G00 X40.0 Z5.0 T0606 M08
G41 Z1.0
G01 Z-15.0 F0.2
X35.0 Z-24.33
Z-42.0
X29.0
G40 G00 Z10.0
X200.0 Z200.0 T0600 M09
M30

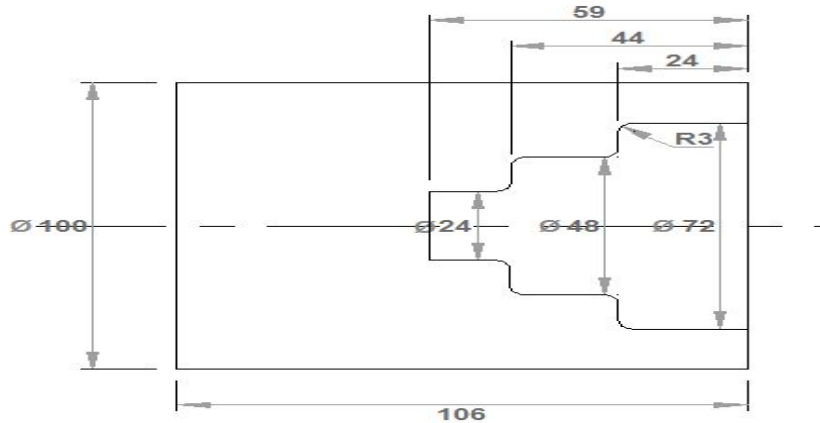
Fanuc Lathe Programming Example Using G70, G71, G74 for ID Machining

Fanuc lathe programming example which uses Fanuc canned cycle

- G71 Turning Cycle
- G70 Finish Cycle
- G74 Peck Drilling Cycle

This Fanuc lathe programming example illustrates when to use G71 Turning cycle, G70 Finish cycle, G74 Peck drilling cycle for ID machining (Inside machining, boring operations)

Fanuc Lathe Programming Example



Fanuc Lathe Programming Example Using G71, G70 for ID Machining

```
N10 G40 G00
N20 G99
N30 M5
N40 M9
N60 T0101
N70 G50 S3500
N80 G96 S0240 M4
N90 G00 X102. Z0.
N100 G01 X-2. F0.15 M7
N110 G00 X150. Z150.
N120 M9
```

N130 T0707
N140 G97 S0950 M3
N150 G00 Z3.
N160 G00 X0.
N170 G74 R1.0
N171 G74 X0.0 Z-59.0 Q12000 R0.0 F0.2
N350 G00 X150. Z150.
N360 T0505
N370 G50 S3500
N380 G96 S0200 M4
N390 G00 X23. Z2.
N400 G71 U3. R1
N410 G71 P420 Q530 U-0.5 W0.2 F0.3
N420 G41 G00 X72.
N440 G01 Z-21.
N450 G03 X66. Z-24. I-3. K0.
N460 G01 X54.
N470 G02 X48. Z-27. I0. K-3.
N480 G01 Z-41.
N490 G03 X42. Z-44. I-3. K0.
N500 G01 X30.
N510 G02 X24. Z-47. I0. K-3.
N520 G01 Z-59.
N530 G01 X23.
N540 G40
N550 G00 X150. Z150. F0.3
N560 T1111
N570 G50 S4500
N580 G96 S0380 M4
N590 G41 G00 X72. Z2.
N600 G70 P420 Q530
N610 G40
N620 G00 X23. Z2.
N630 G00 X200. Z150.
N640 M5
N650 M30

Tools & Operations

T0101 Turning Tool – Facing Operation

T0707 Tip Drill Tool – Drilling

T0505 Boring Bar – Internal machining (ID machining)

T1111 Boring Bar – Internal finish machining

CNC Lathe Programming Exercise

Fanuc G71 Turning Cycle, G74 Peck Drilling Cycle

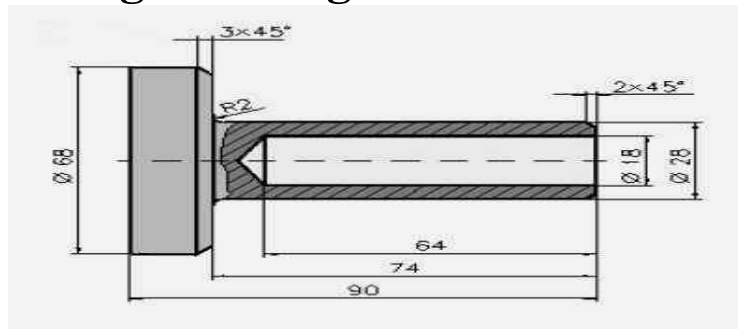
Drilling Cycle

CNC programming exercise for cnc lathe machinists who work on Fanuc cnc control (or similar cnc control).

This cnc programming exercise use

- Fanuc G71 Turning Cycle
- Fanuc G74 Peck Drilling Cycle

CNC Lathe Programming Exercise



CNC Lathe Programming Exercise Fanuc G71, G74 Cycles

N10 G40 G00

N20 G99

N60 T0101

N70 G50 S3500

N80 G96 S0240 M4

N90 G00 X72. Z0.1

N100 G01 X-1.6 F0.12 M7

N110 G00 X150. Z150.

N120 M5

N130 M9

N140 T0303

N150 G97 S2500 M3

N160 G00 X0. Z3.

N170 G01 Z-6. F0.1 M7

N180 G00 Z2.

N190 G00 X150. Z150.

N210 T0707 M7

N220 G97 S0884 M3

N230 G00 Z3.

N240 G00 X0.

N250 G74 R1.0

N260 G74 X0.0 Z-68.326 Q18000 F0.22
N380 G00 X200.
N400 G00 Z100.
N500 T0404 M7
N510 G50 S3500
N520 G96 S0240 M4
N530 G00 Z1.
N540 G00 X70.
N550 G71 U4. R1
N560 G71 P570 Q650 U0.6 W0.2 F0.35
N570 G42 G00 X24.
N580 G01 Z0.
N590 G01 X28. Z-2.
N600 G01 Z-72.
N610 G02 X32. Z-74. I2. K0.
N620 G01 X62.
N630 G01 X68. Z-77.
N640 G01 Z-90.
N650 G40
N660 G00 X150.
N680 G00 Z70.
N690 T0202 M7
N700 G50 S4500
N710 G96 S0380 M4
N720 G00 X16. Z3.
N730 G42 G01 Z0. F0.1
N740 G01 X24.
N750 G01 X28. Z-2.
N760 G01 Z-72.
N770 G02 X32. Z-74. I2. K0.
N780 G01 X62.
N790 G01 X68. Z-77.
N800 G01 Z-90.
N810 G40
N820 G00 X150. Z150.
N830 M5
N840 M9
N850 M30

Used Tools & Operations

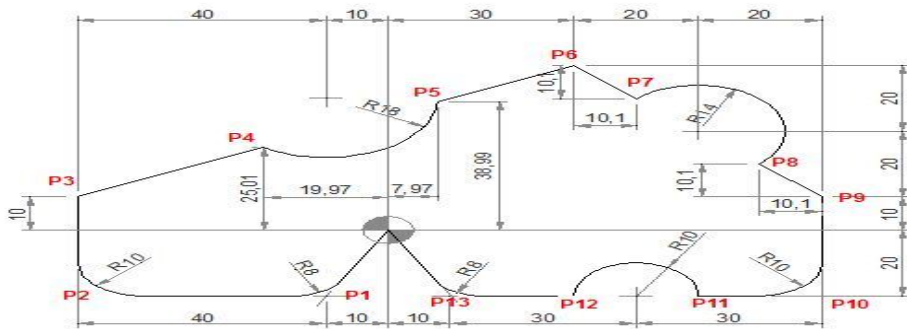
- T0101 Turning Tool – Rough Facing
- T0303 Center Drill – Center Drilling
- T0707 Twist Drill – Drilling
- T0404 Turning Tool – Rough Turning
- T0202 Turning Tool – Finish Contour Cutting

Chamfer Radius

CNC Mill Example Program G01 G02 G03 G90 G91

A cnc mill program for cnc machinists programmers, who have started to learning basic cnc programming techniques.

CNC Mill Example Program



CNC Program

```
N40 G90 G00 X0 Y0
N50 G01 X-10 Y-20 R8 (P1)
N60 G01 X-50 R10 (P2)
N70 Y10 (P3)
N80 X-19.97 Y25.01 (P4)
N90 G03 X7.97 Y38.99 R18 (P5)
N100 G01 X30 Y50 (P6)
N110 G91 X10.1 Y-10.1(P7)
N120 G90 G02 X59.9 Y20.1 R14 (P8)
N130 G01 X70 Y10 (P9)
N140 Y-20 R10(P10)
N150 X50(P11)
N160 G03 X30 R10 (P12)
N170 G01 X10 R8(P13)
N180 X0 Y0
```

G M S T Codes Explanation

Code	Description

G00	Rapid traverse
G01	Linear interpolation
G02	Circular interpolation CW
G03	Circular interpolation CCW
G90	Absolute command
G91	Increment command

Haas Corner Rounding and Chamfering Example G01 C R

Haas Corner Rounding and Chamfering

Haas CNC program example to show how Chamfer and Corner Radius can be programmed.

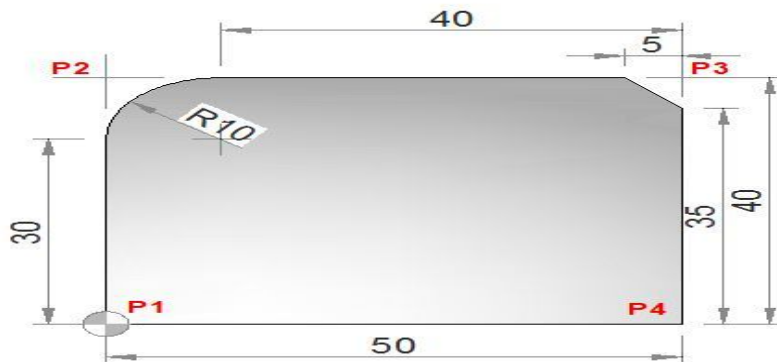
Haas Chamfering

To program Chamfer
 N10 G01 X20 Y30 ,C3

Haas Corner Rounding

To program Radius
 N10 G01 X20 Y30 ,R3

Haas Corner Rounding and Chamfering Example



Haas CNC Program

O1234 (Corner Rounding and Chamfering Example);

```
T1 M6;  
G00 G90 G54 X0. Y0. S3000 M3; (P1)  
G43 H01 Z0.1 M08;  
G01 Z-0.5 F20.;  
Y40. ,R10.:(P2)  
X50. ,C5.:(P3)  
Y0.:(P4)  
G00 Z0.1 M09;  
G53 G49 Z0.;  
G53 Y0.;  
M30;
```

Haas G M S T Codes

Code	Description
G00	Rapid Motion
G01	Linear Interpolation Motion
G43	Tool Length Compensation +
G49	G43/G44 Cancel
G53	Non-Modal Machine Coordinate Selection
G54	Select Work Coordinate System 1
G90	Incremental Programming
M3	Spindle On, Clockwise (S)
M6	Tool Change (T)
M08	Coolant On
M09	Coolant Off
M30	Program End and Reset
S	Spindle speed
T	Tool

Chamfer and Radius Program Example with G01

CNC programming example code to demonstrate, how to program a Chamfer and Radius (Corner Rounding) with G01 G-code.

Easy to program and understand that even a beginner level CNC machinist can understand and program such sample codes.

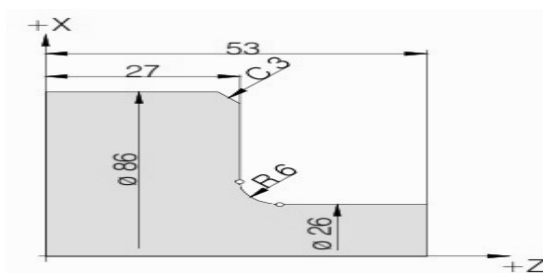
Mostly works on Fanuc and similar CNC controls.

No extra G-code or technique is required. Just have to put a “C” for chamfer and an “R” for Radius in a cnc program block with G01 G-code. Chamfer is at 45° (45 degrees).

For a brief description of how Chamfer and Corner Radius are programmed with G01 G code read following cnc programming article Chamfer and Radius Programming with G01 G code.

Another such program example is here G01 Chamfer and Corner Rounding a CNC Program Example

Chamfer and Radius with G01 G-Code



Chamfer and Radius Program Example with G01

```
N40 G01 X26 Z53  
N50 G01 X26 Z27 R6  
N60 G01 X86 Z27 C3  
N70 G01 X86 Z0
```

G01 Chamfer and Corner Rounding a CNC Program Example

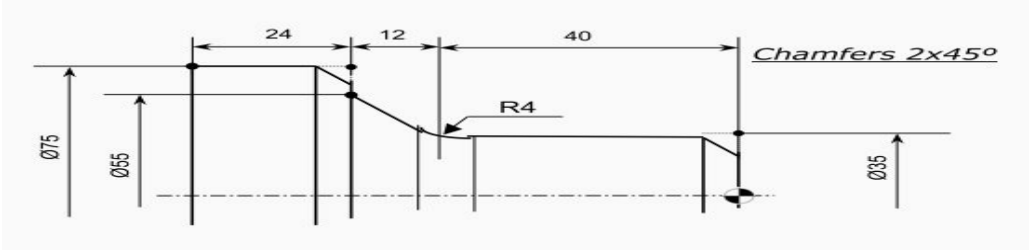
For a brief description of how Chamfer and Corner Radius are programmed with G01 G code read following cnc programming article Chamfer and Radius Programming with G01 G code.

You might read other cnc example program articles, these cnc programming articles are a good cnc programming reference for cnc programmers/cnc machinists working on the shop floor.

- CNC Programming for Beginners a Simple CNC Programming Example
- CNC Programming Example in Inch Simple CNC Lathe Program
- Lathe CNC Programming Example
- CNC Milling Machine Programming Example for Beginners
- CNC G02 Circular Interpolation Clockwise CNC Milling Sample Program

The following cnc program example shows how actually Chamfer and Radius are programmed with G01 in a cnc lathe program.

G01 Chamfer and Corner Rounding



CNC Programming Example of Chamfer and Corner Rounding with G01 G Code
N5

```
N6 G00 X0 Z3
N7 G01 Z0 F0.2
N8 X35 C2
N9 Z-40 R4
N10 X55 Z-52 F0.1
N11 X75 C2
N12 Z-76
N13 G00 X100 Z50
N14 .....
```

Haas G71 Example Program

Haas cnc lathe uses one-line syntax of G71 roughing canned cycle. This cnc program example shows the use of G71 turning cycle for ID roughing (Inside roughing).

You might like

- G71 Rough Turning Cycle One-line Format
- CNC Fanuc G71 Turning Cycle or Stock Removal Canned Cycle (Two-line format)
- Fanuc G70 G71 Rough and Finish Turning Cycle Program Example
- CNC Programming Example with Fanuc G71 Rough Turning Cycle and G70

In the below cnc programming example

1 – A boring bar is used for the whole the roughing operation with G71 Rough Turning Cycle.

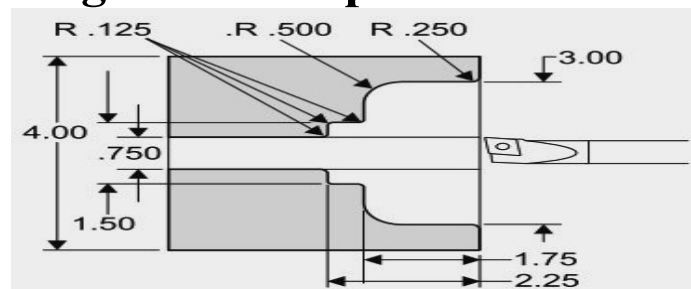
2 – Same boring bar is used for finish cut with G70 Finishing Cycle.

Example of using a Haas G71 for I.D. Roughing and Finishing.

Contents

- Haas CNC Program Example
- Haas CNC Program Explanation

Haas CNC Program Example



Haas G71 Example Program

O1136

N1 T101

N2 G97 S2000 M03

N3 G54 G00 X0.7 Z0.1 M08

N4 G71 P5 Q12 U-0.01 W0.005 D0.08 F0.01

N5 G00 X4.5

N6 G01 X3. R.25 F.005

N7 Z-1.75 R.5

N8 X1.5 R.125

N9 Z-2.25 R.125

N10 X.75 R.125

N11 Z-3.

N12 X0.73

N13 G70 P5 Q12

N14 M09

N15 G53 X0

G53 Z0

M30

Haas CNC Program Explanation

N1 – Tool 1 Offset 1

N3 – Rapid to start position

N4 – U is a minus for G71 I.D. Roughing

N5 – N5 is start of part path geometry defined by P6 in G71 line

N12 – N12 is end of part path geometry defined by Q12 in G71 line

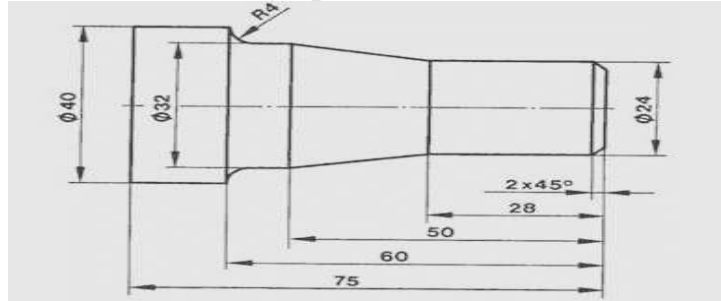
N13 – G70 Defines a finish pass for lines P5 through Q12

N15 – To send machine home for a tool change

Fanuc CNC Program Example

Here is a cnc program example for Fanuc cnc control. This is a very simple and easy cnc program example also shows

- Use of G02 Arc/Radius in Fanuc cnc program
- Use of Chamfer in Fanuc cnc program
- Use of G42/G40 Tool Nose Compensation
- Use of G92 Maximum Spindle Speed
- Use of G96 Constant Cutting Speed



Fanuc CNC Program Example

Fanuc CNC Program Code

N10 T2

```
N20 G92 S1200 M42
N30 G96 S150 M04
N40 G00 X-1 Z5 M08
N50 G01 Z0 G42 F0.2
N60 G01 X24 C2
N70 G01 Z-28
N80 G01 X32 Z-50
N90 G01 Z-56
N100 G02 X40 Z-60 R4
N110 G01 Z-75
N120 G01 X60 G40
N130 G00 X150 Z100
N140 M30
```

CNC Lathe Programming Example

A simple cnc lathe programming example. This cnc programming example will show how to program contours like *chamfer taper grooves and arc*.

This cnc program will use two tools for machining. The first tool used in this cnc programming example is a **turning tool** and the second one is a **grooving tool**.

Contents

- CNC Machining
- CNC Program

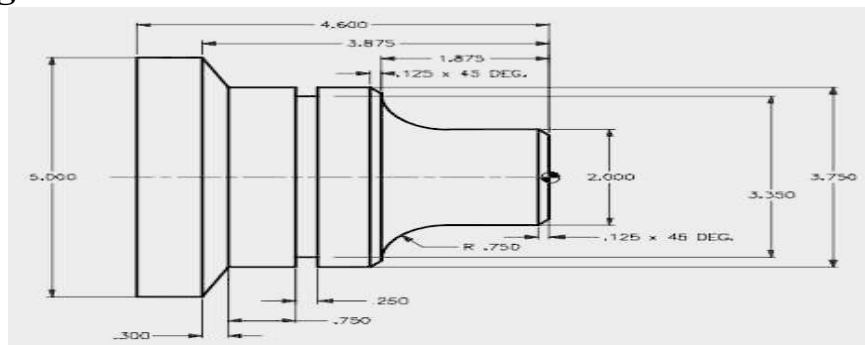
CNC Machining

The **turning tool** will first *face the component* then it will make a *chamfer*, after that there is *straight turning* and then there is *arc machining* the arc may be machined with **R (Radius of arc)** or arc can be machined with **I and K values of the arc**, at the top of arc there is a chamfer, then again straight turning, now there comes the *groove* but **we can't make grooves with turning tools because of their shape** so we will machine the groove with our next tool which is solely made for grooving operations, but at the time we are machining with turning tool so we will just skip this groove and will just machine in straight line, after that there is a taper and again a straight line to be machined.

Now with our **grooving tool** the groove machining task is just easy.

if the *grooving insert* is of the same size as the dimension of the groove then grooving is even more easy, we will make the groove in one go, but if the *grooving insert* is of smaller width than we have to take depth more than one time.

CNC Program



cnc lathe program example

N1 T01 D01 M491

N2 G00 X0 Z1

N3 G01 G96 G41 Z0 F2 S140

N4 G01 X2 CHF=0.125 F0.2

N5 G01 Z-1.125

N6 G02 X3.5 Z-1.875 CR=0.75

N7 G01 X3.75 CHF=0.125

N8 G01 Z-3.575

N9 G01 X5 Z-3.875

N10 G01 Z-4.6

N11 G00 X20 Z20 G40

N12 T02 D02 M491

N13 G00 G97 S500 X4 Z-2.825

N14 G01 X3.85 F1

N15 G01 X3.35 F0.15

N16 G01 X3.85 F0.5

N17 G00 X4

N18 G00 X20 Z20

N19 M30

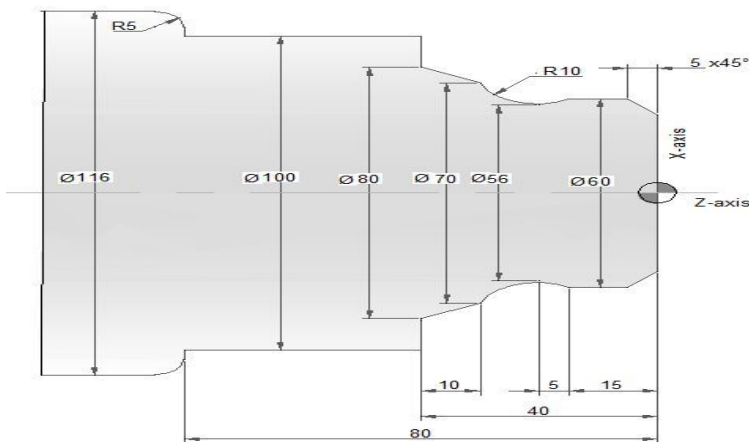
CNC Lathe Machine

CNC Turning Center Programming Example

Easy to understand CNC turning center/cnc lathe programming example for cnc machinists who work on cnc turning centers/cnc lathe machines.

This cnc programming example can be used as a cnc learning programming exercise for beginners level cnc programmers/machinists.

CNC Turning Center Programming Example

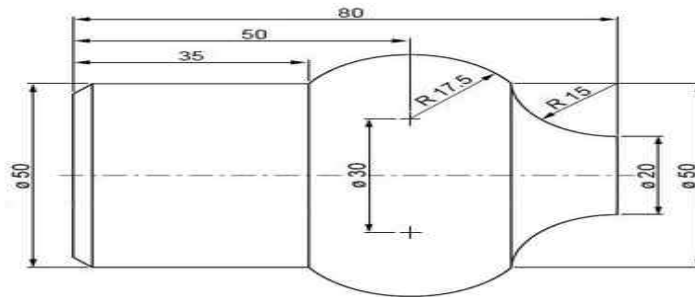


```
N10 T03 D03 M06
N20 G97 S900 M04
N30 G00 G42 X40 Z5
N40 G92 S3500
N50 G96 S250
N60 G01 X60 Z-5 F0.1
N70 Z-15
N80 X56 Z-20
N90 G02 X70 Z-30 R10
N100 G01 X80 Z-40
N110 X100
N120 Z-80
N130 X106
N140 G03 X116 Z-85 R5
N150 G01 X120
N160 G00 X150 Z100 G40 G97 S900 M05
N170 M30
```

G02 G03 Programming Example

CNC program example to show how two combined arc can be programmed with G02 G03 for a cnc lathe.

First arc is programmed using G02 R, but the second arc is programmed using G03 I K



G02 G03 Programming Example

G02 G03 Programming Example

```
N50 G00 X20 Z85  
N60 G96 S200  
N70 G01 Z80  
N80 G02 X50 Z65 R15  
N90 G03 X50 Z35 I-10 K-15  
N100 G00 X80  
N110 G97 S900 M05  
N120 M30
```

Fanuc G71 Turning Cycle

Fanuc G71 Turning Cycle

G71 turning cycle is used for rough-material removal from a cnc lathe component. G71 turning cycle makes large diameter cutting easy. Cutting can be done in simple straight line or a complex contour can also be machined very easily.

Through G71 turning cycle parameters cnc machinists can control

Depth of cut.
Retract height.
Finishing allowance in x-axis and z-axis.
Cycle cutting-feed, spindle speed.

Programming

G71 U... R...
G71 P... Q... U... W... F... S...

Parameters

First block

Parameter	Description
U	Depth of cut.
R	Retract height.

Second block

Parameter	Description
P	Contour start block number.
Q	Contour end block number.
U	Finishing allowance in x-axis.
W	Finishing allowance in z-axis.
F	Feedrate during G71 cycle.
S	Spindle speed during G71 cycle.

G71 Turning Cycle Overview

- G71 turning cycle cuts the whole contour repeatedly which is given in P Q blocks.
- Depth of every cut can be controlled by first-block U value.
- Second-block U W are the finishing allowances which can be given if you want to make a finish cut with G70 finishing cycle.
- F is cutting feed and S is spindle speed (given in second-block) which are used during G71 turning cycle.

Note – The F and S given inside P Q block will not be used during G71 turning cycle, they are used with G70 finishing cycle if later called.

G71 Turning Cycle Working

```
N60 G71 U10 R10  
N70 G71 P80 Q90 U3 W0 F0.25  
N80 G00 X60  
N90 G01 Z-75
```

When G71 turning cycle is run the whole operation will be done in following sequence,

First-cut

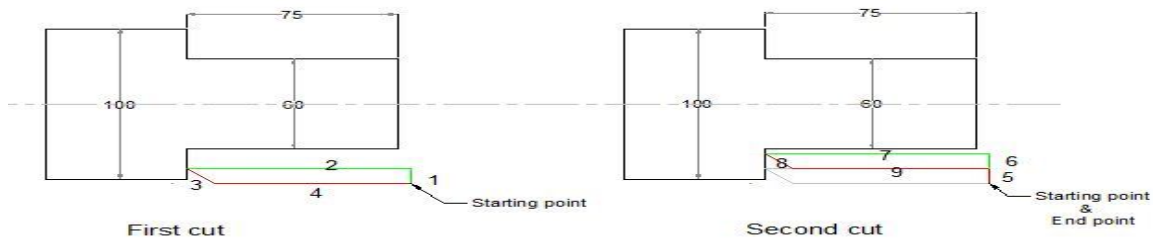
- 1 – Tool will move in x-axis U (depth of cut) deep with programmed feed from starting-point.
- 2 – Tool will travel with feed in z-axis (destination point in z-axis is given in P Q blocks)
- 3 – Tool rapidly retracts R amount in both x-axis and z-axis (at 45 degrees).
- 4 – Tool rapidly travel in z-axis to start-point

Later-cuts

- 5 – Tool rapidly moves to last cut depth.
- 6 – Tool moves with feed in x-axis U deep (first-block U depth of cut).
- 7 – Tool with feed moves in z-axis (destination point given in P Q blocks).
- 8 – Tool rapidly retracts in x-axis and z-axis R amount (45 degrees).
- 9 – Tool rapidly moves to start-point only in z-axis.

This whole sequence of operation keep on going, until the destination point in x-axis is met.

If finishing allowance is given tool will not make the exact diameter and length given in P Q blocks but will leave that much allowance, This finishing allowance can be later machined by calling G70 finishing cycle.



Fanuc G71 Turning Cycle
Fanuc G71 Turning Cycle

Fanuc G71 Example

Here is a cnc part-program which shows how G71 turning cycle can be used, this is the program for the drawing given above

```
N50 G00 X106 Z5 M3 S800
```

```
N60 G71 U10 R10
N70 G71 P80 Q90 U3 W0 F0.25
N80 G00 X60
N90 G01 Z-75
```

In this program G71 turning cycle will keep repeating the contour given inside P Q blocks shown below

```
N80 G00 X60
N90 G01 Z-75
```

These two cnc program blocks tell us that we want to remove material till X60 deep and in Z-75 in length.

The depth of cut is given in first-block U10 retract amount is also given R10. Finishing allowance in x-axis is U3 but there is no finishing allowance given in z-axis W0.

G70 Finishing Cycle

If you programmed G71 turning cycle with finishing allowances then that finish allowances can be removed with G70 finishing cycle.

G70 finishing cycle repeats the whole contour the G71 way, but in just one-cut removing the finishing allowances.

Why Use G70 Finishing Cycle

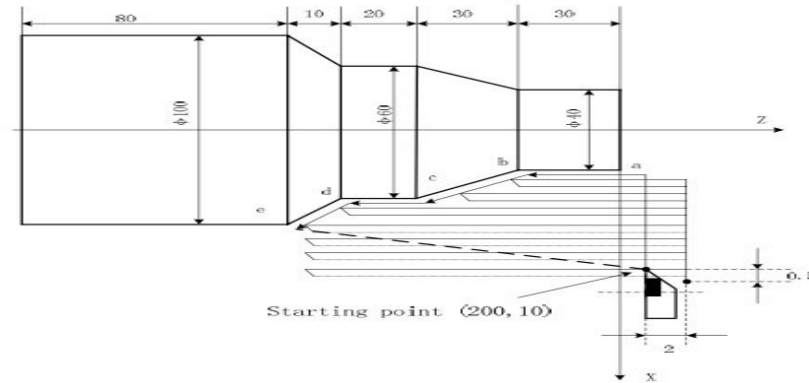
As material can be removed with G71 turning cycle, but if you want a different cutting-feed and spindle speed for the last cut, then it is recommended that you use G70 finishing cycle.

G70 finishing cycle use F and S values which are given inside P Q programmed blocks. (G71 use F S values which are given inside G71 second block.)

Fanuc G70 Example

```
N50 G00 X106 Z5 M3 S800
N60 G71 U10 R10
N70 G71 P80 Q90 U3 W0 F0.25
N80 G00 X60
N90 G01 Z-75 F0.15
N100 G00 X200 Z100
N110 G92 S1200
N120 T3 G96 S150 M03
N130 G00 X106 Z5
N140 G70 P80 Q90
N150 G00 X200 Z100
N160 M30
```

G70 G71 Example



G71 Rough Turning Cycle Example

```

O0004
G00 X200 Z10 M3 S800
G71 U2 R1 F200
G71 P80 Q120 U0.5 W0.2
N80 G00 X40 S1200
G01 Z-30 F100
X60 W-30
W-20
N120 X100 W-10
G70 P80 Q120
M30

```

Fanuc G71 G72 G70 Canned Cycle CNC Lathe Internal Machining Example (Boring & Facing)

Fanuc programming example which shows the use of multiple fanuc canned cycle in cnc programming, Following canned cycle are used in this cnc lathe programming example

- G71 Rough Turning Cycle
- G72 Facing Cycle
- G70 Finish Cycle

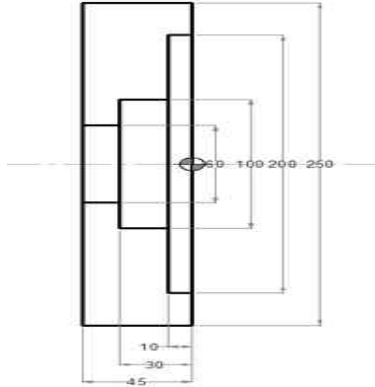
Contents

- Fanuc G71 for Boring Opertaion
- Fanuc Programming Example

Fanuc G71 for Boring Operation

This programming example also illustrates how G71 turning cycle can be used for internal machining (boring operation). So if you want to remove extra stock from inside of a component, you can use Fanuc G71 turning cycle for internal stock removal as well.

Fanuc Programming Example



Fanuc Canned Cycle Example

```
G28 U0 W0
G50 S1500
N1 T0101 M8 (OD FACING)
G96 S180 M3
G0 X255.0 Z5.0
G72 W2.0 R0.5
G72 P100 Q200 F0.25
N100 G0 Z0 G41
G1 X-2.0 F0.18
N200 G0 Z5.0
G40
X255.0
G28 U0 W0
N2 T0404 M8 (ID ROUGH)
G96 S180 M3
G0 X50.0 Z5.0
G71 U2.0 R0.5
G71 P500 Q600 U-0.5 W0.1 F0.25
N500 G0 X202.0 G41
G1 Z0 F0.12
X200.0
Z-10.0
X100.0
Z-30.0
X60.0
Z-45.0
N600 X50.0
G40
G0 Z5.0
G28 U0 W0
```

```

N3 T0505 M8 (ID FINISH)
G96 S220 M3
G0 X50.0 Z5.0
G70 P500 Q600
G0 X50.0 Z5.0
G28 U0 W0
M5 M9
M30

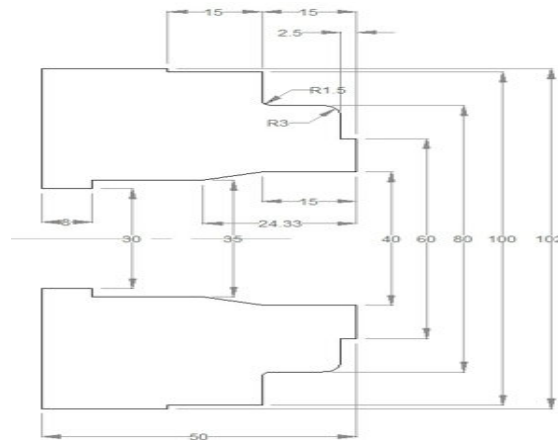
```

CNC Lathe Basic Programming Example ID/OD Turning/Boring Operations (No Canned Cycle Used)

A full CNC programming example with ID/OD (Turning/Boring operations) for cnc machinists who work on a cnc lathe machine. A must to learn/practice for those who are learning cnc programming.

The added benefit of this programming example is that no cnc lathe canned cycle is used in this programming example.

CNC Lathe Basic Programming Example (Turning Boring Operations)



CNC Lathe Example Turning Boring

```

O0000
N10 (ø30 DRILL)
G50 T0200
G97 S250 M03
G00 X0 Z5.0 T0202 M08

```

G01 Z-5.0 F0.07
 W1.0
 Z-40.0 F0.25
 G00 Z5.0
 Z-39.0
 G01 Z-60.0
 G00 Z10.0
 X200.0 Z200.0 T0200
 M01
 N20 (Outside diameter stock removal)
 G50 S1500 T0100
 G96 S180 M03
 G00 X94.0 Z5.0 T0101 M08
 G01 Z-14.8 F0.27
 G00 U2.0 Z0.5
 G01 X28.0 F0.23
 G00 X87.0 W1.0
 G01 Z-14.8 F0.27
 G00 U2.0 Z1.0
 X80.5
 G01 Z-14.1
 G02 X81.9 Z-14.8 R0.7
 G00 X100.5 W1.0
 G01 Z-29.8
 G00 U2.0 Z-1.0
 G01 X60.5 F0.23
 G00 X82.0 W1.0
 Z-2.4
 G01 X60.5
 X72.9
 G03 X80.5 Z-6.2 R3.8
 G00 U2.0 Z5.0
 X200.0 Z200.0 T0100
 M01
 N30 (Inside diameter stock removal)
 G50 S1500 T0400
 G96 S180 M03
 G00 X34.5 Z3.0 T0404 M08
 G01 Z-41.8 F0.27
 G00 U-0.5 Z1.0
 X39.5
 G01 Z-15.0
 X34.5 Z-24.3
 G00 Z10.0
 X200.0 Z200.0 T0400
 M01
 N40 (Out diameter finishing)
 G50 S1800 T0500
 G96 S200 M03
 G00 X63.0 Z5.0 T0505 M08
 Z0
 G01 X38.0 F0.2
 G00 X60.0 Z3.0
 G42 Z1.0

```
G01 Z-2.5 F0.2
X74.0
G03 X80.0 Z-5.5 R3.0
G01 Z-13.5
G02 X83.0 Z-15.0 R1.5
G01 X100.0
Z-30.0
X102.0
G40 G00 U2.0 W1.0
G00 Z10.0
X200.0 Z200.0 T0500
M01
N50 (Inside diameter finishing)
G50 S1800 T0600
G96 S200 M03
G00 X40.0 Z5.0 T0606 M08
G41 Z1.0
G01 Z-15.0 F0.2
X35.0 Z-24.33
Z-42.0
X29.0
G40 G00 Z10.0
X200.0 Z200.0 T0600 M09
M30
```

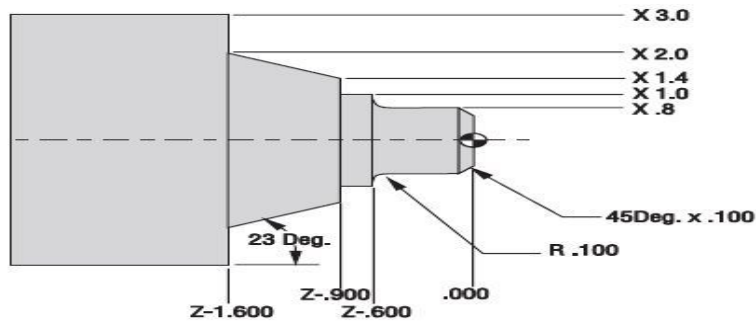
Haas G72 Type I Rough and G70 Finish Facing Cycle Program Example – Fanuc Compatible

Haas lathe programming example to illustrate the use and programming of Haas G72 Type I Rough Facing Cycle/ G70 Finish Cycle.

The above code will also work on cnc lathe machines with Fanuc cnc control with little or no change.

All the G-code / M-code which are used in this Haas lathe programming example are given below with description.

Haas G72 Type I Rough G70 Finish Facing Cycle Program Example



Haas G72 Type I Rough and G70 Finish Facing Program Example

```

O00096
N1 (G72 ROUGHING FACE)
N2 G53 G00 X0. Z0. T0 (Sending home for a tool change)
N3 T101 (O.D. TOOL x .031 TNR) (Select Tool 1 Offset 1)
N4 G50 S3000
N5 G97 S450 M03
N6 G54 G00 X3.1 Z0.1 M08 (Rapid to start point above part stock)
N7 G96 S370
N8 G72 P9 Q18 U0.01 W0.01 D0.06 F0.012 (G72 Rough Facing Cycle with TNC)
N9 G41 G00 Z-1.6 (Starting sequence number defined by P8 in G72 and G70)
N10 G01 X2. F0.008
N11 X1.4 Z-0.9
N12 X1.
N13 Z-0.6
N14 G03 X0.8 Z-0.5 R0.1
N15 G01 Z-0.1
N16 X0.6 Z0.
N17 X-0.062
N18 G40 G00 Z0.1 (End of part path geometry defined with P17 in G72 and G70)
N19 G97 S450 M09
N20 G53 G00 X0. Z0. T0 (Sending home for a tool change)
N21 M01 (Optional Stop)
N22 (G70 FINISHING FACE)
N23 G53 G00 X0. Z0. T0 (Sending home for a tool change)
N24 T202 (O.D. TOOL x .015 TNR) (Select Tool 2 Offset 2)
N25 G50 S3000
N26 G97 S450 M03
N27 G54 G00 X3.1 Z0.1 M08 (Rapid to start point)
N28 G96 S420
N29 G70 P9 Q18 (Finish Facing with G70 Finish Cycle)
N30 G97 S450 M09
N31 G53 G00 X0. Z0. T0 (Sending home for a tool change)
N32 M30 ( End of Program and Reset)

```

G-Codes

G00 Rapid traverse motion.

G01 Linear interpolation motion.

G03 Circular Interpolation – Counterclockwise.

G41 Tool Nose Compensation LEFT of the programmed path.

G40 Tool Nose Compensation CANCEL.
G50 Spindle Maximum RPM Limit.
G53 Machine Zero Positioning Coordinate Shift.
G54 Work Offset Positioning Coordinate #1 (Setting 56)
G72 End Face Stock Removal Cycle.
G70 Finishing Cycle.
G96 Constant Surface Speed On.
G97 Constant Surface Speed Cancel.

M-Codes

M01 Optional Program Stop.
M03 Starts the Spindle FORWARD.
M30 Program End and Reset to the beginning of program.

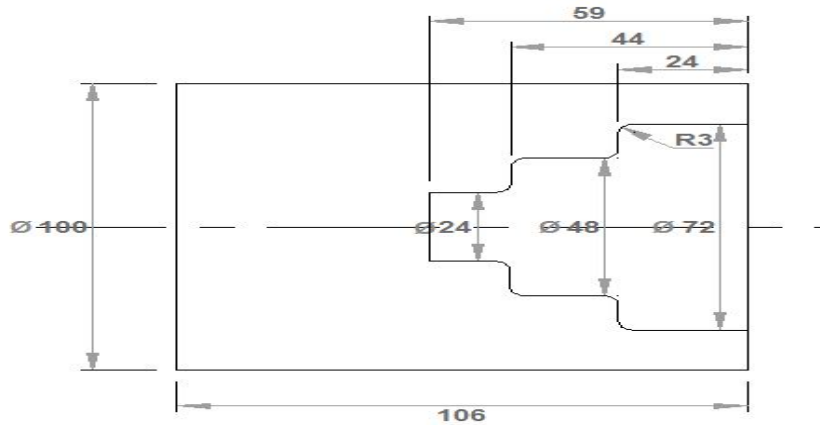
Fanuc Lathe Programming Example Using G70, G71, G74 for ID Machining

Fanuc lathe programming example which uses Fanuc canned cycle

- G71 Turning Cycle
- G70 Finish Cycle
- G74 Peck Drilling Cycle

This Fanuc lathe programming example illustrates when to use G71 Turning cycle, G70 Finish cycle, G74 Peck drilling cycle for ID machining (Inside machining, boring operations)

Fanuc Lathe Programming Example



Fanuc Lathe Programming Example Using G71, G70 for ID Machining

```

N10 G40 G00
N20 G99
N30 M5
N40 M9
N60 T0101
N70 G50 S3500
N80 G96 S0240 M4
N90 G00 X102. Z0.
N100 G01 X-2. F0.15 M7
N110 G00 X150. Z150.
N120 M9
N130 T0707
N140 G97 S0950 M3
N150 G00 Z3.
N160 G00 X0.
N170 G74 R1.0
N171 G74 X0.0 Z-59.0 Q12000 R0.0 F0.2
N350 G00 X150. Z150.
N360 T0505
N370 G50 S3500
N380 G96 S0200 M4
N390 G00 X23. Z2.
N400 G71 U3. R1
N410 G71 P420 Q530 U-0.5 W0.2 F0.3
N420 G41 G00 X72.
N440 G01 Z-21.
N450 G03 X66. Z-24. I-3. K0.
N460 G01 X54.
N470 G02 X48. Z-27. I0. K-3.
N480 G01 Z-41.
N490 G03 X42. Z-44. I-3. K0.
N500 G01 X30.
N510 G02 X24. Z-47. I0. K-3.
N520 G01 Z-59.
N530 G01 X23.
N540 G40
N550 G00 X150. Z150. F0.3
N560 T1111
N570 G50 S4500

```

N580 G96 S0380 M4
N590 G41 G00 X72. Z2.
N600 G70 P420 Q530
N610 G40
N620 G00 X23. Z2.
N630 G00 X200. Z150.
N640 M5
N650 M30

Tools & Operations

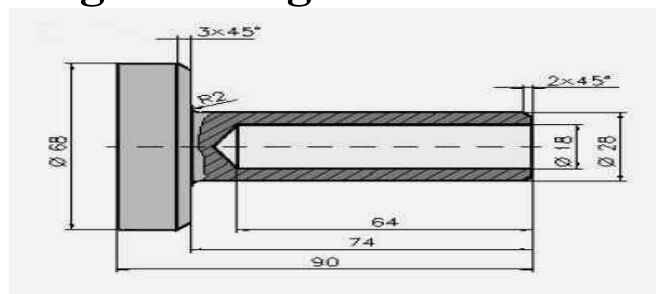
T0101 Turning Tool – Facing Operation
T0707 Tip Drill Tool – Drilling
T0505 Boring Bar – Internal machining (ID machining)
T1111 Boring Bar – Internal finish machining

CNC Lathe Programming Exercise Fanuc G71 Turning Cycle, G74 Peck Drilling Cycle

CNC programming exercise for cnc lathe machinists who work on Fanuc cnc control (or similar cnc control).

This cnc programming exercise use
Fanuc G71 Turning Cycle
Fanuc G74 Peck Drilling Cycle

CNC Lathe Programming Exercise



CNC Lathe Programming Exercise Fanuc G71, G74 Cycles

N10 G40 G00
N20 G99
N60 T0101

N70 G50 S3500
N80 G96 S0240 M4
N90 G00 X72. Z0.1
N100 G01 X-1.6 F0.12 M7
N110 G00 X150. Z150.
N120 M5
N130 M9
N140 T0303
N150 G97 S2500 M3
N160 G00 X0. Z3.
N170 G01 Z-6. F0.1 M7
N180 G00 Z2.
N190 G00 X150. Z150.
N210 T0707 M7
N220 G97 S0884 M3
N230 G00 Z3.
N240 G00 X0.
N250 G74 R1.0
N260 G74 X0.0 Z-68.326 Q18000 F0.22
N380 G00 X200.
N400 G00 Z100.
N500 T0404 M7
N510 G50 S3500
N520 G96 S0240 M4
N530 G00 Z1.
N540 G00 X70.
N550 G71 U4. R1
N560 G71 P570 Q650 U0.6 W0.2 F0.35
N570 G42 G00 X24.
N580 G01 Z0.
N590 G01 X28. Z-2.
N600 G01 Z-72.
N610 G02 X32. Z-74. I2. K0.
N620 G01 X62.
N630 G01 X68. Z-77.
N640 G01 Z-90.
N650 G40
N660 G00 X150.
N680 G00 Z70.
N690 T0202 M7
N700 G50 S4500
N710 G96 S0380 M4
N720 G00 X16. Z3.
N730 G42 G01 Z0. F0.1
N740 G01 X24.
N750 G01 X28. Z-2.
N760 G01 Z-72.
N770 G02 X32. Z-74. I2. K0.
N780 G01 X62.
N790 G01 X68. Z-77.
N800 G01 Z-90.
N810 G40
N820 G00 X150. Z150.
N830 M5

N840 M9
N850 M30

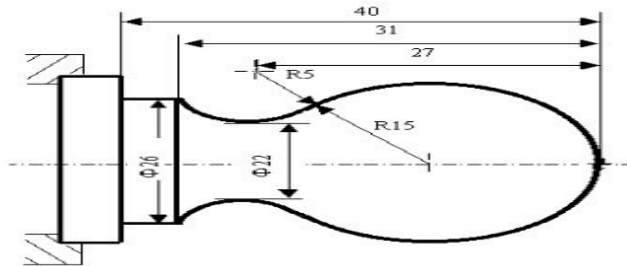
Used Tools & Operations

- T0101 Turning Tool – Rough Facing
- T0303 Center Drill – Center Drilling
- T0707 Twist Drill – Drilling
- T0404 Turning Tool – Rough Turning
- T0202 Turning Tool – Finish Contour Cutting

CNC Arc Programming G02 G03 Example

CNC arc programming example this cnc program shows how two arcs G03 G02 can be joint together.

CNC Arc Programming G02 G03 Example



CNC Arc Programming G02 G03 Example

```
O0001
N001 G0 X40 Z5 i¼> i¼^Rapid positioni¼%o
N002 M03 S200i¼> i¼^Start spindlei¼%o
N003 G01 X0 Z0 F900i¼> (Approach workpiece)
N005 G03 U24 W-24 R15i¼> i¼^Cut R15 arci¼%o
N006 G02 X26 Z-31 R5i¼>i¼^Cut R5 arci¼%o
N007 G01 Z-40i¼> (CutÑ,,26)
N008 X40 Z5i¼> (Return to starting point)
N009 M30i¼>i¼^End of programi¼%o
```

G71 Rough Turning Cycle Example Code – CNC Lathe Programming

G71 Turning Cycle

G71 Rough Turning Cycle example code. This cnc program code works on Fanuc and similar cnc controls.

G71 Rough Turning Cycle parameters explanation

CNC Fanuc G71 Turning Cycle or Stock Removal Canned Cycle

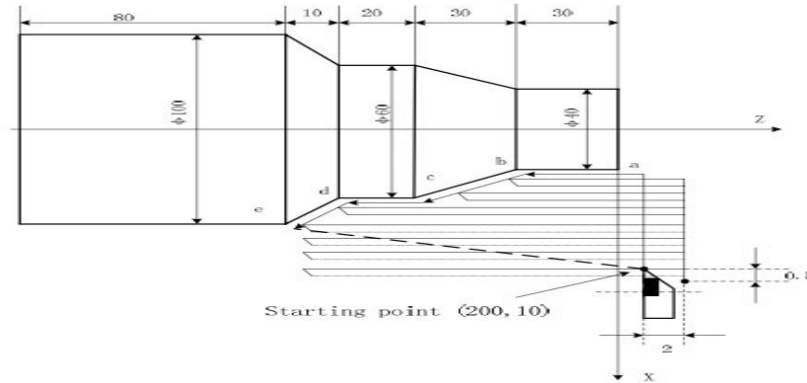
Other G71 Rough Turning Cycle Examples

Fanuc G70 G71 Rough and Finish Turning Cycle Program Example

G71 Longitudinal Roughing Cycle Mazak CNC Basic Programming Example

CNC Programming Example with Fanuc G71 Rough Turning Cycle and G70

Example Program



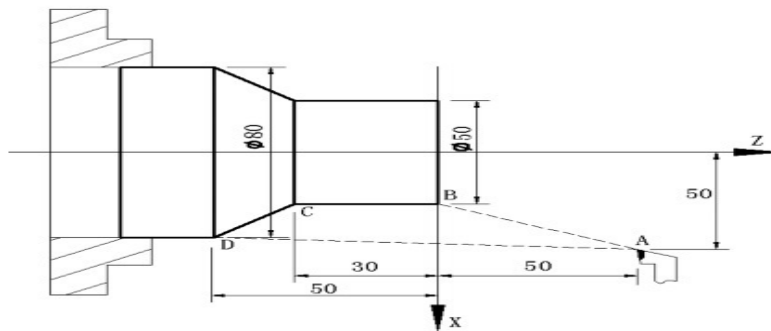
G71 Rough Turning Cycle Example

```
O0004
G00 X200 Z10 M3 S800
G71 U2 R1 F200
G71 P80 Q120 U0.5 W0.2
N80 G00 X40 S1200
G01 Z-30 F100
X60 W-30
W-20
N120 X100 W-10
G70 P80 Q120
M30
```

CNC Lathe Simple G Code Example – G code Programming for Beginners

CNC lathe simple g code example for CNC machinists/CNC programmers who just started their journey to learn CNC programming.

Simple G Code Example CNC Lathe



CNC Lathe Simple G Code Example

```
O0001  
N5 M12  
N10 T0101  
N15 G0 X100 Z50  
N20 M3 S600  
N25 M8  
N30 G1 X50 Z0 F600  
N40 W-30 F200  
N50 X80 W-20 F150  
N60 G0 X100 Z50  
N70 T0100  
N80 M5  
N90 M9  
N100 M13  
N110 M30  
N120 %
```

Code Explanation

N5 Clamping workpiece
N10 Changing No.1 tool and executing its offset
N15 Rapidly positioning to A point
N20 Starting the spindle with 600 r/min

N25 Cooling ON
N30 Approaching B point with 600mm/min
N40 Cutting from B point to C point
N50 Cutting from C point to D point
N60 Rapidly retracting to A point
N70 Canceling the tool offset
N80 Stopping the spindle
N90 Cooling OFF
N100 Releasing workpiece
N110 End of program, spindle stopping and Cooling OFF

Fanuc Circular Interpolation G02 G03 Code Example

Fanuc Circular Interpolation

Fanuc Circular Interpolation commands (G02, G03 G Codes) are used to move a tool along a circular arc.

G02 – Circular Interpolation Clockwise.

G03 – Circular Interpolation Anti-Clockwise.

How to Program Circular Interpolation?

On CNC lathe machines with Fanuc control Clockwise Circular Interpolation (Arc) can be programmed in following ways.

N10 G02 X Z R

or

N10 G02 X Z I K

Explanation

N10 G02 X Z R

X – Arc end-point in X-axis

Z – Arc end-point in Z-axis

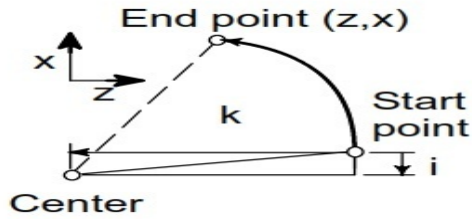
R – Arc Radius.

N10 G02 X Z I K

X- Arc end-point in X-axis.

Z – Arc end-point in Z-axis.

I – Distance from arc-start-point to arc-center-point in X-axis.
 K – Distance from arc-start-point to arc-center-point in Z-axis.

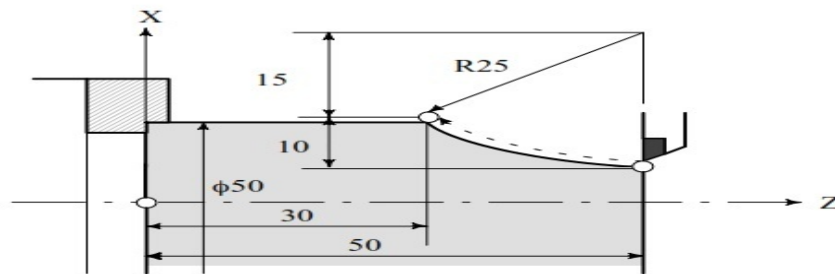


Fanuc Circular Interpolation G02

Fanuc Circular Interpolation G02 G Code Example

The following Fanuc circular interpolation program example is programmed in multiple different ways.

CNC programmers/machinists can use G02/G03 X Z R method or G02/G03 X Z I K method if control allows.



Fanuc Circular Interpolation G02 G Code Example

(Diameter programming)

```
G02 X50 Z30 I25 F0.3
G02 U20 W-20 I25 F0.3
G02 X50 Z30 R25 F0.3
G02 U20 W-20 R25 F0.3
```

Newbie CNC Machinists a Basic CNC Canned Cycle Example G90

Basic CNC Canned Cycle Example Program for Newbie CNC Machinists

A basic cnc programming exercise for newbie cnc machinists / newbie cnc programmers who have just to start their professional career.
 This cnc program example shows the use of G90 rough turning cycle in a very extensive way.
 Newbie cnc machinists must learn such cnc programming techniques and then master them.

Basic CNC Program

A program example for Fanuc cnc control.

This cnc program uses

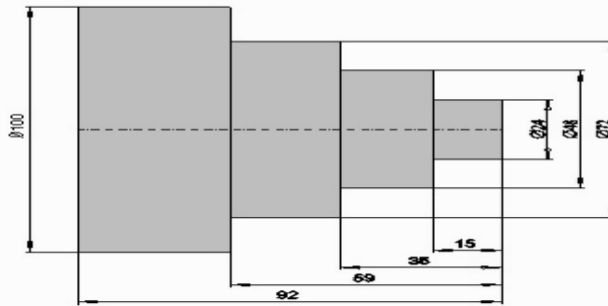
- G90 Rough Turning Cycle.
- G50 Spindle Speed Limitation.

Two tools are used to complete machining of component.

First tool makes roughing cuts with G90 rough turning cycle.

Second tool makes the finish cut.

Basic CNC Canned Cycle Example Program Code



Basic CNC Canned Cycle Program

```

N10 G40 G00
N20 G99
N30 M5
N40 M9
N50 G54
N60 T0101
N70 G50 S3500
N80 G96 S0240 M4
N90 G00 X102. Z0.1
N100 G01 X-2. F0.15 M7
N110 G00 X101. Z2.
N115 G90 X90.0 Z-58.8 F0.3
N120 X80.0
N125 X73.0
N126 G00 X73
N127 G90 X63 Z-34.8 F0.3
N128 X55.
N130 X49.
  
```

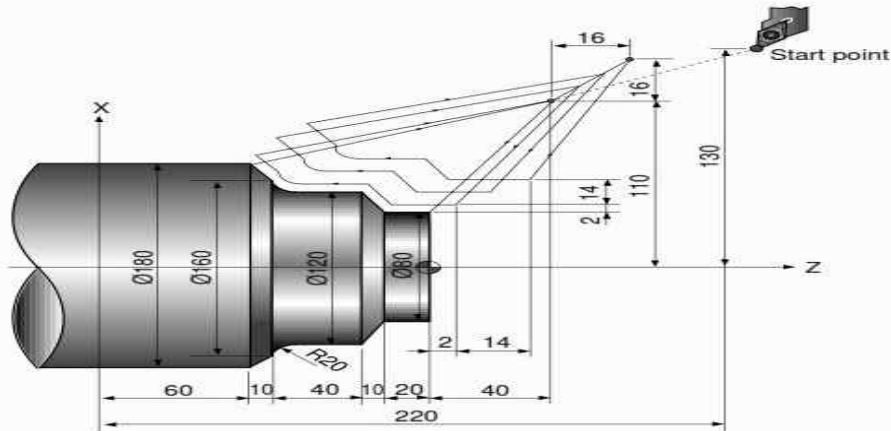
```
N140 G00 X49.  
N150 G90 X39. Z-14.8 F0.3  
N160 X31.  
N170 X25.  
N430 G00 X150. Z100.  
N440 T0202  
N450 G50 S4500  
N460 G96 S0380 M4  
N470 G00 X-1.6 Z2.  
N480 G42 G01 Z0. F0.1  
N490 G01 X24.  
N500 G01 Z-15.  
N510 G01 X48.  
N520 G01 Z-35.  
N530 G01 X72.  
N540 G01 Z-59.  
N550 G01 X102.  
N560 G40  
N570 G00 X150. Z100.  
N580 M5  
N590 M9  
N600 M30
```

Fanuc G73 Pattern Repeating Cycle CNC Program Example Code

CNC programming example for Fanuc G73 pattern repeating cycle.
Fanuc G73 Pattern Repeating Cycle has already been described here
CNC Fanuc G73 Pattern Repeating Cycle
You might like other Fanuc G73 pattern repeating cycle examples
CNC Fanuc G73 Pattern Repeating Cycle CNC Program Example
Fanuc G73 Pattern Repeating Canned Cycle Basic CNC Sample Program

Fanuc G73 Pattern Repeating Cycle Programming Example

This cnc program example also shows how cnc machinists can use ‘W’ instead of ‘Z’ for z-axis movements.



Fanuc G73 Pattern Repeating Cycle Program Example

```

N010 G00 X260.0 Z80.0
N011 G00 X220.0 Z40.0
N012 G73 U14.0 W14.0 R3
N013 G73 P014 Q020 U4.0 W2.0 F0.3 S0180
N014 G00 G42 X80.0 Z2.0
N015 G01 W-20.0 F0.15 S0600
N016 X120.0 W-10.0
N017 W-20.0 S0400
N018 G02 X160.0 W-20.0 R20.0
N019 G01 X180.0 W-10.0 S0280
N020 G40
N021 G70 P014 Q020
N022 G00 X260.0 Z80.0
N023 M30

```

Fanuc G73 Pattern Repeating Canned Cycle Basic CNC Sample Program

Fanuc G73 pattern repeating cycle helps cnc machinists to program/maintain/debug rough material removal programs easy.

Other Fanuc canned cycle like
 G71 Longitudinal cutting cycle or
 G72 Facing Cycle

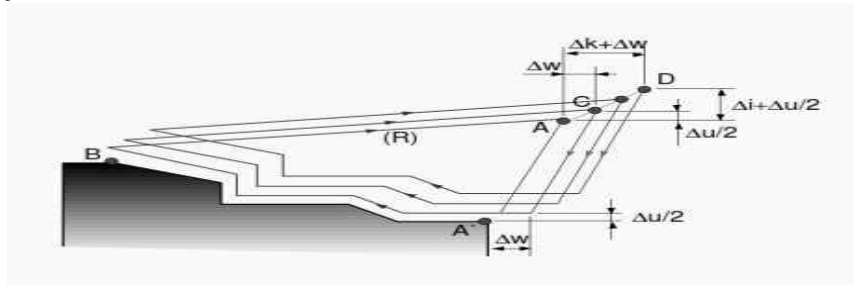
removes the rough material by taking straight cuts.

But Fanuc G73 Pattern repeating cycle removes the rough material by taking cuts along the contour.

So from the first cut G73 pattern repeating cycle cuts the same shape as the finish component.

Fanuc G73 Pattern Repeating Cycle Format

G73 U W R
G73 P Q U W F



Fanuc G73 Pattern Repeating Canned Cycle

G73 First CNC Program Block

U – Escape distance and direction in X axis (radius value). This is the amount of material which will be cut in x-axis.

W – Escape distance and direction in Z axis. This is the amount of material which will be cut in z-axis.

R – Number of divisions. The number the contour will be repeated.

G73 Second CNC Program Block.

P – Start sequence No.

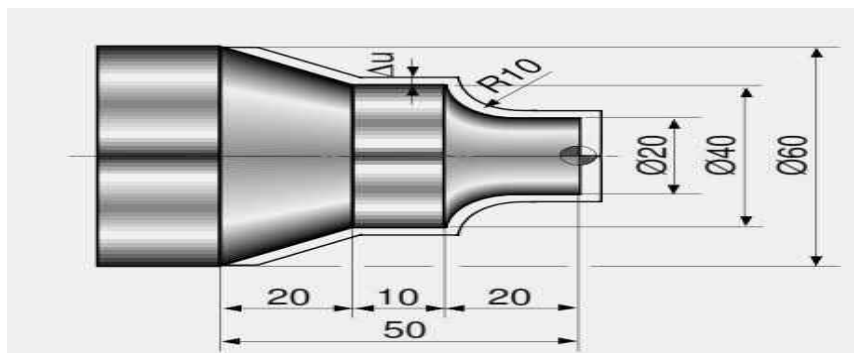
Q – End sequence No.

U – Finishing allowance in x-axis.

W – Finishing allowance in z-axis.

F – Cutting Feedrate (during G73 canned cycle).

Fanuc G73 Pattern Repeating Cycle Program Example



Fanuc G73 Pattern Repeating Canned Cycle Basic CNC Sample Program

```
N10 G50 S2000 T0300
G96 S200 M03
G00 X35.0 Z5.0 T0303
Z0
G01 X-1.6 F0.2
G00 X70.0 Z10.0
G73 U3.0 W2.0 R2
G73 P12 Q16 U0.5 W0.1 F0.25
N12 G00 G42 X20.0 Z2.0
G01 Z-10.0 F0.15
G02 X40.0 Z-20.0 R10.0
G01 Z-30.0
X60.0 Z-50.0
N16 G40 U1.0
G70 P12 Q16
G00 X200.0 Z200.0 T0300
M30
```

G28 Reference Point Return – CNC Lathe

G28 Reference Point Return

G28 reference point return G-code is used to approach the reference point via an intermediate position.

The intermediate position can be specified absolute X, Z or relative U, W.

What is reference point read CNC Zero Return or Reference Point Return

During G28 reference point return command machine,

first reaches intermediate-point rapidly (G00),

then it moves rapidly to reference-point position.

Programming

G28 X(U) Z(W)

X, Z absolute intermediate point position.

U, W incremental intermediate point position.

Examples

For following examples suppose actual reference-point position on your machine is X330 and Z529.

Example 1

G28 U0 W0

the machine will directly move to reference-point (as no intermediate point is given).

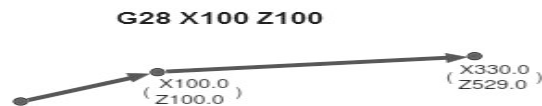


G28 Reference Point Return G28 U0 W0

Example 2

G28 X100 Z100

the machine will first move to intermediate position X100 Z100 and then will move to reference-point.



G71 Longitudinal Roughing Cycle

Mazak CNC Basic Programming

Example

CNC basic programming example for CNC programmers/machinists who work on Mazak / FANUC CNC controls. This CNC code example shows the use of G71 Longitudinal Roughing Cycle.

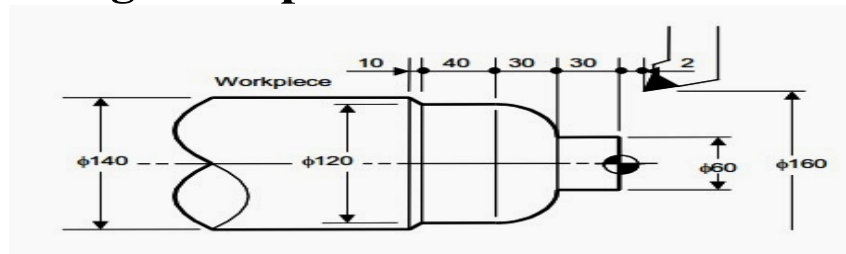
G71 Turning Cycle

For complete parameter explanation for G71 canned cycle read CNC Fanuc G71 Turning Cycle or Stock Removal Canned Cycle you might like G71 Rough Turning Cycle One-line Format.

Other such programming examples are here

CNC Programming Example with Fanuc G71 Rough Turning Cycle and G70 Fanuc G70 G71 Rough and Finish Turning Cycle Program Example

Programming Example



G71 Longitudinal Roughing Cycle Mazak

```
N001 G00 G96 G98;  
N002 G28 U0 W0;  
N003 X160. Z2.;  
N010 G71 U5. R1.;  
N011 G71 P012 Q016 U4. W2. F150 S150 M03;  
N012 G00 X60. S200;  
N013 G01 Z-30. F100;  
N014 G03 X120. Z-60. R30.;  
N015 G01 W-40.;  
N016 X140.W-10.;  
N017 G70 P012 Q016;  
N018 G28 U0 W0 M05;  
N019 M30;
```

Fanuc G72 Facing Canned Cycle Example Program

Here is another cnc programming example which shows the use of Fanuc G72 Facing Cycle.

Fanuc G72 facing cycle comes in two format for different fanuc cnc controls.

Fanuc G72 Facing Cycle One-line Format

If your Fanuc control use One-line format canned cycle then here is full description of Fanuc G72 Facing Cycle

G72 Facing Cycle One-line Format for Fanuc 10T 11T 15T

Here is an example of One-line (single-line) format

Sample Program Example Fanuc G72 Facing Cycle Single-line-format

Fanuc G72 Facing Cycle Two-line Format

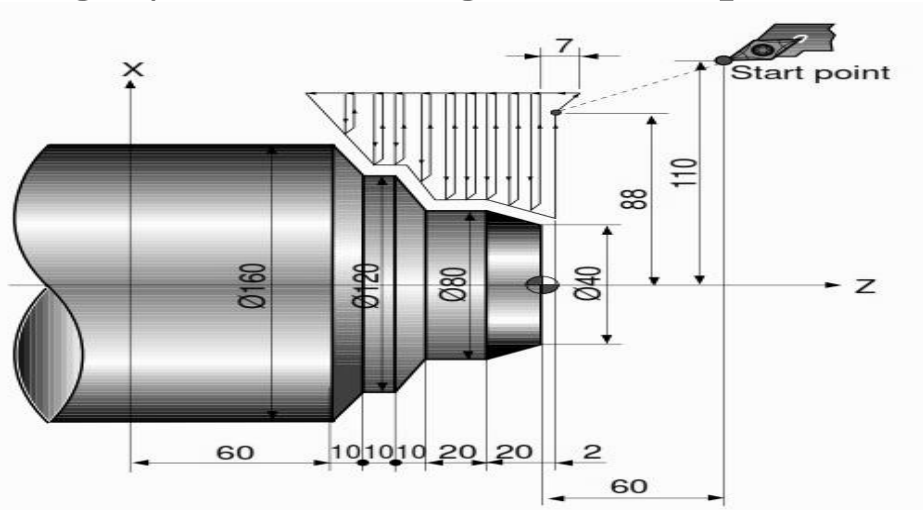
For Double-line format canned cycle of G72 facing cycle read

Fanuc G72 Facing Cycle – Stock Removal in Facing

CNC Fanuc G72 Canned Cycle Facing

Below is the cnc program example for two-line format of Fanuc G72 facing canned cycle.

G72 Facing Cycle CNC Program Example



Fanuc G72 Facing Canned Cycle Example Program

```
N010 G00 X220.0 Z60.0
N011 G00 X176.0 Z2.0
N012 G72 W7.0 R1.0
N013 G72 P014 Q021 U4.0 W2.0 F0.3 S550
N014 G00 G41 Z-70.0 S700
N015 X160.0
N016 G01 X120.0 Z-60.0 F0.15
N017 W10.0
N018 X80.0 W10.0
N019 W20.0
N020 X36.0 W22.0
N021 G40
N022 G70 P014 Q021
N023 G00 X220.0 Z60.0
N024 M30
```

Same program example for G72 one-line format read Sample Program Example
Fanuc G72 Facing Cycle Single-line-format

Sample Program Example Fanuc G72 Facing Cycle Single-line-format

As canned cycle for cnc machines looks difficult to learn and program for
beginner level cnc machinists, but they pay off in long run.

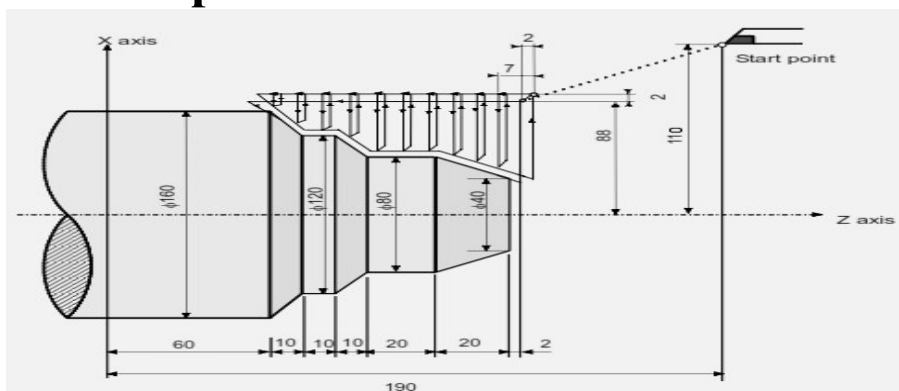
As canned cycle makes cnc machinists life easy because with the change of few
parameter you can entirely change their behaviors according to your machining
needs.

Here is a cnc program example code which demonstrate the use and programming
of Fanuc G72 Facing Cycle.

Fanuc G72 Facing Cycle

Fanuc G72 facing cycle is used to remove extra material from component face by
making cuts in traverse (X-axis) direction.

Program Example



Sample Program Fanuc G72 Facing Cycle

```
N011 G50 X220.0 Z190.0  
N012 G00 X176.0 Z132.0  
N013 G72 P014 Q019 U4.0 W2.0 D7000 F0.3 S550  
N014 G00 Z56.0 S700  
N015 G01 X120.0 W14.0 F0.15
```

```
N016 W10.0
N017 X80.0 W10.0
N018 W20.0
N019 X36.0 W22.0
N020 G70 P014 Q019
```

As above code shows machining contour is from N014 till N019 as given with G72 P014 Q019

The G72 canned cycle will take D7000 (7mm) cut each.

Finishing allowance in X-axis is U4.0 and in Z-axis is W0.2

During the canned cycle the tool feed will be F0.3 and spindle speed will be S550 as given in block N013

After canned cycle completion G70 finishing cycle is called with the same block numbers as P014 Q019.

The feed and spindle speed will act normally as normal cnc program,

As spindle speed will be S700 given in block N014

and tool feed will be F0.15 as given in block N015 (this feed will active throughout G70 as no other feed-rate is given in later blocks)

Chamfer and Radius Program Example with G01

CNC programming example code to demonstrate, how to program a Chamfer and Radius (Corner Rounding) with G01 G-code.

Easy to program and understand that even a beginner level CNC machinist can understand and program such sample codes.

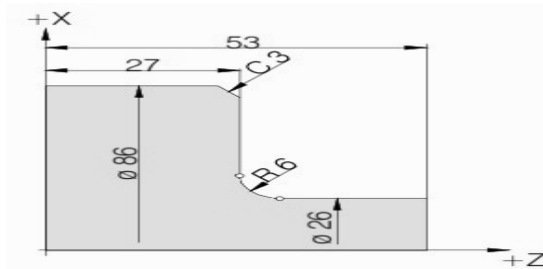
Mostly works on Fanuc and similar CNC controls.

No extra G-code or technique is required. Just have to put a “C” for chamfer and an “R” for Radius in a cnc program block with G01 G-code. Chamfer is at 45° (45 degrees).

For a brief description of how Chamfer and Corner Radius are programmed with G01 G code read following cnc programming article Chamfer and Radius Programming with G01 G code.

Another such program example is here G01 Chamfer and Corner Rounding a CNC Program Example

Chamfer and Radius with G01 G-Code



Chamfer and Radius Program Example with G01

```
N40 G01 X26 Z53  
N50 G01 X26 Z27 R6  
N60 G01 X86 Z27 C3  
N70 G01 X86 Z0
```

Fanuc G94 Facing Cycle CNC Example Program

Fanuc G94 One Pass Facing Cycle

Fanuc G94 G code is used for rough facing.

Fanuc G94 facing cycle is used for simple facing (one-pass facing) however multiple passes are possible by specifying the Z-axis location of additional passes.

Related: Fanuc G72 Facing Canned Cycle

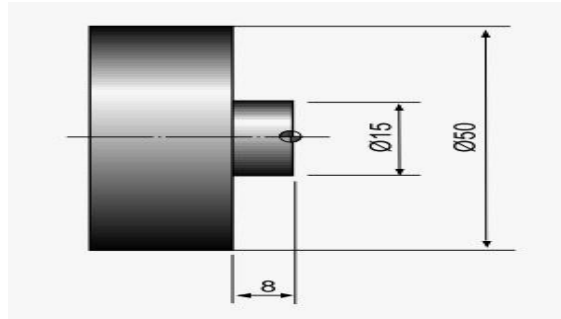
Fanuc G94 facing cycle is very simple to program and use. G94 G code parameters are explained below,

G94 X... Z...

X: End point in X-axis.

Z: End point in Z-axis.

CNC Program Code using Fanuc G94 Facing Cycle



Fanuc G94 Facing Cycle CNC Example Program

```

N10 G50 S2500
N20 G96 S180 M03
N30 T0100
N40 G00 X55.0 Z2.0 T0101
N50 G94 X15.0 Z-2.0 F0.2
N60 Z-4.0
N70 Z-6.0
N80 Z-8.0
N90 G00 X200.0 Z200.0 T0100
N95 M30

```

Explanation of CNC Program Code using Fanuc G94 Facing Cycle

N40 : shows the starting position of the tool.

N50 : the values of x and z with G94 facing cycle are the destination values for the facing cut.

N60 : G94 is a modal G code. “Modal” G code meaning that they stay in effect until they are cancelled or replaced by a contradictory G code. It means G90 turning cycle remains active until another motion command is given like G00, G01 etc

But as you can see the cnc program block N60 only show the value for z-axis, so it means that the G94 facing cycle will continue to work and now G94 facing cycle will make the second facing cut, whose x-axis value will remain the same but the z-axis value will now be -4.0

N70 : the third facing cut will be taken at -6.0

N80 : fourth cut will be taken at -8.0

N90 : this cnc program block show a contradictory G-code for G94 which is G00, so this mean that the Fanuc G94 facing cycle has been ended. The tool will move rapidly to x200 z200.

Internal Threading on Fanuc 21i 18i 16i with G76 Threading Cycle

CNC program for the internal threading with G76 threading cycle on fanuc controls 21i/18i/16i.

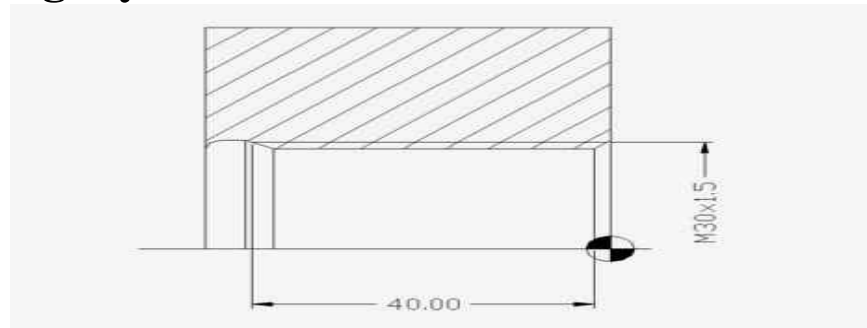
For an example of external threading with G76 threading cycle read External Thread Cutting with G76 Threading Cycle on Fanuc 21i 18i 16i CNC
Fanuc 21i/18i/16i use two block format of G76 threading cycle.

Related: G76 Threading Cycle One Line Format for Fanuc 10/11/15T

Fanuc G76 threading cycle has multiple parameters making it difficult to remember, but at the same time those multiple parameters of G76 thread cycle give the cnc programmer/cnc machinist multiple options to control thread cutting, some are listed below.

G76 thread cutting cycle allow cnc machinist to control number of idle cuts, thread run-out, infeed angle.

CNC Program of Internal Threading with G76 Threading Cycle



Internal Threading on Fanuc 21i 18i 16i with G76 Threading Cycle

```
N17 T101  
N18 G54  
N19 G97 S800 M3  
N20 G0 X25 Z6 M8  
N21 G76 P010060 Q100 R0.02  
N22 G76 X30 Z-40 P919 Q250 F1.5  
N23 G0 X150 Z100
```

External Thread Cutting with G76 Threading Cycle on Fanuc 21i 18i 16i CNC

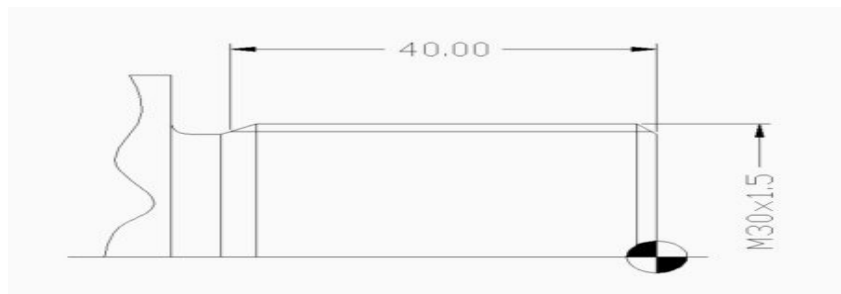
Fanuc Series 21i 18i 16i Thread Cutting with G76 Threading Cycle

Fanuc G76 cycle can be used for external and internal thread cutting. Fanuc series 21i 18i 16i cnc controls use two program block format of G76 threading cycle.

For internal theading example of threading with G76 on Fanuc 21i/18i/16i read Internal Threading on Fanuc 21i 18i 16i with G76 Threading Cycle

Related: G76 Threading Cycle One Line Format for Fanuc 10/11/15T

CNC Program of External Metric Thread Cutting with G76



External Thread Cutting with G76 Threading Cycle on Fanuc 21i 18i 16i CNC

```
N17 T101  
N18 G54  
N19 G97 S800 M3
```

```
N20 G0 X32 Z6 M8
N21 G76 P010060 Q100 R0.02
N22 G76 X28.161 Z-50 P919 Q250 F1.5
N23 G0 X150 Z100
```

G01 Chamfer and Corner Rounding a CNC Program Example

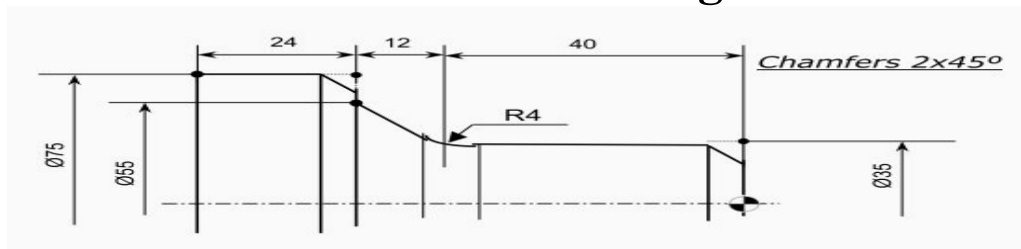
For a brief description of how Chamfer and Corner Radius are programmed with G01 G code read following cnc programming article Chamfer and Radius Programming with G01 G code.

You might read other cnc example program articles, these cnc programming articles are a good cnc programming reference for cnc programmers/cnc machinists working on the shop floor.

- CNC Programming for Beginners a Simple CNC Programming Example
- CNC Programming Example in Inch Simple CNC Lathe Program
- Lathe CNC Programming Example
- CNC Milling Machine Programming Example for Beginners
- CNC G02 Circular Interpolation Clockwise CNC Milling Sample Program

The following cnc program example shows how actually Chamfer and Radius are programmed with G01 in a cnc lathe program.

G01 Chamfer and Corner Rounding



CNC Programming Example of Chamfer and Corner Rounding with G01 G Code

```
N5 .....
N6 G00 X0 Z3
N7 G01 Z0 F0.2
N8 X35 C2
N9 Z-40 R4
N10 X55 Z-52 F0.1
N11 X75 C2
```

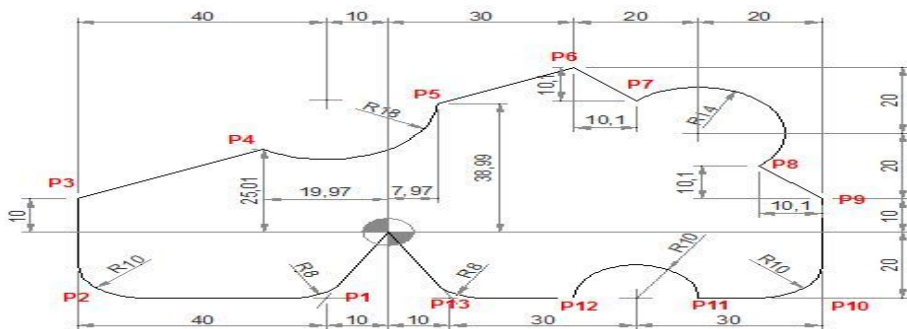
N12 Z-76
N13 G00 X100 Z50
N14

CNC Milling Machine

CNC Mill Example Program G01 G02 G03 G90 G91

A cnc mill program for cnc machinists programmers, who have started to learning basic cnc programming techniques.

CNC Mill Example Program



CNC Program

```
N40 G90 G00 X0 Y0
N50 G01 X-10 Y-20 R8 (P1)
N60 G01 X-50 R10 (P2)
N70 Y10 (P3)
N80 X-19.97 Y25.01 (P4)
N90 G03 X7.97 Y38.99 R18 (P5)
N100 G01 X30 Y50 (P6)
N110 G91 X10.1 Y-10.1(P7)
N120 G90 G02 X59.9 Y20.1 R14 (P8)
N130 G01 X70 Y10 (P9)
N140 Y-20 R10(P10)
N150 X50 (P11)
N160 G03 X30 R10 (P12)
N170 G01 X10 R8 (P13)
N180 X0 Y0
```

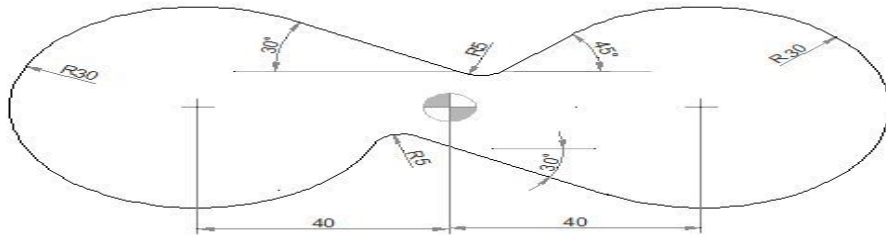
G M S T Codes Explanation

Code	Description

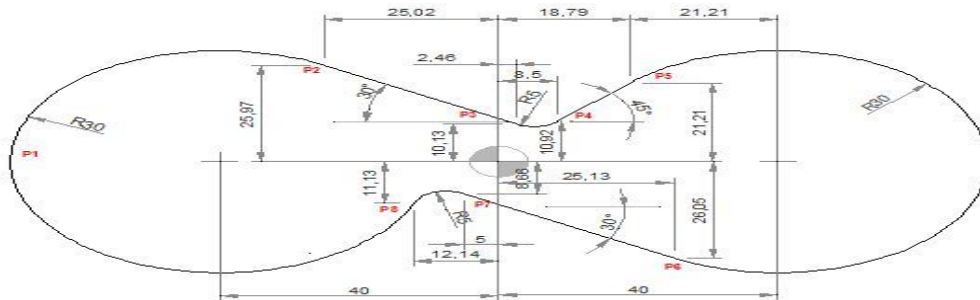
G00	Rapid traverse
G01	Linear interpolation
G02	Circular interpolation CW
G03	Circular interpolation CCW
G90	Absolute command
G91	Increment command

CNC Mill Programming Example

CNC mill program example for cnc machinists.



CNC Mill Programming Example



CNC Program

```

G0 X-60 Y0
G1 X-70 (P1)
G2 X-25.02 Y25.97 R30(P2)
G1 X2.46 Y10.13 (P3)
G3 X8.5 Y10.92 R5(P4)
G1 X18.79 Y21.21 (P5)
G2 X25.13 Y-26.05 I21.21 J-21.21 (P6)
G1 X-5 Y-8.66(P7)

```

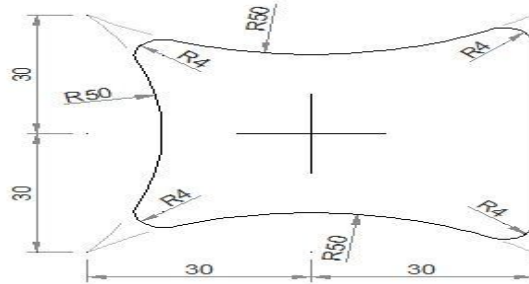
G3 X-12.14 Y-11.13 R5(P8)
 G2 X-70 Y0 R30 (P1)
 G1 X-60

G M S T Codes Explanation

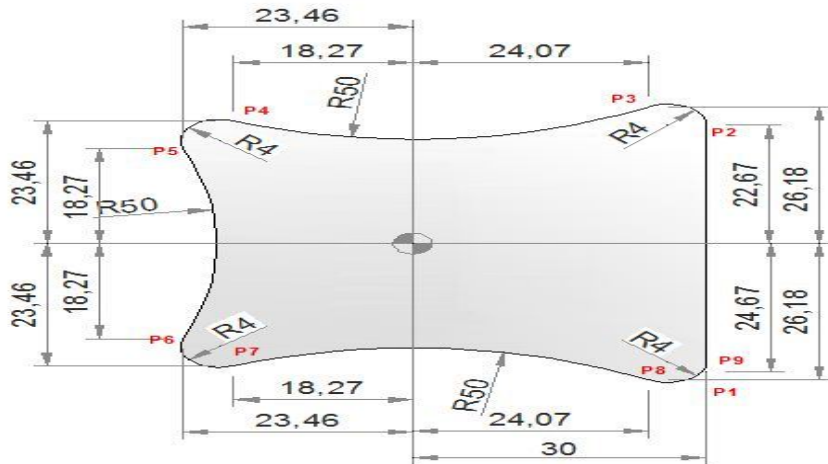
Code	Description
G00	Rapid traverse
G01	Linear interpolation
G02	Circular interpolation CW
G03	Circular interpolation CCW

G02 G03 Example CNC Mill

G02 G03 Circular interpolation CNC mill example program.



G02 G03 Example CNC Mill



CNC Part Program

```
G0 X30 Y-30 (P1)
G1 Y22.67 (P2)
G3 X24.07 Y26.18 R4 (P3)
G2 X-18.27 Y23.46 R50 (P4)
G3 X-23.46 Y18.27 R4(P5)
G2 X-23.46 Y-18.27 R50 (P6)
G3 X-18.27 Y-23.46 R4 (P7)
G2 X24.07 Y-26.18 R50 (P8)
G3 X30 Y-24.67 R4 (P9)
G1 X33
```

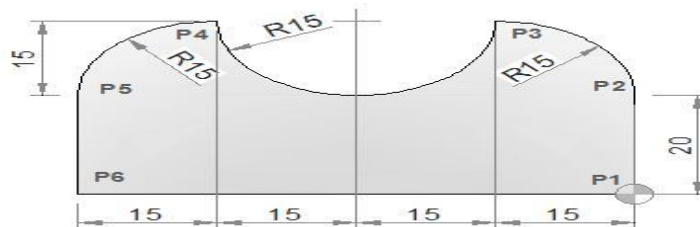
G M S T Codes Explanation

Code	Description
G0	Rapid traverse
G1	Linear interpolation
G2	Circular interpolation CW
G3	Circular interpolation CCW
M30	End of program (Reset)

Multiple Arc CNC Mill Program G2 G3 I J

CNC milling machine program which combines/joins multiple arcs.

Multiple Arc CNC Mill Program G2 G3 I J



CNC Part Program

N10 M6 T1 G43 H1 M3
 N15 S500 F120
 N20 G0 X0 Y0 (P1)
 N25 G1 Y20(P2)
 N30 G3 X-15 Y35 I-15 J0 (P3)
 N35 G2 X-45 Y35 I-15 J0 (P4)
 N40 G3 X-60 Y20 I0 J-15 (P5)
 N45 G1 Y0 (P6)
 N50 G1 X0 (P1)
 N55 M30

G M S T Codes Explanation

Code	Description
G0	Rapid traverse
G1	Linear interpolation
G2	Circular interpolation CW
G3	Circular interpolation CCW
G43	Tool length compensation + direction
M3	Spindle start forward CW
M6	Tool change
M30	End of program (Reset)
T	Tool
S	Speed
F	Feed

Haas Corner Rounding and Chamfering Example G01 C R

Haas Corner Rounding and Chamfering

Haas CNC program example to show how Chamfer and Corner Radius can be programmed.

Haas Chamfering

To program Chamfer

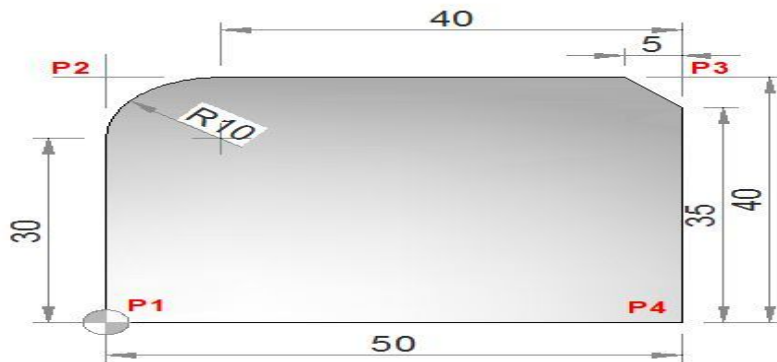
N10 G01 X20 Y30 ,C3

Haas Corner Rounding

To program Radius

N10 G01 X20 Y30 ,R3

Haas Corner Rounding and Chamfering Example



Haas CNC Program

O1234 (Corner Rounding and Chamfering Example);

```
T1 M6;
G00 G90 G54 X0. Y0. S3000 M3; (P1)
G43 H01 Z0.1 M08;
G01 Z-0.5 F20.;
Y40. ,R10.:(P2)
X50. ,C5.:(P3)
Y0.:(P4)
G00 Z0.1 M09;
G53 G49 Z0.;
G53 Y0.;
M30;
```

Haas G M S T Codes

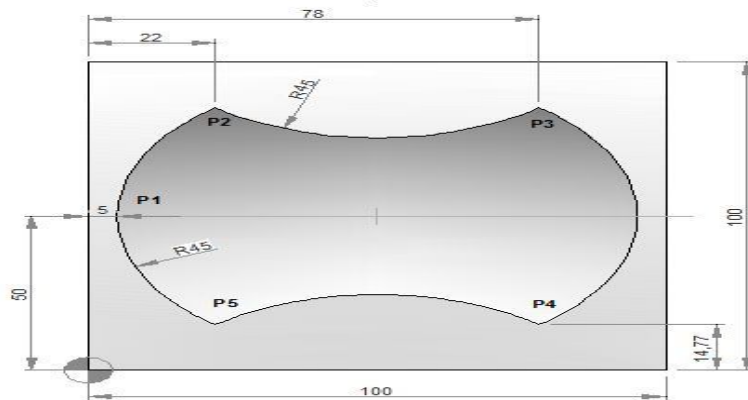
Code	Description
G00	Rapid Motion
G01	Linear Interpolation Motion
G43	Tool Length Compensation +
G49	G43/G44 Cancel
G53	Non-Modal Machine Coordinate Selection
G54	Select Work Coordinate System I
G90	Incremental Programming
M3	Spindle On, Clockwise (S)
M6	Tool Change (T)
M08	Coolant On
M09	Coolant Off
M30	Program End and Reset
S	Spindle speed
T	Tool

CNC Mill Subprogram Example

Joining Multiple Arcs G02 G03 G41

CNC milling program to describe how two or more radii can be joint together in a cnc mill program.

CNC Mill Subprogram Example



CNC Part Program

```
N10 T1 H1 M6 G43 M3
N20 F150 S250
N30 G0 X-21 Y50 Z0.5
N40 G0 Z0
N50 M98 P040050
N60 G49
N70 G0 Z50
N80 M30
```

Subprogram

```
O0050
N10 F160 S400
N20 G0 Z-2.5 G91
N30 G1 G90 X5 Y50 G41 (P1)
N40 G2 X22 Y85.23 I45 J0 (P2)
N50 G3 X78 Y85.23 R45 (P3)
N60 G2 X78 Y14.77 R45 (P4)
N70 G3 X22 Y14.77 R45 (P5)
N80 G2 X5 Y50 R45 (P1)
N90 G0 G40 X-21
N100 M99
```

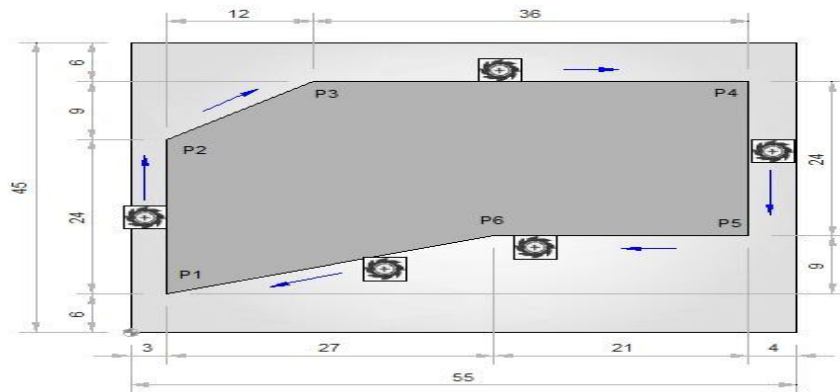
G M S T Codes Explanation

Code	Description
G00	Rapid traverse
G01	Linear interpolation
G02	Circular interpolation CW
G03	Circular interpolation CCW
G40	Cutter compensation cancel
G41	Tool nose radius compensation left
G43	Tool length compensation + direction
G49	Tool length compensation cancel
G90	Absolute command
G91	Increment command
M03	Spindle start forward CW
M06	Tool change
M30	End of program (Reset)
M98	Subprogram call
M99	End of subprogram
T	Tool
S	Speed
F	Feed

CNC Mill Program G91 G41 G43

CNC milling program examples shows the use of G91 G41 G43 G-codes.

CNC Mill Program G91 G41 G43



CNC Part Program

```
N05 G54  
N10 M6 T1 G43 H1 M3  
N15 S500 F120  
N20 G0 X-22 Y-22  
N25 Z-3  
N30 G1 X3 Y6 G41 H2 (P1)  
N35 G91 X0 Y24(P2)  
N40 X12 Y9(P3)  
N45 X36 (P4)  
N50 Y-24 (P5)  
N55 X-21 (P6)  
N60 G90 X3 Y6 (P1)  
N65 G0 X-21 G40
```

G M S T Codes Explanation

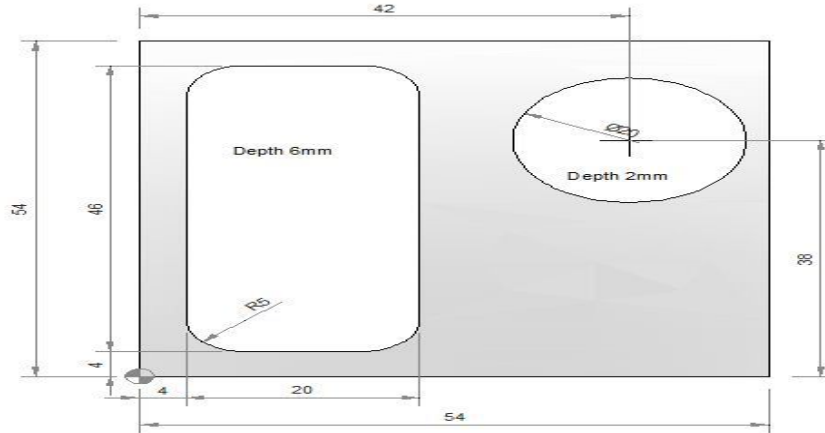
Code	Description
G00	Rapid traverse
G01	Linear interpolation
G40	Cutter compensation cancel
G41	Tool nose radius compensation left
G43	Tool length compensation + direction
G54	Workpiece coordinate system 1 selection
G90	Absolute command
G91	Incremental command
M06	Tool change
T	Tool
S	Speed
F	Feed

CNC Pocket Milling Program Example – Peck Milling

CNC milling program example which shows how a cnc program can be made to machine Pockets on a cnc mill.

This program example uses Peck milling to cut material to machine a rectangular and one round pocket.

CNC Pocket Milling Program Example



Main Program

Milling cutter diameter: 10mm

```
N05 G55
N10 M6 T2 H3 G43 M3
N15 S1000 F60
N20 G0 X9 Y9 Z1
N25 G1 Z0
N30 M98 P030035
N35 G0 Z1 G90
N40 X42 Y38
N45 G1 Z-2 F30
N50 X47 F300
N55 G3 X47 Y38 I-5 J0
N60 G0 Z100
N65 G49
N70 M30
```

Subprogram

```
O0035
N05 G1 Z-2 G91 F30
N10 X10 F100
N15 Y36
N20 X-10
N25 Y-36
N30 M99
```

Explanation

Although this cnc mill program is self explanatory

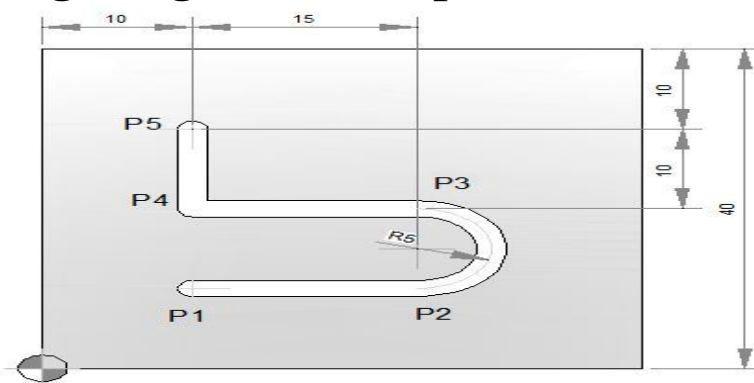
```
M98 P030035
```

this code mean call Subprogram No. 0035 three times.

CNC Milling Program Example G03 G90 G91

CNC milling program example, which illustrates how a cnc program can be easily converted from G90 Absolute program mode to G91 Incremental program mode.

CNC Milling Program Example



G90 Absolute Program	G91 Incremental Program
<pre> N10 T1 M6 M3 G43 N15 S2000 F80 N20 G0 X10 Y10 Z1 (P1) N25 F30 N30 G1 Z-2 N35 X25 F80 (P2) N40 G3 X25 Y20 I0 J5 (P3) N45 G1 X10 (P4) N50 Y30 (P5) N55 G0 Z100 G49 N60 M30 </pre>	

Fanuc G72.1 G72.2 Figure Copy Program Example (Bolt Hole Circle)

This CNC program example shows how both G72.1 and G72.2 figure copy functions can call one-another in one part program, read Programming

Notes below carefully.

Programming

Fanuc G72.1 Rotational Copy

Using G72.1 Rotational Copy G-code a figure specified by a subprogram can be repeatedly produced with **Rotational movement**.

Read G72.1 definition with program example Fanuc G72.1 Rotational Copy (Figure Copy Function CNC Mill)

Fanuc G72.2 Linear Copy

Using G72.2 Linear Copy G-code a figure specified by a subprogram can be repeatedly produced with **Linear movement**.

Read G72.2 definition with program example Fanuc G72.2 Linear Copy (Figure Copy Function CNC Mill)

Fanuc G81 Drilling Cycle

G81 drilling cycle is used for simple drilling/spot drilling operations.

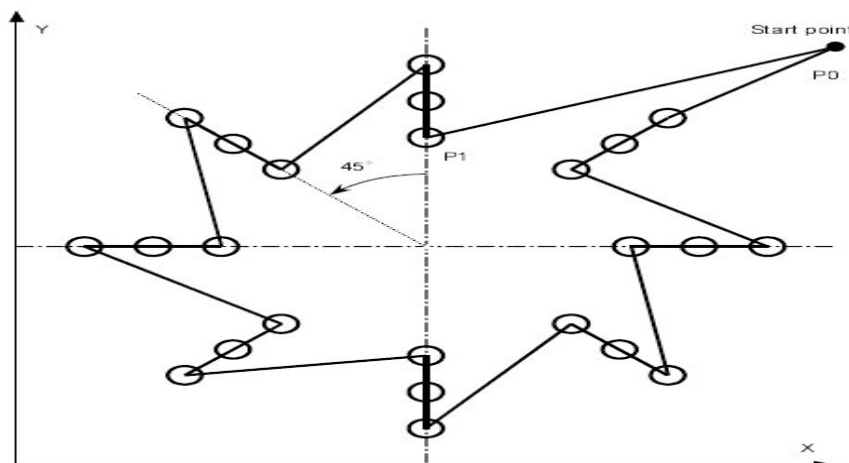
Read G81 drilling cycle definition with program examples Fanuc G81 Drilling Cycle

Programming Notes

Specifying two or more commands to copy a figure

- G72.1 cannot be specified more than once in a subprogram for making a rotational copy (If this is attempted, alarm PS0900 will occur).
- G72.2 cannot be specified more than once in a subprogram for making a linear copy (If this is attempted, alarm PS0901 will occur).
- In a subprogram that specifies rotational copy, however, linear copy can be specified. Similarly, in a subprogram that specifies linear copy, rotational copy can be specified.

Fanuc G71.2 G72.2 Program Example



Main program

```
O4000 ;  
N10 G90 G00 G17 X240. Y230. Z100. ; (P0)  
N20 G72.1 P4100 X120. Y120. L8 R45. ;  
N30 G80 G00 X240. Y230. ; (P0)  
N40 M30 ;
```

Sub program_ Rotation copy_ (G72.1)

```
O4100 N100 G72.2 P4200 I0 J20. L3 ;  
N200 M99 ;
```

Sub program_ Linear copy (G72.2_)

```
O4200 N110 G90 G81 X120. Y180. R60. Z10. F200. ; (P1)  
N210 M99 ;
```

Fanuc G72.2 Linear Copy (Figure Copy Function CNC Mill)

Fanuc G72.2 Linear Copy

Using G72.2 Linear Copy G-code a figure specified by a subprogram can be repeatedly produced with **Linear** movement.

Programming

G72.2 P... L... I... J...

Parameters

Parameter	Description
P	Subprogram number
L	Number of times the operation is repeated
I	Shift along X-axis
J	Shift along Y-axis

G-Code Data

Modal/Non-Modal	G-Code Group
Non-Modal	00

Programming Notes

Notes

- In the G72.2 block, addresses other than P, L, I and J are ignored.
- P, I and J must always be specified.
- If L is not specified, the figure is copied once.

- For shifts (I, J), specify increments. The n-th geometric shift is equal to the specified shift times (n – 1).

First block of the subprogram

Always specify a move command in the first block of a subprogram that performs a linear copy. If the first block contains only the program number such as O00001234; and does not have a move command, movement may stop at the start point of the figure made by the n-th (n = 1,2, 3, ...) copying.

Example of an incorrect program

```
O00001234 ;
G00 G90 X100.0 Y200.0 ;
____;
____;
M99 ;
```

Example of a correct program

```
O00001000 G00 G90 X100.0 Y200.0 ;
____;
____;
M99 ;
```

Limitation

Specifying two or more commands to copy a figure

G72.2 cannot be specified more than once in a subprogram for making a linear copy (If this is attempted, alarm PS0901 will occur).

In a subprogram that specifies linear copy, however, rotational copy (G72.1) can be specified. Similarly, in a subprogram that specifies rotational copy, linear copy can be specified.

Commands that must not be specified

Within a program that performs a linear copy, the following must not be specified:

Command for changing the selected plane (G17 to G19)

Command for specifying polar coordinates (G16)

Reference position return command(G28)

Axis switching

Coordinate system rotation (G68)

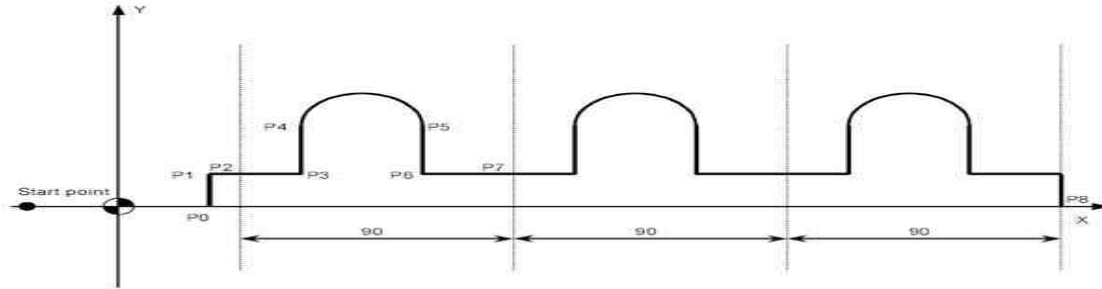
scaling (G51)

programmable mirror image (G51.1)

Single block

Single-block stops are not performed in a block with G721.1 or G72.2.

G72.2 Programming Example



Main program

```
O3000 ;
N10 G90 G00 X-30. Y0 ;
N20 X0 ;
N30 G01 G17 G41 X30. D01 F100 ; (P0)
N40 Y20. ; (P1)
N50 X40. ; (P2)
N60 G72.2 P3100 L3 I90.0 J0 ;
N70 G90 X310. Y0 ; (P8)
N80 X0 ;
N90 G40 G00 X-30.0 ;
N100 M30 ;
```

Sub program

```
O3100 G91 G01 X20. ; (P3)
N100 Y30. ; (P4)
N200 G02 X40. I20. ; (P5)
N300 G01 Y-30. ; (P6)
N400 X30. ; (P7)
N500 M99 ;
```

Milling Arc using G2 I & J CNC Program Example

CNC Program example code with drawing to show how an arc can be milled on a cnc milling machine.

G02 Circular interpolation Clockwise with I & J is used for arc machining.

G2 Circular Interpolation CW

G2 X... Y... I... J...

Parameter	Description
X	Coordinates of the arc end point in x-axis.
Y	Coordinates of the arc end point in y-axis.
I	Distance from arc start-point to arc-center-point in X-

	axis.
J	Distance from arc start-point to arc-center-point in Y-axis.

Program Example

```

Milling Arc using G2 I & J
G0 X0 Y0 (P1)
G1 X0 Y-1.493 F...(P2)
X1.939(P3)
G2 X1.5 Y-2.5 I1.061 J-1.061 (P4)
G1 X0 (P5)

```

G2 I & J Calculation Method

The image below shows how I & J values are calculated for Arc machining with G2 circular interpolation.

Fanuc G81 Drilling Cycle

G81 drilling cycle is used for simple drilling/spot drilling operations.

Syntax

G81 X... Y... Z... R... K... F...

Parameter	Description
X	Hole position in x-axis.
Y	Hole position in y-axis.
Z	Depth, tool will travel with feed to Z-depth starting from R plane.
R	Position of the R plane.
K	Number of cycle repetitions (if required).
F	Feedrate.

Once G81 drilling cycle is defined, the canned cycle is repeated at every X-Y position in sequential blocks. So G81 drilling cycle must be cancelled with G80.

Usage

```

N30 G81 X10 Y30 Z-17 R2 F75
N40 Y10
N50 X30
N60 Y30
N70 X90
N80 Y10
N90 G80

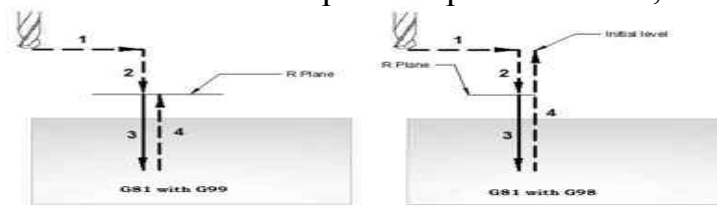
```

In the above example drilling will start with G81 drilling cycle at X10 Y30, so first drill will be at X10 Y30, then second at Y10, third at X30, fourth at Y30, fifth at X90 and the last one at Y10, because next block have G80 code, so drilling cycle will no more be repeated.

Working

Here is briefly described how G81 drilling cycle operates,

- 1- Rapid traverse to the specified x,y axis position (drilling position).
- 2- Rapid traverse to the R plane position.
- 3- Drilling with specified Feed from R-plane position to Z-depth position.
- 4- Rapid traverse to Initial level or R-plane depends on G98, G99 modes.



G81 drilling cycle working

G98 G99 Modes

After completing drilling depth the return is made with Rapid feed, the return height can be controlled through using G98 or G99.

G98 Drill will return to the Initial level

G99 Drill will return to R-plane.

G98, G99 can be used multiple times during G81 drilling cycle.

Example

```

N30 G81 X10 Y30 Z-17 R2 F75
N40 Y10
N50 G98 X30
N60 G99 Y30
N70 X90
N80 Y10
N90 G80

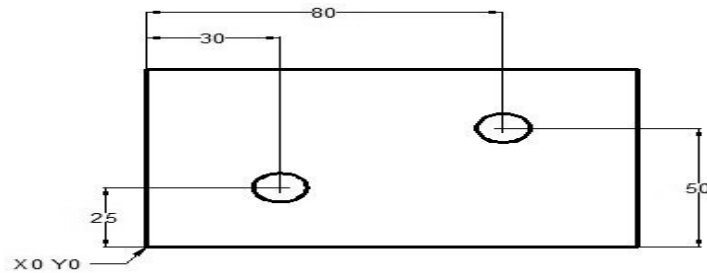
```

Repeat Drilling

With G81 drilling cycle drilling operation can be repeated multiple times. The drilling is repeated K times when that parameter is given with G81 drilling cycle.

Repeat drilling is normally used with G91 Incremental mode, and a good example of repeated drilling is Grid-plate drilling. the example for repeat drilling is given below.

Working Examples

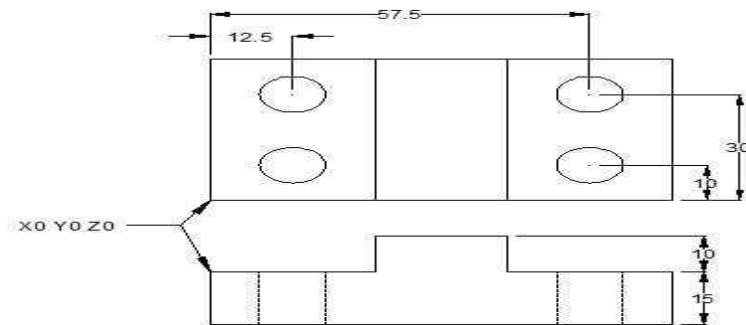


G81 Drilling Cycle Example

```

N10 T1 M06
N20 G90 G54 G00 X30 Y25
N30 S1200 M03
N40 G43 H01 Z5 M08
N50 G81 Z-10 R2 F75
N60 X80 Y50
N70 G80 G00 Z100 M09
N80 M30
    
```

G98 G99 Example

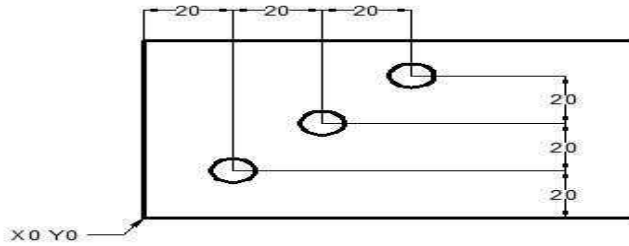


G81 drilling cycle usage with G98 G99

```

N10 M06 T1
N20 G90 G00 X12.5 Y10 Z12 S1000 M03
N30 G99 G81 X12.5 Y10 Z-17 R2 F75
N40 Y30
N50 G98 X57.5
N60 G99 Y10
N70 G91 G80 G28 X0 Y0 Z0 M05
N80 M30
    
```

Repeat Drilling Example



Repeat drilling with G81 Drilling Cycle

```

T1 M6
G00 G90 G40 G21 G17 G94
G54 X0 Y0 S1000 M03
G43 H1 Z100
Z3
G81 G99 G91 X20 Y20 R3 Z-20 K3 F100 M08
G80
G00 G90 Z100
M30

```

OR

```

T1 M6
G00 G90 G40 G21 G17 G94
G54 X20 Y20 S1000 M03
G43 H1 Z100
Z3
G81 G99 R3 Z-20 F100 M08
G91 X20 Y20 K2
G80
G00 G90 Z100
M30

```

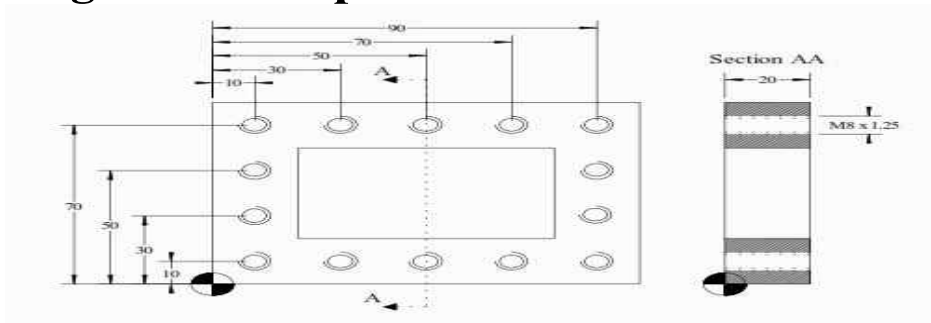
G81 Drilling Cycle G84 Tapping Cycle CNC Program Example

CNC program which shows the use of G81 Drilling Cycle G84 Tapping Cycle. As G81 drilling cycle has already been briefly explained here G81 Drilling Canned Cycle – CNC Mill Programming.

G81 drilling cycle program examples <http://www.helmancnc.com/fanuc/fanuc-g81-drilling-cycle/>.

G84 tapping cycle can be read here G84 Tapping Cycle – CNC Mill Programming with multiple G84 tapping cycle examples programs <http://www.helmancnc.com/fanuc/fanuc-g84-tapping-cycle/>.

CNC Program Example



G81 Drilling Cycle G84 Tapping Cycle Program Example

```

O1000 ;
T1 M6 ;
G0 G90 G40 G21 G17 G94 G80 ;
G54 X10 Y10 S? M3 ;
G43 Z100 H1 ;
Z5 ;
G81 R3 Z-20 F? M8 ;
Y30 ;
Y50 ;
Y70 ;
X30 ;
X50 ;
X70 ;
X90 ;
Y50 ;
Y30 ;
Y10 ;
X70 ;
X50 ;
X30 ;
G80 ;
G0 G90 Z100
T2 M6 ;
G0 G90 G40 G21 G17 G94 G80 ;
G54 X10 Y10 S? M3 ;
G43 Z100 H1 ;
Z5 ;
G84 G99 G95 R3 Z-20 F1.25 M8 ;
Y30 ;
Y50 ;
Y70 ;
X30 ;
X50 ;
X70 ;
X90 ;
Y50 ;
Y30 ;
Y10 ;
X70 ;
X50 ;
X30 ;

```

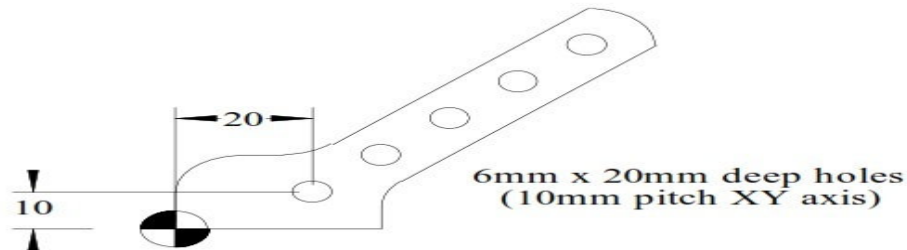
```
G80 ;  
G0 G90 Z100 ;  
T0 M6 ;  
M30 ;
```

Repeat Drilling with G81 Drilling Cycle and G91 Example Program

Here is another G81 drilling cycle programming example which illustrates the use of repeat drilling in G91 Incremental Programming Mode.

Other CNC program example is here [G81 Drilling Cycle – Repeat Drilling in G91 Incremental Mode Example Code](#)

Repeat Drilling with G81 Drilling Cycle



Repeat Drilling with G81 Drilling Cycle and G91

```
O1000  
T1 M6  
G00 G90 G40 G21 G17 G94  
G54 X20 Y10 S1000 M03  
G43 H1 Z100  
Z3  
G81 G99 R3 Z-20 F350 M08  
G91 X10 Y10 K4  
G80  
G00 G90 Z100  
M30
```

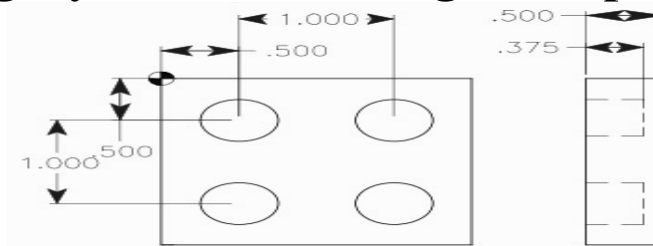
G82 Drilling Canned Cycle with Dwell CNC Milling Example Program

G82 drilling cycle with dwell can be used for normal drilling where bottom of the hole need more accurate machining.

Otherwise if you just want a drilling cycle without a dwell time at the bottom of the hole G81 drilling cycle is a big time saver.

Following is a cnc programming example which illustrates the use of G82 Drill cycle.

G82 Drilling Cycle CNC Milling Example Program



G82 Drilling Canned Cycle with Dwell CNC Milling Example Program

O10076

```
N10 T11 M06
N20 G90 G54 G00 X0.5 Y-0.5
N30 S1200 M03
N40 G43 H11 Z1. M08
N50 G82 G99 Z-0.375 P1 R0.1 F7.5
N60 X1.5
N70 Y-1.5
N80 X0.5
N90 G80 G00 Z1. M09
N100 G53 G49 Z0. M05
N110 M30
```

Note N50 – CNC machines with Fanuc cnc control will use P1000 instead of P1 which is used for Haas CNC machines

G81 Drilling Cycle G83 Peck Drilling with G98 G99 Example Program

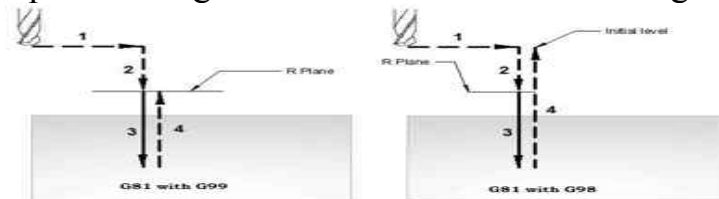
A complete cnc part-program which shows how G98 and G99 (canned cycle return level) work with G81 drilling cycle and G83 peck drilling cycle for drilling of a component which have different heights.

G98 G99 Summary

G98 and G99 are modal commands that change the way canned cycles (G81,G83 etc.) operate.

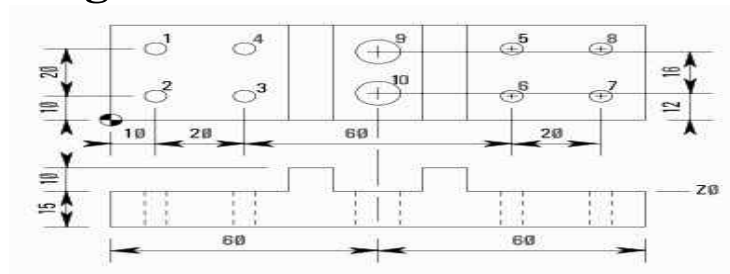
When G98 is active, the Z-axis will return to the start position (initial plane) when it completes an single operation.

When G99 is active, the Z-axis will be returned to the R point (plane) when the canned cycle completes a single hole. Then the machine will go to the next hole.



G81 drilling cycle working – G98 G99 return level

CNC Part Program



G81 Drilling Cycle with G98 G99 G code Example Program

```

N10 M06 T01
N20 G90 G00 X10 Y30 Z12 S1000 M03
N30 G99 G81 X10 Y30 Z-17 R2 F75 (Hole 1)
N40 Y10 (Hole 2)
N50 X30 (Hole 3)
N60 Y30 (Hole 4)
N70 G98 X90 (Hole 5)
N80 G99 Y10 (Hole 6)
N90 X110(Hole 7)
N100 G98 Y30 (Hole 8)
N110 G91 G80 G28 X0 Y0 Z0 M05
N120 M06 T02
N130 G90 G00 X60 Y28 Z12 S750 M03
N140 G99 G83 X60 Y28 Z-17 Q6 R2 F60 (Hole 9)
N150 G98 Y12 (Hole 10)
N160 G91 G80 G28 X0 Y0 Z0 M05
N170 M30
    
```

Explanation

N10- Tool change (M06) to tool no.1
 N20- Rapid traverse to X10 Y30 Z12, Spindle started clockwise (M03) with 1000rpm (S1000).
 N30- Drilling starts (G81) at X10 Y30 with cutting-feed (F75) drill will retract to R-plane after drilling operation.
 N40- Next drilling position Y10 (as G99 is a modal g-code drill will keep on retracting to R-plane until G98 is given).
 N50- Next drill at X30.
 N60- Drill at Y30
 N70- Drill at X90 & Retract to Initial-plane.
 N80- Drill at Y10 & Retract to R-plane.
 N90- Drill at X110
 N100- Drill at Y30 & Retract to Initial-plane.
 N110- Drilling cycle is cancelled (G80), return to reference point (G28) for tool change, stop spindle (M05).
 N120- Tool change (M06) to tool number 2.
 N130- Rapid traverse to X60 Y28 Z12, start spindle at 750rpm (S750) clockwise (M03).
 N140- G83 Peck drilling starts at X60 Y28, drill depth is Z-17 and drill peck size is Q6, drilling feed is F60
 N150- Next deep drill at Y12 (return to initial point).
 N160- G83 Peck drilling cycle cancelled with G80, tool returned to reference point (G28), spindle stopped (M05).
 N170- Part-program end with return to program start (M30)

G & M Codes

Code	Description
T	Tool no. used.
M06	Tool change command.
G90	Absolute programming
G00	Rapid traverse
S	Cutter speed
M03	Cutter rotation Clockwise
M08	Coolant on.
G81	Fanuc drilling cycle.
G83	Fanuc peck drilling cycle.
G98	Return to initial point in canned cycle.
G99	Return to R point in canned cycle.
F	Cutting feed.
G80	Canned cycle cancel.

M09	Coolant off.
G28	Return to reference position.
G91	Incremental programming.
M05	Cutter rotation stop.
M30	CNC part-program end with return to program-start.

CNC Milling Machine Programming Example for Beginners

CNC Mill Program Example

A very simple cnc milling machine programming tutorial for beginner level cnc machinists.

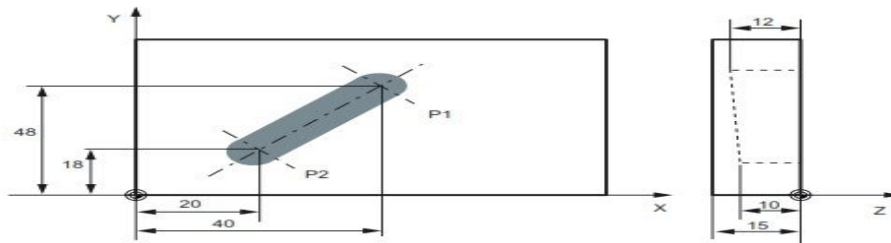
An easy to understand cnc mill programming code. This is a cnc g code example without the use of any cnc canned cycle.

Related cnc mill program examples

- CNC G02 Circular Interpolation Clockwise CNC Milling Sample Program
- CNC Milling Circular Interpolation G02 G03 G-Code Program Example

Beginner level cnc program examples for **CNC Lathe**

- Fanuc CNC Program Example
- CNC Programming for Beginners a Simple CNC Programming Example
- CNC Programming for Beginners a CNC Programming Example



CNC Milling Machine Programming Example for Beginners

```

N05 G0 G90 X40 Y48 Z2 S500 M3
N10 G1 Z-12 F100
N15 X20 Y18 Z-10
N20 G0 Z100
N25 X-20 Y80
N30 M2

```

N05 The tool traverses in rapid traverse on P1, three axes concurrently, spindle speed = 500 rpm, clockwise

N10 Infeed on Z-12, feed 100 mm/min

N15 Tool travels on a straight line in space on P2
N20 Retraction in rapid traverse
N30 End of program

Drilling

Fanuc G72.1 G72.2 Figure Copy Program Example (Bolt Hole Circle)

This CNC program example shows how both G72.1 and G72.2 figure copy functions can call one-another in one part program, read Programming Notes below carefully.

Programming

Fanuc G72.1 Rotational Copy

Using G72.1 Rotational Copy G-code a figure specified by a subprogram can be repeatedly produced with **Rotational movement**.

Read G72.1 definition with program example Fanuc G72.1 Rotational Copy (Figure Copy Function CNC Mill)

Fanuc G72.2 Linear Copy

Using G72.2 Linear Copy G-code a figure specified by a subprogram can be repeatedly produced with **Linear movement**.

Read G72.2 definition with program example Fanuc G72.2 Linear Copy (Figure Copy Function CNC Mill)

Fanuc G81 Drilling Cycle

G81 drilling cycle is used for simple drilling/spot drilling operations.

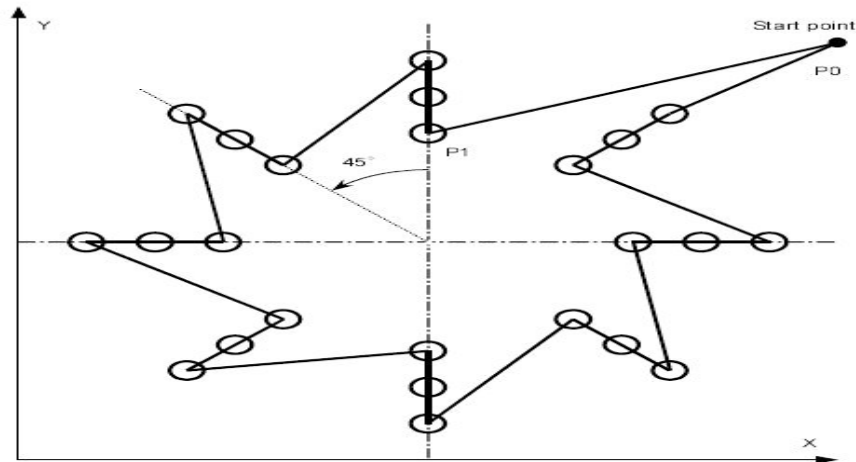
Read G81 drilling cycle definition with program examples Fanuc G81 Drilling Cycle

Programming Notes

Specifying two or more commands to copy a figure

- 1 G72.1 cannot be specified more than once in a subprogram for making a rotational copy (If this is attempted, alarm PS0900 will occur).
- 2 G72.2 cannot be specified more than once in a subprogram for making a linear copy (If this is attempted, alarm PS0901 will occur).
- 3 In a subprogram that specifies rotational copy, however, linear copy can be specified. Similarly, in a subprogram that specifies linear copy, rotational copy can be specified.

Fanuc G71.2 G72.2 Program Example



Main program

```
O4000 ;  
N10 G90 G00 G17 X240. Y230. Z100. ; (P0)  
N20 G72.1 P4100 X120. Y120. L8 R45. ;  
N30 G80 G00 X240. Y230. ; (P0)  
N40 M30 ;
```

Sub program_ Rotation copy_ (G72.1)

```
O4100 N100 G72.2 P4200 I0 J20. L3 ;  
N200 M99 ;
```

Sub program_ Linear copy_ (G72.2_)

```
O4200 N110 G90 G81 X120. Y180. R60. Z10. F200. ; (P1)  
N210 M99 ;
```

Fanuc G81 Drilling Cycle

G81 drilling cycle is used for simple drilling/spot drilling operations.

Contents

- Syntax
- Usage
- Working
- G98 G99 Modes
 - Example
- Repeat Drilling
- Working Examples
- G98 G99 Example
- Repeat Drilling Example

Syntax

G81 X... Y... Z... R... K... F...

Parameter	Description
X	Hole position in x-axis.
Y	Hole position in y-axis.
Z	Depth, tool will travel with feed to Z-depth starting from R plane.
R	Position of the R plane.
K	Number of cycle repetitions (if required)_____.
F	Feedrate.

Once G81 drilling cycle is defined, the canned cycle is repeated at every X-Y position in sequential blocks. So G81 drilling cycle must be cancelled with G80.

Usage

```
N30 G81 X10 Y30 Z-17 R2 F75
```

```
N40 Y10
```

```
N50 X30
```

```
N60 Y30
```

```
N70 X90
```

```
N80 Y10
```

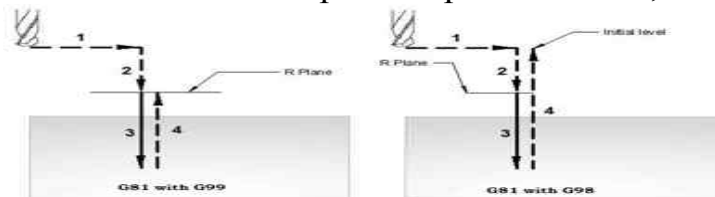
```
N90 G80
```

In the above example drilling will start with G81 drilling cycle at X10 Y30, so first drill will be at X10 Y30, then second at Y10, third at X30, fourth at Y30, fifth at X90 and the last one at Y10, because next block have G80 code, so drilling cycle will no more be repeated.

Working

Here is briefly described how G81 drilling cycle operates,

- 1- Rapid traverse to the specified x,y axis position (drilling position).
- 2- Rapid traverse to the R plane position.
- 3- Drilling with specified Feed from R-plane position to Z-depth position.
- 4- Rapid traverse to Initial level or R-plane depends on G98, G99 modes.



G81 drilling cycle working

G98 G99 Modes

After completing drilling depth the return is made with Rapid feed, the return height can be controlled through using G98 or G99.

G98 Drill will return to the Initial level

G99 Drill will return to R-plane.

G98, G99 can be used multiple times during G81 drilling cycle.

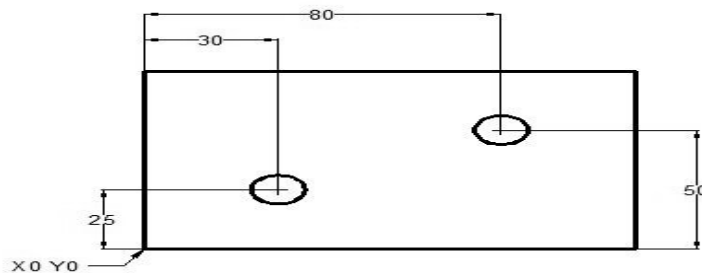
Example

```
N30 G81 X10 Y30 Z-17 R2 F75
N40 Y10
N50 G98 X30
N60 G99 Y30
N70 X90
N80 Y10
N90 G80
```

Repeat Drilling

With G81 drilling cycle drilling operation can be repeated multiple times. The drilling is repeated K times when that parameter is given with G81 drilling cycle. Repeat drilling is normally used with G91 Incremental mode, and a good example of repeated drilling is Grid-plate drilling. the example for repeat drilling is given below.

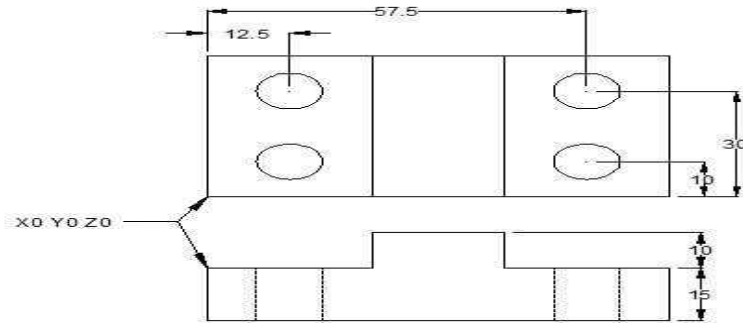
Working Examples



G81 Drilling Cycle Example

```
N10 T1 M06
N20 G90 G54 G00 X30 Y25
N30 S1200 M03
N40 G43 H01 Z5 M08
N50 G81 Z-10 R2 F75
N60 X80 Y50
N70 G80 G00 Z100 M09
N80 M30
```

G98 G99 Example



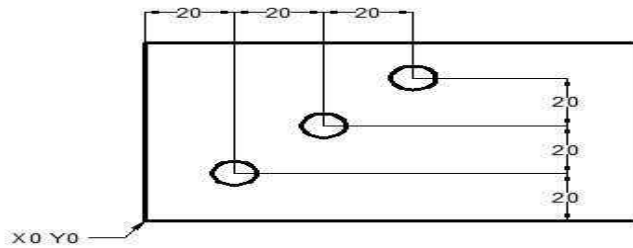
G81 drilling cycle usage with G98 G99

```

N10 M06 T1
N20 G90 G00 X12.5 Y10 Z12 S1000 M03
N30 G99 G81 X12.5 Y10 Z-17 R2 F75
N40 Y30
N50 G98 X57.5
N60 G99 Y10
N70 G91 G80 G28 X0 Y0 Z0 M05
N80 M30

```

Repeat Drilling Example



Repeat drilling with G81 Drilling Cycle

```

T1 M6
G00 G90 G40 G21 G17 G94
G54 X0 Y0 S1000 M03
G43 H1 Z100
Z3
G81 G99 G91 X20 Y20 R3 Z-20 K3 F100 M08
G80
G00 G90 Z100
M30

```

OR

```

T1 M6
G00 G90 G40 G21 G17 G94
G54 X20 Y20 S1000 M03
G43 H1 Z100
Z3
G81 G99 R3 Z-20 F100 M08
G91 X20 Y20 K2
G80
G00 G90 Z100
M30

```

Fanuc G82 Drilling Cycle

G82 drilling cycle is also called G82 counter boring cycle.

G82 is a normal drilling cycle the only difference is that it dwell for specified time at the bottom of the hole, normally used for accurate depth drilling.

Syntax

G82 X... Y... Z... R... P... F... K...

Parameter	Description
X	Hole position in x-axis.
Y	Hole position in y-axis.
Z	Depth, tool will travel with feed to Z-depth starting from R plane.
R	Position of the R plane.
P	Dwell at the bottom of hole.
K	Number of cycle repetitions (if required) _____.
F	Feedrate.

Usage

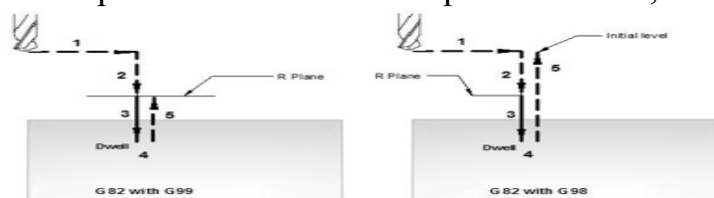
```
N30 G82 X10 Y30 Z-17 R2 P1000 F75
N40 Y10
N50 X30
N60 Y30
N70 G80
```

Once G82 drilling cycle is specified with it's parameters in a program block, this will keep drilling at every axis movement, until cycle is ended with G80

Working

How G82 drilling cycle works

- 1- Rapid traverse to x, y position
- 2- Rapid traverse to R-plane position
- 3- Drilling with feed from R-plane to Z-depth position.
- 4- Dwell for specified time at hole bottom.
- 5- Rapid traverse to R-plane or Initial-level depends on G99, G98 mode.



G82 drilling cycle working

G98 G99 Modes

How G82 drilling cycle behaves upon G98 or G99 mode,

G98 Drill will return to the Initial level

G99 Drill will return to R-plane.

For a working example see G81 drilling cycle.

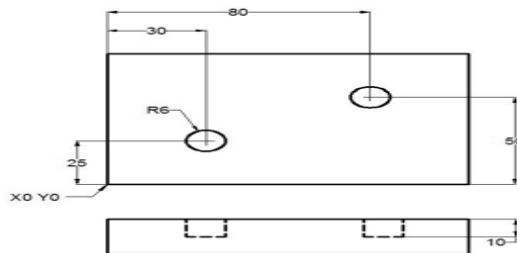
Example

```
N30 G82 X10 Y30 Z-17 R2 P2000 F75  
N40 Y10  
N50 G98 X30  
N60 G99 Y30  
N70 X90  
N80 Y10  
N90 G80
```

Repeat Drilling

If K parameter value is given with G82 drilling cycle, then drilling will repeat the number of times given with K. An effective use of repeat drilling is while drilling multiple same distance holes, this way G82 cycle is used in G91 incremental mode. See G81 drilling cycle for repeat drilling example.

Working Example



G82 drilling cycle example

```
N10 T1 M06  
N20 G90 G54 G00 X30 Y25  
N30 S1200 M03  
N40 G43 H01 Z5 M08  
N50 G82 Z-10 R2 P1000 F75  
N60 X80 Y50  
N70 G80 G00 Z100 M09  
N80 M30
```

Fanuc G83 Peck Drilling Cycle

G83 peck drilling cycle perform the drilling operation in multiple pecks, this technique makes deep-hole drilling easy and economical.

Cutting feed is performed intermittently to the bottom of the hole while chips are discharged.

As the drilling is performed to the bottom of the hole with feed in multiple small steps, every time a specified depth is made and then drill retracts, then drill makes the next peck, this operation is repeated again and again until the drill depth is reached.

Syntax

G83 X... Y... Z... R... Q... F... K...

Parameter	Description
X	Hole position in x-axis.
Y	Hole position in y-axis.
Z	Depth, tool will travel with feed to Z-depth starting from R plane.
R	Position of the R plane.
Q	Depth of cut for each cutting feed (Peck).
K	Number of cycle repetitions (if required)
F	Feedrate.

Once given in program G83 peck drilling cycle is repeated at every axis movement until G80 is given in program to end peck drilling cycle.

Usage

```
N150 M06 T02
N160 G90 G00 X60 Y28 Z12 S750 M03
N170 G99 G83 X60 Y28 Z-17 Q6 R2 F60
N180 G98 Y12
N190 G91 G80 G28 X0 Y0 Z0 M05
N200 M30
```

In the above example code first drill is done at X60 Y28 and second at Y12 and then peck drilling is cycle is ended with G80.

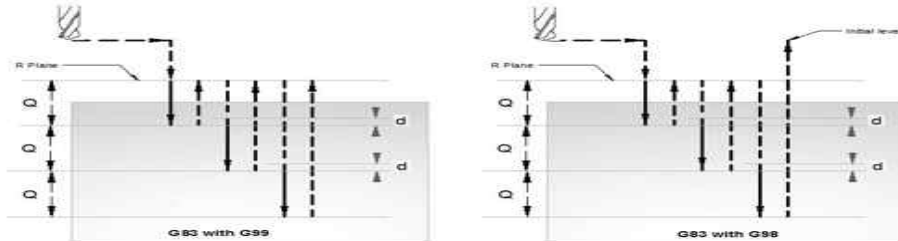
6mm pecks are taken to complete total drilling depth of 17mm.

Working

Here is briefly described how G83 peck drilling cycle works,

- 1- Rapid traverse to X, Y drilling position.
- 2- Rapid traverse to R-plane.
- 3- Drilling with feed Q deep.
- 4- Retraction with Rapid traverse to R-plane.

- 5- Rapid traverse to Q-d deep (d value is specified in parameters).
- 6- Drilling with feed Q+d deep.
- 7- Retraction with Rapid traverse to R-plane
 - this whole procedure is repeated until drill reaches Z-depth position,
 - then drill is retracted to R-plane or Initial-level depends on G99 or G98 which one is given in program.



G83 peck drilling cycle working

G98 G99 Modes

After completing drilling depth the return is made with Rapid feed, the return height can be controlled through using G98 or G99.

G98 Drill will return to the Initial level

G99 Drill will return to R-plane.

G98, G99 can be used multiple times during G83 peck drilling cycle.

Example

```

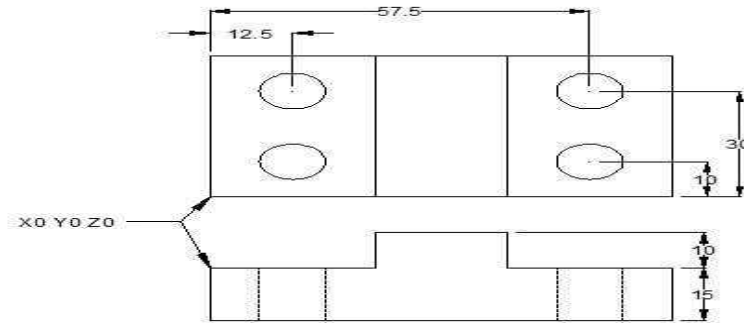
N30 G83 X10 Y30 Z-17 Q5 R2 F75
N40 Y10
N50 G98 X30
N60 G99 Y30
N70 X90
N80 Y10
N90 G80
  
```

Repeat Drilling

G83 peck drilling cycle, drilling operation can be repeated multiple times. The drilling is repeated K times if K value is given with G83.

Repeat drilling is normally used with G91 Incremental mode, and a good example of repeated drilling is Grid-plate drilling. For working example see G81 drilling cycle.

Working Example



G83 Peck drilling cycle Example

```

N10 M06 T1
N20 G90 G00 X12.5 Y10 Z12 S1000 M03
N30 G99 G83 X12.5 Y10 Z-17 R2 Q4 F75
N40 Y30
N50 G98 X57.5
N60 G99 Y10
N70 G91 G80 G28 X0 Y0 Z0 M05
N80 M30

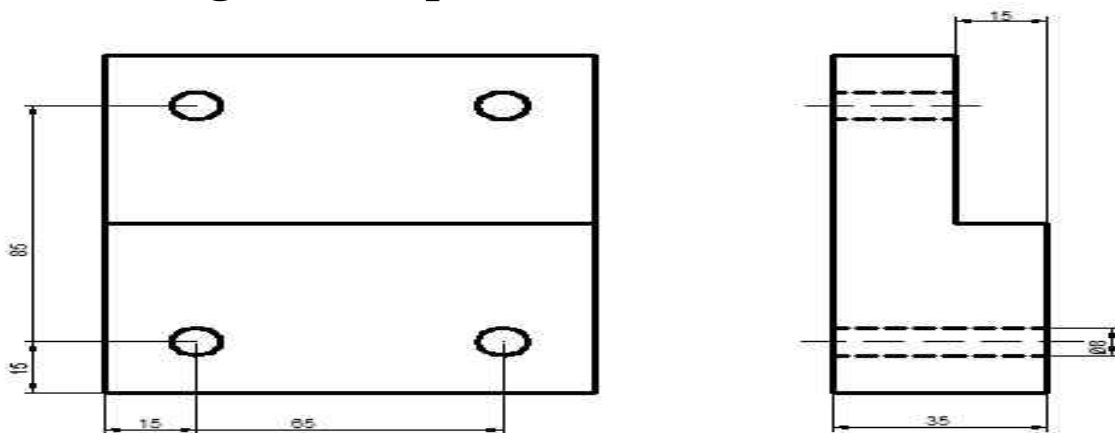
```

Drilling a Two Step Block with G81 Drilling Cycle

G81 drilling cycle program example which shows how a step block can be drilled economically.

The cnc program code is self explanatory, no complex technique is used.

G81 Drilling Two Step Block



G81 Drilling Two Step Block

```

N10 T4 M6 (TWIST DRILL 8")
N15 G90 G54 G00 X15 Y15

```

N20 S1000 M3 F100
N25 G43 H01 Z2 M8
N30 G81 R2 Z-42
N35 X65
N40 Y85 R-13
N45 X15
N50 G80 Z50 M5
N55 M30

CNC Lathe Programming Exercise

Fanuc G71 Turning Cycle, G74 Peck Drilling Cycle

CNC programming exercise for cnc lathe machinists who work on Fanuc cnc control (or similar cnc control).

This cnc programming exercise use

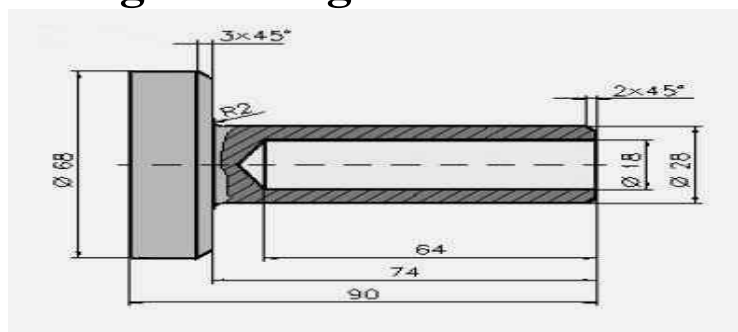
Fanuc G71 Turning Cycle

Fanuc G74 Peck Drilling Cycle

Contents

- CNC Lathe Programming Exercise
Used Tools & Operations

CNC Lathe Programming Exercise



CNC Lathe Programming Exercise Fanuc G71, G74 Cycles

N10 G40 G00
N20 G99
N60 T0101
N70 G50 S3500
N80 G96 S0240 M4
N90 G00 X72. Z0.1
N100 G01 X-1.6 F0.12 M7
N110 G00 X150. Z150.

N120 M5
N130 M9
N140 T0303
N150 G97 S2500 M3
N160 G00 X0. Z3.
N170 G01 Z-6. F0.1 M7
N180 G00 Z2.
N190 G00 X150. Z150.
N210 T0707 M7
N220 G97 S0884 M3
N230 G00 Z3.
N240 G00 X0.
N250 G74 R1.0
N260 G74 X0.0 Z-68.326 Q18000 F0.22
N380 G00 X200.
N400 G00 Z100.
N500 T0404 M7
N510 G50 S3500
N520 G96 S0240 M4
N530 G00 Z1.
N540 G00 X70.
N550 G71 U4. R1
N560 G71 P570 Q650 U0.6 W0.2 F0.35
N570 G42 G00 X24.
N580 G01 Z0.
N590 G01 X28. Z-2.
N600 G01 Z-72.
N610 G02 X32. Z-74. I2. K0.
N620 G01 X62.
N630 G01 X68. Z-77.
N640 G01 Z-90.
N650 G40
N660 G00 X150.
N680 G00 Z70.
N690 T0202 M7
N700 G50 S4500
N710 G96 S0380 M4
N720 G00 X16. Z3.
N730 G42 G01 Z0. F0.1
N740 G01 X24.
N750 G01 X28. Z-2.
N760 G01 Z-72.
N770 G02 X32. Z-74. I2. K0.
N780 G01 X62.
N790 G01 X68. Z-77.
N800 G01 Z-90.
N810 G40
N820 G00 X150. Z150.
N830 M5
N840 M9
N850 M30

Used Tools & Operations

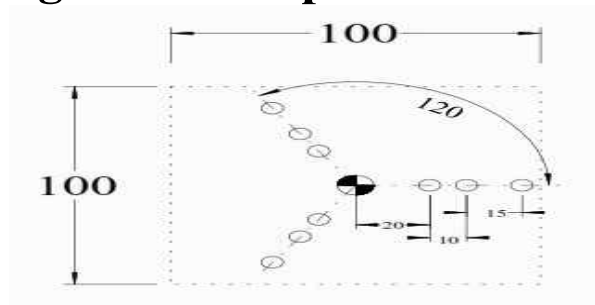
- T0101 Turning Tool – Rough Facing

- T0303 Center Drill – Center Drilling
- T0707 Twist Drill – Drilling
- T0404 Turning Tool – Rough Turning
- T0202 Turning Tool – Finish Contour Cutting

Fanuc G68 Coordinate Rotation Program Example

Fanuc G68 Coordinate Rotation G-Code makes it easy for cnc machinist to run a pattern of operations in a rotated angle.
Here is a basic cnc programming Example which helps to understand the actual working of G68 coordinate rotation.

Fanuc G68 Program Example



```

T1 M6
G0 G90 G40 G21 G17 G94 G80
G54 X20 Y0 S1500 M3
G43 Z100 H1
Z5
G81 R3 Z-20 F? M8
X30
X45
G68 X0 Y0 R120
X20 Y0
X30
X45
G68 X0 Y0 R240
X20 Y0
X30
X45
G69 G80
G0 G90 Z100 M30

```

Fanuc Subprogram Example

Here is a Fanuc subprogram example which shows, how cnc machinists can make their cnc programs easier to write and maintain by using sub-programming method.

Contents

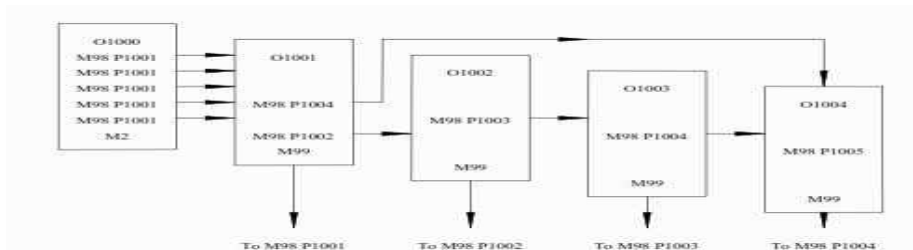
- CNC Sub Programs
- Example Without Fanuc Sub-Program Call
- Example with Fanuc Sub-Program Call
- Fanuc Subprogram Example
Sub Program

CNC Sub Programs

Sub-programming is a way in which cnc machinists can call other cnc programs stored in cnc machines from their programs (called as main-programs).

Read article about How to do Fanuc Sub Programming.

This Sub-programming process makes it easier for cnc programmers/machinists to keep their cnc programs small by dividing programs into small pieces of repeatable useful code.



Fanuc Sub Programming

As the below cnc program example shows, cnc machinist want to drill some holes and then he want to tap the same holes.

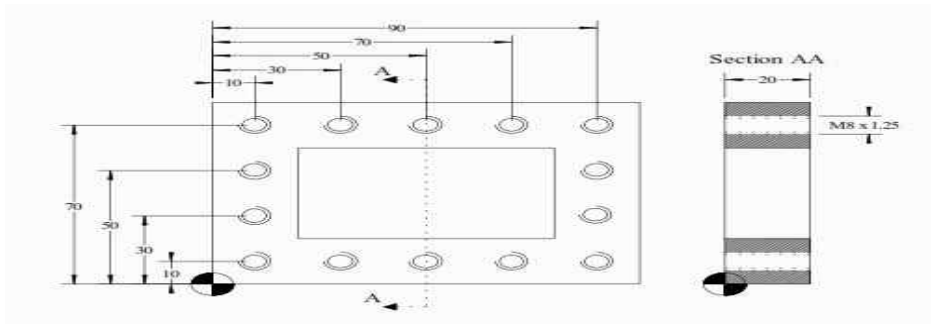
Example Without Fanuc Sub-Program Call

The simplest way to program can be read here G81 Drilling Cycle G84 Tapping Cycle CNC Program Example.

Example with Fanuc Sub-Program Call

Below is the same program example but this time we are using Sub-programming technique.

Fanuc Subprogram Example



Fanuc Subprogram Example

```

O1000 ;
N1 T1 M6 ;
N2 G0 G90 G40 G21 G17 G94 G80 ;
N3 G54 X10 Y10 S? M3 ;
N4 G43 Z100 H1 ;
N5 Z5 ;
N6 G81 R3 Z-20 F? M8 ;
N7 M98 P1001 ;
N8 G0 G90 Z100
N9 T2 M6 ;
N10 G0 G90 G40 G21 G17 G94 G80 ;
N11 G54 X10 Y10 S? M3 ;
N12 G43 Z100 H1 ;
N13 Z5 ;
N14 G84 G99 G95 R3 Z-20 F1.25 M8 ;
N15 M98 P1001 ;
N16 G0 G90 Z100 ;
N17 T0 M6 ;
N18 M30 ;

```

Sub Program

```

O1001 ;
N101 Y30 ;
N102 Y50 ;
N103 Y70 ;
N104 X30 ;
N105 X50 ;
N106 X70 ;
N107 X90 ;
N108 Y50 ;
N109 Y30 ;
N110 Y10 ;
N111 X70 ;
N112 X50 ;
N113 X30 ;
N114 G80 ;
N115 M99

```

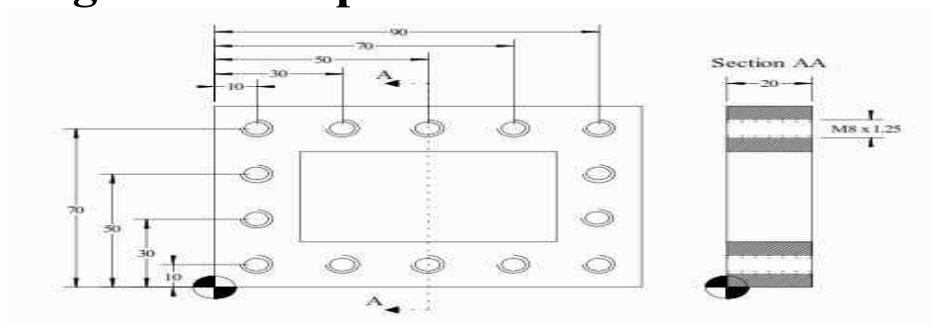
G81 Drilling Cycle G84 Tapping Cycle CNC Program Example

CNC program which shows the use of G81 Drilling Cycle G84 Tapping Cycle.
As G81 drilling cycle has already been briefly explained here G81 Drilling
Canned Cycle – CNC Mill Programming.

G81 drilling cycle program examples <http://www.helmancnc.com/fanuc/fanuc-g81-drilling-cycle/>.

G84 tapping cycle can be read here G84 Tapping Cycle – CNC Mill
Programming with multiple G84 tapping cycle examples
programs <http://www.helmancnc.com/fanuc/fanuc-g84-tapping-cycle/>.

CNC Program Example



G81 Drilling Cycle G84 Tapping Cycle Program Example

```
O1000 ;  
T1 M6 ;  
G0 G90 G40 G21 G17 G94 G80 ;  
G54 X10 Y10 S? M3 ;  
G43 Z100 H1 ;  
Z5 ;  
G81 R3 Z-20 F? M8 ;  
Y30 ;  
Y50 ;  
Y70 ;  
X30 ;  
X50 ;  
X70 ;  
X90 ;  
Y50 ;  
Y30 ;  
Y10 ;  
X70 ;  
X50 ;  
X30 ;  
G80 ;  
G0 G90 Z100
```

```
T2 M6 ;
G0 G90 G40 G21 G17 G94 G80 ;
G54 X10 Y10 S? M3 ;
G43 Z100 H1 ;
Z5 ;
G84 G99 G95 R3 Z-20 F1.25 M8 ;
Y30 ;
Y50 ;
Y70 ;
X30 ;
X50 ;
X70 ;
X90 ;
Y50 ;
Y30 ;
Y10 ;
X70 ;
X50 ;
X30 ;
G80 ;
G0 G90 Z100 ;
T0 M6 ;
M30 ;
```

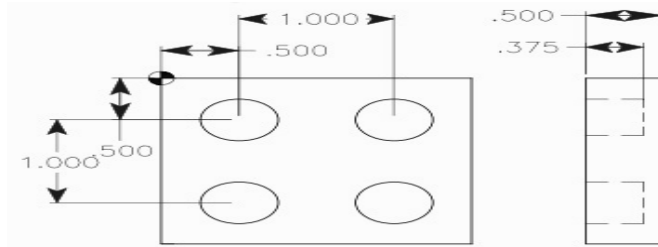
G82 Drilling Canned Cycle with Dwell CNC Milling Example Program

G82 drilling cycle with dwell can be used for normal drilling where bottom of the hole need more accurate machining.

Otherwise if you just want a drilling cycle without a dwell time at the bottom of the hole G81 drilling cycle is a big time saver.

Following is a cnc programming example which illustrates the use of G82 Drill cycle.

G82 Drilling Cycle CNC Milling Example Program



G82 Drilling Canned Cycle with Dwell CNC Milling Example Program

O10076

N10 T11 M06

N20 G90 G54 G00 X0.5 Y-0.5

N30 S1200 M03

N40 G43 H11 Z1. M08

N50 G82 G99 Z-0.375 P1 R0.1 F7.5

N60 X1.5

N70 Y-1.5

N80 X0.5

N90 G80 G00 Z1. M09

N100 G53 G49 Z0. M05

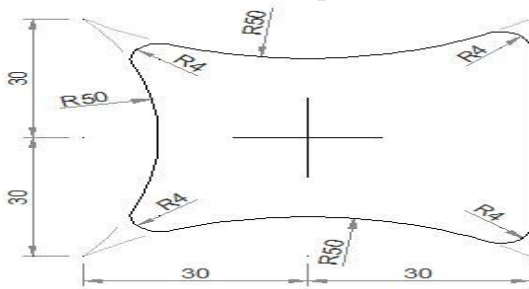
N110 M30

Note N50 – CNC machines with Fanuc cnc control will use P1000 instead of P1 which is used for Haas CNC machines.

G00	Rapid traverse
G01	Linear interpolation
G02	Circular interpolation CW
G03	Circular interpolation CCW

G02 G03 Example CNC Mill

G02 G03 Circular interpolation CNC mill example program.



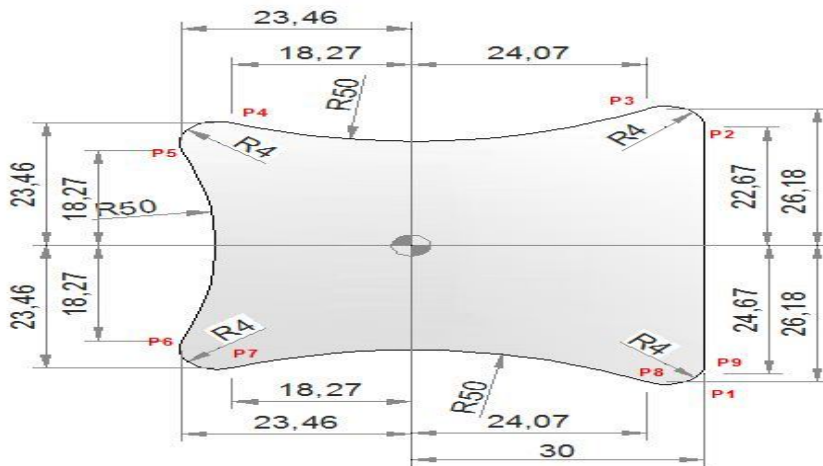
Contents

4 G02 G03 Example CNC Mill

CNC Part Program

G M S T Codes Explanation

G02 G03 Example CNC Mill



CNC Part Program

G0 X30 Y-30 (P1)

G1 Y22.67 (P2)

G3 X24.07 Y26.18 R4 (P3)

G2 X-18.27 Y23.46 R50 (P4)

G3 X-23.46 Y18.27 R4(P5)

G2 X-23.46 Y-18.27 R50 (P6)
 G3 X-18.27 Y-23.46 R4 (P7)
 G2 X24.07 Y-26.18 R50 (P8)
 G3 X30 Y-24.67 R4 (P9)
 G1 X33

G M S T Codes Explanation

Code	Description
G0	Rapid traverse
G1	Linear interpolation
G2	Circular interpolation CW
G3	Circular interpolation CCW
M30	End of program (Reset)

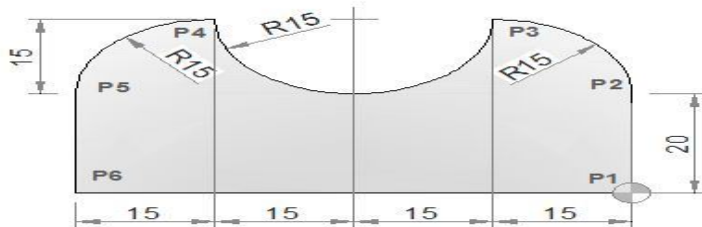
Multiple Arc CNC Mill Program G2 G3 I J

CNC milling machine program which combines/joins multiple arcs.

Contents

- Multiple Arc CNC Mill Program G2 G3 I J
 - CNC Part Program
 - G M S T Codes Explanation

Multiple Arc CNC Mill Program G2 G3 I J



CNC Part Program

```
N10 M6 T1 G43 H1 M3
N15 S500 F120
N20 G0 X0 Y0 (P1)
N25 G1 Y20(P2)
N30 G3 X-15 Y35 I-15 J0 (P3)
N35 G2 X-45 Y35 I-15 J0 (P4)
N40 G3 X-60 Y20 I0 J-15 (P5)
N45 G1 Y0 (P6)
N50 G1 X0 (P1)
```

G M S T Codes Explanation

Code	Description
G0	Rapid traverse
G1	Linear interpolation
G2	Circular interpolation CW
G3	Circular interpolation CCW
G43	Tool length compensation + direction
M3	Spindle start forward CW
M6	Tool change
M30	End of program (Reset)
T	Tool
S	Speed
F	Feed

CNC Mill Subprogram Example

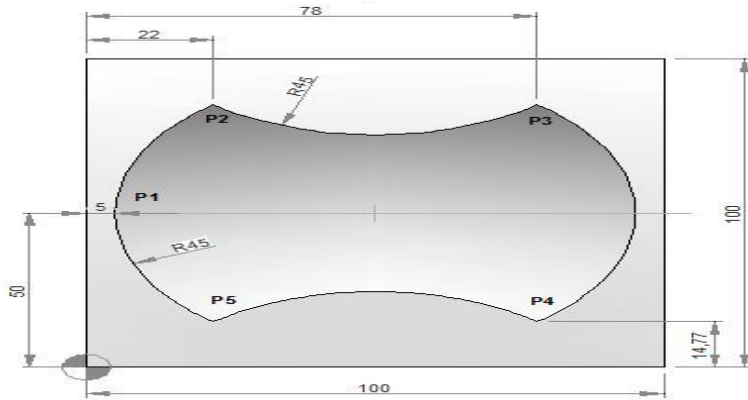
Joining Multiple Arcs G02 G03 G41

CNC milling program to describe how two or more radii can be joint together in a cnc mill program.

Contents

- CNC Mill Subprogram Example
 - CNC Part Program
 - Subprogram
 - G M S T Codes Explanation

CNC Mill Subprogram Example



CNC Part Program

```

N10 T1 H1 M6 G43 M3
N20 F150 S250
N30 G0 X-21 Y50 Z0.5
N40 G0 Z0
N50 M98 P040050
N60 G49
N70 G0 Z50
N80 M30

```

Subprogram

```

O0050
N10 F160 S400
N20 G0 Z-2.5 G91
N30 G1 G90 X5 Y50 G41 (P1)
N40 G2 X22 Y85.23 I45 J0 (P2)
N50 G3 X78 Y85.23 R45 (P3)
N60 G2 X78 Y14.77 R45 (P4)
N70 G3 X22 Y14.77 R45 (P5)
N80 G2 X5 Y50 R45 (P1)
N90 G0 G40 X-21
N100 M99

```

G M S T Codes Explanation

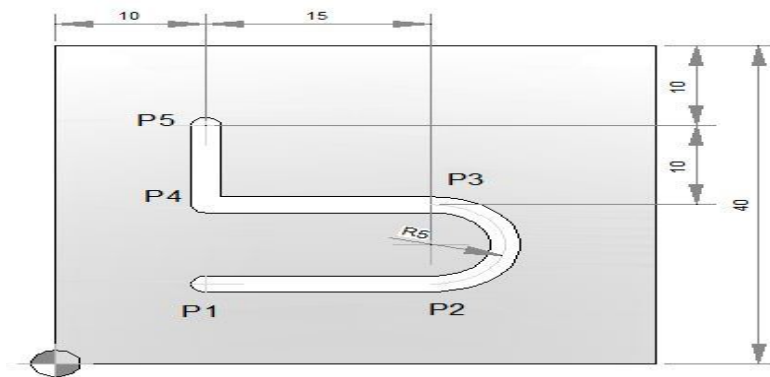
Code	Description
G00	Rapid traverse
G01	Linear interpolation
G02	Circular interpolation CW
G03	Circular interpolation CCW
G40	Cutter compensation cancel
G41	Tool nose radius compensation left
G43	Tool length compensation + direction
G49	Tool length compensation cancel
G90	Absolute command
G91	Increment command

M03	Spindle start forward CW
M06	Tool change
M30	End of program (Reset)
M98	Subprogram call
M99	End of subprogram
T	Tool
S	Speed
F	Feed

CNC Milling Program Example G03 G90 G91

CNC milling program example, which illustrates how a cnc program can be easily converted from G90 Absolute program mode to G91 Incremental program mode.

CNC Milling Program Example



G90 Absolute Program	G91 Incremental Program
N10 T1 M6 M3 G43	N10 T1 M6 M3 G43
N15 S2000 F80	N15 S2000 F80
N20 G0 X10 Y10 Z1 (P1)	N20 G0 X10 Y10 Z1
N25 F30	N25 F30
N30 G1 Z-2	N30 G1 Z-2
N35 X25 F80 (P2)	N35 G91 X15 F80
N40 G3 X25 Y20 I0 J5 (P3)	N40 G3 X0 Y10 I0 J5
N45 G1 X10 (P4)	N45 G1 X-15
N50 Y30 (P5)	N50 Y10
N55 G0 Z100 G49	N55 G90 G0 Z100 G49
N60 M30	N60 M30

Fanuc G72.2 Linear Copy (Figure Copy Function CNC Mill)

Fanuc G72.2 Linear Copy

Using G72.2 Linear Copy G-code a figure specified by a subprogram can be repeatedly produced with **Linear** movement.

Contents

- Programming
 - Parameters
- Programming Notes
 - Notes
 - First block of the subprogram
 - Limitation
- G72.2 Programming Example

Programming

G72.2 P... L... I... J...

Parameters

Parameter	Description
P	Subprogram number
L	Number of times the operation is repeated
I	Shift along X-axis
J	Shift along Y-axis

G-Code Data

Modal/Non-Modal	G-Code Group
Non-Modal	00

Programming Notes

Notes

- In the G72.2 block, addresses other than P, L, I and J are ignored.
- P, I and J must always be specified.
- If L is not specified, the figure is copied once.
- For shifts (I, J), specify increments. The n-th geometric shift is equal to the specified shift times (n – 1).

First block of the subprogram

Always specify a move command in the first block of a subprogram that performs a linear copy. If the first block contains only the program number such as O00001234; and does not have a move command, movement may stop at the start point of the figure made by the n-th (n = 1,2, 3, ...) copying.

Example of an incorrect program

```
O00001234 ;
G00 G90 X100.0 Y200.0 ;
____;
____;
____;
M99 ;
```

Example of a correct program

```
O00001000 G00 G90 X100.0 Y200.0 ;
____;
____;
____;
M99 ;
```

Limitation

Specifying two or more commands to copy a figure

G72.2 cannot be specified more than once in a subprogram for making a linear copy (If this is attempted, alarm PS0901 will occur).

In a subprogram that specifies linear copy, however, rotational copy (G72.1) can be specified. Similarly, in a subprogram that specifies rotational copy, linear copy can be specified.

Commands that must not be specified

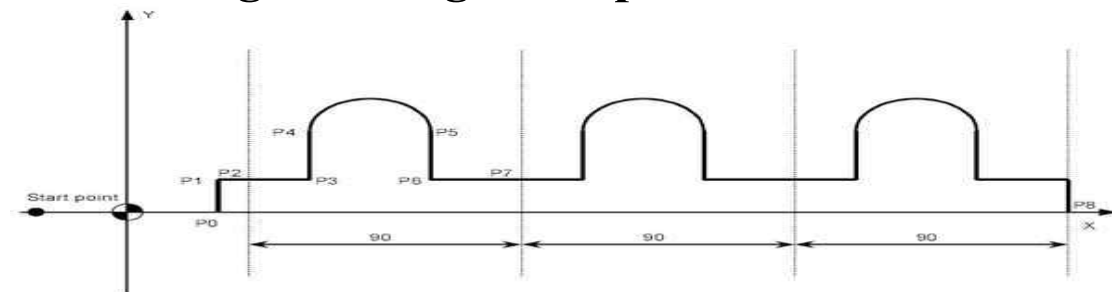
Within a program that performs a linear copy, the following must not be specified:

- Command for changing the selected plane (G17 to G19)
- Command for specifying polar coordinates (G16)
- Reference position return command(G28)
- Axis switching
- Coordinate system rotation (G68)
- scaling (G51)
- programmable mirror image (G51.1)

Single block

Single-block stops are not performed in a block with G721.1 or G72.2.

G72.2 Programming Example



Main program

```

O3000 ;
N10 G90 G00 X-30. Y0 ;
N20 X0 ;
N30 G01 G17 G41 X30. D01 F100 ; (P0)
N40 Y20. ; (P1)
N50 X40. ; (P2)
N60 G72.2 P3100 L3 I90.0 J0 ;
N70 G90 X310. Y0 ; (P8)
N80 X0 ;
N90 G40 G00 X-30.0 ;
N100 M30 ;

```

Sub program

```

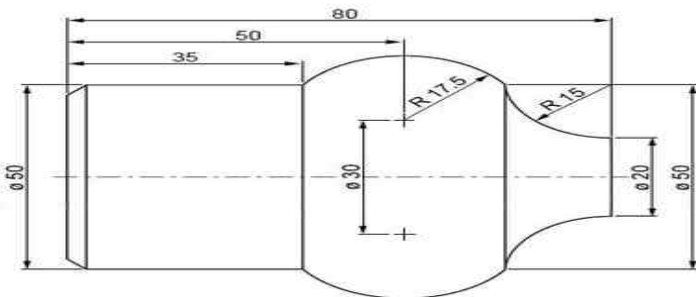
O3100 G91 G01 X20. ; (P3)
N100 Y30. ; (P4)
N200 G02 X40. I20. ; (P5)
N300 G01 Y-30. ; (P6)
N400 X30. ; (P7)
N500 M99 ;

```

G02 G03 Programming Example

CNC program example to show how two combined arc can be programmed with G02 G03 for a cnc lathe.

First arc is programmed using G02 R, but the second arc is programmed using G03 I K



G02 G03 Programming Example

G02 G03 Programming Example

```

N50 G00 X20 Z85
N60 G96 S200
N70 G01 Z80
N80 G02 X50 Z65 R15
N90 G03 X50 Z35 I-10 K-15
N100 G00 X80
N110 G97 S900 M05
N120 M30

```

Quincunx a CNC Plasma Cutting Controller Program Example

CNC program example to cut Quincunx on a CNC Plasma Cutting Controller. This program example is programmed in G91 Incremental programming.

Contents

- What is a CNC Plasma Cutting Machine
 - Plasma Cutting
 - CNC Plasma Cutter
- Plasma Cutting Controller Program Example

What is a CNC Plasma Cutting Machine

Plasma Cutting

Plasma cutting is a process that is used to cut steel and other metals of different thicknesses (or sometimes other materials) using a plasma torch. In this process, an inert gas (in some units, compressed air) is blown at high speed out of a nozzle; at the same time an electrical arc is formed through that gas from the nozzle to the surface being cut, turning some of that gas to plasma. The plasma is sufficiently hot to melt the metal being cut and moves sufficiently fast to blow molten metal away from the cut.

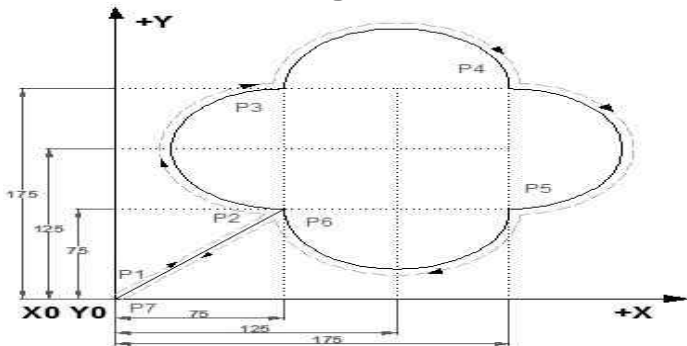


CNC Plasma Cutting Machine

CNC Plasma Cutter

A “CNC plasma” system is a machine that carries a plasma torch, and can move that torch in a path directed by CNC (Computer Numerical Control), which means that a computer is used to direct the machines motion based on numerical codes in a program.

Plasma Cutting Controller Program Example



Quincunx a CNC Plasma Cutting Controller Program Example

Relative coordinate programming

```
G92 X0 Y0(P1)
G00 X75 Y75 (P2)
G02 Y100 I0 J50 (P3)
G02 X100 I50 J0 (P4)
G02 Y-100 I0 J-50(P5)
G02 X-100 I-50 J0(P6)
G00 X-75 Y-75(P7)
M02
```

Wire EDM Programming Example

Complete cnc programming example for machinists who work on a Wire EDM machine (Electrical Discharge Machining)

Contents

- CNC Wire Cut Introduction
- Wire EDM Programming Example

CNC Wire Cut Introduction

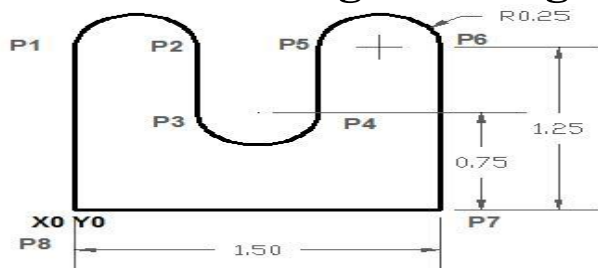
Wire EDM cutting, also known as electrical discharge machining, is a process that uses an electrically energized thin wire to slice through metal. Wire EDM cutting uses rapid, controlled, repetitive spark discharges from the wire to the workpiece, thereby eroding the metal away. The workpiece must be electrically conductive.



wire edm machine

Wire EDM cutting can provide high dimensional accuracy for close fitting parts. The process can make sharp inside corners.

Wire EDM Programming Example



Wire EDM Programming Example

```
G92 X0. Y0.  
G01 Y1.25 (P1)  
G02 X.5 I.25 J0.(P2)  
G01 Y.75(P3)  
G03 X1. I.25 J0.(P4)  
G01 Y1.25 (P5)  
G02 X1.5 I.25 J0. (P6)  
G01 Y0. (P7)  
X0. (P8)  
M02
```

G92 command is used to assign program zero. You will include the distances and directions from the program zero point to the wire start point in your G92 command.

CNC Program Example G03 Circular Interpolation

Now a day CAD and CAM software make our lives just easier and easier, you just not have to calculate a single point in drawing, every thing is just ready. But it is a good practice to know actually how the calculations are being done, if not all but some which are necessary for a CNC programmer.



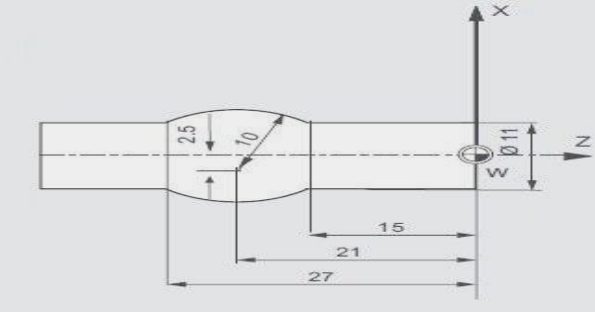
CNC Program Example G03 Circular Interpolation

The calculation of G02 G03 circular interpolation values is a good technique to learn. This CNC program tells not all but a little of course enough for the beginner CNC programmer and CNC students.

Contents

- CNC Program With G03 Circular Interpolation
 - Description
 - I Calculation for G03 Circular Interpolation
 - K Calculation for G03 Circular Interpolation

CNC Program With G03 Circular Interpolation



CNC Program Example G03 Circular Interpolation

```
N1 G00 X0 Z0
N2 G01 X11
N3 G01 Z-15
N4 G03 X11 Z-27 I-8 K-6
```

Description

The I and K for the G03.

I Calculation for G03 Circular Interpolation

I is the distance from the arc start point to arc center point in x-axis. This value is not given in diameter but as a radius value, so

- Find the start diameter it is 11
- Now make it radius value by dividing it by 2 such as $11 / 2 = 5.5$
- This is a easy example in way that the arc center point distance from the workpiece center point is given other wise we have to calculate it.
- So now just add 2.5 in 5.5 earlier calculated $2.5 + 5.5 = 8$
- This is the value of I 8 this is the distance from the arc start point to the arc center point.

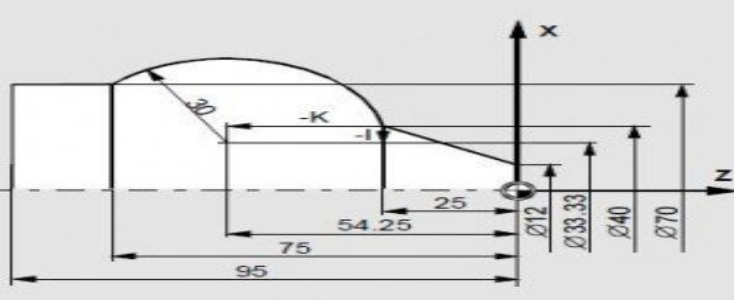
K Calculation for G03 Circular Interpolation

K is the distance from the arc start point to arc center point in z-axis.

- In this example it is just easy as you just have to subtract 21 from 15 such as $21 - 15 = 6$.
- So the value from the arc start point to the arc center point in z-axis is 6

CNC Arc Programming Exercise

CNC arc programming exercise, shows how to cnc program an arc with I and K. CNC G-code G03 is used to program this arc.



CNC Arc Programming Exercise

```

N10 G00 X0 Z0
N20 G01 X12 F0.3
N30 G01 X40 Z-25
N40 G03 X70 Z-75 I-3.335 K-29.25
N50 G01 Z-95
N60 G00 X200 Z200

```

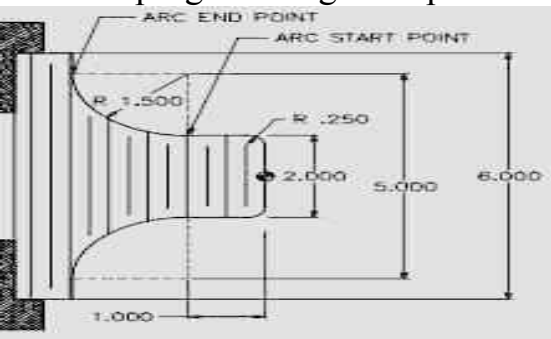
Fanuc G20 Measuring in Inches with CNC Program Example

Fanuc G20 Measuring in Inches or Programming in inches. As cnc machines gives us ease to program and machine any type of component.

The same way the cnc machine controls also give us the utmost flexibility that we can program not only in **metric system** but also in **inches system**.

On Fanuc cnc control **G20** and **G21** are used to change between the inches and metric system.

This cnc programming example demonstrate the usage of Fanuc G20 G code.



Fanuc G20 Program Example

Fanuc G20 CNC Program Example

```

N1 T0101
N2 G97 S800 M03

```

N3 G96 S150 G20
N4 G00 X0 Z1
N5 G01 Z0 G95 F0.3
N6 G01 X2 R0.25
N7 G01 Z-1
N8 G02 X5 Z-2.5 I1.5 K0
N9 G01 X6
N10 G00 X10 Z10
N11 M30

G02 G03 R

CNC Mill Example Program G01 G02 G03 G90 G91

A cnc mill program for cnc machinists programmers, who have started to learning basic cnc programming techniques.

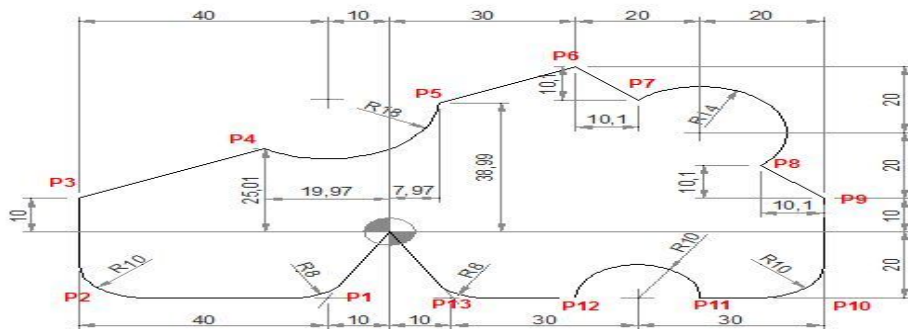
Contents

2 CNC Mill Example Program

CNC Program

G M S T Codes Explanation

CNC Mill Example Program



CNC Program

```
N40 G90 G00 X0 Y0
N50 G01 X-10 Y-20 R8 (P1)
N60 G01 X-50 R10 (P2)
N70 Y10 (P3)
N80 X-19.97 Y25.01 (P4)
N90 G03 X7.97 Y38.99 R18 (P5)
N100 G01 X30 Y50 (P6)
N110 G91 X10.1 Y-10.1(P7)
N120 G90 G02 X59.9 Y20.1 R14 (P8)
N130 G01 X70 Y10 (P9)
N140 Y-20 R10(P10)
N150 X50 (P11)
N160 G03 X30 R10 (P12)
N170 G01 X10 R8 (P13)
N180 X0 Y0
```

G M S T Codes Explanation

--	--

Code	Description
G00	Rapid traverse
G01	Linear interpolation
G02	Circular interpolation CW
G03	Circular interpolation CCW
G90	Absolute command
G91	Increment command

CNC Mill Subprogram Example

Joining Multiple Arcs G02 G03 G41

CNC milling program to describe how two or more radii can be joint together in a cnc mill program.

Contents

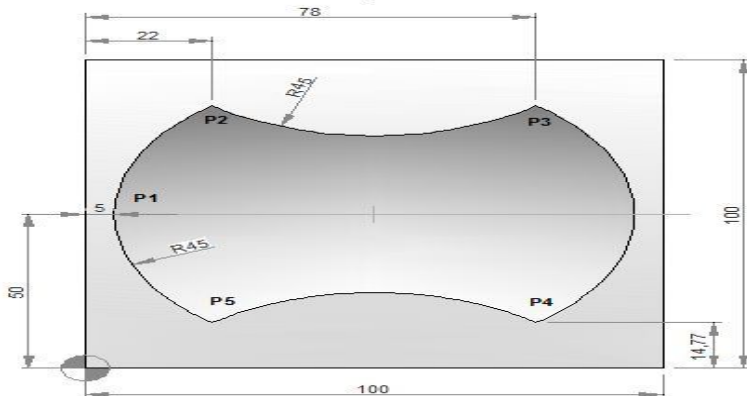
5 CNC Mill Subprogram Example

CNC Part Program

Subprogram

G M S T Codes Explanation

CNC Mill Subprogram Example



CNC Part Program

```

N10 T1 H1 M6 G43 M3
N20 F150 S250
N30 G0 X-21 Y50 Z0.5
N40 G0 Z0
N50 M98 P040050
N60 G49

```

N70 G0 Z50

N80 M30

Subprogram

O0050

N10 F160 S400

N20 G0 Z-2.5 G91

N30 G1 G90 X5 Y50 G41 (P1)

N40 G2 X22 Y85.23 I45 J0 (P2)

N50 G3 X78 Y85.23 R45 (P3)

N60 G2 X78 Y14.77 R45 (P4)

N70 G3 X22 Y14.77 R45 (P5)

N80 G2 X5 Y50 R45 (P1)

N90 G0 G40 X-21

N100 M99

G M S T Codes Explanation

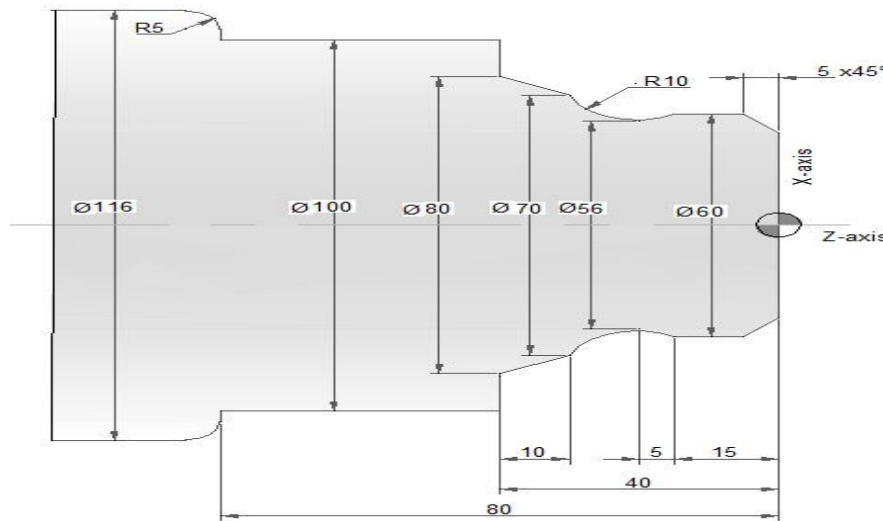
Code	Description
G00	Rapid traverse
G01	Linear interpolation
G02	Circular interpolation CW
G03	Circular interpolation CCW
G40	Cutter compensation cancel
G41	Tool nose radius compensation left
G43	Tool length compensation + direction
G49	Tool length compensation cancel
G90	Absolute command
G91	Increment command
M03	Spindle start forward CW
M06	Tool change
M30	End of program (Reset)
M98	Subprogram call
M99	End of subprogram
T	Tool
S	Speed
F	Feed

CNC Turning Center Programming Example

Easy to understand CNC turning center/cnc lathe programming example for cnc machinists who work on cnc turning centers/cnc lathe machines.

This cnc programming example can be used as a cnc learning programming exercise for beginners level cnc programmers/machinists.

CNC Turning Center Programming Example

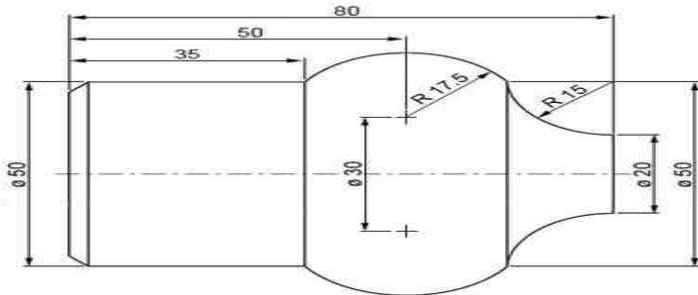


```
N10 T03 D03 M06
N20 G97 S900 M04
N30 G00 G42 X40 Z5
N40 G92 S3500
N50 G96 S250
N60 G01 X60 Z-5 F0.1
N70 Z-15
N80 X56 Z-20
N90 G02 X70 Z-30 R10
N100 G01 X80 Z-40
N110 X100
N120 Z-80
N130 X106
N140 G03 X116 Z-85 R5
N150 G01 X120
N160 G00 X150 Z100 G40 G97 S900 M05
N170 M30
```

G02 G03 Programming Example

CNC program example to show how two combined arc can be programmed with G02 G03 for a cnc lathe.

First arc is programmed using G02 R, but the second arc is programmed using G03 I K



G02 G03 Programming Example

G02 G03 Programming Example

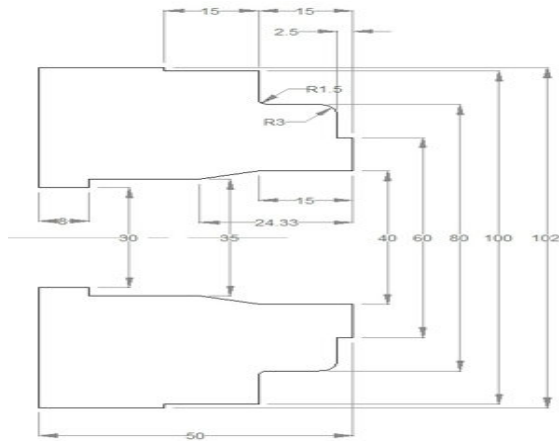
```
N50 G00 X20 Z85  
N60 G96 S200  
N70 G01 Z80  
N80 G02 X50 Z65 R15  
N90 G03 X50 Z35 I-10 K-15  
N100 G00 X80  
N110 G97 S900 M05  
N120 M30
```

CNC Lathe Basic Programming Example ID/OD Turning/Boring Operations (No Canned Cycle Used)

A full CNC programming example with ID/OD (Turning/Boring operations) for cnc machinists who work on a cnc lathe machine. A must to learn/practice for those who are learning cnc programming.

The added benefit of this programming example is that no cnc lathe canned cycle is used in this programming example.

CNC Lathe Basic Programming Example (Turning Boring Operations)



CNC Lathe Example Turning Boring

O0000

N10 (ø30 DRILL)

G50 T0200

G97 S250 M03

G00 X0 Z5.0 T0202 M08

G01 Z-5.0 F0.07

W1.0

Z-40.0 F0.25

G00 Z5.0

Z-39.0

G01 Z-60.0

G00 Z10.0

X200.0 Z200.0 T0200

M01

N20 (Outside diameter stock removal)

G50 S1500 T0100

G96 S180 M03

G00 X94.0 Z5.0 T0101 M08

G01 Z-14.8 F0.27

G00 U2.0 Z0.5

G01 X28.0 F0.23

G00 X87.0 W1.0

G01 Z-14.8 F0.27

G00 U2.0 Z1.0

X80.5

G01 Z-14.1

G02 X81.9 Z-14.8 R0.7

G00 X100.5 W1.0

G01 Z-29.8

G00 U2.0 Z-1.0

G01 X60.5 F0.23

G00 X82.0 W1.0

Z-2.4

G01 X60.5

X72.9

G03 X80.5 Z-6.2 R3.8

G00 U2.0 Z5.0

X200.0 Z200.0 T0100

M01

N30 (Inside diameter stock removal)

G50 S1500 T0400

G96 S180 M03

G00 X34.5 Z3.0 T0404 M08

G01 Z-41.8 F0.27

G00 U-0.5 Z1.0

X39.5

G01 Z-15.0

X34.5 Z-24.3

G00 Z10.0

X200.0 Z200.0 T0400

M01

N40 (Out diameter finishing)

G50 S1800 T0500

G96 S200 M03

G00 X63.0 Z5.0 T0505 M08

Z0

G01 X38.0 F0.2

G00 X60.0 Z3.0

G42 Z1.0

G01 Z-2.5 F0.2

X74.0

G03 X80.0 Z-5.5 R3.0

G01 Z-13.5

G02 X83.0 Z-15.0 R1.5

G01 X100.0

Z-30.0

X102.0

G40 G00 U2.0 W1.0

G00 Z10.0

X200.0 Z200.0 T0500

M01

N50 (Inside diameter finishing)

G50 S1800 T0600

G96 S200 M03

G00 X40.0 Z5.0 T0606 M08

G41 Z1.0

G01 Z-15.0 F0.2

X35.0 Z-24.33

Z-42.0

X29.0

G40 G00 Z10.0

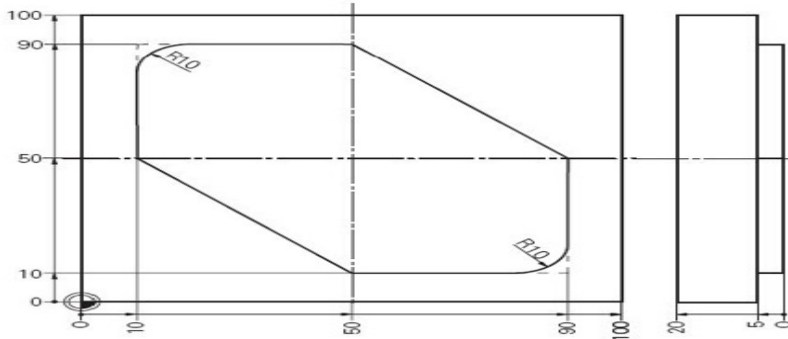
X200.0 Z200.0 T0600 M09

M30

Vertical Machining Center Programming Example CNC

Vertical machining centers are a vital part of a cnc machine workshop, So here is a Vertical machining center programming example for cnc machinists who have just started learning cnc programming.

Vertical Machining Center Programming Example



Vertical Machining Center Programming Example

```

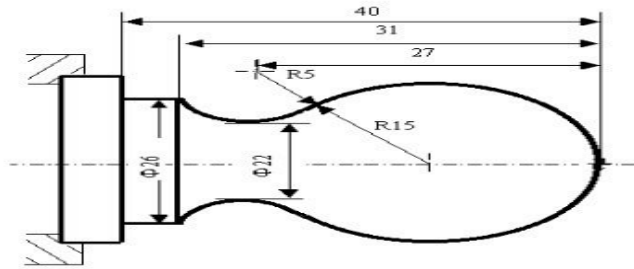
N10 T1
N20 S1000 M3
N30 G00 G40 G90 G17 X-30 Y60
N40 G43 Z50 H1
N50 Z-5
N60 G01 G41 X10 Y60 D1 F200
N70 Y80
N80 G02 X20 Y90 R10
N90 G01 X50
N100 X90 Y50
N110 Y20
N120 G02 X80 Y10 R10
N130 G01 X50
N140 X10 Y50
N150 Y65
N160 G40 X-30 Y60
N170 G00 Z200 M02

```

CNC Arc Programming G02 G03 Example

CNC arc programming example this cnc program shows how two arcs G03 G02 can be joint together.

CNC Arc Programming G02 G03 Example



CNC Arc Programming G02 G03 Example

O0001

N001 G0 X40 Z5 (Rapid position)

N002 M03 S200 (Start spindle)

N003 G01 X0 Z0 F900 (Approach workpiece)

N005 G03 U24 W-24 R15 (Cut R15 arc)

N006 G02 X26 Z-31 R5 (Cut R5 arc)

N007 G01 Z-40 (Cut Z,26)

N008 X40 Z5 (Return to starting point)

N009 M30 (End of program)

Fanuc G73 Pattern Repeating Cycle CNC Program Example Code

CNC programming example for Fanuc G73 pattern repeating cycle.

Fanuc G73 Pattern Repeating Cycle has already been described here

CNC Fanuc G73 Pattern Repeating Cycle

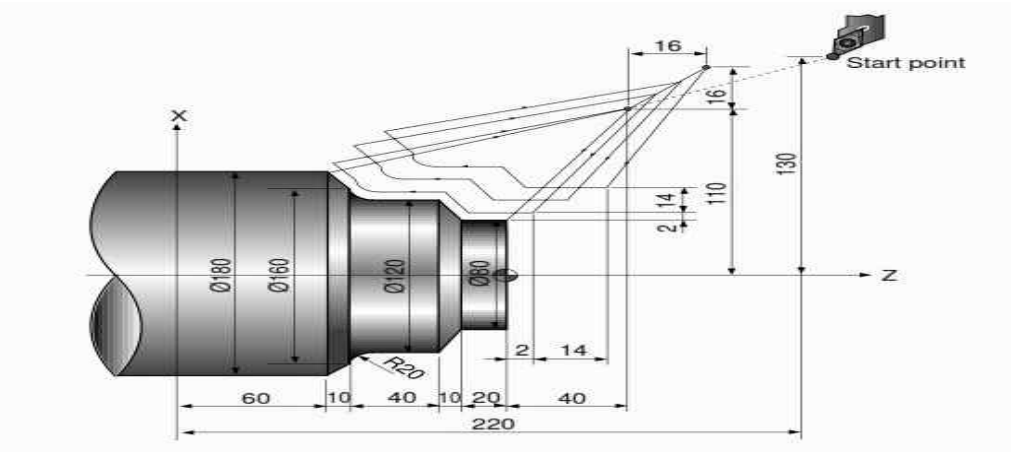
You might like other Fanuc G73 pattern repeating cycle examples

CNC Fanuc G73 Pattern Repeating Cycle CNC Program Example

Fanuc G73 Pattern Repeating Canned Cycle Basic CNC Sample Program

Fanuc G73 Pattern Repeating Cycle Programming Example

This cnc program example also shows how cnc machinists can use 'W' instead of 'Z' for z-axis movements.



Fanuc G73 Pattern Repeating Cycle Program Example

```

N010 G00 X260.0 Z80.0
N011 G00 X220.0 Z40.0
N012 G73 U14.0 W14.0 R3
N013 G73 P014 Q020 U4.0 W2.0 F0.3 S0180
N014 G00 G42 X80.0 Z2.0
N015 G01 W-20.0 F0.15 S0600
N016 X120.0 W-10.0
N017 W-20.0 S0400
N018 G02 X160.0 W-20.0 R20.0
N019 G01 X180.0 W-10.0 S0280
N020 G40
N021 G70 P014 Q020
N022 G00 X260.0 Z80.0
N023 M30

```

Fanuc G73 Pattern Repeating Canned Cycle Basic CNC Sample Program

Fanuc G73 pattern repeating cycle helps cnc machinists to program/maintain/debug rough material removal programs easy.

Other Fanuc canned cycle like
 G71 Longitudinal cutting cycle or
 G72 Facing Cycle

removes the rough material by taking straight cuts.

But Fanuc G73 Pattern repeating cycle removes the rough material by taking cuts along the contour.

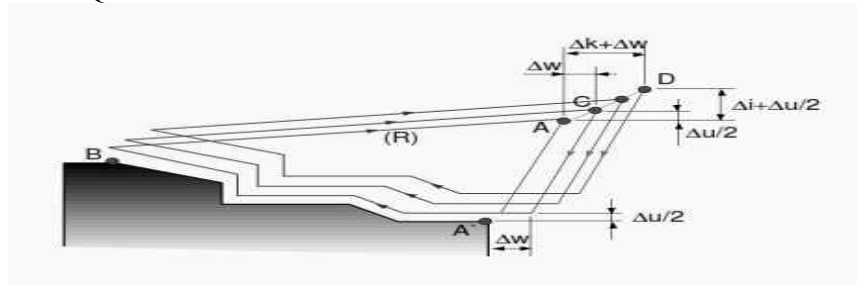
So from the first cut G73 pattern repeating cycle cuts the same shape as the finish component.

Contents

- FANUC G73 Pattern Repeating Cycle Format
- FANUC G73 Pattern Repeating Cycle Program Example

FANUC G73 Pattern Repeating Cycle Format

G73 U W R
G73 P Q U W F



FANUC G73 Pattern Repeating Canned Cycle

G73 First CNC Program Block

U – Escape distance and direction in X axis (radius value). This is the amount of material which will be cut in x-axis.

W – Escape distance and direction in Z axis. This is the amount of material which will be cut in z-axis.

R – Number of divisions. The number the contour will be repeated.

G73 Second CNC Program Block.

P – Start sequence No.

Q – End sequence No.

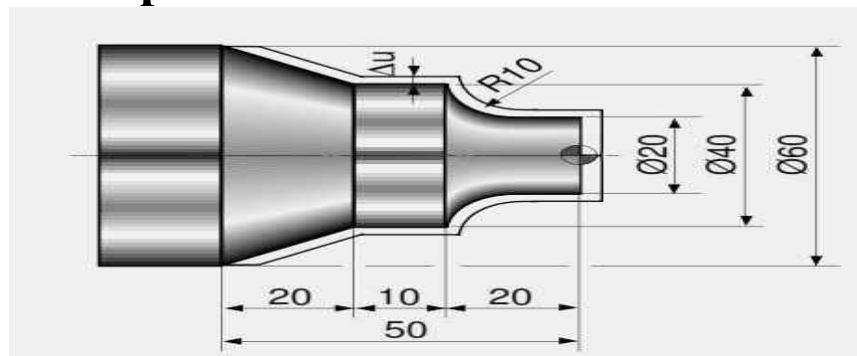
U – Finishing allowance in x-axis.

W – Finishing allowance in z-axis.

F – Cutting Feedrate (during G73 canned cycle).

FANUC G73 Pattern Repeating Cycle Program

Example



FANUC G73 Pattern Repeating Canned Cycle Basic CNC Sample Program

N10 G50 S2000 T0300

G96 S200 M03

G00 X35.0 Z5.0 T0303

```

Z0
G01 X-1.6 F0.2
G00 X70.0 Z10.0
G73 U3.0 W2.0 R2
G73 P12 Q16 U0.5 W0.1 F0.25
N12 G00 G42 X20.0 Z2.0
G01 Z-10.0 F0.15
G02 X40.0 Z-20.0 R10.0
G01 Z-30.0
X60.0 Z-50.0
N16 G40 U1.0
G70 P12 Q16
G00 X200.0 Z200.0 T0300
M30

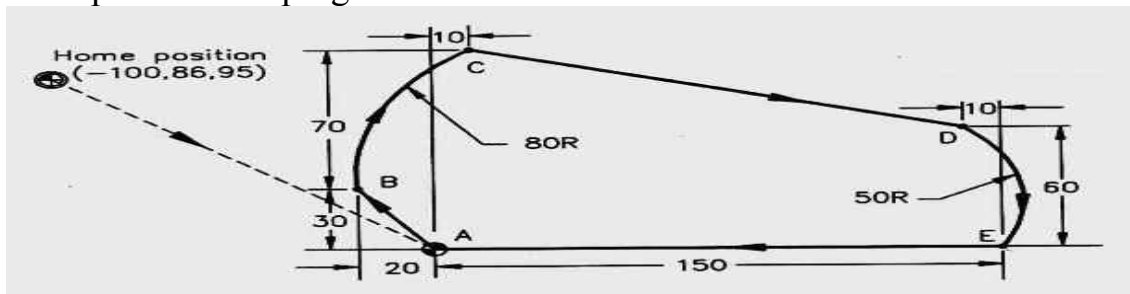
```

CNC Milling Sample Program

CNC milling sample program with program code explanation.

This cnc milling machine programming tutorial shows how Circular Interpolation G2 / G3 G-code are programmed. As well as programming of straight lines for cnc milling machine.

This cnc milling sample program is easy to understand without the help of any CNC professional programmer.



CNC Milling Sample Program

```

N5 G90 G71
N10 T1 M6
N15 G92 X-100 Y86 Z95
N20 G0 X0 Y0 S2500 M3
N25 Z12.5
N30 G1 Z-12.5 F150
N35 X-20 Y30
N40 G2 X10 Y100 R80
N45 G1 X140 Y60
N50 G2 X150 Y0 R50
N55 G1 X0 Y0
N60 G0 Z12.5
N65 G91 G28 Z0 M5
N70 G91 G28 X0 Y0

```

N75 M30
N5 absolute positioning, metric unit
N10 tool change to T1
N15 define work zero point at A
N20 rapid traverse to A, spindle on (2500 RPM, CW)
N25 rapid plunge to 12.5 mm above Z0
N30 feed to Z-12.5, feed rate 150 MMPM
N35 cut line AB to B
N40 cut arc BC to C
N45 cut line CD to D
N50 cut arc DE to E
N55 cut line EA to A
N60 rapid retract to Z12.5
N65 reference point return in Z direction, spindle off
N70 reference point return in X and Y directions
N75 end of program

CNC Mill Programming Absolute Incremental G90 G91 Example Code

Following is a cnc program example code for cnc milling machine.

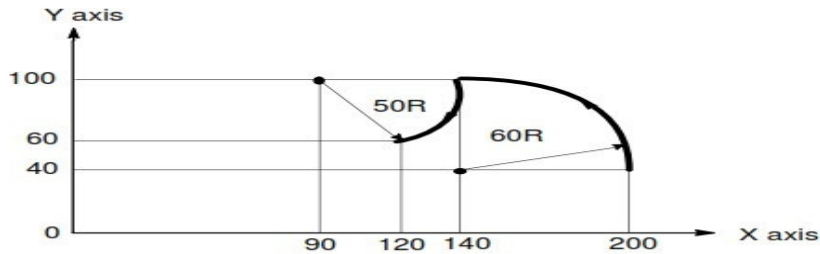
This cnc example code illustrates the usage of CNC Absolute Programming G90 G-Code and Incremental Programming G91 G-Code, as well as the usage of Circular Interpolation G-Code (G02/G03).

This cnc program draws (machines) cnc mill Arc with R as well as Arc with I.

Contents

- CNC Absolute Programming G90 Example Code
- CNC Incremental Programming G91 Example Code

CNC Absolute Programming G90 Example Code



CNC Mill Programming Absolute Incremental G90 G91 Example Code

Mill Circular Interpolation G02 G03 with R

```
G92 X200 Y40 Z0
G90 G03 X140 Y100 R60 F300
G02 X120 Y60 R50
```

Mill Circular Interpolation G02 G03 with I

```
G92 X200 Y40 Z0
G90 G03 X140 Y100 I-60 F300
G02 X120 Y60 I-50
```

CNC Incremental Programming G91 Example Code

Mill Circular Interpolation G02 G03 with R

```
G91 G03 X-60 Y60 R60 F300
G02 X-20 Y-40 R50
```

Mill Circular Interpolation G02 G03 with I

```
G91 G03 X-60 Y60 I-60 F300
G02 X-20 Y-40 I-50
```

CNC Circular Interpolation Tutorial

G02 G03

This cnc circular interpolation tutorial will easily make you learn how to use G02 G03 G-code.

CNC programming of a component is easy if you divide the component on drawing in small pieces, and cnc program every piece one by one.

At the end you will see that whole the process is same, whether you are machining a simple component or a very complex one.

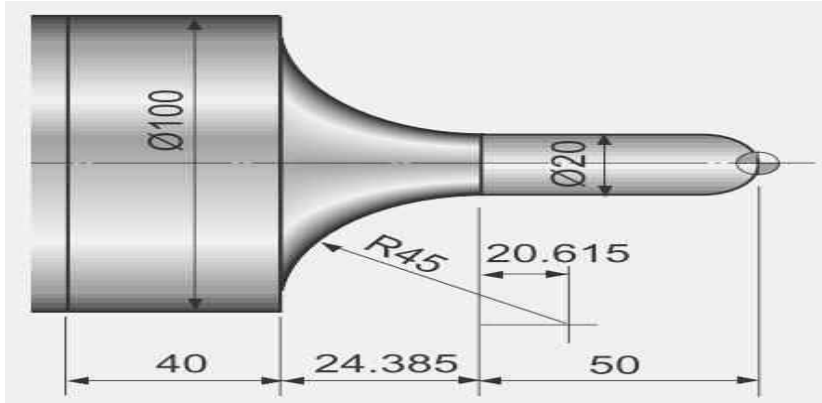
The idea is same you will just use G00, G01, G02, G03 and so on.

Other related cnc programming examples with circular interpolation G02, G03

- CNC Programming for Beginners a CNC Programming Example
- CNC Lathe Programming Example
- CNC Arc Programming Exercise

The below is a cnc circular interpolation tutorial which uses G02 and G03 cnc g-code.

CNC Programming Example



CNC Programming Example with Circular Interpolation G02 G03

```

N20 G50 S2000 T0300
N30 G96 S200 M03
N40 G00 X0 Z3.0 T0303 M08
N50 G42 G01 Z0 F0.2
N60 G03 X20.0 Z-10.0 R10.0
N70 G01 Z-50.0
N80 G02 X100.0 Z-74.385 I40.0 K20.615
N80 G01 Z-125.0
N90 G40 U2.0 W1.0
N100 G00 X200.0 Z200.0 M09 T0300
N110 M30

```

You can use R or I K with G02/G03 G-codes see N80 in both cnc program examples

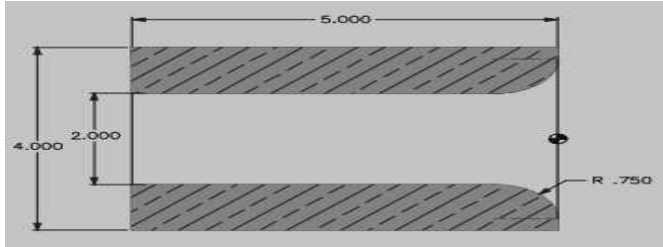
```

N20 G50 S2000 T0300
N30 G96 S200 M03
N40 G00 X0 Z3.0 T0303 M08
N50 G42 G01 Z0 F0.2
N60 G03 X20.0 Z-10.0 R10.0
N70 G01 Z-50.0
N80 G02 X100.0 Z-74.385 R45.0
N80 G01 Z-125.0
N90 G40 U2.0 W1.0
N100 G00 X200.0 Z200.0 M09 T0300
N110 M30

```

CNC Programming for Beginners a CNC Programming Example

Here is a cnc programming example for beginners, this cnc programming example is a starting step for **cnc learning** or **CNC Programming for Beginners**. Here you will find plenty of *free cnc programming examples* with component drawings. This cnc programming example explains the cnc boring with cnc boring bar tool.



CNC Programming for Beginners a CNC Programming Example CNC Program Example

```
N1 T01 G20
N2 G00 X3.5 Z0.5
N3 G01 G96 S120 Z0 F.5
N4 G02 X2 Z-.75 R0.75 F0.15
N5 G01 Z-5 F0.2
N6 G01 X1.85
N7 G00 Z20
N8 M30
```

CNC Lathe Programming Example

A simple cnc lathe programming example. This cnc programming example will show how to program contours like *chamfer taper grooves and arc*. This cnc program will use two tools for machining. The first tool used in this cnc programming example is a **turning tool** and the second one is a **grooving tool**.

Contents

- CNC Machining
- CNC Program

CNC Machining

The **turning tool** will first *face the component* then it will make a *chamfer*, after that there is *straight turning* and then there is *arc machining* the arc may be machined with **R (Radius of arc)** or arc can be machined with **I and K values of the arc**, at the top of arc there is a chamfer, then again straight turning, now there comes the *groove* but **we can't make grooves with turning tools because of their shape** so we will machine the groove with our next tool which is solely made for grooving operations, but at the time we are machining with turning tool

so we will just skip this groove and will just machine in straight line, after that there is a taper and again a straight line to be machined.

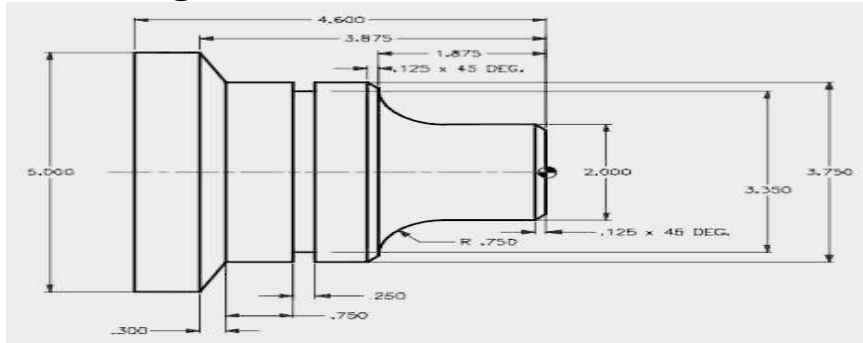
Now with our **grooving tool** the groove machining task is just easy.

if the *grooving insert* is of the same size as the dimension of the groove then

grooving is even more easy, we will make the groove in one go, but if

the *grooving insert* is of smaller width than we have to take depth more than one time.

CNC Program



cnc lathe program example

```
N1 T01 D01 M491
N2 G00 X0 Z1
N3 G01 G96 G41 Z0 F2 S140
N4 G01 X2 CHF=0.125 F0.2
N5 G01 Z-1.125
N6 G02 X3.5 Z-1.875 CR=0.75
N7 G01 X3.75 CHF=0.125
N8 G01 Z-3.575
N9 G01 X5 Z-3.875
N10 G01 Z-4.6
N11 G00 X20 Z20 G40
N12 T02 D02 M491
N13 G00 G97 S500 X4 Z-2.825
N14 G01 X3.85 F1
N15 G01 X3.35 F0.15
N16 G01 X3.85 F0.5
N17 G00 X4
N18 G00 X20 Z20
N19 M30
```


G40 G41 G42

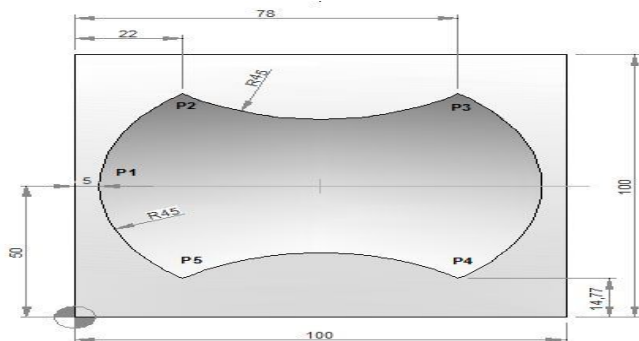
CNC Mill Subprogram Example Joining Multiple Arcs G02 G03 G41

CNC milling program to describe how two or more radii can be joint together in a cnc mill program.

Contents

- 3 CNC Mill Subprogram Example
 - CNC Part Program
 - Subprogram
 - G M S T Codes Explanation

CNC Mill Subprogram Example



CNC Part Program

```
N10 T1 H1 M6 G43 M3
N20 F150 S250
N30 G0 X-21 Y50 Z0.5
N40 G0 Z0
N50 M98 P040050
N60 G49
N70 G0 Z50
N80 M30
```

Subprogram

```
O0050
N10 F160 S400
N20 G0 Z-2.5 G91
N30 G1 G90 X5 Y50 G41 (P1)
N40 G2 X22 Y85.23 I45 J0 (P2)
N50 G3 X78 Y85.23 R45 (P3)
```

N60 G2 X78 Y14.77 R45 (P4)
N70 G3 X22 Y14.77 R45 (P5)
N80 G2 X5 Y50 R45 (P1)
N90 G0 G40 X-21
N100 M99

G M S T Codes Explanation

Code	Description
G00	Rapid traverse
G01	Linear interpolation
G02	Circular interpolation CW
G03	Circular interpolation CCW
G40	Cutter compensation cancel
G41	Tool nose radius compensation left
G43	Tool length compensation + direction
G49	Tool length compensation cancel
G90	Absolute command
G91	Increment command
M03	Spindle start forward CW
M06	Tool change
M30	End of program (Reset)
M98	Subprogram call
M99	End of subprogram
T	Tool
S	Speed
F	Feed

CNC Mill Program G91 G41 G43

CNC milling program examples shows the use of G91 G41 G43 G-codes.

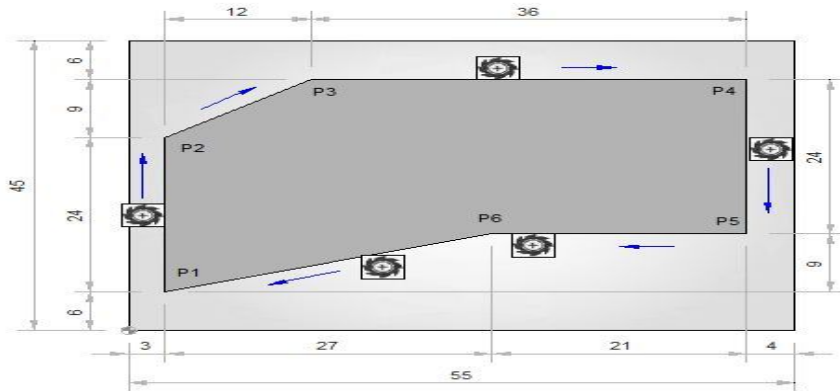
Contents

6 CNC Mill Program G91 G41 G43

CNC Part Program

G M S T Codes Explanation

CNC Mill Program G91 G41 G43



CNC Part Program

```

N05 G54
N10 M6 T1 G43 H1 M3
N15 S500 F120
N20 G0 X-22 Y-22
N25 Z-3
N30 G1 X3 Y6 G41 H2 (P1)
N35 G91 X0 Y24(P2)
N40 X12 Y9(P3)
N45 X36 (P4)
N50 Y-24 (P5)
N55 X-21 (P6)
N60 G90 X3 Y6 (P1)
N65 G0 X-21 G40

```

G M S T Codes Explanation

Code	Description
G00	Rapid traverse
G01	Linear interpolation
G40	Cutter compensation cancel
G41	Tool nose radius compensation left
G43	Tool length compensation + direction
G54	Workpiece coordinate system 1 selection
G90	Absolute command
G91	Incremental command
M06	Tool change
T	Tool
S	Speed
F	Feed

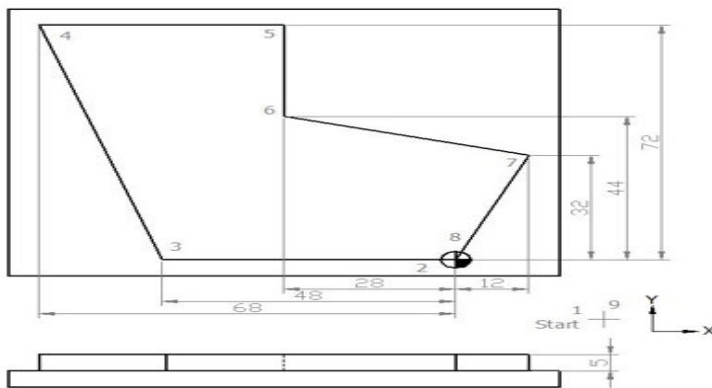
G41 G40 Cutter Radius Compensation Example CNC Mill Program

Cutter Radius Compensation Example program shows how G41, G40 can be used in a cnc mill program.

Cutter Compensation code used in this program are,

- G41 Cutter Radius Compensation Left
- G40 Cutter Radius Compensation Cancel

Cutter Radius Compensation Example



G41 G40 Cutter Radius Compensation Example

```
N5 G00 G54 G64 G90 G17 X20 Y-20 Z50
```

```
N10 S450 M03 F250 D01 (12.5 MM DIA)
```

```
N15 C0
```

```
N20 Z5
```

```
N25 G01 Z0
```

```
N30 Z-5
```

```
N35 G41 X0 Y0
```

```
N40 X-48
```

```
N45 X-68 Y72
```

```
N50 X-28
```

```
N55 Y44
```

```
N60 X12 Y32
```

```
N65 X0 Y0
```

```
N70 G40 X20 Y-20
```

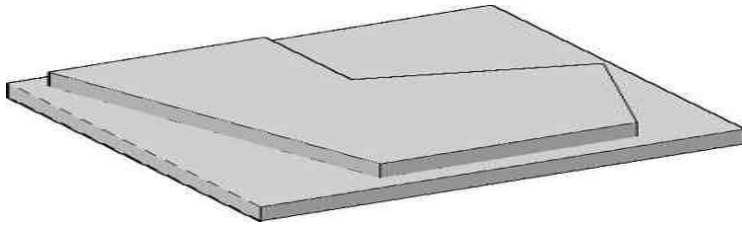
```
N75 G00 Z50
```

```
N80 Y100
```

```
N85 M30
```

Finished Part

After machining process completion, component will look like



Cutter Radius Compensation Example Finished Part

Explanation of CNC G-Code

G00 : Rapid traverse.

G54 : Zero Offset no. 1.

G64 : Continuous-path mode.

G90 : Absolute dimensioning system.

G17 : X-Y plan selection.

G41 : Cutter radius compensation activation (left hand side movement)

G40 : Cutter radius compensation de-active

S : Spindle speed

F : Axis motion feed

M : Cutter rotation (3=clockwise, 4=anti-clockwise)

D : Tool offset no

Siemens Sinumerik Milling Programming Example

A very simple cnc milling program example which will show cnc machinists the use of Siemens Sinumerik milling programming concepts.

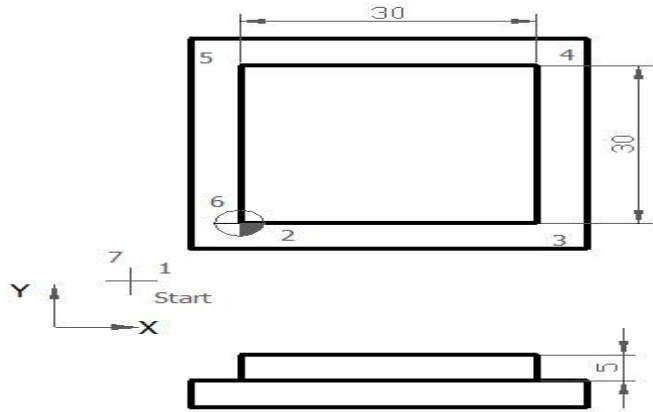
This program is written for 4-axis cnc mill, where C is used for rotary table.

But a simple cnc mill can also run this program just remove program block N15

Contents

- Sinumerik Milling Program
Finished Part
Explanation of G-Code

Sinumerik Milling Program



Sinumerik Mill Programming Example

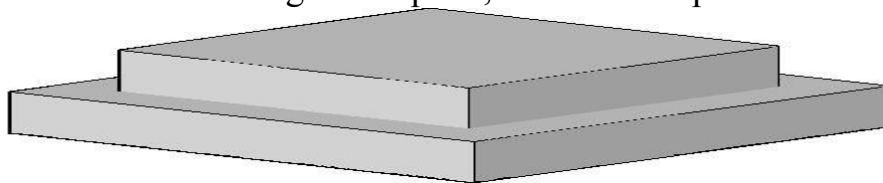
```

N5 G00 G54 G64 G90 G17 X-20 Y-20 Z50
N10 S450 M03 F250 D01 (12.5 MM DIA)
N15 C0
N20 Z5
N25 G01 Z0
N30 Z-5
N35 G42 X0 Y0
N40 X30
N45 Y30
N50 X0
N55 Y0
N60 G40 X-20 Y-20
N65 G00 Z50
N70 Y100
N75 M30

```

Finished Part

After the machining is complete, this finished part will look like this



Finished Part

Explanation of G-Code

- G00 – Rapid traverse.
- G54 – Zero Offset no 1.
- G64 – Continuous-path mode.
- G90 – Absolute dimensioning system.
- G17 – X-Y plan selection.
- G42 – Cutter radius compensation activation
- G40 – Cutter radius compensation cancel
- M03 – Cutter rotation clockwise
- S – Spindle speed

F – Axis motion feed
D – Tool no

CNC Lathe Programming Exercise

Fanuc G71 Turning Cycle, G74 Peck Drilling Cycle

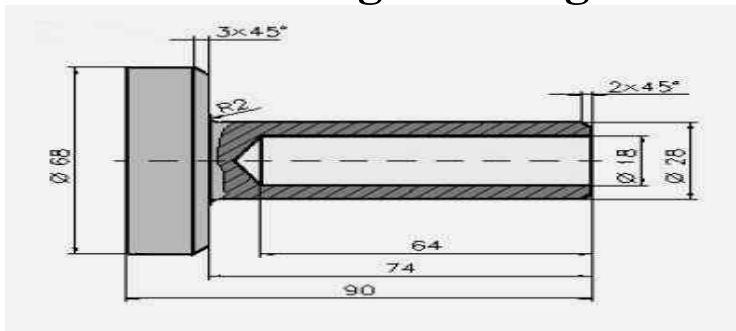
CNC programming exercise for cnc lathe machinists who work on Fanuc cnc control (or similar cnc control).

This cnc programming exercise use
Fanuc G71 Turning Cycle
Fanuc G74 Peck Drilling Cycle

Contents

- CNC Lathe Programming Exercise
 - Used Tools & Operations

CNC Lathe Programming Exercise



CNC Lathe Programming Exercise Fanuc G71, G74 Cycles

N10 G40 G00

N20 G99

N60 T0101

N70 G50 S3500

N80 G96 S0240 M4

N90 G00 X72. Z0.1

N100 G01 X-1.6 F0.12 M7

N110 G00 X150. Z150.

N120 M5

N130 M9

N140 T0303

N150 G97 S2500 M3

N160 G00 X0. Z3.

N170 G01 Z-6. F0.1 M7
N180 G00 Z2.
N190 G00 X150. Z150.
N210 T0707 M7
N220 G97 S0884 M3
N230 G00 Z3.
N240 G00 X0.
N250 G74 R1.0
N260 G74 X0.0 Z-68.326 Q18000 F0.22
N380 G00 X200.
N400 G00 Z100.
N500 T0404 M7
N510 G50 S3500
N520 G96 S0240 M4
N530 G00 Z1.
N540 G00 X70.
N550 G71 U4. R1
N560 G71 P570 Q650 U0.6 W0.2 F0.35
N570 G42 G00 X24.
N580 G01 Z0.
N590 G01 X28. Z-2.
N600 G01 Z-72.
N610 G02 X32. Z-74. I2. K0.
N620 G01 X62.
N630 G01 X68. Z-77.
N640 G01 Z-90.
N650 G40
N660 G00 X150.
N680 G00 Z70.
N690 T0202 M7
N700 G50 S4500
N710 G96 S0380 M4
N720 G00 X16. Z3.
N730 G42 G01 Z0. F0.1
N740 G01 X24.
N750 G01 X28. Z-2.
N760 G01 Z-72.
N770 G02 X32. Z-74. I2. K0.
N780 G01 X62.
N790 G01 X68. Z-77.
N800 G01 Z-90.
N810 G40
N820 G00 X150. Z150.
N830 M5
N840 M9
N850 M30

Used Tools & Operations

- T0101 Turning Tool – Rough Facing
- T0303 Center Drill – Center Drilling
- T0707 Twist Drill – Drilling
- T0404 Turning Tool – Rough Turning
- T0202 Turning Tool – Finish Contour Cutting

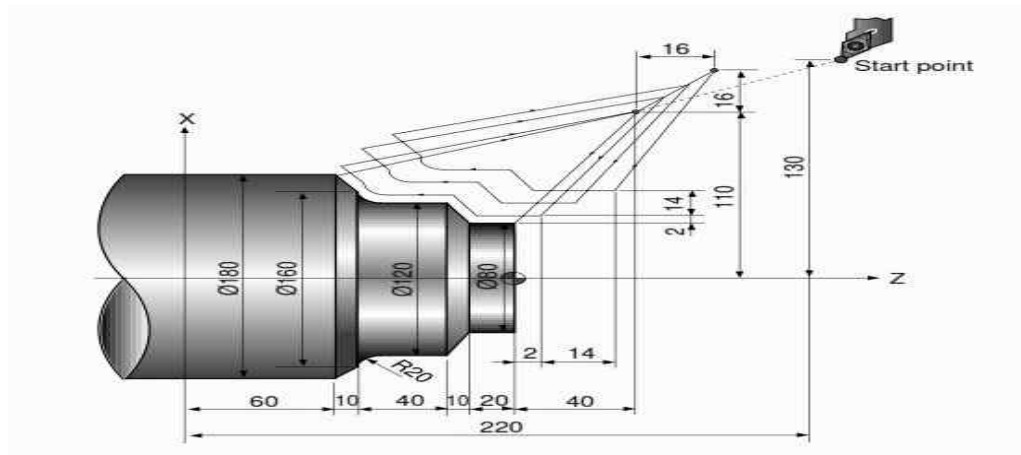
Fanuc G73 Pattern Repeating Cycle CNC Program Example Code

CNC programming example for Fanuc G73 pattern repeating cycle.
Fanuc G73 Pattern Repeating Cycle has already been described here
CNC Fanuc G73 Pattern Repeating Cycle

You might like other Fanuc G73 pattern repeating cycle examples
CNC Fanuc G73 Pattern Repeating Cycle CNC Program Example
Fanuc G73 Pattern Repeating Canned Cycle Basic CNC Sample Program

Fanuc G73 Pattern Repeating Cycle Programming Example

This cnc program example also shows how cnc machinists can use 'W' instead of 'Z' for z-axis movements.



Fanuc G73 Pattern Repeating Cycle Program Example

```
N010 G00 X260.0 Z80.0
N011 G00 X220.0 Z40.0
N012 G73 U14.0 W14.0 R3
N013 G73 P014 Q020 U4.0 W2.0 F0.3 S0180
N014 G00 G42 X80.0 Z2.0
N015 G01 W-20.0 F0.15 S0600
N016 X120.0 W-10.0
N017 W-20.0 S0400
N018 G02 X160.0 W-20.0 R20.0
N019 G01 X180.0 W-10.0 S0280
N020 G40
N021 G70 P014 Q020
N022 G00 X260.0 Z80.0
```

Fanuc G73 Pattern Repeating Canned Cycle Basic CNC Sample Program

Fanuc G73 pattern repeating cycle helps cnc machinists to program/maintain/debug rough material removal programs easy.

Other Fanuc canned cycle like
G71 Longitudinal cutting cycle or
G72 Facing Cycle

removes the rough material by taking straight cuts.

But Fanuc G73 Pattern repeating cycle removes the rough material by taking cuts along the contour.

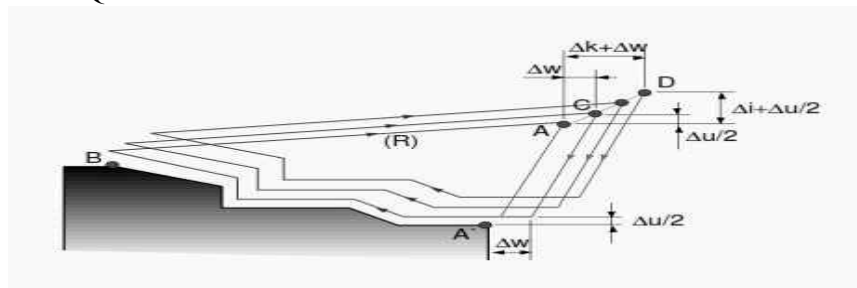
So from the first cut G73 pattern repeating cycle cuts the same shape as the finish component.

Contents

- Fanuc G73 Pattern Repeating Cycle Format
- Fanuc G73 Pattern Repeating Cycle Program Example

Fanuc G73 Pattern Repeating Cycle Format

G73 U W R
G73 P Q U W F



Fanuc G73 Pattern Repeating Canned Cycle

G73 First CNC Program Block

U – Escape distance and direction in X axis (radius value). This is the amount of material which will be cut in x-axis.

W – Escape distance and direction in Z axis. This is the amount of material which will be cut in z-axis.

R – Number of divisions. The number the contour will be repeated.

G73 Second CNC Program Block.

P – Start sequence No.

Q – End sequence No.

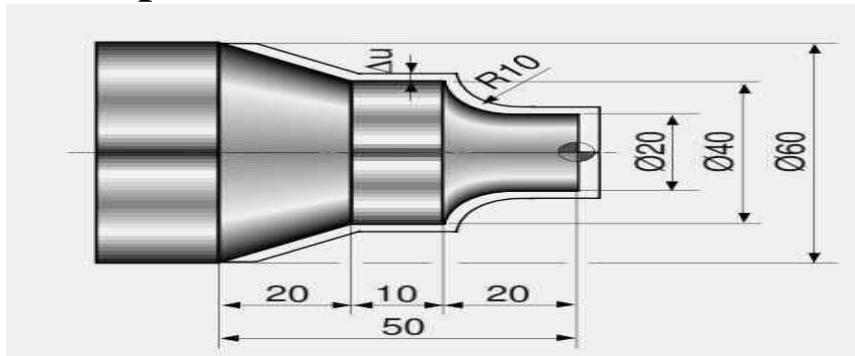
U – Finishing allowance in x-axis.

W – Finishing allowance in z-axis.

F – Cutting Feedrate (during G73 canned cycle).

Fanuc G73 Pattern Repeating Cycle Program

Example



Fanuc G73 Pattern Repeating Canned Cycle Basic CNC Sample Program

```
N10 G50 S2000 T0300
```

```
G96 S200 M03
```

```
G00 X35.0 Z5.0 T0303
```

```
Z0
```

```
G01 X-1.6 F0.2
```

```
G00 X70.0 Z10.0
```

```
G73 U3.0 W2.0 R2
```

```
G73 P12 Q16 U0.5 W0.1 F0.25
```

```
N12 G00 G42 X20.0 Z2.0
```

```
G01 Z-10.0 F0.15
```

```
G02 X40.0 Z-20.0 R10.0
```

```
G01 Z-30.0
```

```
X60.0 Z-50.0
```

```
N16 G40 U1.0
```

```
G70 P12 Q16
```

```
G00 X200.0 Z200.0 T0300
```

```
M30
```

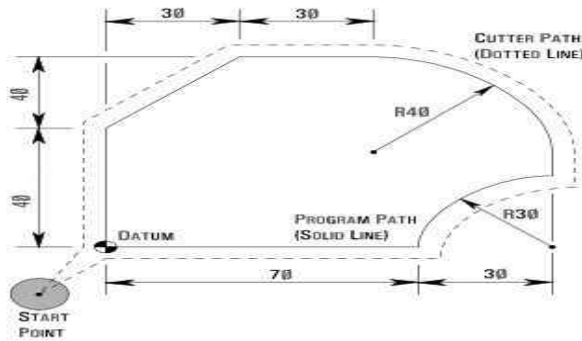
CNC Machining Center Programming

Example

This is a CNC machining center programming example or Vertical machining centers (VMCs) programming exercise.

Basic and easy to understand for beginners level cnc machinists / cnc programmers.

Also illustrates the use of CNC G-code **G41 Cutter Radius Compensation Left**.
CNC Machining Center Programming Example



cnc machining center programming example

```
O0010
N10 G00 X-15 Y-15
N20 G41 G01 X0 Y0 F100
N30 Y40
N40 X30 Y80
N50 X60
N60 G02 X100 Y40 R40
N70 G01 Y30
N80 G03 X70 Y0 R30
N90 G01 X0
N100 X-15 Y-15
```

G02 G03 G Code Circular Interpolation Example Program

This article explains the use of G02 G03 G code. The G code G02, G03 are used for Circular Interpolation in cnc programming.

Contents

- G02 G03 G Code Circular Interpolation
- G02 G03 G Code Example CNC Programs (G code Arc Examples)
- G02 G03 G Code Example Program
- G Code G02 G03 I & K Example Program

G02 G03 G Code Circular Interpolation

G02 G Code Clock wise Circular Interpolation.

G03 G Code Counter Clock wise Circular Interpolation.

There are multiple articles/cnc program examples about G code circular interpolation, here is the list of few articles so that cnc machinists can easily

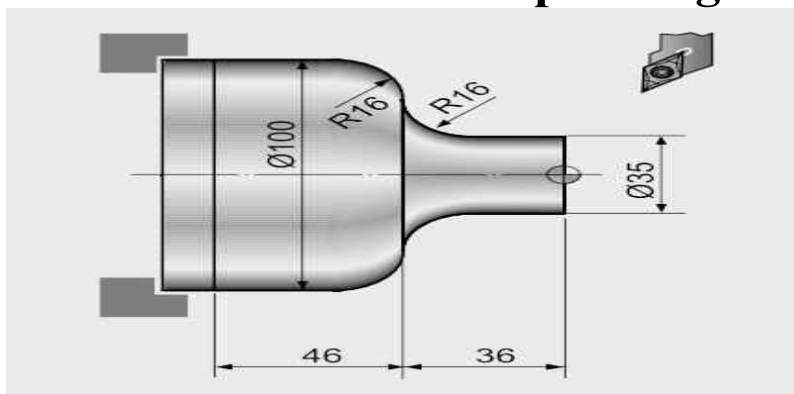
navigate through different cnc programming articles.

G02 G03 G Code Example CNC Programs (G code Arc Examples)

- CNC Circular Interpolation Tutorial G02 G03
- Fancu CNC Lathe Programming Example
- CNC Programming Example G Code G02 Circular Interpolation Clockwise
- Fancu G20 Measuring in Inches with CNC Program Example
- CNC Arc Programming Exercise
- CNC Programming for Beginners a CNC Programming Example
- CNC Lathe Programming Example

Here is a new cnc programming examples which shows the use of G02 G03 G code circular interpolation.

G02 G03 G Code Example Program



G02 G03 G Code Circular Interpolation Example Program

```
N20 G50 S2000 T0300
G96 S200 M03
G42 G00 X35.0 Z5.0 T0303 M08
G01 Z-20.0 F0.2
G02 X67.0 Z-36.0 R16.0
G01 X68.0 :
G03 X100.0 Z-52.0 R16.0
G01 Z-82.0
G40 G00 X200.0 Z200.0 M09 T0300
M30
```

G Code G02 G03 I & K Example Program

G02 G03 G Code Circular Interpolation can be programmed in two ways,

```
G02 X... Z... R...
G02 X... Z... I... K...
```

The below is the same cnc program but this version uses I & K with G02 G03 G code.

```
N20 G50 S2000 T0300
G96 S200 M03
G42 G00 X35.0 Z5.0 T0303 M08
```

```
G01 Z-20.0 F0.2
G02 X67.0 Z-36.0 I16.0 K0
G01 X68.0 :
G03 X100.0 Z-52.0 I0 K-16.0
G01 Z-82.0
G40 G00 X200.0 Z200.0 M09 T0300
M30
```

CNC Circular Interpolation Tutorial

G02 G03

This cnc circular interpolation tutorial will easily make you learn how to use G02 G03 G-code.

CNC programming of a component is easy if you divide the component on drawing in small pieces, and cnc program every piece one by one.

At the end you will see that whole the process is same, whether you are machining a simple component or a very complex one.

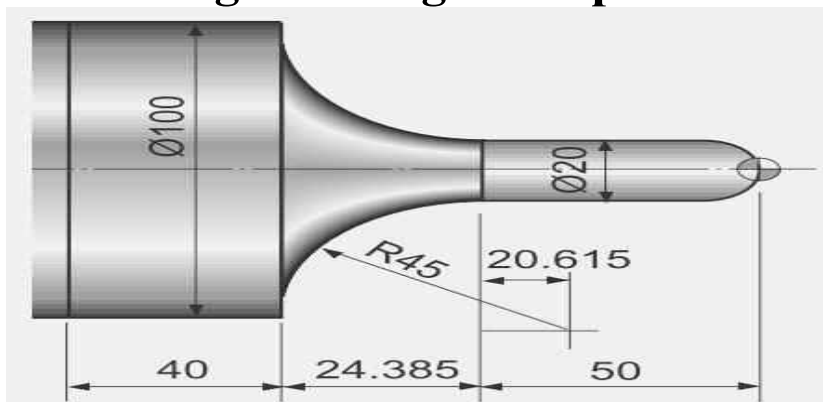
The idea is same you will just use G00, G01, G02, G03 and so on.

Other related cnc programming examples with circular interpolation G02, G03

- CNC Programming for Beginners a CNC Programming Example
- CNC Lathe Programming Example
- CNC Arc Programming Exercise

The below is a cnc circular interpolation tutorial which uses G02 and G03 cnc g-code.

CNC Programming Example



CNC Programming Example with Circular Interpolation G02 G03

```
N20 G50 S2000 T0300
N30 G96 S200 M03
N40 G00 X0 Z3.0 T0303 M08
N50 G42 G01 Z0 F0.2
```

```

N60 G03 X20.0 Z-10.0 R10.0
N70 G01 Z-50.0
N80 G02 X100.0 Z-74.385 I40.0 K20.615
N80 G01 Z-125.0
N90 G40 U2.0 W1.0
N100 G00 X200.0 Z200.0 M09 T0300
N110 M30

```

You can use R or I K with G02/G03 G-codes see N80 in both cnc program examples

```

N20 G50 S2000 T0300
N30 G96 S200 M03
N40 G00 X0 Z3.0 T0303 M08
N50 G42 G01 Z0 F0.2
N60 G03 X20.0 Z-10.0 R10.0
N70 G01 Z-50.0
N80 G02 X100.0 Z-74.385 R45.0
N80 G01 Z-125.0
N90 G40 U2.0 W1.0
N100 G00 X200.0 Z200.0 M09 T0300
N110 M30

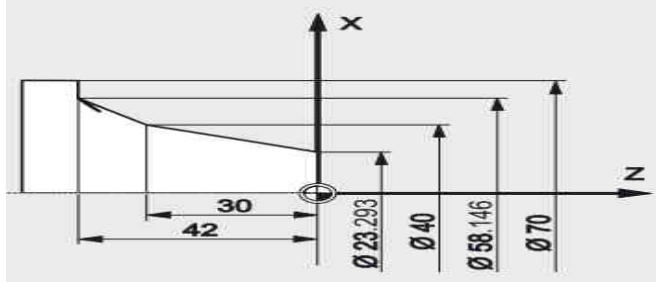
```

Fanuc G21 Measuring in Millimeter with CNC Lathe Programming Example

Fanuc G21 Measuring in Millimeter or Programming in Millimeter. CNC gives us the flexibility to program in **metric system** or in **inches system**.

Fanuc **G21** G-code changes cnc to metric system.

So here is another cnc programming example, for *beginner level cnc programmers*. This cnc programming example uses **Fanuc G21 G-code**.



Fanuc G21 Programming Example

```

N1 T0505
N2 G92 S1500 M03

```

```

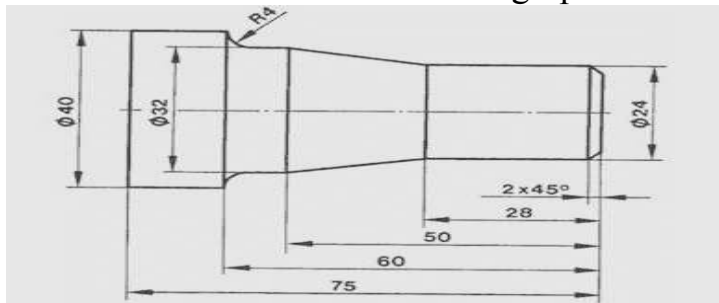
N3 G21 G96 S150
N4 G00 X0 Z5
N5 G42 G01 Z0 G95 F0.3
N6 G01 X23.293
N7 G01 X40 Z-30
N8 G01 X58.146 Z-42
N9 G01 X70
N10 G40 G00 X100 Z100 G97 S500

```

Fanuc CNC Program Example

Here is a cnc program example for Fanuc cnc control. This is a very simple and easy cnc program example also shows

- Use of G02 Arc/Radius in Fanuc cnc program
- Use of Chamfer in Fanuc cnc program
- Use of G42/G40 Tool Nose Compensation
- Use of G92 Maximum Spindle Speed
- Use of G96 Constant Cutting Speed



Fanuc CNC Program Example

Fanuc CNC Program Code

```

N10 T2
N20 G92 S1200 M42
N30 G96 S150 M04
N40 G00 X-1 Z5 M08
N50 G01 Z0 G42 F0.2
N60 G01 X24 C2
N70 G01 Z-28
N80 G01 X32 Z-50
N90 G01 Z-56
N100 G02 X40 Z-60 R4
N110 G01 Z-75
N120 G01 X60 G40
N130 G00 X150 Z100
N140 M30

```


G81 Drilling Cycle

Fanuc G72.1 G72.2 Figure Copy Program Example (Bolt Hole Circle)

This CNC program example shows how both G72.1 and G72.2 figure copy functions can call one-another in one part program, read Programming Notes below carefully.

Programming

Fanuc G72.1 Rotational Copy

Using G72.1 Rotational Copy G-code a figure specified by a subprogram can be repeatedly produced with **Rotational movement**.

Read G72.1 definition with program example Fanuc G72.1 Rotational Copy (Figure Copy Function CNC Mill)

Fanuc G72.2 Linear Copy

Using G72.2 Linear Copy G-code a figure specified by a subprogram can be repeatedly produced with **Linear movement**.

Read G72.2 definition with program example Fanuc G72.2 Linear Copy (Figure Copy Function CNC Mill)

Fanuc G81 Drilling Cycle

G81 drilling cycle is used for simple drilling/spot drilling operations.

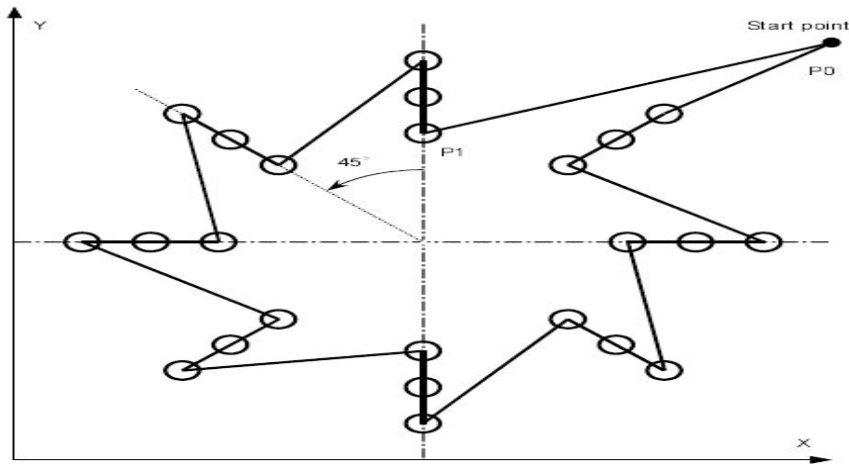
Read G81 drilling cycle definition with program examples Fanuc G81 Drilling Cycle

Programming Notes

Specifying two or more commands to copy a figure

- G72.1 cannot be specified more than once in a subprogram for making a rotational copy (If this is attempted, alarm PS0900 will occur).
- G72.2 cannot be specified more than once in a subprogram for making a linear copy (If this is attempted, alarm PS0901 will occur).
- In a subprogram that specifies rotational copy, however, linear copy can be specified. Similarly, in a subprogram that specifies linear copy, rotational copy can be specified.

Fanuc G71.2 G72.2 Program Example



Main program

```
O4000 ;  
N10 G90 G00 G17 X240. Y230. Z100. ; (P0)  
N20 G72.1 P4100 X120. Y120. L8 R45. ;  
N30 G80 G00 X240. Y230. ; (P0)  
N40 M30 ;
```

Sub program_ Rotation copy_ (G72.1)

```
O4100 N100 G72.2 P4200 I0 J20. L3 ;  
N200 M99 ;
```

Sub program_ Linear copy_ (G72.2_)

```
O4200 N110 G90 G81 X120. Y180. R60. Z10. F200. ; (P1)  
N210 M99 ;
```

Fanuc G72.1 Rotational Copy Program Example

Fanuc G72.1 Rotational copy programming example, G72.1 G-code is used to repeatedly produce a figure with rotational movement.

Contents

- Fanuc G72.1 Rotational Copy
- Fanuc G81 Drilling Cycle
- Fanuc G72.1 Program Example

Fanuc G72.1 Rotational Copy

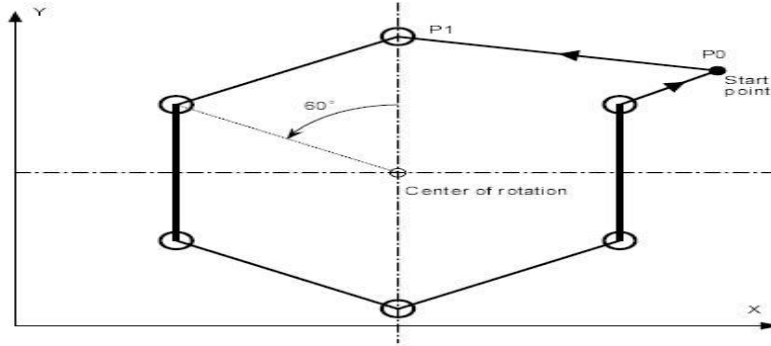
Using G72.1 Rotational Copy G-code a figure specified by a subprogram can be repeatedly produced with Rotational movement.

Read more Fanuc G72.1 Rotational Copy (Figure Copy Function CNC Mill)

Fanuc G81 Drilling Cycle

G81 drilling cycle is used for simple drilling/spot drilling operations.
 Read complete article with program examples Fanuc G81 Drilling Cycle

Fanuc G72.1 Program Example



Main program

```
O2000 ;
N10 G90 G00 G17 X250. Y100. Z100. ; (P0)
N20 G72.1 P2100 L6 X100. Y50. R60. ;
N30 G80 G00 X250. Y100. ; (P0)
N40 M30 ;
```

Sub program

```
O2100 N100 G90 G81 X100. Y150. R60. Z10. F200. ; (P1)
N200 M99 ;
```

Fanuc G81 Drilling Cycle

G81 drilling cycle is used for simple drilling/spot drilling operations.

Contents

- Syntax
- Usage
- Working
- G98 G99 Modes
 - Example
- Repeat Drilling
- Working Examples
- G98 G99 Example
- Repeat Drilling Example

Syntax

G81 X... Y... Z... R... K... F...

Parameter	Description

X	Hole position in x-axis.
Y	Hole position in y-axis.
Z	Depth, tool will travel with feed to Z-depth starting from R plane.
R	Position of the R plane.
K	Number of cycle repetitions (if required)_____.
F	Feedrate.

Once G81 drilling cycle is defined, the canned cycle is repeated at every X-Y position in sequential blocks. So G81 drilling cycle must be cancelled with G80.

Usage

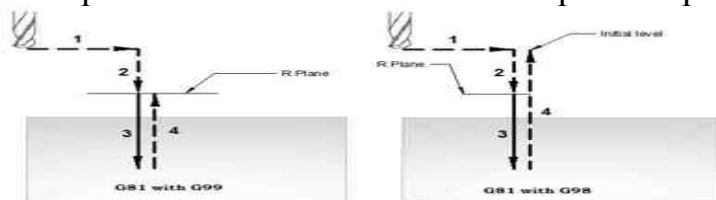
```
N30 G81 X10 Y30 Z-17 R2 F75
N40 Y10
N50 X30
N60 Y30
N70 X90
N80 Y10
N90 G80
```

In the above example drilling will start with G81 drilling cycle at X10 Y30, so first drill will be at X10 Y30, then second at Y10, third at X30, fourth at Y30, fifth at X90 and the last one at Y10, because next block have G80 code, so drilling cycle will no more be repeated.

Working

Here is briefly described how G81 drilling cycle operates,

- 1- Rapid traverse to the specified x,y axis position (drilling position).
- 2- Rapid traverse to the R plane position.
- 3- Drilling with specified Feed from R-plane position to Z-depth position.
- 4- Rapid traverse to Initial level or R-plane depends on G98, G99 modes.



G81 drilling cycle working

G98 G99 Modes

After completing drilling depth the return is made with Rapid feed, the return height can be controlled through using G98 or G99.

G98 Drill will return to the Initial level

G99 Drill will return to R-plane.

G98, G99 can be used multiple times during G81 drilling cycle.

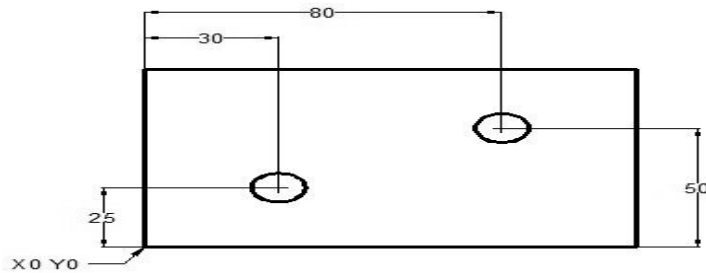
Example

```
N30 G81 X10 Y30 Z-17 R2 F75  
N40 Y10  
N50 G98 X30  
N60 G99 Y30  
N70 X90  
N80 Y10  
N90 G80
```

Repeat Drilling

With G81 drilling cycle drilling operation can be repeated multiple times. The drilling is repeated K times when that parameter is given with G81 drilling cycle. Repeat drilling is normally used with G91 Incremental mode, and a good example of repeated drilling is Grid-plate drilling. the example for repeat drilling is given below.

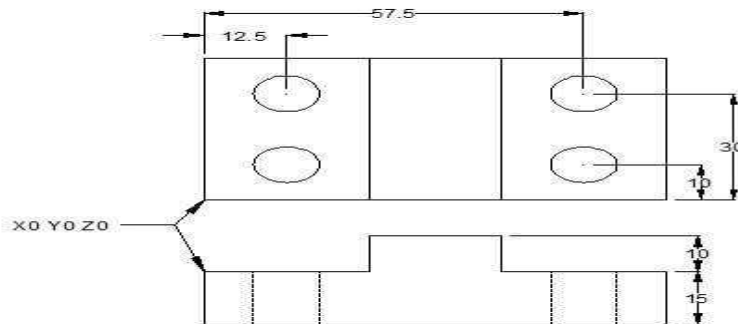
Working Examples



G81 Drilling Cycle Example

```
N10 T1 M06  
N20 G90 G54 G00 X30 Y25  
N30 S1200 M03  
N40 G43 H01 Z5 M08  
N50 G81 Z-10 R2 F75  
N60 X80 Y50  
N70 G80 G00 Z100 M09  
N80 M30
```

G98 G99 Example



G81 drilling cycle usage with G98 G99

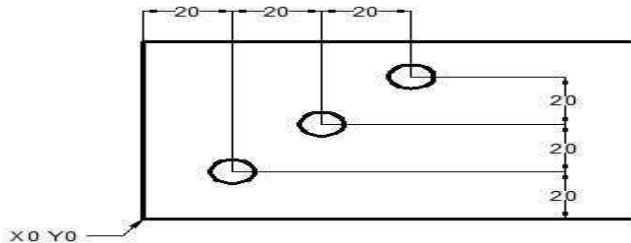
```
N10 M06 T1  
N20 G90 G00 X12.5 Y10 Z12 S1000 M03
```

```

N30 G99 G81 X12.5 Y10 Z-17 R2 F75
N40 Y30
N50 G98 X57.5
N60 G99 Y10
N70 G91 G80 G28 X0 Y0 Z0 M05
N80 M30

```

Repeat Drilling Example



Repeat drilling with G81 Drilling Cycle

```

T1 M6
G00 G90 G40 G21 G17 G94
G54 X0 Y0 S1000 M03
G43 H1 Z100
Z3
G81 G99 G91 X20 Y20 R3 Z-20 K3 F100 M08
G80
G00 G90 Z100
M30

```

OR

```

T1 M6
G00 G90 G40 G21 G17 G94
G54 X20 Y20 S1000 M03
G43 H1 Z100
Z3
G81 G99 R3 Z-20 F100 M08
G91 X20 Y20 K2
G80
G00 G90 Z100
M30

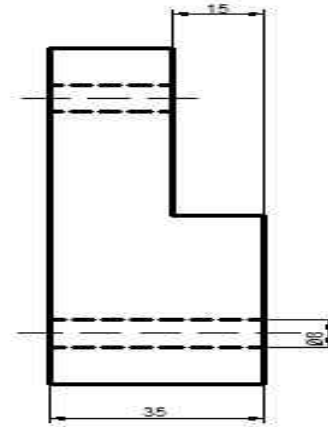
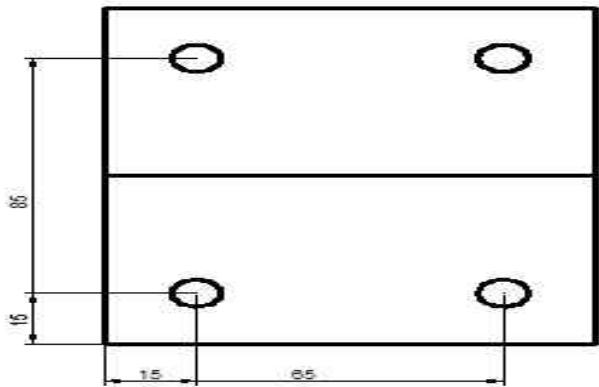
```

Drilling a Two Step Block with G81 Drilling Cycle

G81 drilling cycle program example which shows how a step block can be drilled economically.

The cnc program code is self explanatory, no complex technique is used.

G81 Drilling Two Step Block



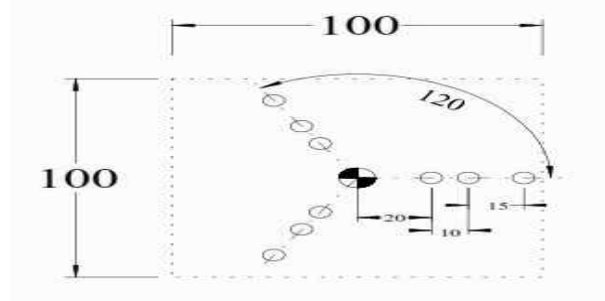
G81 Drilling Two Step Block

N10 T4 M6 (TWIST DRILL 8")
 N15 G90 G54 G00 X15 Y15
 N20 S1000 M3 F100
 N25 G43 H01 Z2 M8
 N30 G81 R2 Z-42
 N35 X65
 N40 Y85 R-13
 N45 X15
 N50 G80 Z50 M5
 N55 M30

Fanuc G68 Coordinate Rotation Program Example

Fanuc G68 Coordinate Rotation G-Code makes it easy for cnc machinist to run a pattern of operations in a rotated angle. Here is a basic cnc programming Example which helps to understand the actual working of G68 coordinate rotation.

Fanuc G68 Program Example



T1 M6

```
G0 G90 G40 G21 G17 G94 G80
G54 X20 Y0 S1500 M3
G43 Z100 H1
Z5
G81 R3 Z-20 F? M8
X30
X45
G68 X0 Y0 R120
X20 Y0
X30
X45
G68 X0 Y0 R240
X20 Y0
X30
X45
G69 G80
G0 G90 Z100 M30
```

Fanuc Subprogram Example

Here is a Fanuc subprogram example which shows, how cnc machinists can make their cnc programs easier to write and maintain by using sub-programming method.

Contents

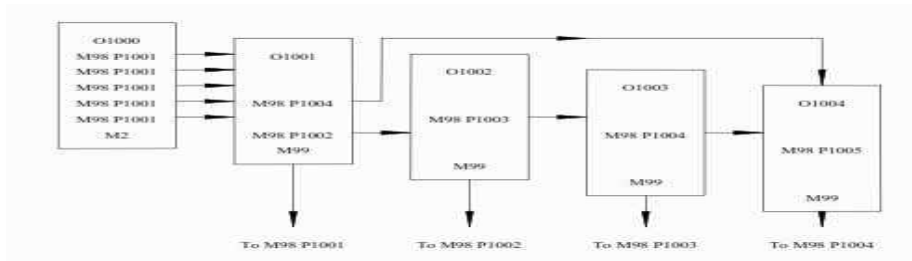
- CNC Sub Programs
- Example Without Fanuc Sub-Program Call
- Example with Fanuc Sub-Program Call
- Fanuc Subprogram Example
 - Sub Program

CNC Sub Programs

Sub-programming is a way in which cnc machinists can call other cnc programs stored in cnc machines from their programs (called as main-programs).

Read article about How to do Fanuc Sub Programming.

This Sub-programming process makes it easier for cnc programmers/machinists to keep their cnc programs small by dividing programs into small pieces of repeatable useful code.



Fanuc Sub Programming

As the below cnc program example shows, cnc machinist want to drill some holes and then he want to tap the same holes.

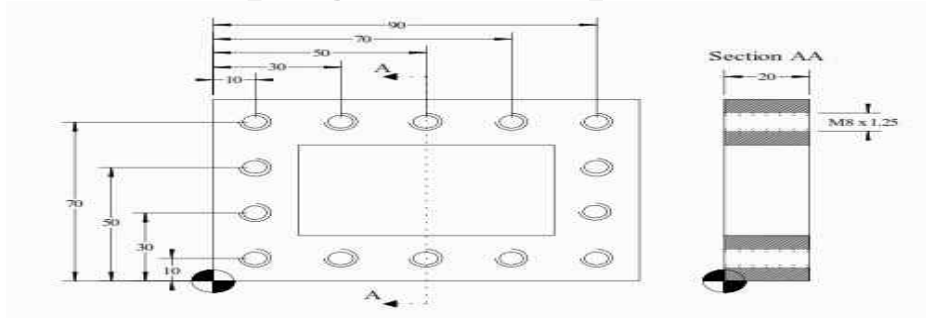
Example Without Fanuc Sub-Program Call

The simplest way to program can be read here G81 Drilling Cycle G84 Tapping Cycle CNC Program Example.

Example with Fanuc Sub-Program Call

Below is the same program example but this time we are using Sub-programming technique.

Fanuc Subprogram Example



Fanuc Subprogram Example

```

O1000 ;
N1 T1 M6 ;
N2 G0 G90 G40 G21 G17 G94 G80 ;
N3 G54 X10 Y10 S? M3 ;
N4 G43 Z100 H1 ;
N5 Z5 ;
N6 G81 R3 Z-20 F? M8 ;
N7 M98 P1001 ;
N8 G0 G90 Z100
N9 T2 M6 ;
N10 G0 G90 G40 G21 G17 G94 G80 ;
N11 G54 X10 Y10 S? M3 ;
N12 G43 Z100 H1 ;
N13 Z5 ;
N14 G84 G99 G95 R3 Z-20 F1.25 M8 ;
N15 M98 P1001 ;
N16 G0 G90 Z100 ;
N17 T0 M6 ;
N18 M30 ;

```

Sub Program

```
O1001 ;  
N101 Y30 ;  
N102 Y50 ;  
N103 Y70 ;  
N104 X30 ;  
N105 X50 ;  
N106 X70 ;  
N107 X90 ;  
N108 Y50 ;  
N109 Y30 ;  
N110 Y10 ;  
N111 X70 ;  
N112 X50 ;  
N113 X30 ;  
N114 G80 ;  
N115 M99
```

G81 Drilling Cycle G83 Peck Drilling with G98 G99 Example Program

A complete cnc part-program which shows how G98 and G99 (canned cycle return level) work with G81 drilling cycle and G83 peck drilling cycle for drilling of a component which have different heights.

Contents

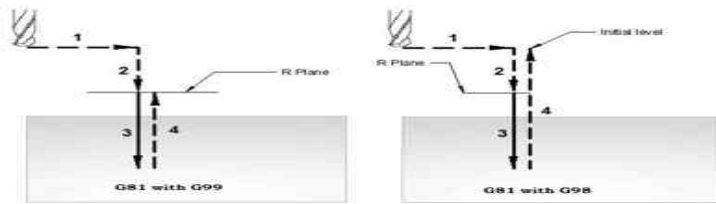
- G98 G99 Summary
- CNC Part Program
- Explanation
- G & M Codes

G98 G99 Summary

G98 and G99 are modal commands that change the way canned cycles (G81,G83 etc.) operate.

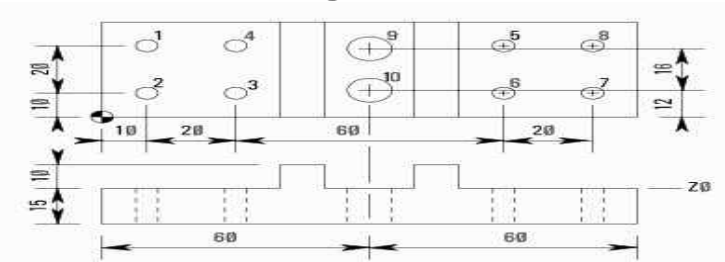
When G98 is active, the Z-axis will return to the start position (initial plane) when it completes an single operation.

When G99 is active, the Z-axis will be returned to the R point (plane) when the canned cycle completes a single hole. Then the machine will go to the next hole.



G81 drilling cycle working – G98 G99 return level

CNC Part Program



G81 Drilling Cycle with G98 G99 G code Example Program

```

N10 M06 T01
N20 G90 G00 X10 Y30 Z12 S1000 M03
N30 G99 G81 X10 Y30 Z-17 R2 F75 (Hole 1)
N40 Y10 (Hole 2)
N50 X30 (Hole 3)
N60 Y30 (Hole 4)
N70 G98 X90 (Hole 5)
N80 G99 Y10 (Hole 6)
N90 X110(Hole 7)
N100 G98 Y30 (Hole 8)
N110 G91 G80 G28 X0 Y0 Z0 M05
N120 M06 T02
N130 G90 G00 X60 Y28 Z12 S750 M03
N140 G99 G83 X60 Y28 Z-17 Q6 R2 F60 (Hole 9)
N150 G98 Y12 (Hole 10)
N160 G91 G80 G28 X0 Y0 Z0 M05
N170 M30

```

Explanation

N10- Tool change (M06) to tool no.1

N20- Rapid traverse to X10 Y30 Z12, Spindle started clockwise (M03) with 1000rpm (S1000).

N30- Drilling starts (G81) at X10 Y30 with cutting-feed (F75) drill will retract to R-plane after drilling operation.

N40- Next drilling position Y10 (as G99 is a modal g-code drill will keep on retracting to R-plane until G98 is given).

N50- Next drill at X30.

N60- Drill at Y30

N70- Drill at X90 & Retract to Initial-plane.

N80- Drill at Y10 & Retract to R-plane.

- N90- Drill at X110
 N100- Drill at Y30 & Retract to Initial-plane.
 N110- Drilling cycle is cancelled (G80), return to reference point (G28) for tool change, stop spindle (M05).
 N120- Tool change (M06) to tool number 2.
 N130- Rapid traverse to X60 Y28 Z12, start spindle at 750rpm (S750) clockwise (M03).
 N140- G83 Peck drilling starts at X60 Y28, drill depth is Z-17 and drill peck size is Q6, drilling feed is F60
 N150- Next deep drill at Y12 (return to initial point).
 N160- G83 Peck drilling cycle cancelled with G80, tool returned to reference point (G28), spindle stopped (M05).
 N170- Part-program end with return to program start (M30)

G & M Codes

Code	Description
T	Tool no. used.
M06	Tool change command.
G90	Absolute programming
G00	Rapid traverse
S	Cutter speed
M03	Cutter rotation Clockwise
M08	Coolant on.
G81	Fanuc drilling cycle.
G83	Fanuc peck drilling cycle.
G98	Return to initial point in canned cycle.
G99	Return to R point in canned cycle.
F	Cutting feed.
G80	Canned cycle cancel.
M09	Coolant off.
G28	Return to reference position.
G91	Incremental programming.
M05	Cutter rotation stop.
M30	CNC part-program end with return to program-start.

G81 Drilling Cycle – Repeat Drilling in G91 Incremental Mode Example Code

This cnc program example explains the use of G81 Drilling cycle but this time the tool is not positioned in the usual way (G90 absolute programming mode) but this time G91 Incremental Programming Mode is used.

This programming example code also explains the use of G81 drilling cycle parameter K (number of repeats), which is not normally used.

Contents

- G81 Drilling Cycle Format
- CNC Code G81 Drilling Cycle with G91 Incremental Programming
 - Explanation

G81 Drilling Cycle Format

G81 X_ Y_ Z_ R_ F_ K_ ;

X_ Y_ : Hole position data

Z_ : Z-depth (tool will travel with feed to Z-depth starting from R plane)

R_ : The distance from the initial level to point R level

F_ : Cutting feedrate

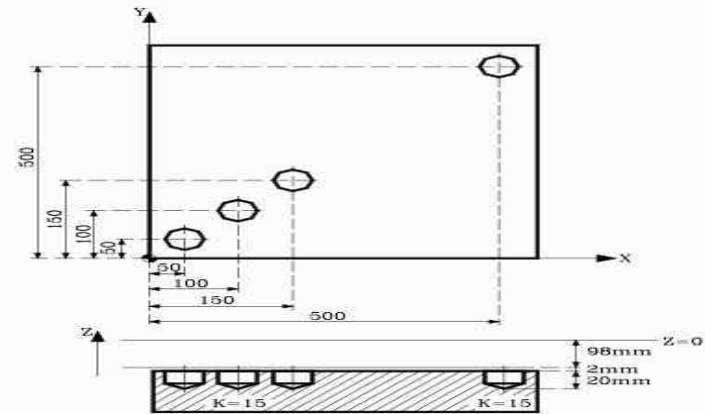
K_ : Number of repeats (if required)

G81 drilling cycle parameter K (number of repeats). On different cnc controls this parameter has different letter assign to it such as on,

- Fanuc uses letter K for number of repeats.
- Haas CNC uses letter L for number of repeats.
- Fagor CNC Control uses letter N for number of repeats.

Incremental motion in canned cycle is often useful as a loop count, which can be used to repeat the operation with an incremental X or Y move between each cycle.

CNC Code G81 Drilling Cycle with G91 Incremental Programming



G81 Drilling Cycle – Repeat Drilling in G91 Incremental Mode Example Code

```

N10 T1 M6
N20 G00 G90 X0 Y0 Z0
N30 S1450 M03
N40 G81 G99 G91 X50 Y50 Z-120 R-98 K3 F350
N50 G98 G90 G00 X500 Y500
N60 G80
N70 G90 X0 Y0
N80 M30

```

Explanation

N40 G81 G99 G91 X50 Y50 Z-120 R-98 K3 F350

K3 means that the G81 drilling cycle will repeat three times.

G91 Incremental mode makes the tool to move every time X50 and Y50

So first hole will be at X50 Y50

Second hole will be at X100 Y100 because

$X100 = X50(\text{previous value}) + X50 (\text{increment})$

$Y100 = Y50(\text{previous value}) + Y50 (\text{increment})$

Third hole will be at X150 Y150 because

$X150 = X100(\text{previous value}) + X50 (\text{increment})$

$Y150 = Y100(\text{previous value}) + Y50 (\text{increment})$

N50 G98 G90 G00 X500 Y500

The fourth hole position is given with G90 Absolute Programming Mode.

The above code is only possible if you use G91 Incremental Programming Mode with G81 Drilling cycle, but if you try to use G90 Absolute Programming Mode with G81 as above you will see the tool will repeat drilling at the same position.

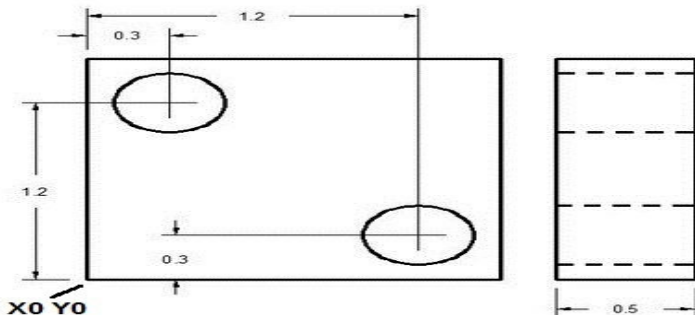
CNC G81 Drilling Cycle Program Example

Fanuc G81 drilling cycle is used for simple drilling operations, so here is a cnc program example which demonstrates how G81 drilling cycle can be used.

Contents

- CNC Program
- Explanation
- G & M Codes

CNC Program



Fanuc G81 Drilling Cycle Example Program

```
N1 T1 M06
N2 G90 G54 G00 X.3 Y1.2
N3 S1200 M03
N4 G43 H01 Z1. M08
N5 G81 Z-.6 R.1 F10
N6 X1.2 Y.3
N7 G80 G00 Z1. M09
N8 G28 G91 Z0. M05
N9 M30
```

Explanation

Once G81 drilling canned cycle is defined, the canned cycle is repeated at every X-Y position in sequential blocks, G80 code is used to cancel G81 drilling cycle.

N1- Tool change to tool no.1

N2- Tool rapidly moves to first drilling position X0.3 Y1.2 while taking into account Zero-offset-no. 1

N3- Drill starts rotating clockwise with 1200 rpm.

N4- Drill takes depth Z1. taking into account tool length compensation (G43 H01), coolant is turned on.

N5- Drilling cycle parameters, drill depth and cutting feed are given, with this command first drill is made at current position (X0.3 Y1.2).

N6- As drilling cycle continues it's work with every axis movement so next drill is done at X1.2 Y0.3

N7- Drilling cycle is canceled with G80 command, Coolant is turned off.

N8- As operation for this component is finished, so cutting tool is taken to reference position, and cutter rotation is stopped.

N9- CNC part-program is ended.

G & M Codes

Code	Description
T	Tool no. used.
M06	Tool change command.
G90	Absolute programming
G54	Zero offset no.1
G00	Rapid traverse
S	Cutter speed
M03	Cutter rotation Clockwise
G43	Tool length compensation.
M08	Coolant on.
G81	Fanuc drilling cycle.
F	Cutting feed.
G80	Canned cycle cancel.
M09	Coolant off.
G28	Return to reference position.
G91	Incremental programming.
M05	Cutter rotation stop.
M30	CNC part-program end with return to program-start.

G81 Drilling Canned Cycle CNC Milling Program Example

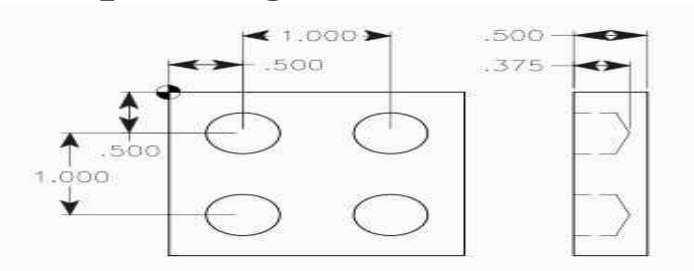
CNC program example for cnc vertical machining centers illustrates the use of G81 drilling cycle.

CNC milling machine controls like Fanuc, Haas etc, has many standard canned cycles built-in the cnc control, these canned cycles make the cnc programmer/ setter/ operator's life easier as canned cycles are easy to program and debug.

Contents

- Sample Program
- Explanation
- G & M Codes

Sample Program



G81 Drilling Canned Cycle CNC Milling Program Example Code

```
O10075
N1 T16 M06
N2 G90 G54 G00 X0.5 Y-0.5
N3 S1450 M03
N4 G43 H16 Z1. M08
N5 G81 G99 Z-0.375 R0.1 F9.
N6 X1.5
N7 Y-1.5
N8 X0.5
N9 G80 G00 Z1. M09
N10 G53 G49 Z0. M05
N11 M30
```

Explanation

O10075 Program number (there are many programs in cnc memory so they are distinguished by their number).

N1- Tool change (M06) to tool no.16

N2- Tool rapidly moves (G00) to first drilling position X0.5 Y-0.5 while taking into account Zero-offset-no. 1 (G54)

N3- Drill starts rotating clockwise (M03) with 1450 rpm (S1450).

N4- Drill takes depth Z1. taking into account tool length compensation (G43 H16), coolant is turned on (M08).

N5- Drilling cycle (G81) parameters, drill depth (Z) and cutting feed (F) are given, with this command first drill is made at current position (X0.5 Y-0.5).

N6- As drilling cycle continues it's work with every axis movement so next drill is done at X1.5

N7- Third drilling hole at Y-1.5

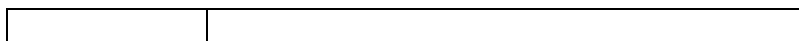
N8- Fourth drill at X0.5

N9- Drilling cycle is canceled (G80), Coolant is turned off (M09).

N10- Taking Machine-coordinate-system (G53) into account the drill is taken to Z0 position. Tool length compensation is canceled (G49), cutter rotation is stopped (M05).

N11- CNC part-program is ended.

G & M Codes



Code	Description
T	Tool no. used.
M06	Tool change command.
G90	Absolute programming
G54	Zero offset no.1
G00	Rapid traverse
S	Cutter speed
M03	Cutter rotation Clockwise
G43	Tool length compensation.
M08	Coolant on.
G81	Fanuc drilling cycle.
F	Cutting feed.
G80	Canned cycle cancel.
M09	Coolant off.
G53	Machine coordinate system selection.
G49	Tool length compensation cancel.
M05	Cutter rotation stop.
M30	CNC part-program end.

G91 Incremental Programming

Face Grooving with G74 Peck Drilling Cycle CNC Programming Tutorial

G74 Peck Drilling Cycle

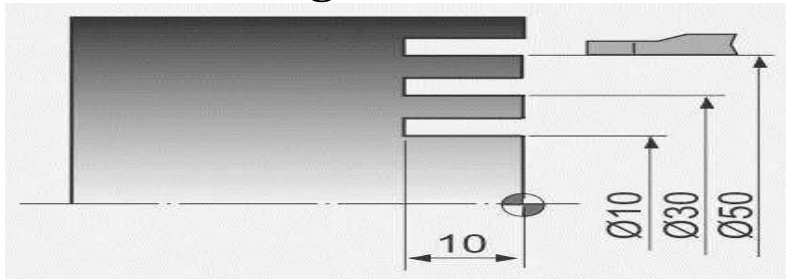
G74 peck drilling cycle be used in variety of ways, from peck drilling to face grooving.

The G74 Peck drilling in already discussed here Simple CNC Lathe Drilling with Fanuc G74 Peck Drilling Cycle.

The cnc programming example below shows how face grooving can be machined with the help of G74 peck drilling canned cycle.

With face grooving operations the tool is fed axially rather than radially toward the end surface of the workpiece.

Face Grooving with G74 Peck Drilling Cycle



Face Grooving with G74 G Code a CNC Programming Tutorial

```
N10 G50 S2000 T0100  
N20 G96 S80 M03  
N30 G00 X50.0 Z1.0 T0101  
N40 G74 R1.0  
N50 G74 X10.0 Z-10.0 P10000 Q3000 F0.1  
N60 G00 X200.0 Z200.0 T0100  
N70 M30
```

G75 Canned Cycle Grooving CNC Programming Example

G75 is the grooving cycle in x-axis.

For a full description of G75 canned cycle grooving read this G75 Grooving Cycle.

For one-line format (one-block format) of Fanuc G75 read Fanuc G75 Grooving Cycle One-Line Format.

You might find another G75 grooving cycle cnc programming example here Fanuc G75 Grooving Cycle CNC Program Example.

Contents

7 Explanation of Parameters of Fanuc G75 Grooving Cycle

8 G75 Canned Cycle Grooving CNC Programming Example

Explanation of Parameters of Fanuc G75 Grooving Cycle

N10 G75 R

N20 G75 X Z P Q R

G75 First CNC Programming Block

R = Return amount

G75 Second CNC Programming Block

X = Groove Depth.

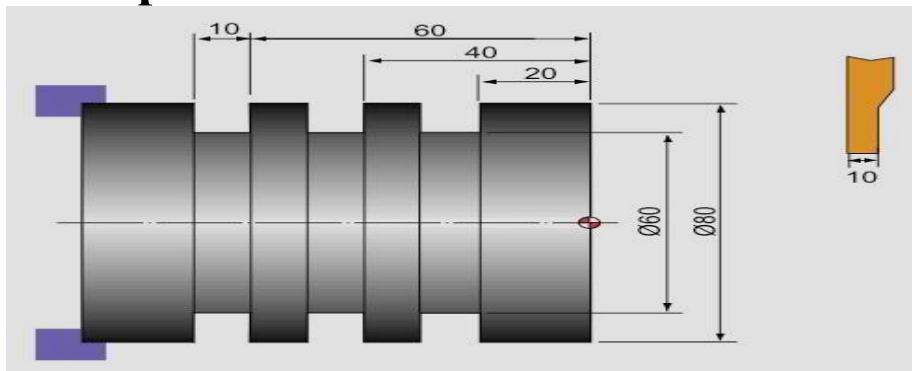
Z = Last groove position in z-axis.

P = Peck increment in x-axis

Q = Stepping in z-axis.

R = Relief amount at end of the cut.

G75 Canned Cycle Grooving CNC Programming Example



G75 Canned Cycle Grooving CNC Programming Example

N10 G50 S500 T0100

N20 G97 S400 M03

N30 G00 X90.0 Z1.0 T0101

N40 X82.0 Z-60.0

N50 G75 R1.0

N60 G75 X60.0 Z-20.0 P3000 Q20000 F0.1

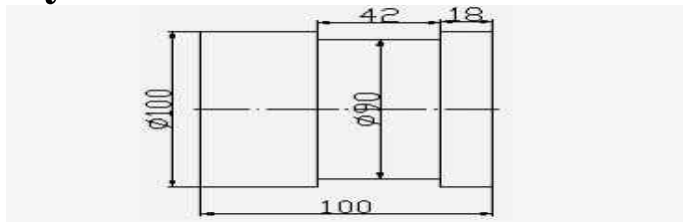
N70 G00 X90.0
N80 X200.0 Z200.0 T0

Fanuc G75 Grooving Cycle CNC Program Example

I have posted about Fanuc G75 Grooving Cycle. The **Fanuc G75 grooving cycle** has multiple parameters, which can be set according to your needs. In this post here is a cnc program example for the G75 Fanuc grooving cycle. Although it is a simple cnc program example for grooving, but this cnc program can be easily altered to your needs.

The *fanuc G75 grooving cycle* is briefly described in this article Fanuc G75 Grooving Cycle, so here is the cnc program example and some explanation of the cnc program.

CNC Programming Example of Fanuc G75 Grooving Cycle



Fanuc G75 Grooving Cycle CNC Program Example

```
N10 T0202  
N20 G92 S500 M42  
N30 G97 S400 M03  
N40 G00 X110 Z0 M08  
N50 G01 Z-22 F0.5  
N60 G75 R1  
N70 G75 X90 Z-60 P2000 Q3000 R0 F0.1  
N80 G00 X120 Z100  
N90 M30
```

Note: The grooving tool is 4mm wide, so I started from z-22.

Every time the grooving tool will take 2mm (P2000) cut in x-axis, and it will retract 1mm (Pecking, First R1)

After a groove in x-axis is complete it will start the next groove by moving the grooving tool by 3mm (Q3000) in z-axis, and it will repeat it.

CNC Lathe Programming Example

A simple cnc lathe programming example. This cnc programming example will show how to program contours like *chamfer taper grooves and arc*.

This cnc program will use two tools for machining. The first tool used in this cnc programming example is a **turning tool** and the second one is a **grooving tool**.

Contents

- CNC Machining
- CNC Program

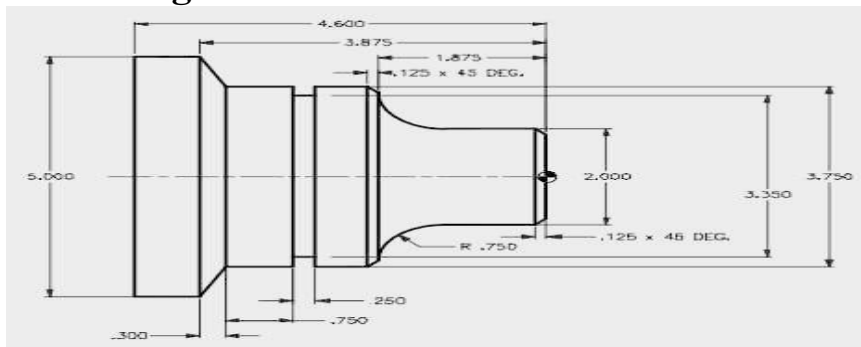
CNC Machining

The **turning tool** will first *face the component* then it will make a *chamfer*, after that there is *straight turning* and then there is *arc machining* the arc may be machined with **R (Radius of arc)** or arc can be machined with **I and K values of the arc**, at the top of arc there is a chamfer, then again straight turning, now there comes the *groove* but **we can't make grooves with turning tools because of their shape** so we will machine the groove with our next tool which is solely made for grooving operations, but at the time we are machining with turning tool so we will just skip this groove and will just machine in straight line, after that there is a taper and again a straight line to be machined.

Now with our **grooving tool** the groove machining task is just easy.

if the *grooving insert* is of the same size as the dimension of the groove then grooving is even more easy, we will make the groove in one go, but if the *grooving insert* is of smaller width than we have to take depth more than one time.

CNC Program



cnc lathe program example

```
N1 T01 D01 M491
N2 G00 X0 Z1
N3 G01 G96 G41 Z0 F2 S140
N4 G01 X2 CHF=0.125 F0.2
N5 G01 Z-1.125
N6 G02 X3.5 Z-1.875 CR=0.75
```

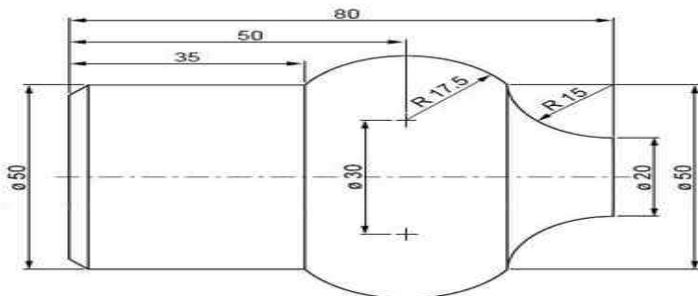
N7 G01 X3.75 CHF=0.125
N8 G01 Z-3.575
N9 G01 X5 Z-3.875
N10 G01 Z-4.6
N11 G00 X20 Z20 G40
N12 T02 D02 M491
N13 G00 G97 S500 X4 Z-2.825
N14 G01 X3.85 F1
N15 G01 X3.35 F0.15
N16 G01 X3.85 F0.5
N17 G00 X4
N18 G00 X20 Z20
N19 M30

Grooving

G02 G03 Programming Example

CNC program example to show how two combined arc can be programmed with G02 G03 for a cnc lathe.

First arc is programmed using G02 R, but the second arc is programmed using G03 I K



G02 G03 Programming Example

G02 G03 Programming Example

```
N50 G00 X20 Z85  
N60 G96 S200  
N70 G01 Z80  
N80 G02 X50 Z65 R15  
N90 G03 X50 Z35 I-10 K-15  
N100 G00 X80  
N110 G97 S900 M05  
N120 M30
```

Quincunx a CNC Plasma Cutting Controller Program Example

CNC program example to cut Quincunx on a CNC Plasma Cutting Controller. This program example is programmed in G91 Incremental programming.

What is a CNC Plasma Cutting Machine

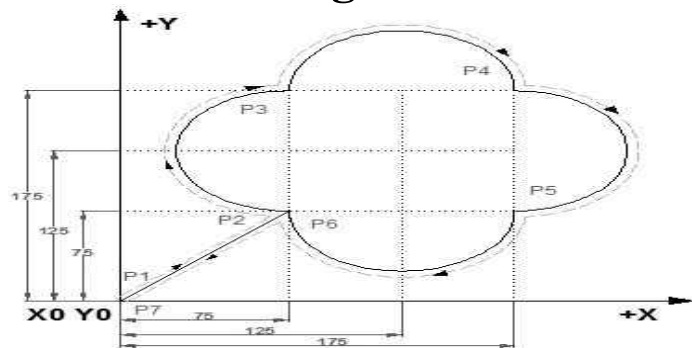
Plasma Cutting

Plasma cutting is a process that is used to cut steel and other metals of different thicknesses (or sometimes other materials) using a plasma torch. In this process, an inert gas (in some units, compressed air) is blown at high speed out of a nozzle; at the same time an electrical arc is formed through that gas from the nozzle to the surface being cut, turning some of that gas to plasma. The plasma is sufficiently hot to melt the metal being cut and moves sufficiently fast to blow molten metal away from the cut.

CNC Plasma Cutter

A “CNC plasma” system is a machine that carries a plasma torch, and can move that torch in a path directed by CNC (Computer Numerical Control), which means that a computer is used to direct the machines motion based on numerical codes in a program.

Plasma Cutting Controller Program Example



Quincunx a CNC Plasma Cutting Controller Program Example

Relative coordinate programming

```
G92 X0 Y0 (P1)
G00 X75 Y75 (P2)
G02 Y100 I0 J50 (P3)
G02 X100 I50 J0 (P4)
G02 Y-100 I0 J-50 (P5)
G02 X-100 I-50 J0 (P6)
G00 X-75 Y-75(P7)
M02
```

Fanuc G83 Peck Drilling Cycle

G83 peck drilling cycle perform the drilling operation in multiple pecks, this technique makes deep-hole drilling easy and economical.

Cutting feed is performed intermittently to the bottom of the hole while chips are discharged.

As the drilling is performed to the bottom of the hole with feed in multiple small steps, every time a specified depth is made and then drill retracts, then drill makes the next peck, this operation is repeated again and again until the drill depth is reached.

Syntax

G83 X... Y... Z... R... Q... F... K...

Parameter	Description
X	Hole position in x-axis.
Y	Hole position in y-axis.
Z	Depth, tool will travel with feed to Z-depth starting from R plane.
R	Position of the R plane.
Q	Depth of cut for each cutting feed (Peck).
K	Number of cycle repetitions (if required)
F	Feedrate.

Once given in program G83 peck drilling cycle is repeated at every axis movement until G80 is given in program to end peck drilling cycle.

Usage

```
N150 M06 T02
N160 G90 G00 X60 Y28 Z12 S750 M03
N170 G99 G83 X60 Y28 Z-17 Q6 R2 F60
N180 G98 Y12
N190 G91 G80 G28 X0 Y0 Z0 M05
N200 M30
```

In the above example code first drill is done at X60 Y28 and second at Y12 and then peck drilling is cycle is ended with G80.

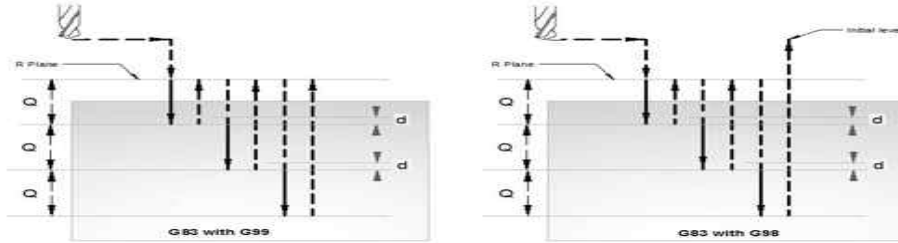
6mm pecks are taken to complete total drilling depth of 17mm.

Working

Here is briefly described how G83 peck drilling cycle works,

- 1- Rapid traverse to X, Y drilling position.
 - 2- Rapid traverse to R-plane.
 - 3- Drilling with feed Q deep.
 - 4- Retraction with Rapid traverse to R-plane.
 - 5- Rapid traverse to Q-d deep (d value is specified in parameters).
 - 6- Drilling with feed Q+d deep.
 - 7- Retraction with Rapid traverse to R-plane
- this whole procedure is repeated until drill reaches Z-depth position,

– then drill is retracted to R-plane or Initial-level depends on G99 or G98 which one is given in program.



G83 peck drilling cycle working

G98 G99 Modes

After completing drilling depth the return is made with Rapid feed, the return height can be controlled through using G98 or G99.

G98 Drill will return to the Initial level

G99 Drill will return to R-plane.

G98, G99 can be used multiple times during G83 peck drilling cycle.

Example

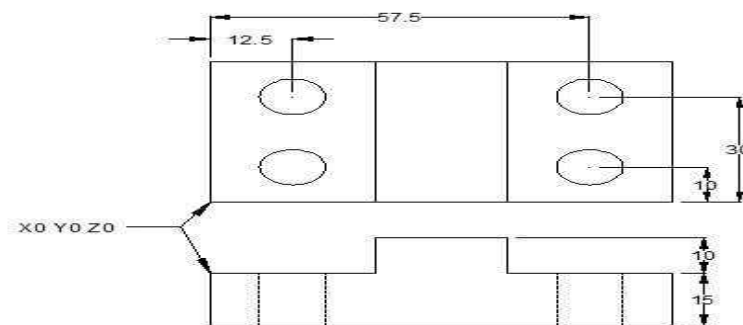
```
N30 G83 X10 Y30 Z-17 Q5 R2 F75
N40 Y10
N50 G98 X30
N60 G99 Y30
N70 X90
N80 Y10
N90 G80
```

Repeat Drilling

G83 peck drilling cycle, drilling operation can be repeated multiple times. The drilling is repeated K times if K value is given with G83.

Repeat drilling is normally used with G91 Incremental mode, and a good example of repeated drilling is Grid-plate drilling. For working example see G81 drilling cycle.

Working Example



G83 Peck drilling cycle Example

```
N10 M06 T1
N20 G90 G00 X12.5 Y10 Z12 S1000 M03
```

```

N30 G99 G83 X12.5 Y10 Z-17 R2 Q4 F75
N40 Y30
N50 G98 X57.5
N60 G99 Y10
N70 G91 G80 G28 X0 Y0 Z0 M05
N80 M30

```

Fanuc G71 G72 G70 Canned Cycle CNC Lathe Internal Machining Example (Boring & Facing)

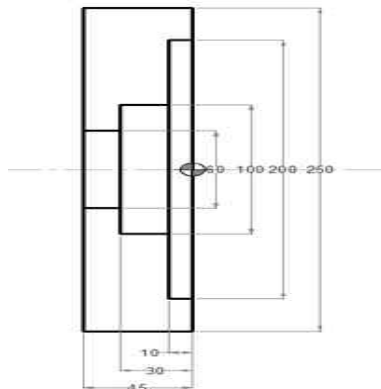
Fanuc programming example which shows the use of multiple fanuc canned cycle in cnc programming, Following canned cycle are used in this cnc lathe programming example

- G71 Rough Turning Cycle
- G72 Facing Cycle
- G70 Finish Cycle

Fanuc G71 for Boring Opertaion

This programming example also illustrate how G71 turning cycle can be used for internal machining (boring operation). So if you want to remove extra stock from inside of a component, you can use Fanuc G71 turning cycle for internal stock removal as well.

Fanuc Programming Example



Fanuc Canned Cycle Example

```

G28 U0 W0
G50 S1500
N1 T0101 M8 (OD FACING)
G96 S180 M3

```

```
G0 X255.0 Z5.0
G72 W2.0 R0.5
G72 P100 Q200 F0.25
N100 G0 Z0 G41
G1 X-2.0 F0.18
N200 G0 Z5.0
G40
X255.0
G28 U0 W0
N2 T0404 M8 (ID ROUGH)
G96 S180 M3
G0 X50.0 Z5.0
G71 U2.0 R0.5
G71 P500 Q600 U-0.5 W0.1 F0.25
N500 G0 X202.0 G41
G1 Z0 F0.12
X200.0
Z-10.0
X100.0
Z-30.0
X60.0
Z-45.0
N600 X50.0
G40
G0 Z5.0
G28 U0 W0
N3 T0505 M8 (ID FINISH)
G96 S220 M3
G0 X50.0 Z5.0
G70 P500 Q600
G0 X50.0 Z5.0
G28 U0 W0
M5 M9
M30
```

Internal Threading on Fanuc 21i 18i 16i with G76 Threading Cycle

CNC program for the internal threading with G76 threading cycle on fanuc controls 21i/18i/16i.

For an example of external threading with G76 threading cycle read External Thread Cutting with G76 Threading Cycle on Fanuc 21i 18i 16i CNC
Fanuc 21i/18i/16i use two block format of G76 threading cycle.

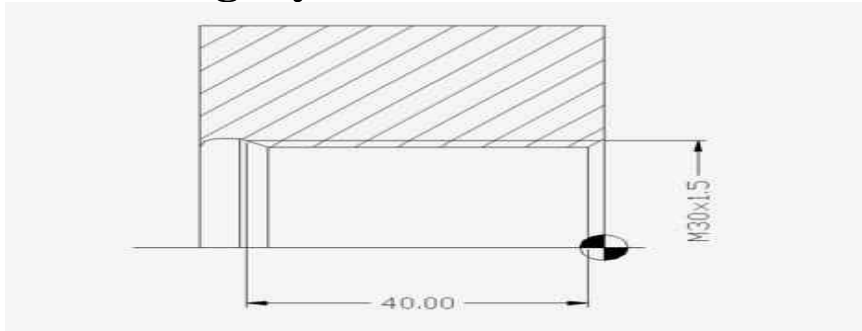
Related: G76 Threading Cycle One Line Format for Fanuc 10/11/15T

Fanuc G76 threading cycle has multiple parameters making it difficult to remember, but at the same time those multiple parameters of G76 thread cycle

give the cnc programmer/cnc machinist multiple options to control thread cutting, some are listed below.

G76 thread cutting cycle allow cnc machinist to control number of idle cuts, thread run-out, infeed angle.

CNC Program of Internal Threading with G76 Threading Cycle



Internal Threading on Fanuc 21i 18i 16i with G76 Threading Cycle

```
N17 T101  
N18 G54  
N19 G97 S800 M3  
N20 G0 X25 Z6 M8  
N21 G76 P010060 Q100 R0.02  
N22 G76 X30 Z-40 P919 Q250 F1.5  
N23 G0 X150 Z100
```

You might like other cnc threading options on cnc machines with fanuc control

G75 Canned Cycle Grooving CNC Programming Example

G75 is the grooving cycle in x-axis.

For a full description of G75 canned cycle grooving read this G75 Grooving Cycle.

For one-line format (one-block format) of Fanuc G75 read Fanuc G75 Grooving Cycle One-Line Format.

You might find another G75 grooving cycle cnc programming example here Fanuc G75 Grooving Cycle CNC Program Example.

Contents

- Explanation of Parameters of Fanuc G75 Grooving Cycle
- G75 Canned Cycle Grooving CNC Programming Example

Explanation of Parameters of Fanuc G75 Grooving Cycle

N10 G75 R

N20 G75 X Z P Q R

G75 First CNC Programming Block

R = Return amount

G75 Second CNC Programming Block

X = Groove Depth.

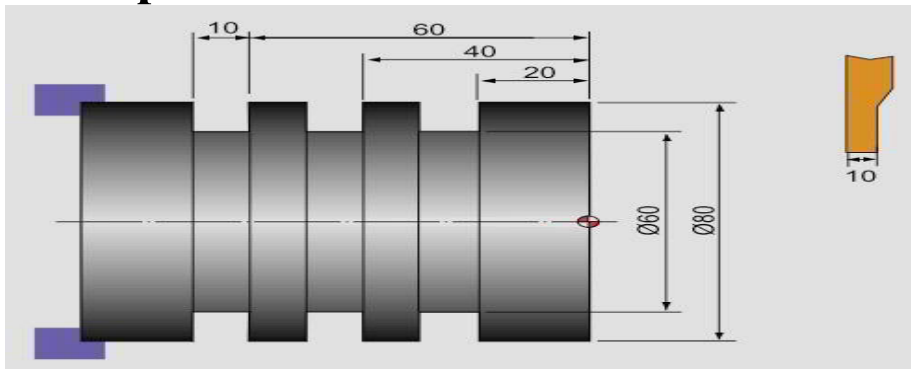
Z = Last groove position in z-axis.

P = Peck increment in x-axis

Q = Stepping in z-axis.

R = Relief amount at end of the cut.

G75 Canned Cycle Grooving CNC Programming Example



G75 Canned Cycle Grooving CNC Programming Example

N10 G50 S500 T0100

N20 G97 S400 M03

N30 G00 X90.0 Z1.0 T0101

N40 X82.0 Z-60.0

N50 G75 R1.0

N60 G75 X60.0 Z-20.0 P3000 Q20000 F0.1

N70 G00 X90.0

N80 X200.0 Z200.0 T0100

N90 M30

Multi Start Threads with Fanuc G76 Threading Cycle

Fanuc cnc controls has no direct threading cycle for cutting **multi start threads** on cnc. But you can cut multi start threads on a cnc with fanuc control by using Fanuc G76 Threading Cycle.

Related: Fanuc G76 Thread Cycle for Dummies



Multi Start Threads with Fanuc G76 Threading Cycle

There are multiple techniques for cutting multi-start threads on cnc with Fanuc G76 threading cycle. Here is one of them.

This cnc programming example uses cnc subprogram call to cut multi-start threads on Fanuc cnc.

Contents

- CNC Main program
- CNC Sub Program

CNC Main program

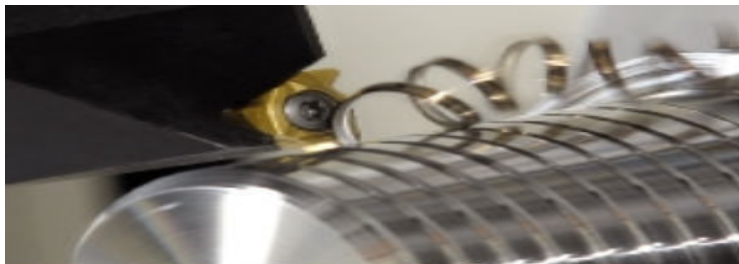
```
N50 G00 X48 Z5  
N60 M98 P0034713
```

CNC Sub Program

```
N10 G76 P020000 Q100 R0.05  
N20 G76 X42 Z-15 P974 Q200 F4.5  
N30 G00 W1.5  
N40 M99
```

The above cnc main program calls the 4713 sub-program three times.

Fanuc G76 Thread Cycle for Dummies



Fanuc G76 Thread Cycle for Dummies

Fanuc G76 Thread Cycle for Dummies explains Fanuc G76 threading cycle briefly. Fanuc G76 gives cnc machinist full control over thread turning. Fanuc G76 threading cycle has multiple parameters but the same way Fanuc G76 gives full flexibility in thread cutting.

This article is actually to help cnc machinists to easily navigate through multiple articles explaining Fanuc G76 threading cycle.

Below are quick links,

- Fanuc G76 Threading Cycle
- G76 Threading Cycle One Line Format for Fanuc 10/11/15T
- Tapered Threading with Fanuc G76 threading cycle
- Multi-Start Threading with Fanuc G76 threading cycle
- Controlling Thread Infeed with Fanuc G76 threading cycle
- How to Fully Control G76 Threading Cycle Number of Pass and Depth of Cut

For explanation of all the variations of Fanuc G76 see below

Contents

- CNC Fanuc G76 Threading Cycle
- One Line Format for Fanuc 10/11/15T
- Tapered Threading
- Multi Start Threads
- Controlling Threading Infeed Angle
- Controlling Number of Pass and Depth of Cut

CNC Fanuc G76 Threading Cycle

CNC Fanuc G76 Threading Cycle this article briefly **explains all the parameters** of Fanuc G76 threading cycle, like the following cnc programming code for fanuc g76 threading cycle

```
N5 G76 P010060 Q100 R0.05  
N6 G76 X30 Z-20 P1024 Q200 F2
```

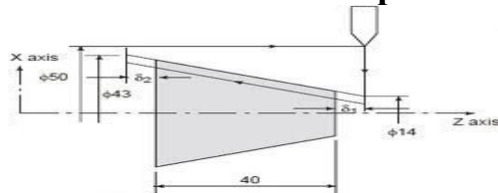
One Line Format for Fanuc 10/11/15T

G76 Threading Cycle One Line Format for Fanuc 10/11/15T, Fanuc control models 10/11/15 use a single-block format for G76 threading cycle.

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

Tapered Threading

Tapered Threading with Fanuc G76 Threading Cycle this post explained how a cnc machinist can cut **Tapered Threads** with Fanuc G76 threading cycle.



Tapered Threading with Fanuc G76 Threading Cycle

The following cnc programming code is explained in the above post.

```
N5 G00 X50 Z5
```

```
  N6 G76 P010060 Q100 R0.05
```

```
  N7 G76 X43 Z-45 P1024 Q200 R-14.5 F2
```

Multi Start Threads

Multi Start Threads with Fanuc G76 Threading Cycle this article fully describes how to cut **Multi-Start threads** on cnc machine with Fanuc G76 threading cycle.

Controlling Threading Infeed Angle

Controlling Threading Infeed Angle with Fanuc G76 Threading Cycle this article explains how a cnc machinist can control **Thread Infeed Angle** with Fanuc G76 threading cycle.

Controlling Number of Pass and Depth of Cut

How to Fully Control G76 Threading Cycle Number of Pass and Depth of Cut Explained this articles tells how a cnc machinist can control

- Depth of cut for First pass
- Depth of cut for normal passes
- Depth of cut for Last pass
- Control number of Spring passes

Intermediate Level

Fanuc G81 Drilling Cycle

G81 drilling cycle is used for simple drilling/spot drilling operations.

Syntax

G81 X... Y... Z... R... K... F...

Parameter	Description
X	Hole position in x-axis.
Y	Hole position in y-axis.
Z	Depth, tool will travel with feed to Z-depth starting from R plane.
R	Position of the R plane.
K	Number of cycle repetitions (if required)
F	Feedrate.

Once G81 drilling cycle is defined, the canned cycle is repeated at every X-Y position in sequential blocks. So G81 drilling cycle must be cancelled with G80.

Usage

```
N30 G81 X10 Y30 Z-17 R2 F75
N40 Y10
N50 X30
N60 Y30
N70 X90
N80 Y10
N90 G80
```

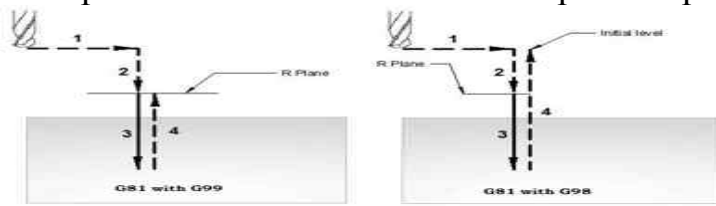
In the above example drilling will start with G81 drilling cycle at X10 Y30, so first drill will be at X10 Y30, then second at Y10, third at X30, fourth at Y30, fifth at X90 and the last one at Y10, because next block have G80 code, so drilling cycle will no more be repeated.

Working

Here is briefly described how G81 drilling cycle operates,

- 1- Rapid traverse to the specified x,y axis position (drilling position).
- 2- Rapid traverse to the R plane position.
- 3- Drilling with specified Feed from R-plane position to Z-depth position.

4- Rapid traverse to Initial level or R-plane depends on G98, G99 modes.



G81 drilling cycle working

G98 G99 Modes

After completing drilling depth the return is made with Rapid feed, the return height can be controlled through using G98 or G99.

G98 Drill will return to the Initial level

G99 Drill will return to R-plane.

G98, G99 can be used multiple times during G81 drilling cycle.

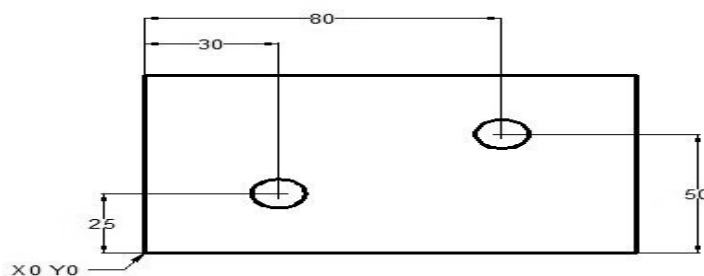
Example

```
N30 G81 X10 Y30 Z-17 R2 F75
N40 Y10
N50 G98 X30
N60 G99 Y30
N70 X90
N80 Y10
N90 G80
```

Repeat Drilling

With G81 drilling cycle drilling operation can be repeated multiple times. The drilling is repeated K times when that parameter is given with G81 drilling cycle. Repeat drilling is normally used with G91 Incremental mode, and a good example of repeated drilling is Grid-plate drilling. the example for repeat drilling is given below.

Working Examples

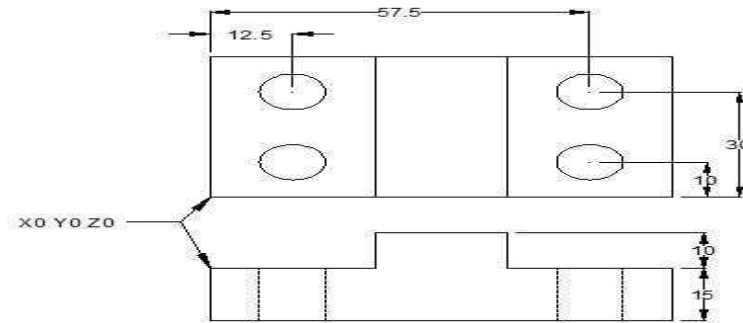


G81 Drilling Cycle Example

```
N10 T1 M06
N20 G90 G54 G00 X30 Y25
N30 S1200 M03
N40 G43 H01 Z5 M08
N50 G81 Z-10 R2 F75
N60 X80 Y50
N70 G80 G00 Z100 M09
```

N80 M30

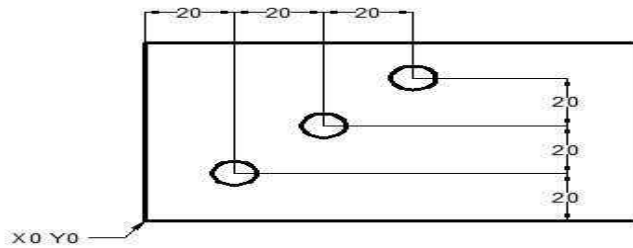
G98 G99 Example



G81 drilling cycle usage with G98 G99

```
N10 M06 T1
N20 G90 G00 X12.5 Y10 Z12 S1000 M03
N30 G99 G81 X12.5 Y10 Z-17 R2 F75
N40 Y30
N50 G98 X57.5
N60 G99 Y10
N70 G91 G80 G28 X0 Y0 Z0 M05
N80 M30
```

Repeat Drilling Example



Repeat drilling with G81 Drilling Cycle

```
T1 M6
G00 G90 G40 G21 G17 G94
G54 X0 Y0 S1000 M03
G43 H1 Z100
Z3
G81 G99 G91 X20 Y20 R3 Z-20 K3 F100 M08
G80
G00 G90 Z100
M30
OR
T1 M6
G00 G90 G40 G21 G17 G94
G54 X20 Y20 S1000 M03
G43 H1 Z100
Z3
G81 G99 R3 Z-20 F100 M08
G91 X20 Y20 K2
G80
G00 G90 Z100
```

Drilling Grid Plate with G81 Drilling Cycle – Sample CNC Program

CNC machine workshops deal with variety of components, of course cnc machinists program and machine them.

But a cnc machinists also should understand and practice new and economical ways to machine a component.

The following cnc programming example can be programmed in variety of ways, the method of cnc programming used in this cnc program sample uses G81 drilling cycle with G91 Incremental Programming mode, which makes programming easy.

Fanuc cnc control uses K with G81 drilling cycle to repeat drilling cycle.

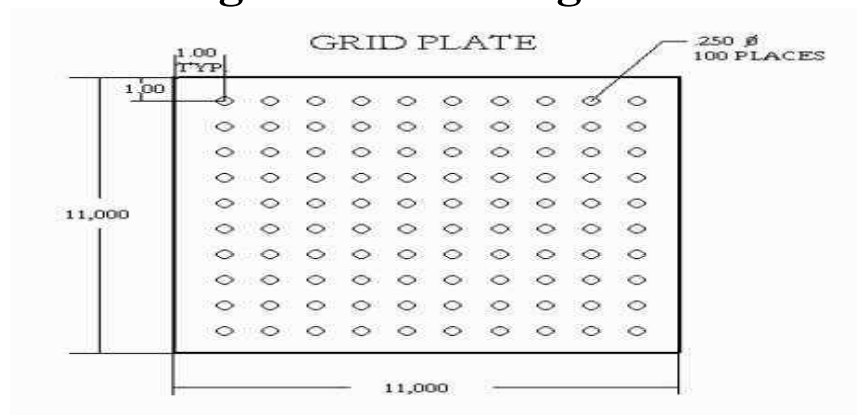
Haas cnc controls use L to repeat G81 drilling cycle.

Following cnc program is written for haas cnc machine but can easily be converted for Fanuc cnc controls.

You might read other cnc programming example which shows the same technique for Fanuc cnc controls

- G81 Drilling Cycle – Repeat Drilling in G91 Incremental Mode Example Code
- Repeat Drilling with G81 Drilling Cycle and G91 Example Program

CNC Program of Drilling Grid Plate



Drilling Grid Plate with G81 Drilling Cycle

03400 (Drilling grid plate)

```

T1 M06
G00 G90 G54 X1.0 Y-1.0 S2500 M03
G43 H01 Z.1 M08
G81 Z-1.5 F15. R.1
G91 X1.0 L9
G90 Y-2.0 (Or stay in G91 and repeat Y-1.0)
G91 X-1.0 L9
G90 Y-3.0
G91 X1.0 L9
G90 Y-4.0
G91 X-1.0 L9
G90 Y-5.0
G91 X1.0 L9
G90 Y-6.0
G91 X-1.0 L9
G90 Y-7.0
G91 X1.0 L9
G90 Y-8.0
G91 X-1.0 L9
G90 Y-9.0
G91 X1.0 L9
G90 Y-10.0
G91 X-1.0 L9
G00 G90 G80 Z1.0 M09
G28 G91 Y0 Z0
M30

```

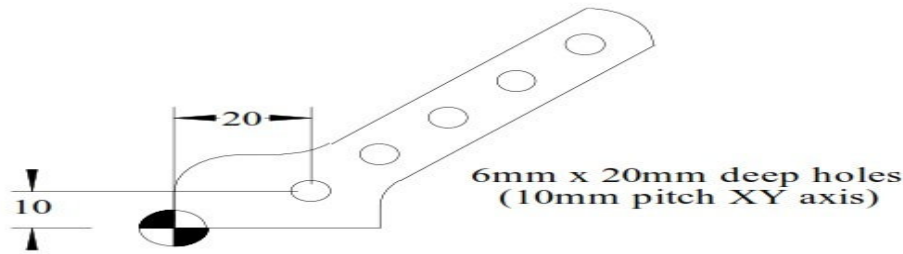
In above cnc program G90 Absolute programming mode is used while starting a new row for drilling. Although G91 Incremental programming can be used but using G90 there makes this program easy to read understand and debug (if anything goes wrong).

Repeat Drilling with G81 Drilling Cycle and G91 Example Program

Here is another G81 drilling cycle programming example which illustrates the use of repeat drilling in G91 Incremental Programming Mode.

Other CNC program example is here G81 Drilling Cycle – Repeat Drilling in G91 Incremental Mode Example Code

Repeat Drilling with G81 Drilling Cycle



Repeat Drilling with G81 Drilling Cycle and G91

```
O1000
T1 M6
G00 G90 G40 G21 G17 G94
G54 X20 Y10 S1000 M03
G43 H1 Z100
Z3
G81 G99 R3 Z-20 F350 M08
G91 X10 Y10 K4
G80
G00 G90 Z100
M30
```

G81 Drilling Cycle – Repeat Drilling in G91 Incremental Mode Example Code

This cnc program example explains the use of G81 Drilling cycle but this time the tool is not positioned in the usual way (G90 absolute programming mode) but this time G91 Incremental Programming Mode is used.

This programming example code also explains the use of G81 drilling cycle parameter K (number of repeats), which is not normally used.

G81 Drilling Cycle Format

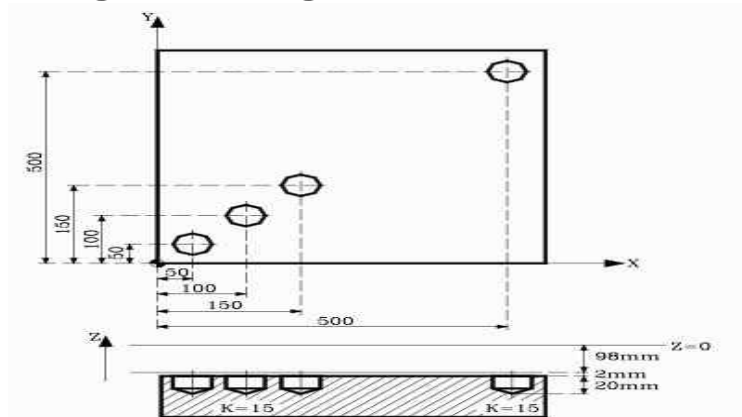
- G81 X_ Y_ Z_ R_ F_ K_ ;
- X_ Y_ : Hole position data
- Z_ : Z-depth (tool will travel with feed to Z-depth starting from R plane)
- R_ : The distance from the initial level to point R level
- F_ : Cutting feedrate
- K_ : Number of repeats (if required)

G81 drilling cycle parameter K (number of repeats). On different cnc controls this parameter has different letter assign to it such as on,

- Fanuc uses letter K for number of repeats.
- Haas CNC uses letter L for number of repeats.
- Fagor CNC Control uses letter N for number of repeats.

Incremental motion in canned cycle is often useful as a loop count, which can be used to repeat the operation with an incremental X or Y move between each cycle.

CNC Code G81 Drilling Cycle with G91 Incremental Programming



G81 Drilling Cycle – Repeat Drilling in G91 Incremental Mode Example Code

```

N10 T1 M6
N20 G00 G90 X0 Y0 Z0
N30 S1450 M03
N40 G81 G99 G91 X50 Y50 Z-120 R-98 K3 F350
N50 G98 G90 G00 X500 Y500
N60 G80
N70 G90 X0 Y0
N80 M30
    
```

Explanation

N40 G81 G99 G91 X50 Y50 Z-120 R-98 K3 F350

K3 means that the G81 drilling cycle will repeat three times.

G91 Incremental mode makes the tool to move every time X50 and Y50

So first hole will be at X50 Y50

Second hole will be at X100 Y100 because

$X100 = X50(\text{previous value}) + X50 (\text{increment})$

$Y100 = Y50(\text{previous value}) + Y50 (\text{increment})$

Third hole will be at X150 Y150 because

$X150 = X100(\text{previous value}) + X50 (\text{increment})$

$Y150 = Y100(\text{previous value}) + Y50 (\text{increment})$

N50 G98 G90 G00 X500 Y500

The fourth hole position is given with G90 Absolute Programming Mode.

The above code is only possible if you use G91 Incremental Programming Mode with G81 Drilling cycle, but if you try to use G90 Absolute Programming Mode with G81 as above you will see the tool will repeat drilling at the same position

Pattern Drilling

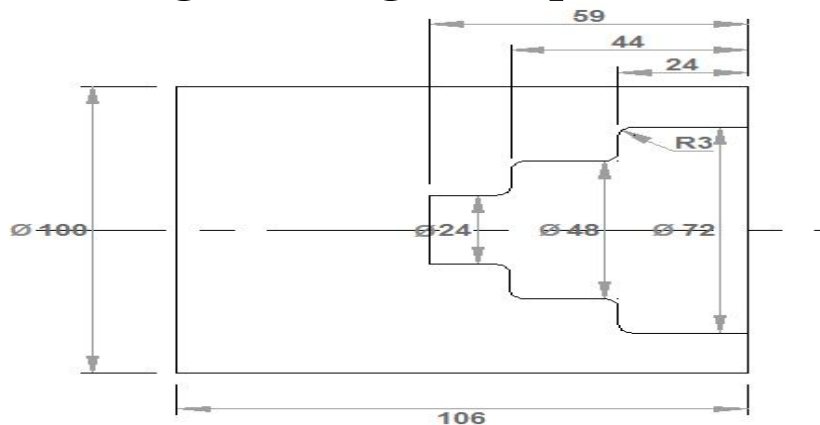
Fanuc Lathe Programming Example Using G70, G71, G74 for ID Machining

Fanuc lathe programming example which uses Fanuc canned cycle

- G71 Turning Cycle
- G70 Finish Cycle
- G74 Peck Drilling Cycle

This Fanuc lathe programming example illustrates who to use G71 Turning cycle, G70 Finish cycle, G74 Peck drilling cycle for ID machining (Inside machining, boring operations)

Fanuc Lathe Programming Example



Fanuc Lathe Programming Example Using G71, G70 for ID Machining

```
N10 G40 G00  
N20 G99  
N30 M5  
N40 M9  
N60 T0101  
N70 G50 S3500  
N80 G96 S0240 M4  
N90 G00 X102. Z0.  
N100 G01 X-2. F0.15 M7
```

N110 G00 X150. Z150.
N120 M9
N130 T0707
N140 G97 S0950 M3
N150 G00 Z3.
N160 G00 X0.
N170 G74 R1.0
N171 G74 X0.0 Z-59.0 Q12000 R0.0 F0.2
N350 G00 X150. Z150.
N360 T0505
N370 G50 S3500
N380 G96 S0200 M4
N390 G00 X23. Z2.
N400 G71 U3. R1
N410 G71 P420 Q530 U-0.5 W0.2 F0.3
N420 G41 G00 X72.
N440 G01 Z-21.
N450 G03 X66. Z-24. I-3. K0.
N460 G01 X54.
N470 G02 X48. Z-27. I0. K-3.
N480 G01 Z-41.
N490 G03 X42. Z-44. I-3. K0.
N500 G01 X30.
N510 G02 X24. Z-47. I0. K-3.
N520 G01 Z-59.
N530 G01 X23.
N540 G40
N550 G00 X150. Z150. F0.3
N560 T1111
N570 G50 S4500
N580 G96 S0380 M4
N590 G41 G00 X72. Z2.
N600 G70 P420 Q530
N610 G40
N620 G00 X23. Z2.
N630 G00 X200. Z150.
N640 M5
N650 M30

Tools & Operations

T0101 Turning Tool – Facing Operation

T0707 Tip Drill Tool – Drilling

T0505 Boring Bar – Internal machining (ID machining)

T1111 Boring Bar – Internal finish machining

CNC Lathe Programming Exercise

Fanuc G71 Turning Cycle, G74 Peck

N540 G00 X70.
N550 G71 U4. R1
N560 G71 P570 Q650 U0.6 W0.2 F0.35
N570 G42 G00 X24.
N580 G01 Z0.
N590 G01 X28. Z-2.
N600 G01 Z-72.
N610 G02 X32. Z-74. I2. K0.
N620 G01 X62.
N630 G01 X68. Z-77.
N640 G01 Z-90.
N650 G40
N660 G00 X150.
N680 G00 Z70.
N690 T0202 M7
N700 G50 S4500
N710 G96 S0380 M4
N720 G00 X16. Z3.
N730 G42 G01 Z0. F0.1
N740 G01 X24.
N750 G01 X28. Z-2.
N760 G01 Z-72.
N770 G02 X32. Z-74. I2. K0.
N780 G01 X62.
N790 G01 X68. Z-77.
N800 G01 Z-90.
N810 G40
N820 G00 X150. Z150.
N830 M5
N840 M9
N850 M30

Used Tools & Operations

- T0101 Turning Tool – Rough Facing
- T0303 Center Drill – Center Drilling
- T0707 Twist Drill – Drilling
- T0404 Turning Tool – Rough Turning
- T0202 Turning Tool – Finish Contour Cutting

Face Grooving with G74 Peck Drilling Cycle CNC Programming Tutorial

G74 Peck Drilling Cycle

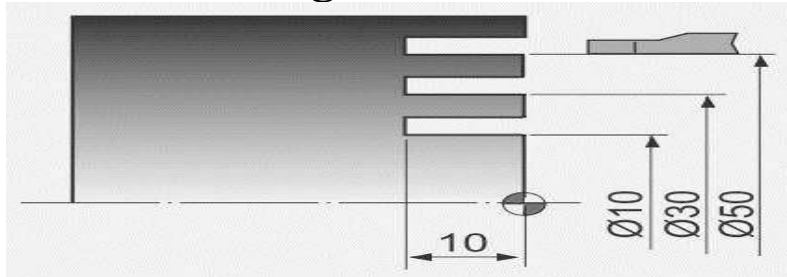
G74 peck drilling cycle be used in variety of ways, from peck drilling to face grooving.

The G74 Peck drilling in already discussed here Simple CNC Lathe Drilling with Fanuc G74 Peck Drilling Cycle.

The cnc programming example below shows how face grooving can be machined with the help of G74 peck drilling canned cycle.

With face grooving operations the tool is fed axially rather than radially toward the end surface of the workpiece.

Face Grooving with G74 Peck Drilling Cycle



Face Grooving with G74 G Code a CNC Programming Tutorial

```
N10 G50 S2000 T0100
N20 G96 S80 M03
N30 G00 X50.0 Z1.0 T0101
N40 G74 R1.0
N50 G74 X10.0 Z-10.0 P10000 Q3000 F0.1
N60 G00 X200.0 Z200.0 T0100
N70 M30
```

Simple CNC Lathe Drilling with Fanuc G74 Peck Drilling Cycle

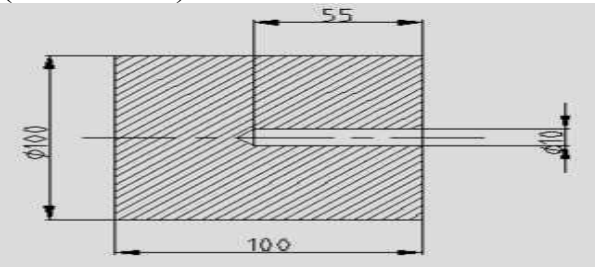
Here is a cnc programming example for simple drilling on a cnc lathe machine. CNC Fanuc control has a very powerful and versatile peck drilling cycle (Fanuc G74) which relieves us from many unwanted chores. Although Fanuc G74 peck drilling cycle for fanuc cnc control can be used in variety of ways but this cnc programming example is just doing a simple peck drilling. One thing for newbies in cnc field is that we can simply drill a component by just giving a feed with G01.

```
N10 G00 X0 Z10
N20 G01 Z-30 F0.2
N30 G01 Z10 F1
```

If we can drill with the above method then why use a peck drilling cycle. Actually peck drilling (Fanuc G74) gives us some hidden benefits like

- Longer drill life
- Proper chip breaking
- Proper chip removal
- Prevents component from heating
- Smooth drilling
- Easy to program

So here is the simple cnc program example which shows simple cnc peck drilling (Fanuc G74) on a cnc lathe machine



Simple CNC Lathe Drilling with Fanuc G74 Peck Drilling Cycle

```
N10 T5  
N20 G97 S500 M03  
N30 G00 X0 Z2  
N40 G74 R1  
N50 G74 Z-60 Q30000 F0.1  
N60 G00 X100 Z100  
N70 M30
```

The above cnc program code shows that the tool no.5 which is a drill, will drill the component with the peck drilling cycle G74.

The R in first block shows the amount (1mm) the drill will get back after it drills 30 mm every time.

Peck Drilling Lathe

Fanuc G83 Peck Drilling Cycle

G83 peck drilling cycle perform the drilling operation in multiple pecks, this technique makes deep-hole drilling easy and economical.

Cutting feed is performed intermittently to the bottom of the hole while chips are discharged.

As the drilling is performed to the bottom of the hole with feed in multiple small steps, every time a specified depth is made and then drill retracts, then drill makes the next peck, this operation is repeated again and again until the drill depth is reached.

Syntax

G83 X... Y... Z... R... Q... F... K...

Parameter	Description
X	Hole position in x-axis.
Y	Hole position in y-axis.
Z	Depth, tool will travel with feed to Z-depth starting from R plane.
R	Position of the R plane.
Q	Depth of cut for each cutting feed (Peck).
K	Number of cycle repetitions (if required) _____.
F	Feedrate.

Once given in program G83 peck drilling cycle is repeated at every axis movement until G80 is given in program to end peck drilling cycle.

Usage

```
N150 M06 T02
N160 G90 G00 X60 Y28 Z12 S750 M03
N170 G99 G83 X60 Y28 Z-17 Q6 R2 F60
N180 G98 Y12
N190 G91 G80 G28 X0 Y0 Z0 M05
N200 M30
```

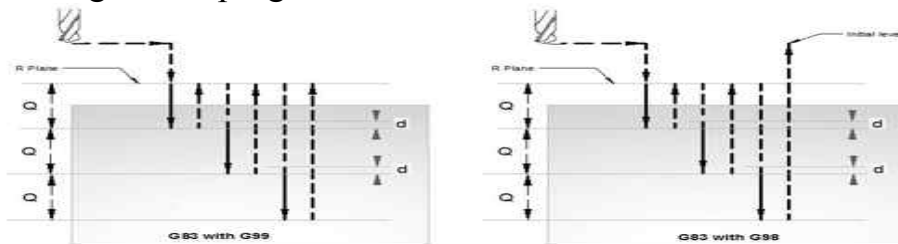
In the above example code first drill is done at X60 Y28 and second at Y12 and then peck drilling is cycle is ended with G80.

6mm pecks are taken to complete total drilling depth of 17mm.

Working

Here is briefly described how G83 peck drilling cycle works,

- 1- Rapid traverse to X, Y drilling position.
 - 2- Rapid traverse to R-plane.
 - 3- Drilling with feed Q deep.
 - 4- Retraction with Rapid traverse to R-plane.
 - 5- Rapid traverse to Q-d deep (d value is specified in parameters).
 - 6- Drilling with feed Q+d deep.
 - 7- Retraction with Rapid traverse to R-plane
- this whole procedure is repeated until drill reaches Z-depth position,
– then drill is retracted to R-plane or Initial-level depends on G99 or G98 which one is given in program.



G83 peck drilling cycle working

G98 G99 Modes

After completing drilling depth the return is made with Rapid feed, the return height can be controlled through using G98 or G99.

G98 Drill will return to the Initial level

G99 Drill will return to R-plane.

G98, G99 can be used multiple times during G83 peck drilling cycle.

Example

```
N30 G83 X10 Y30 Z-17 Q5 R2 F75
N40 Y10
N50 G98 X30
N60 G99 Y30
N70 X90
N80 Y10
N90 G80
```

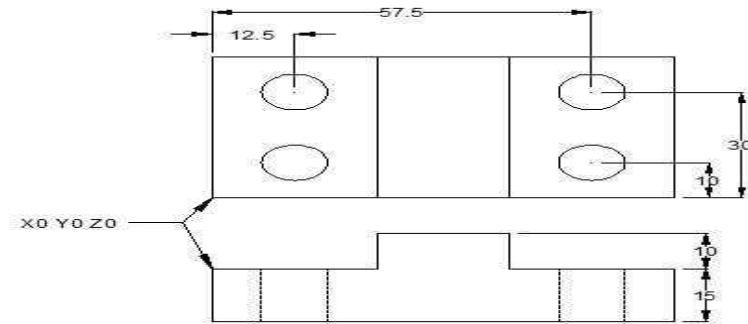
Repeat Drilling

G83 peck drilling cycle, drilling operation can be repeated multiple times. The drilling is repeated K times if K value is given with G83.

Repeat drilling is normally used with G91 Incremental mode, and a good example of repeated drilling is Grid-plate drilling. For working example see G81 drilling

cycle.

Working Example



G83 Peck drilling cycle Example

```
N10 M06 T1  
N20 G90 G00 X12.5 Y10 Z12 S1000 M03  
N30 G99 G83 X12.5 Y10 Z-17 R2 Q4 F75  
N40 Y30  
N50 G98 X57.5  
N60 G99 Y10  
N70 G91 G80 G28 X0 Y0 Z0 M05  
N80 M30
```

G83 Peck Drilling Cycle Example

CNC programming example code for the G83 Peck Drilling Cycle. G83 deep hole peck drilling cycle makes the drilling of deep holes a breeze.

As with G81 drilling cycle you can do simple drilling in a fast and easy way.

But G83 peck drilling due to its specialty of pecking throws away the chips after every cut or peck and this way also the coolant reaches to the bottom of the hole in a free way, so keeps the drill and component cool and clean.

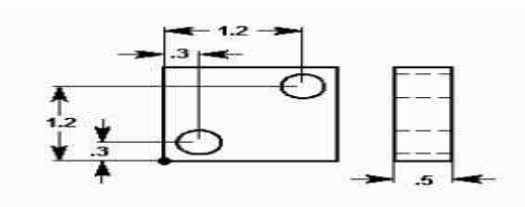
The following G83 peck drilling cycle example code illustrates the use of peck drilling cycle in an easy to understand way, even the beginner level cnc programmers/cnc machinists will understand the working with ease.

For a complete working of G83 Peck Drilling Cycle read

9 G83 Peck Drilling Cycle (Deep Hole) for Fanuc

10 G83 Peck Drilling Cycle (Deep Hole) for Haas CNC

G83 Peck Drilling Cycle Example Program



G83 Peck Drilling Cycle Example

```

N1 T1 M06
N2 G90 G54 G00 X.3 Y.3
N3 S1200 M03
N4 G43 H01 Z1. M08
N5 G83 Z-1.5 Q.5 R.1 F10.
N6 X1.2 Y1.2
N7 G80 G00 Z1. M09
N8 G91 G28 Z0. M05
N9 M30

```

G81 Drilling Cycle G83 Peck Drilling with G98 G99 Example Program

A complete cnc part-program which shows how G98 and G99 (canned cycle return level) work with G81 drilling cycle and G83 peck drilling cycle for drilling of a component which have different heights.

Contents

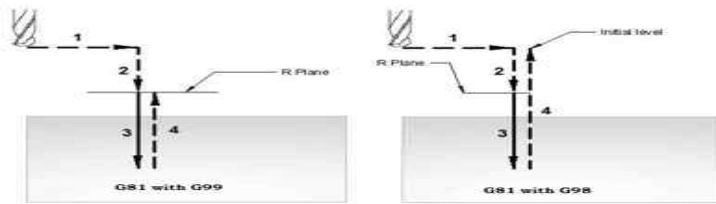
- G98 G99 Summary
- CNC Part Program
- Explanation
- G & M Codes

G98 G99 Summary

G98 and G99 are modal commands that change the way canned cycles (G81,G83 etc.) operate.

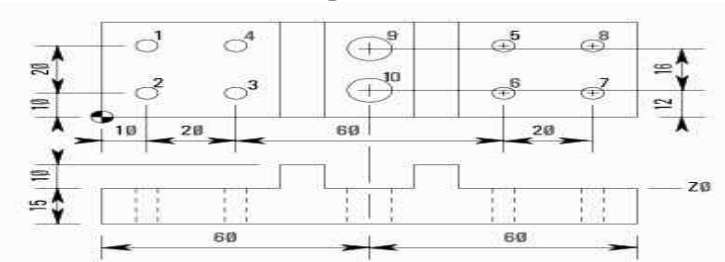
When G98 is active, the Z-axis will return to the start position (initial plane) when it completes an single operation.

When G99 is active, the Z-axis will be returned to the R point (plane) when the canned cycle completes a single hole. Then the machine will go to the next hole.



G81 drilling cycle working – G98 G99 return level

CNC Part Program



G81 Drilling Cycle with G98 G99 G code Example Program

```

N10 M06 T01
N20 G90 G00 X10 Y30 Z12 S1000 M03
N30 G99 G81 X10 Y30 Z-17 R2 F75 (Hole 1)
N40 Y10 (Hole 2)
N50 X30 (Hole 3)
N60 Y30 (Hole 4)
N70 G98 X90 (Hole 5)
N80 G99 Y10 (Hole 6)
N90 X110(Hole 7)
N100 G98 Y30 (Hole 8)
N110 G91 G80 G28 X0 Y0 Z0 M05
N120 M06 T02
N130 G90 G00 X60 Y28 Z12 S750 M03
N140 G99 G83 X60 Y28 Z-17 Q6 R2 F60 (Hole 9)
N150 G98 Y12 (Hole 10)
N160 G91 G80 G28 X0 Y0 Z0 M05
N170 M30
  
```

Explanation

N10- Tool change (M06) to tool no.1

N20- Rapid traverse to X10 Y30 Z12, Spindle started clockwise (M03) with 1000rpm (S1000).

N30- Drilling starts (G81) at X10 Y30 with cutting-feed (F75) drill will retract to R-plane after drilling operation.

N40- Next drilling position Y10 (as G99 is a modal g-code drill will keep on retracting to R-plane until G98 is given).

N50- Next drill at X30.

N60- Drill at Y30

N70- Drill at X90 & Retract to Initial-plane.

N80- Drill at Y10 & Retract to R-plane.

- N90- Drill at X110
 N100- Drill at Y30 & Retract to Initial-plane.
 N110- Drilling cycle is cancelled (G80), return to reference point (G28) for tool change, stop spindle (M05).
 N120- Tool change (M06) to tool number 2.
 N130- Rapid traverse to X60 Y28 Z12, start spindle at 750rpm (S750) clockwise (M03).
 N140- G83 Peck drilling starts at X60 Y28, drill depth is Z-17 and drill peck size is Q6, drilling feed is F60
 N150- Next deep drill at Y12 (return to initial point).
 N160- G83 Peck drilling cycle cancelled with G80, tool returned to reference point (G28), spindle stopped (M05).
 N170- Part-program end with return to program start (M30)

G & M Codes

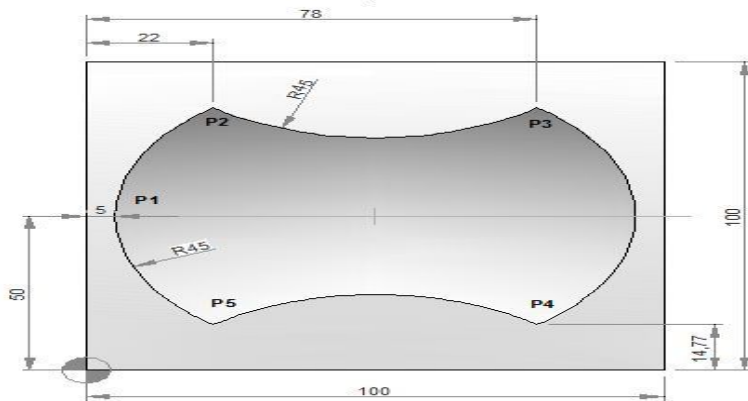
Code	Description
T	Tool no. used.
M06	Tool change command.
G90	Absolute programming
G00	Rapid traverse
S	Cutter speed
M03	Cutter rotation Clockwise
M08	Coolant on.
G81	Fanuc drilling cycle.
G83	Fanuc peck drilling cycle.
G98	Return to initial point in canned cycle.
G99	Return to R point in canned cycle.
F	Cutting feed.
G80	Canned cycle cancel.
M09	Coolant off.
G28	Return to reference position.
G91	Incremental programming.
M05	Cutter rotation stop.
M30	CNC part-program end with return to program-start.

Peck Drilling-Mill

CNC Mill Subprogram Example Joining Multiple Arcs G02 G03 G41

CNC milling program to describe how two or more radii can be joint together in a cnc mill program.

CNC Mill Subprogram Example



CNC Part Program

```
N10 T1 H1 M6 G43 M3
N20 F150 S250
N30 G0 X-21 Y50 Z0.5
N40 G0 Z0
N50 M98 P040050
N60 G49
N70 G0 Z50
N80 M30
```

Subprogram

```
O0050
N10 F160 S400
N20 G0 Z-2.5 G91
N30 G1 G90 X5 Y50 G41 (P1)
N40 G2 X22 Y85.23 I45 J0 (P2)
N50 G3 X78 Y85.23 R45 (P3)
N60 G2 X78 Y14.77 R45 (P4)
N70 G3 X22 Y14.77 R45 (P5)
N80 G2 X5 Y50 R45 (P1)
N90 G0 G40 X-21
```

G M S T Codes Explanation

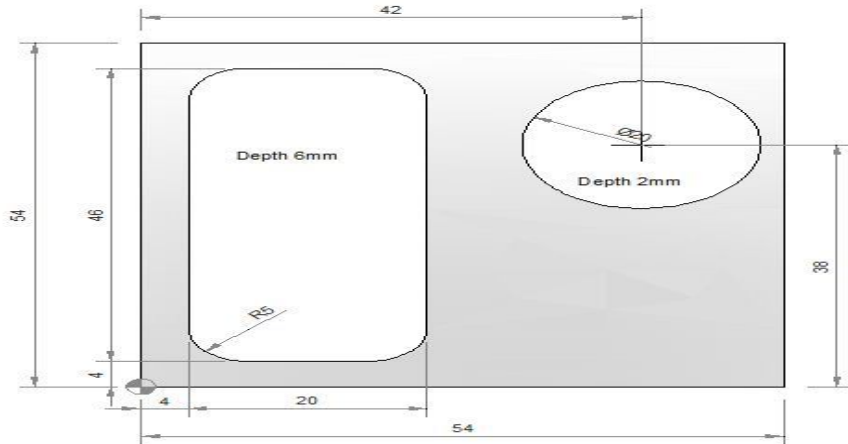
Code	Description
G00	Rapid traverse
G01	Linear interpolation
G02	Circular interpolation CW
G03	Circular interpolation CCW
G40	Cutter compensation cancel
G41	Tool nose radius compensation left
G43	Tool length compensation + direction
G49	Tool length compensation cancel
G90	Absolute command
G91	Increment command
M03	Spindle start forward CW
M06	Tool change
M30	End of program (Reset)
M98	Subprogram call
M99	End of subprogram
T	Tool
S	Speed
F	Feed

CNC Pocket Milling Program Example – Peck Milling

CNC milling program example which shows how a cnc program can be made to machine Pockets on a cnc mill.

This program example uses Peck milling to cut material to machine a rectangular and one round pocket.

CNC Pocket Milling Program Example



Main Program

Milling cutter diameter: 10mm

```

N05 G55
N10 M6 T2 H3 G43 M3
N15 S1000 F60
N20 G0 X9 Y9 Z1
N25 G1 Z0
N30 M98 P030035
N35 G0 Z1 G90
N40 X42 Y38
N45 G1 Z-2 F30
N50 X47 F300
N55 G3 X47 Y38 I-5 J0
N60 G0 Z100
N65 G49
N70 M30

```

Subprogram

```

O0035
N05 G1 Z-2 G91 F30
N10 X10 F100
N15 Y36
N20 X-10
N25 Y-36
N30 M99

```

Explanation

Although this cnc mill program is self explanatory

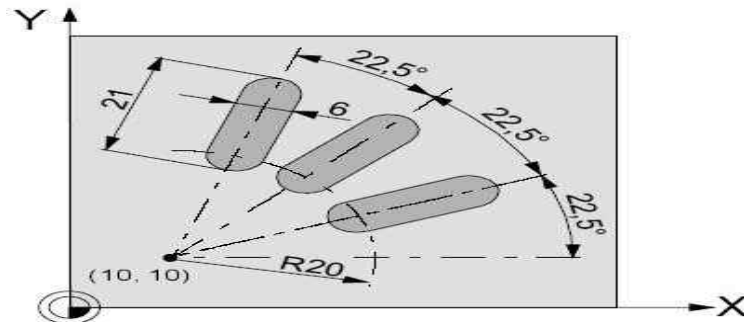
M98 P030035

this code mean call Subprogram No. 0035 three times.

Fanuc G68 Coordinate Rotation – Subprogram Example

CNC program example code to show how Fanuc G68 Coordinate rotation g-code can be used in a subprogram to take most benefit from it's functionality.

G68 Coordinate Rotation Example



Main-program

```
N5 G54  
N10 G43 T10 H10 M6  
N15 S2000 M3 F300  
M98 P030100 ; Subprogram call  
N25 G0 Z50  
N30 M30
```

Subprogram

```
O0100  
N10 G91 G68 X10 Y10 R22.5  
N15 G90 X30 Y10 Z5  
N20 G1 Z-2  
N25 X45  
N30 G0 Z5  
N35 M17  
-----
```

CNC Mill Contour Pecking – Fanuc Subprogram Repeat Example

A very simple cnc programming example which shows how a contour can be repeated multiple times with the added benefit of pecking. So like peck drilling this time cnc machinists are pecking a specific contour of cnc mill.

What is Fanuc Sub programming read Fanuc Sub Programming

CNC Mill Contour Pecking

The subprogram is a simple mix of G-code instructions.

The main program O4000 calls subprogram O4001 to repeat 10 times
M98 P104001

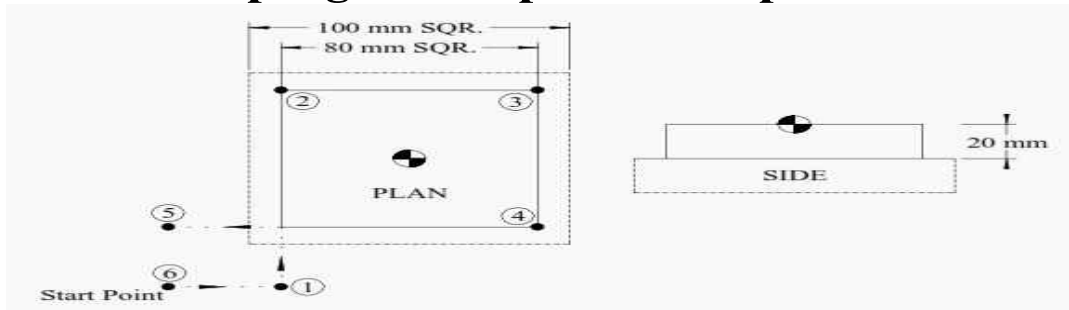
Subprogram cuts contour by making 2mm depth of cut every time (in G91 Incremental Programming)

G1 G91 Z-2 (Incremental peck depth)

Then subprogram switches back to G90 Absolute Programming this makes contour programming easy.

So this program cuts a specific contour 20mm deep by taking 10 pecks of 2mm each.

Fanuc Subprogram Repeat Example



Contour Pecking – Fanuc Mill Subprogram Repeat Example

O4000

T1 M6

G0 G90 G40 G21 G17 G94 G80

G54 X-75 Y-75 S? M3 (Start Point)

G43 Z100 H1

Z5

G1 Z0 F?

M98 P104001 (Call sub-program & repeat 10 times)

G0 G90 Z100

M30

Subprogram

The sub program below takes cut in G91 Incremental Programming Mode
Then switches back to G90 Absolute Programming Mode for the contour
machining.

O4001

G1 G91 Z-2 (Incremental peck depth)

G90 G41 X-40 D? M8 (Absolute Move to position 1 with comp. - Switch on coolant)

Y40 (Move to position 2.)

X40 (Move to position 3.)

Y-40 (Move to position 4.)

X-75 (Move to position 5 - Clear of material - cutter diameter)

G40 Y-75 (Cancel compensation)

Peck Milling

CNC Milling Machine Programming Example for Beginners

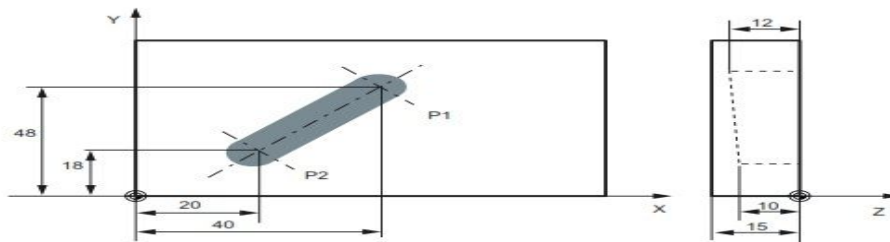
CNC Mill Program Example

A very simple cnc milling machine programming tutorial for beginner level cnc machinists.

An easy to understand cnc mill programming code. This is a cnc g code example without the use of any cnc canned cycle.

Related cnc mill program examples

- CNC G02 Circular Interpolation Clockwise CNC Milling Sample Program
- CNC Milling Circular Interpolation G02 G03 G-Code Program Example
- Beginner level cnc program examples for **CNC Lathe**
- Fanuc CNC Program Example
- CNC Programming for Beginners a Simple CNC Programming Example
- CNC Programming for Beginners a CNC Programming Example



CNC Milling Machine Programming Example for Beginners

```
N05 G0 G90 X40 Y48 Z2 S500 M3
```

```
N10 G1 Z-12 F100
```

```
N15 X20 Y18 Z-10
```

```
N20 G0 Z100
```

```
N25 X-20 Y80
```

```
N30 M2
```

N05 The tool traverses in rapid traverse on P1, three axes concurrently, spindle speed = 500 rpm, clockwise

N10 Infeed on Z-12, feed 100 mm/min

N15 Tool travels on a straight line in space on P2

N20 Retraction in rapid traverse

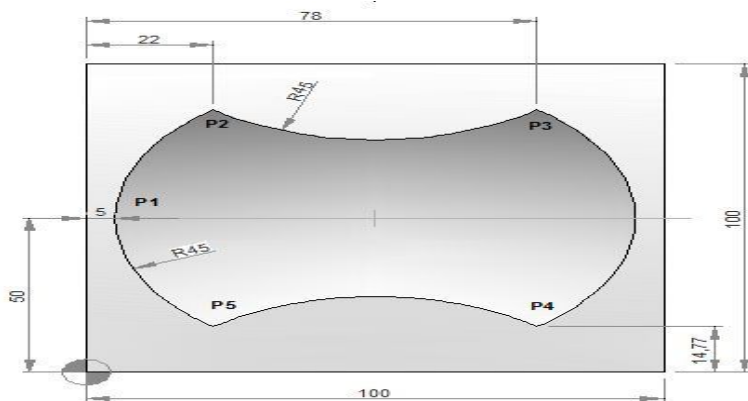
N30 End of program

Ramping Milling

CNC Mill Subprogram Example Joining Multiple Arcs G02 G03 G41

CNC milling program to describe how two or more radii can be joint together in a cnc mill program.

CNC Mill Subprogram Example



CNC Part Program

```
N10 T1 H1 M6 G43 M3
N20 F150 S250
N30 G0 X-21 Y50 Z0.5
N40 G0 Z0
N50 M98 P040050
N60 G49
N70 G0 Z50
N80 M30
```

Subprogram

```
O0050
N10 F160 S400
N20 G0 Z-2.5 G91
N30 G1 G90 X5 Y50 G41 (P1)
N40 G2 X22 Y85.23 I45 J0 (P2)
N50 G3 X78 Y85.23 R45 (P3)
N60 G2 X78 Y14.77 R45 (P4)
N70 G3 X22 Y14.77 R45 (P5)
```

N80 G2 X5 Y50 R45 (P1)
N90 G0 G40 X-21
N100 M99

G M S T Codes Explanation

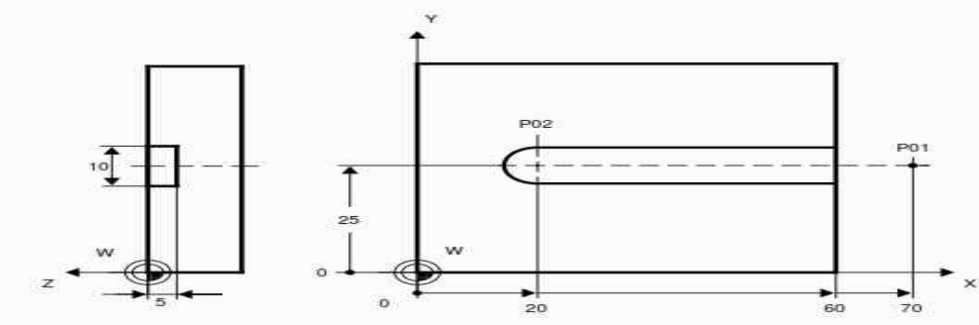
Code	Description
G00	Rapid traverse
G01	Linear interpolation
G02	Circular interpolation CW
G03	Circular interpolation CCW
G40	Cutter compensation cancel
G41	Tool nose radius compensation left
G43	Tool length compensation + direction
G49	Tool length compensation cancel
G90	Absolute command
G91	Increment command
M03	Spindle start forward CW
M06	Tool change
M30	End of program (Reset)
M98	Subprogram call
M99	End of subprogram
T	Tool
S	Speed
F	Feed

Slot Milling a Sample CNC Program Example

A very simple cnc milling program example which shows how a simple slot can be machined.

Another such program example which mills a pocket the same way but in a taper is here [CNC Milling Machine Programming Example for Beginners](#).

Slot Milling Program Example



Slot Milling Sample CNC Program

N10 G00 G90 X70 Y25 Z1 S800 M3

N20 Z-5

N30 G01 X20 F150

N40 G00 Z100

N50 X-25 Y50

N60 M30

N10 Spindle on clockwise rotation at 800 rev/min, tool rapid traverse to P01.

N20 Infeed in Z.

N30 Tool traverse P01 to P02, feedrate 150 mm/min.

N40/N50 Rapid traverse retraction.

N60 End of program.

CNC Milling Machine Programming Example for Beginners

CNC Mill Program Example

A very simple cnc milling machine programming tutorial for beginner level cnc machinists.

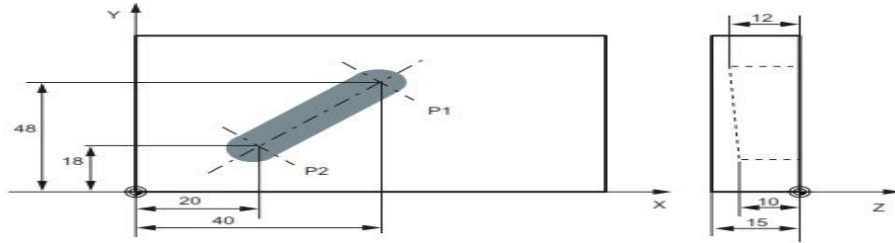
An easy to understand cnc mill programming code. This is a cnc g code example without the use of any cnc canned cycle.

Related cnc mill program examples

- CNC G02 Circular Interpolation Clockwise CNC Milling Sample Program
- CNC Milling Circular Interpolation G02 G03 G-Code Program Example

Beginner level cnc program examples for **CNC Lathe**

- Fancu CNC Program Example
- CNC Programming for Beginners a Simple CNC Programming Example
- CNC Programming for Beginners a CNC Programming Example



CNC Milling Machine Programming Example for Beginners

```
N05 G0 G90 X40 Y48 Z2 S500 M3
```

```
N10 G1 Z-12 F100
```

```
N15 X20 Y18 Z-10
```

```
N20 G0 Z100
```

```
N25 X-20 Y80
```

```
N30 M2
```

N05 The tool traverses in rapid traverse on P1, three axes concurrently, spindle speed = 500 rpm, clockwise

N10 Infeed on Z-12, feed 100 mm/min

N15 Tool travels on a straight line in space on P2

N20 Retraction in rapid traverse

N30 End of program

Slot Milling

Fanuc G71 Turning Cycle

Fanuc G71 Turning Cycle

G71 turning cycle is used for rough-material removal from a cnc lathe component. G71 turning cycle makes large diameter cutting easy. Cutting can be done in simple straight line or a complex contour can also be machined very easily.

Through G71 turning cycle parameters cnc machinists can control

- Depth of cut.
- Retract height.
- Finishing allowance in x-axis and z-axis.
- Cycle cutting-feed, spindle speed.

Programming

G71 U... R...

G71 P... Q... U... W... F... S...

Parameters

First block

Parameter	Description
U	Depth of cut.
R	Retract height.

Second block

Parameter	Description
P	Contour start block number.
Q	Contour end block number.
U	Finishing allowance in x-axis.
W	Finishing allowance in z-axis.
F	Feedrate during G71 cycle.
S	Spindle speed during G71 cycle.

G71 Turning Cycle Overview

- G71 turning cycle cuts the whole contour repeatedly which is given in P Q blocks.
- Depth of every cut can be controlled by first-block U value.

- Second-block U W are the finishing allowances which can be given if you want to make a finish cut with G70 finishing cycle.
- F is cutting feed and S is spindle speed (given in second-block) which are used during G71 turning cycle.

Note – The F and S given inside P Q block will not be used during G71 turning cycle, they are used with G70 finishing cycle if later called.

G71 Turning Cycle Working

```
N60 G71 U10 R10
N70 G71 P80 Q90 U3 W0 F0.25
N80 G00 X60
N90 G01 Z-75
```

When G71 turning cycle is run the whole operation will be done in following sequence,

First-cut

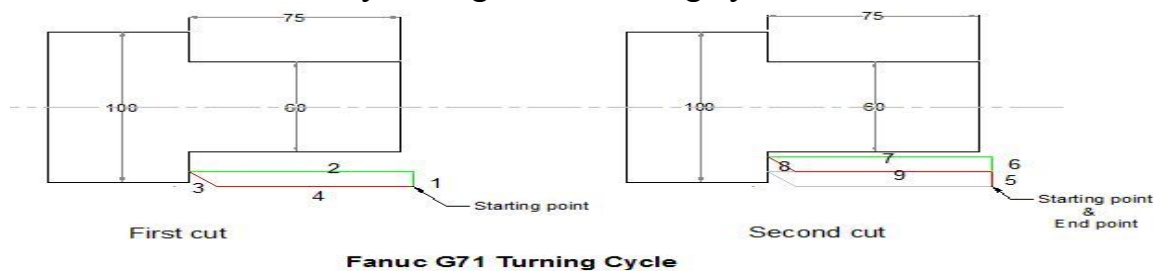
- 1 – Tool will move in x-axis U (depth of cut) deep with programmed feed from starting-point.
- 2 – Tool will travel with feed in z-axis (destination point in z-axis is given in P Q blocks)
- 3 – Tool rapidly retracts R amount in both x-axis and z-axis (at 45 degrees).
- 4 – Tool rapidly travel in z-axis to start-point

Later-cuts

- 5 – Tool rapidly moves to last cut depth.
- 6 – Tool moves with feed in x-axis U deep (first-block U depth of cut).
- 7 – Tool with feed moves in z-axis (destination point given in P Q blocks).
- 8 – Tool rapidly retracts in x-axis and z-axis R amount (45 degrees).
- 9 – Tool rapidly moves to start-point only in z-axis.

This whole sequence of operation keep on going, until the destination point in x-axis is met.

If finishing allowance is given tool will not make the exact diameter and length given in P Q blocks but will leave that much allowance, This finishing allowance can be later machined by calling G70 finishing cycle.



Fanuc G71 Example

Here is a cnc part-program which shows how G71 turning cycle can be used, this is the program for the drawing given above

```
N50 G00 X106 Z5 M3 S800  
N60 G71 U10 R10  
N70 G71 P80 Q90 U3 W0 F0.25  
N80 G00 X60  
N90 G01 Z-75
```

In this program G71 turning cycle will keep repeating the contour given inside P Q blocks shown below

```
N80 G00 X60  
N90 G01 Z-75
```

These two cnc program blocks tell us that we want to remove material till X60 deep and in Z-75 in length.

The depth of cut is given in first-block U10 retract amount is also given R10. Finishing allowance in x-axis is U3 but there is no finishing allowance given in z-axis W0.

G70 Finishing Cycle

If you programmed G71 turning cycle with finishing allowances then that finish allowances can be removed with G70 finishing cycle.

G70 finishing cycle repeats the whole contour the G71 way, but in just one-cut removing the finishing allowances.

Why Use G70 Finishing Cycle

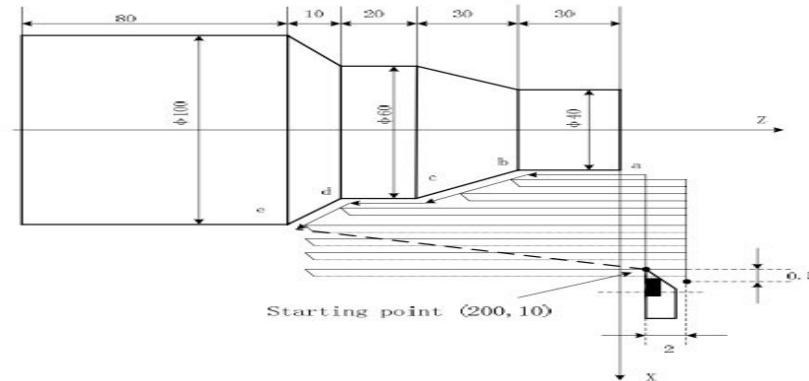
As material can be removed with G71 turning cycle, but if you want a different cutting-feed and spindle speed for the last cut, then it is recommended that you use G70 finishing cycle.

G70 finishing cycle use F and S values which are given inside P Q programmed blocks. (G71 use F S values which are given inside G71 second block.)

Fanuc G70 Example

```
N50 G00 X106 Z5 M3 S800  
N60 G71 U10 R10  
N70 G71 P80 Q90 U3 W0 F0.25  
N80 G00 X60  
N90 G01 Z-75 F0.15  
N100 G00 X200 Z100  
N110 G92 S1200  
N120 T3 G96 S150 M03  
N130 G00 X106 Z5  
N140 G70 P80 Q90  
N150 G00 X200 Z100  
N160 M30
```

G70 G71 Example



G71 Rough Turning Cycle Example

```

O0004
G00 X200 Z10 M3 S800
G71 U2 R1 F200
G71 P80 Q120 U0.5 W0.2
N80 G00 X40 S1200
G01 Z-30 F100
X60 W-30
W-20
N120 X100 W-10
G70 P80 Q120
M30

```

Fanuc G71 G72 G70 Canned Cycle CNC Lathe Internal Machining Example (Boring & Facing)

Fanuc programming example which shows the use of multiple fanuc canned cycle in cnc programming, Following canned cycle are used in this cnc lathe programming example

- G71 Rough Turning Cycle
- G72 Facing Cycle
- G70 Finish Cycle

Contents

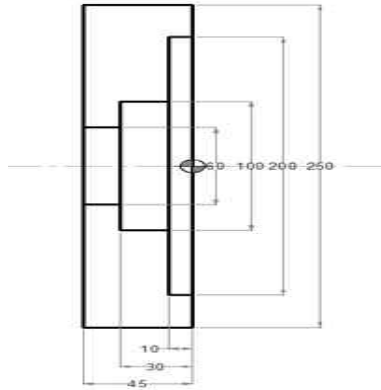
- Fanuc G71 for Boring Opertaion
- Fanuc Programming Example

Fanuc G71 for Boring Opertaion

This programming example also illustrate how G71 turning cycle can be used for internal machining (boring operation). So if you want to remove extra stock from

inside of a component, you can use Fanuc G71 turning cycle for internal stock removal as well.

Fanuc Programming Example



Fanuc Canned Cycle Example

```
G28 U0 W0
G50 S1500
N1 T0101 M8 (OD FACING)
G96 S180 M3
G0 X255.0 Z5.0
G72 W2.0 R0.5
G72 P100 Q200 F0.25
N100 G0 Z0 G41
G1 X-2.0 F0.18
N200 G0 Z5.0
G40
X255.0
G28 U0 W0
N2 T0404 M8 (ID ROUGH)
G96 S180 M3
G0 X50.0 Z5.0
G71 U2.0 R0.5
G71 P500 Q600 U-0.5 W0.1 F0.25
N500 G0 X202.0 G41
G1 Z0 F0.12
X200.0
Z-10.0
X100.0
Z-30.0
X60.0
Z-45.0
N600 X50.0
G40
G0 Z5.0
G28 U0 W0
N3 T0505 M8 (ID FINISH)
G96 S220 M3
G0 X50.0 Z5.0
G70 P500 Q600
G0 X50.0 Z5.0
G28 U0 W0
```

M5 M9
M30

Fanuc G73 Pattern Repeating Cycle CNC Program Example Code

CNC programming example for Fanuc G73 pattern repeating cycle.
Fanuc G73 Pattern Repeating Cycle has already been described here
CNC Fanuc G73 Pattern Repeating Cycle

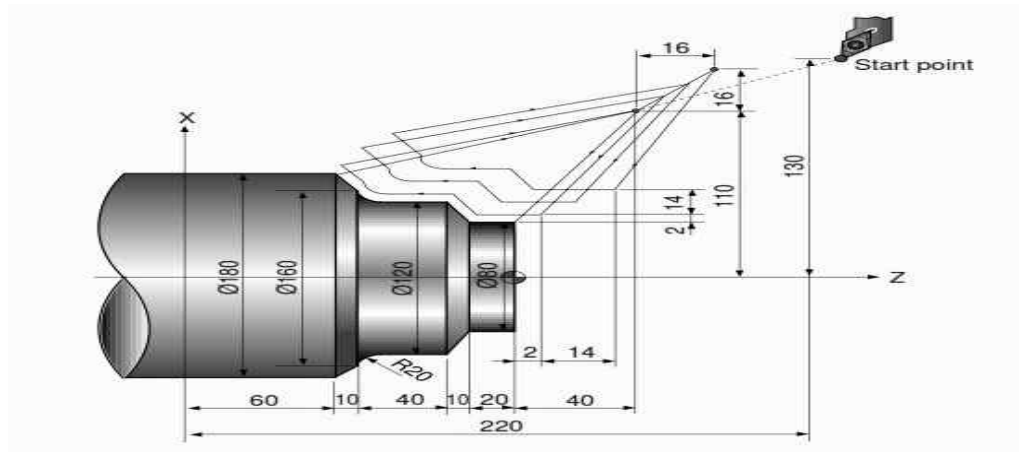
You might like other Fanuc G73 pattern repeating cycle examples

CNC Fanuc G73 Pattern Repeating Cycle CNC Program Example

Fanuc G73 Pattern Repeating Canned Cycle Basic CNC Sample Program

Fanuc G73 Pattern Repeating Cycle Programming Example

This cnc program example also shows how cnc machinists can use 'W' instead of 'Z' for z-axis movements.



Fanuc G73 Pattern Repeating Cycle Program Example

```
N010 G00 X260.0 Z80.0  
N011 G00 X220.0 Z40.0  
N012 G73 U14.0 W14.0 R3  
N013 G73 P014 Q020 U4.0 W2.0 F0.3 S0180  
N014 G00 G42 X80.0 Z2.0  
N015 G01 W-20.0 F0.15 S0600  
N016 X120.0 W-10.0  
N017 W-20.0 S0400  
N018 G02 X160.0 W-20.0 R20.0  
N019 G01 X180.0 W-10.0 S0280  
N020 G40
```

N021 G70 P014 Q020
N022 G00 X260.0 Z80.0
N023 M30

CNC Programming Example with Fanuc G71 Rough Turning Cycle and G70

Here is another **cnc programming example**, this cnc programming example shows the use of **G71 Canned Cycle** and **G70 for Fanuc CNC Control**.

Contents

- G71 Turning Cycle
- Program Example

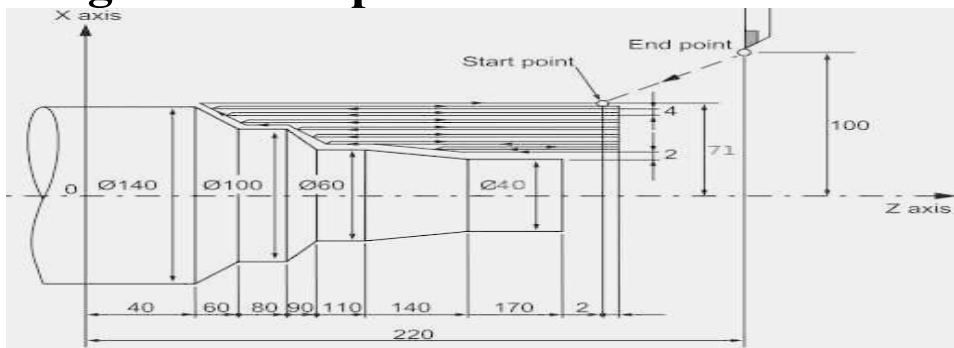
G71 Turning Cycle

Although I already have posted about the G71 Turning Canned Cycle(Rough Turning Cycle), but that blog post just illustrates the use of G71 and G71 parameters.

This cnc programming example shows a complete contour cutting with G71 and finish cut on contour with G70.

G70 finishing cycle for fanuc cnc control can also be used with G72 Facing cycle for fanuc control. Usage of G70 Finishing cycle with *G72 Facing cycle* is same as shows here in the following example.

Program Example



CNC Programming Example with Fanuc G71 Rough Turning Cycle and G70

N10 G00 G90 X142 Z171
N20 G71 U4 R1

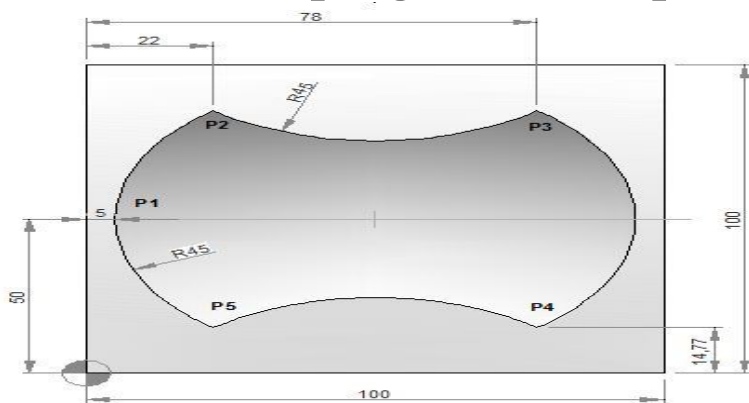
N30 G71 P40 Q110 U4 W2 F0.3
N40 G00 X40
N50 G01 Z140 F0.2
N60 G01 X60 Z110
N70 G01 Z90
N80 G01 X100 Z80
N90 G01 Z60
N110 G01 X140 Z40
N120 G70 P40 Q110
N130 G00 X200 Z220
N140 M30

Step Turning CNC Lathe

CNC Mill Subprogram Example Joining Multiple Arcs G02 G03 G41

CNC milling program to describe how two or more radii can be joint together in a cnc mill program.

CNC Mill Subprogram Example



CNC Part Program

```
N10 T1 H1 M6 G43 M3
N20 F150 S250
N30 G0 X-21 Y50 Z0.5
N40 G0 Z0
N50 M98 P040050
N60 G49
N70 G0 Z50
N80 M30
```

Subprogram

```
O0050
N10 F160 S400
N20 G0 Z-2.5 G91
N30 G1 G90 X5 Y50 G41 (P1)
N40 G2 X22 Y85.23 I45 J0 (P2)
N50 G3 X78 Y85.23 R45 (P3)
N60 G2 X78 Y14.77 R45 (P4)
N70 G3 X22 Y14.77 R45 (P5)
N80 G2 X5 Y50 R45 (P1)
```

N90 G0 G40 X-21
N100 M99

G M S T Codes Explanation

Code	Description
G00	Rapid traverse
G01	Linear interpolation
G02	Circular interpolation CW
G03	Circular interpolation CCW
G40	Cutter compensation cancel
G41	Tool nose radius compensation left
G43	Tool length compensation + direction
G49	Tool length compensation cancel
G90	Absolute command
G91	Increment command
M03	Spindle start forward CW
M06	Tool change
M30	End of program (Reset)
M98	Subprogram call
M99	End of subprogram
T	Tool
S	Speed
F	Feed

CNC Pocket Milling Program Example – Peck Milling

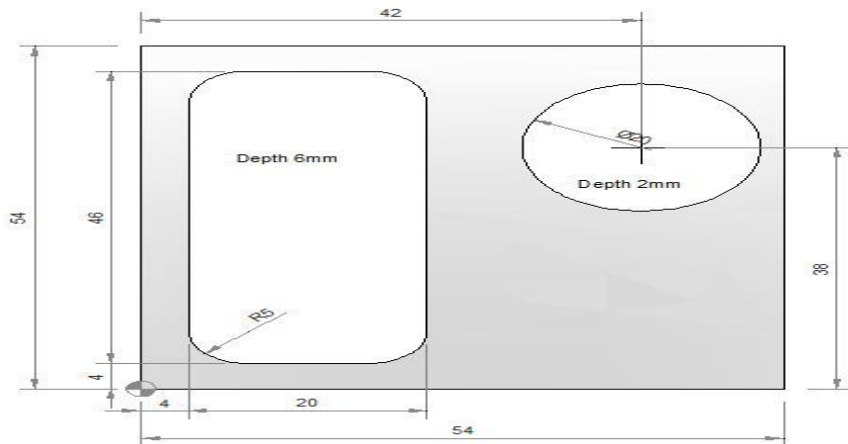
CNC milling program example which shows how a cnc program can be made to machine Pockets on a cnc mill.

This program example uses Peck milling to cut material to machine a rectangular and one round pocket.

Contents

- 11 CNC Pocket Milling Program Example
 - Main Program
 - Subprogram
 - Explanation

CNC Pocket Milling Program Example



Main Program

Milling cutter diameter: 10mm

```
N05 G55
  N10 M6 T2 H3 G43 M3
  N15 S1000 F60
  N20 G0 X9 Y9 Z1
  N25 G1 Z0
  N30 M98 P030035
  N35 G0 Z1 G90
  N40 X42 Y38
  N45 G1 Z-2 F30
  N50 X47 F300
  N55 G3 X47 Y38 I-5 J0
  N60 G0 Z100
  N65 G49
  N70 M30
```

Subprogram

```
O0035
  N05 G1 Z-2 G91 F30
  N10 X10 F100
  N15 Y36
  N20 X-10
  N25 Y-36
  N30 M99
```

Explanation

Although this cnc mill program is self explanatory

M98 P030035

this code mean call Subprogram No. 0035 three times.

Multi Start Threads with Fanuc G76 Threading Cycle

Fanuc cnc controls has no direct threading cycle for cutting **multi start threads** on cnc. But you can cut multi start threads on a cnc with fanuc control by using Fanuc G76 Threading Cycle.

Related: Fanuc G76 Thread Cycle for Dummies



Multi Start Threads with Fanuc G76 Threading Cycle

There are multiple techniques for cutting multi-start threads on cnc with Fanuc G76 threading cycle. Here is one of them.

This cnc programming example uses cnc subprogram call to cut multi-start threads on Fanuc cnc.

Contents

- CNC Main program
- CNC Sub Program

CNC Main program

```
N50 G00 X48 Z5  
N60 M98 P0034713
```

CNC Sub Program

```
N10 G76 P020000 Q100 R0.05  
N20 G76 X42 Z-15 P974 Q200 F4.5  
N30 G00 W1.5  
N40 M99
```

The above cnc main program calls the 4713 sub-program three times.

Subprogram

Taper Threading with G32 a CNC Programming Example

G32 is used for thread cutting, but with G32 we can just make a single threading cut. This all mean that you yourself have to do all the work of threading tool positioning.

For brief introduction of G32, G33 read CNC Fanuc G33 G32 Threading G Code.

Thread Cutting Possibilities with G32

- Longitudinal thread.
- Transverse thread.
- Tapered thread.
- Single and Multiple Start threads with a constant pitch.
- Variable pitch and taper angles by Chaining thread cutting blocks.

For taper threading with G32 first take the tool to the start position.

Then make the thread cut with G32 and give the values of destination point in X, Z axis along with thread pitch F.

The Z value in the G32 thread cutting has the same meaning as the Z value in the G76 cycle or the G92 *threading cycle*. It represents the end position.

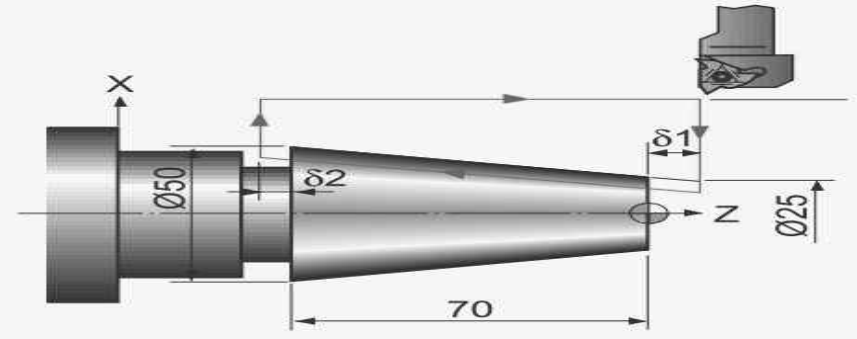
There are multiple possibilities of taper threading on cnc lathe machines.

Taperd threading with G76 threading cycle.

Taper Threading with G92 Threading Cycle.

CNC Programming Example G92 Taper Threading Cycle.

Taper Threading with G32



Taper Threading with G32 a CNC Programming Example

```

N10 G50 S800 T0100
N20 G97 S800 M03
N30 G00 X90.0 Z5.0
N40 X22.026
N50 G32 X49.562 Z-71.5 F3.0
N60 G00 X90.0
N70 Z5.0
N80 X21.052
N90 G32 X48.588 Z-71.5
N100 G00 X90.0
N110 Z5.0
N120 X150.0 Z150.0
N130 M30

```

Here are just two passes of taper threading in this cnc programming example, but you can repeat the thread passes as much you require.

Taper Threading

Fanuc G84 Tapping Cycle

G84 tapping cycle performs tapping operation, tapping is performed by rotating the spindle clockwise, when bottom of the hole has been reached, the spindle is rotated in the reverse direction for retraction. This whole operation creates threads.

Syntax

G84 X... Y... Z... R... P... F... K...

Parameter	Description
X	Hole position in x-axis.
Y	Hole position in y-axis.
Z	Depth, tapping from R-plane to Z-depth.
R	Position of the R plane.
P	Dwell time.
K	Number of cycle repetitions (if required)
F	Feedrate.

Once given in program G84 tapping cycle is repeated at every axis movement until G80 is given in program to end tapping cycle.

Notes

Feedrate override is ignored during tapping.

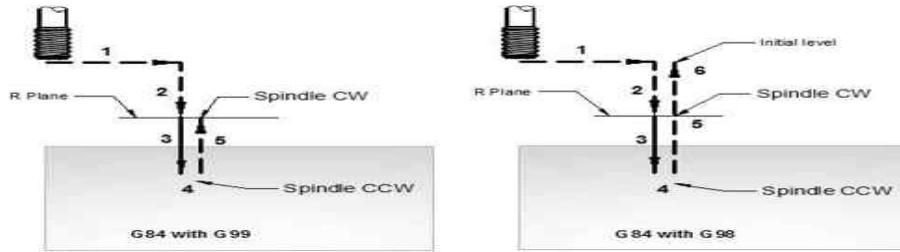
Feed-hold does not stop the machine until the return operation is completed.

Usage

```
N150 M6 T2
N160 G90 G00 X60 Y28 Z12 S100 M03
N170 G99 G84 X60 Y28 Z-17 P300 R2 F120
N180 G98 Y12
N190 G91 G80 G28 X0 Y0 Z0 M05
N200 M30
```

Working

Brief description of how G84 tapping cycle works,



G84 tapping cycle working

- 0- CW Spindle rotation command (M03) must be given in part-program.
- 1- Rapid traverse to X, Y position (pre-drill hole position).
- 2- Rapid traverse to R-plane.
- 3- Tapping operation is done till Z-depth (with tapping feed given with G84).
- 4-1- Dwell for time specified with P
- 4-2- Spindle is rotated CCW
- 5- Tap is Retracted with the specified feed.
- At R-plane spindle is rotated CW.
- If G99 is specified then tapping for this hole ends here,
- 6- if G98 is given in program then tap is moved to Initial-level.

G98 G99 Modes

After completing tapping the return height can be controlled by using G98 or G99.

G98 Tap will return to the Initial level

G99 Tap will return to R-plane.

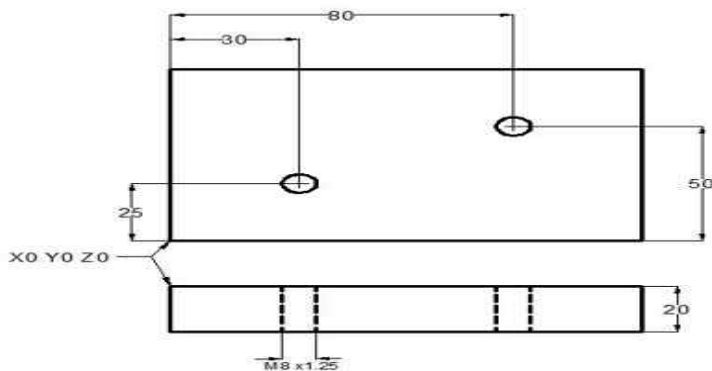
G98, G99 can be used multiple times during G84 tapping cycle.

Repeat Drilling

G84 tapping cycle can be repeated multiple times if K value is given.

For working example see G81 drilling cycle.

Working Example



G84 tapping cycle example

N10 T1 M06

N20 G90 G54 G00 X30 Y25

N30 S100 M03
N40 G43 H01 Z5 M08
N50 G84 Z-20 R2 F1.25
N60 X80 Y50
N70 G80 G00 Z100 M09
N80 M30

Tapping

Internal Threading on Fanuc 21i 18i 16i with G76 Threading Cycle

CNC program for the internal threading with G76 threading cycle on fanuc controls 21i/18i/16i.

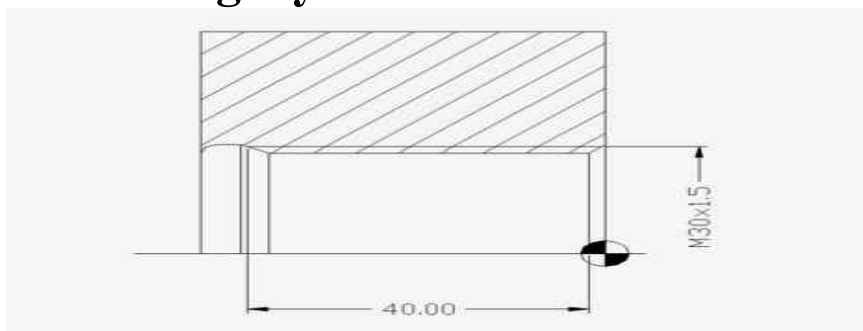
For an example of external threading with G76 threading cycle read External Thread Cutting with G76 Threading Cycle on Fanuc 21i 18i 16i CNC Fanuc 21i/18i/16i use two block format of G76 threading cycle.

Related: G76 Threading Cycle One Line Format for Fanuc 10/11/15T

Fanuc G76 threading cycle has multiple parameters making it difficult to remember, but at the same time those multiple parameters of G76 thread cycle give the cnc programmer/cnc machinist multiple options to control thread cutting, some are listed below.

G76 thread cutting cycle allow cnc machinist to control number of idle cuts, thread run-out, infeed angle.

CNC Program of Internal Threading with G76 Threading Cycle



Internal Threading on Fanuc 21i 18i 16i with G76 Threading Cycle

```
N17 T101  
N18 G54  
N19 G97 S800 M3  
N20 G0 X25 Z6 M8  
N21 G76 P010060 Q100 R0.02  
N22 G76 X30 Z-40 P919 Q250 F1.5  
N23 G0 X150 Z100
```

G76 Thread Cycle a CNC Programming Example

G-code G76 is a cnc cycle which is used for thread cutting on cnc machines.

Threading cycle G76 is explained here **G76 Thread Cycle**.

Taper thread cutting with G76 thread cycle is explained here **G76 Tapered Threading**

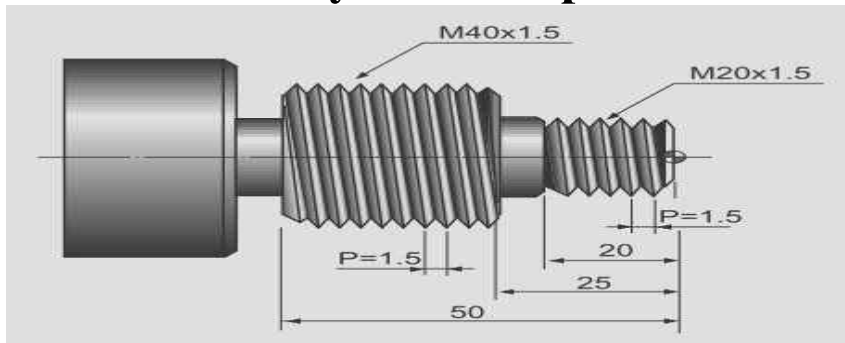
For **Multi-start thread cutting with G76** see **G76 Multi-Start Threading**

For **G76** threading cycle one line see **G76 One-Line Format**.

G76 threading cycle can be used for internal threading on cnc lathe machines.

This **G76** threading example actually cuts external threads on two different diameters.

G76 Thread Cycle Example



G76 Thread Cycle a CNC Programming Example

N10 T3

N20 G97 S800 M03

N30 G00 X30 Z5 T0303

N40 G76 P021060 QI00 R100

N50 G76 X18.2 Z-20 P900 Q200 FI.5

N60 G00 X50 Z-20

N70 G76 P021060 QI00 R100

N80 G76 X38.2 Z-52 P900 Q200 FI .5

N90 G00 X200 Z200

N100 M30

Threading

Fanuc G71 Turning Cycle

Fanuc G71 Turning Cycle

G71 turning cycle is used for rough-material removal from a cnc lathe component. G71 turning cycle makes large diameter cutting easy. Cutting can be done in simple straight line or a complex contour can also be machined very easily.

Through G71 turning cycle parameters cnc machinists can control

- Depth of cut.
- Retract height.
- Finishing allowance in x-axis and z-axis.
- Cycle cutting-feed, spindle speed.

Programming

G71 U... R...

G71 P... Q... U... W... F... S...

Parameters

First block

Parameter	Description
U	Depth of cut.
R	Retract height.

Second block

Parameter	Description
P	Contour start block number.
Q	Contour end block number.
U	Finishing allowance in x-axis.
W	Finishing allowance in z-axis.
F	Feedrate during G71 cycle.
S	Spindle speed during G71 cycle.

G71 Turning Cycle Overview

- G71 turning cycle cuts the whole contour repeatedly which is given in P Q blocks.

- Depth of every cut can be controlled by first-block U value.
- Second-block U W are the finishing allowances which can be given if you want to make a finish cut with G70 finishing cycle.
- F is cutting feed and S is spindle speed (given in second-block) which are used during G71 turning cycle.

Note – The F and S given inside P Q block will not be used during G71 turning cycle, they are used with G70 finishing cycle if later called.

G71 Turning Cycle Working

```
N60 G71 U10 R10
N70 G71 P80 Q90 U3 W0 F0.25
N80 G00 X60
N90 G01 Z-75
```

When G71 turning cycle is run the whole operation will be done in following sequence,

First-cut

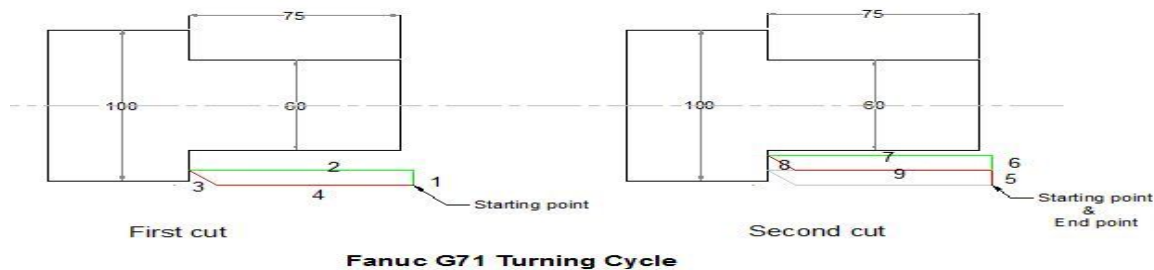
- 1 – Tool will move in x-axis U (depth of cut) deep with programmed feed from starting-point.
- 2 – Tool will travel with feed in z-axis (destination point in z-axis is given in P Q blocks)
- 3 – Tool rapidly retracts R amount in both x-axis and z-axis (at 45 degrees).
- 4 – Tool rapidly travel in z-axis to start-point

Later-cuts

- 5 – Tool rapidly moves to last cut depth.
- 6 – Tool moves with feed in x-axis U deep (first-block U depth of cut).
- 7 – Tool with feed moves in z-axis (destination point given in P Q blocks).
- 8 – Tool rapidly retracts in x-axis and z-axis R amount (45 degrees).
- 9 – Tool rapidly moves to start-point only in z-axis.

This whole sequence of operation keep on going, until the destination point in x-axis is met.

If finishing allowance is given tool will not make the exact diameter and length given in P Q blocks but will leave that much allowance, This finishing allowance can be later machined by calling G70 finishing cycle.



Fanuc G71 Turning Cycle

Fanuc G71 Example

Here is a cnc part-program which shows how G71 turning cycle can be used, this is the program for the drawing given above

```
N50 G00 X106 Z5 M3 S800  
N60 G71 U10 R10  
N70 G71 P80 Q90 U3 W0 F0.25  
N80 G00 X60  
N90 G01 Z-75
```

In this program G71 turning cycle will keep repeating the contour given inside P Q blocks shown below

```
N80 G00 X60  
N90 G01 Z-75
```

These two cnc program blocks tell us that we want to remove material till X60 deep and in Z-75 in length.

The depth of cut is given in first-block U10 retract amount is also given R10. Finishing allowance in x-axis is U3 but there is no finishing allowance given in z-axis W0.

G70 Finishing Cycle

If you programmed G71 turning cycle with finishing allowances then that finish allowances can be removed with G70 finishing cycle.

G70 finishing cycle repeats the whole contour the G71 way, but in just one-cut removing the finishing allowances.

Why Use G70 Finishing Cycle

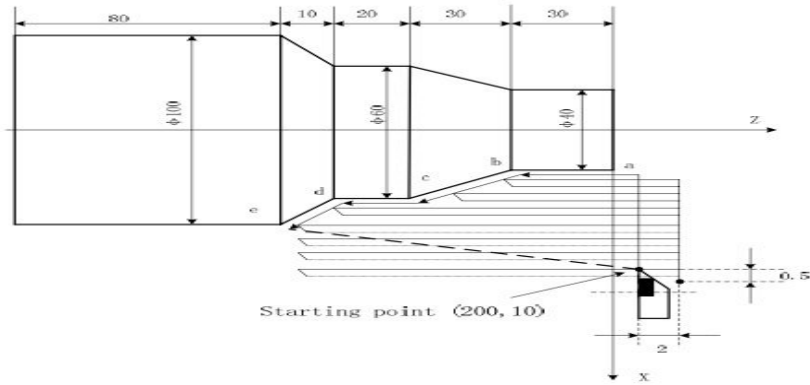
As material can be removed with G71 turning cycle, but if you want a different cutting-feed and spindle speed for the last cut, then it is recommended that you use G70 finishing cycle.

G70 finishing cycle use F and S values which are given inside P Q programmed blocks. (G71 use F S values which are given inside G71 second block.)

Fanuc G70 Example

```
N50 G00 X106 Z5 M3 S800  
N60 G71 U10 R10  
N70 G71 P80 Q90 U3 W0 F0.25  
N80 G00 X60  
N90 G01 Z-75 F0.15  
N100 G00 X200 Z100  
N110 G92 S1200  
N120 T3 G96 S150 M03  
N130 G00 X106 Z5  
N140 G70 P80 Q90  
N150 G00 X200 Z100  
N160 M30
```

G70 G71 Example



G71 Rough Turning Cycle Example

O0004

G00 X200 Z10 M3 S800

G71 U2 R1 F200

G71 P80 Q120 U0.5 W0.2

N80 G00 X40 S1200

G01 Z-30 F100

X60 W-30

W-20

N120 X100 W-10

G70 P80 Q120

M30

