TOYODA

TAKISAWA
TAIWAN
Taiwan TAKISAWA Technology Co., Ltd.

Takisawa M Series

Quick Reference Operations Manual

M Code Function	Main Spindle	Milling spindle
Spindle Forward	M03	M03
Spindle Reverse	M04	M03
Spindle Stop	M05	M05
Spindle Neutral (Gear Box Only)	M40	
Spindle Low Gear (Gear Box Only)	M41	
Spindle High gear (Gear Box Only)	M42	
Spindle Select Turning Mode	M75	
Spindle Select Milling Mode	M76	
Spindle Select Grinding Mode (Run Main and Milling Spindle at the Same Time)		M77 S….. M3/M4 (Milling) G4 U1. (1 Second Dwell) S……M3/M4 (Turning Spindle)
Spindle Orientation	M19	Automatic With Turret Index or Spindle Mode Switch
Spindle Orientation Cancel	M05	In Manual Mode Press Fanuc Rest and Hold for 5-10 Seconds
Chuck O.D Hold	M88	
Chuck I.D Hold	M89	
Chuck Clamp	M68	
Chuck Unclamp	M69	
Chuck Interlock Bypass	M31	
Bypass Off	M32	
C-Axis Brake ON	M50	
C-axis Brake Off	M51	
Rigid Tapping ON	M29 S…..	M29 S…..
Rigid Tapping Reverse Left Hand or Non-Normal Rotation Live Tool	M29; M28;	M29; M28;
Air Blow ON (Option)	M26	
Air Blow Off (Option)	M27	
Safe Start/Tool Change Position *	M101 Z_*	
Chatter Turn	M43 V_F_	

* Custom Macro program Not Supplied From Takisawa Taiwan

M Codes Function	All Modes
Program Stop	M00
Optional Program Stop	M01
High Pressure Coolant On	M07
Coolant On	M08
Coolant Off	M09
Tail Stock Quill Forward	M10
Tail Stock Quill Reverse	M11
Fanuc Parts Count Up	M12
Chuck Pressure High (Option)	M14
Chuck Pressure Low (Option)	M15
Turret Random	M16 (Default)
Turret Forward	M17
Turret Reverse	M18
Bar Feeder Call	M20
End of Bar Call	M33
Chip Conveyor On	M21
Chip Conveyor Off	M23
Open Auto Door	M24
Close Auto Door	M25
End of Program Rewind to Top	M30
Pin Drag Along Tail Stock(Manual Movement)	M37
Pin Drag Along Tail Stock(Programed Movement)	M60 V-(Machine Coordinate)
Parts Catcher Advance	M82
Parts Catcher Retract	M83
Sub-Program Call	M98
Sub-Program From Memory Card Call	M198
End of Sub-Routine /End of Program	M99
Chamfering On	M90
Chamfering Off	M91

* Custom Macro program Not Supplied From Taiwan Takisawa

G Codes	
G0	Positioning Rapid
G1	Linear Interpolation (Cutting Feed)
G2	CW Circular Interpolation or Helical Interpolation
G3	CCW Circular Interpolation or Helical Interpolation
G4	Dwell with U/X
G7.1	Cylindrical Interpolation (Wrapping)
G10	Programmable Data Input
G11	Programmable Data Input Cancel
G12.1	Polar Interpolation On (Face Milling C with X Axis)
G13.1	Polar Interpolation OFF
G17	Xp Yp Plane Selection (Milling on Face with Arcs Not Used with G12.1 or Canned Cycles)
G18	Zp Xp Plane Selection (Default Modal Turning)
G19	Yp Zp Plane Selection (Milling on Side with Arcs Not Used with G7.1 or Canned Cycles)
G28	Return to Reference Use Incremental Axis Variable for Axis X,Y,Z,C = U,V,W,H Exception is A and B Axis. There is not an Incremental Axis Variable for A or B
G30	2nd Reference Use Incremental Variable for Axis
G40	Tool Nose Radius Compensation -Cancel
G41	Tool Nose Radius Compensation -Left
G42	Tool Nose Radius Compensation -Right
G50	Max Spindle Speed Clamp G50S…….
G53	Machine Coordinate System Setting
G54-59	Work piece Coordinate System(Default Modal G54)
G65	Macro Call with Variable Pass
G68.1	3D Coordinate Rotation (Used for Angled Milling /Drilling operations, Fanuc Option)
G69	Coordinate Rotation Cancel

G Codes	
G70	Finishing Cycle
G71	Stock Removal Turning
G72	Stock Removal Facing
G76	Multi-Thread Cutting Cycle 2 Line Format (55° Point Input will Compound Infeed)
G80	Canned Cycle Cancel
G83	Face Drilling (No need for Plain Change)
G84	Face Tapping (No need for Plain Change)
G85	Face Boring (No need for Plain Change)
G87	Side Drilling (No need for Plain Change) X Radial Input
G88	Side Tapping (No need for Plain Change) X Radial Input
G89	Side Boring (No need for Plain Change) X Radial Input
G92	Threading Cycle
G96	Constant Surface Speed Control
G97	Constant Surface Speed Control Cancel
G98	Feed Per Minute
G99	Feed Per Revolution
G130	2nd Reference Set*

* Custom Macro program Not Supplied From Takisawa Taiwan

Spindle Selection Formatting Example

Main Spindle Mode

N100
G50 S2500(DESIRED MAX SPINDLE SPEED)
G0G28 U0
G28 W0
G80G40G13.1G99
M75
T0101
M1
G54 X….Z…. G97/G96 S…..M3/M4
M7/M8

Milling Spindle Mode

N100
G0G28 U0
G28 W0
G80G40G13.1
M76
G28H0
T0101
M1
G54 G99/G98 X….Z….C…. G97 S…..M3/M4
M7/M8

Main Spindle Grinding Mode

N100
G0G28 U0
G28 W0
G80G40G13.1
M75
T0101
M1
G54 G99/G98 X….Z….
M77 S…..M3/M4 (Milling Spindle Speed)
G4 U1. (1 Second Dwell)
S……M3/M4 (Main Spindle Speed)
M7/M8

Tool Call

T0101

Geometry/Wear Offset

Tool Number (Turret Position)

A Tool (Turret position) Is Called By ***T*** Followed by Four Numbers.

The First Two Numbers Equal the Tool (Turret Position) Number.
Example: T**01**01= Tool(Turret position) #1

The Last Two Numbers Equal the Tool Geometry and Wear Offset.
Example T01**01** = Geometry/Wear offset #1

While it is Most Common to Use a Geometry/Wear Offset that Matches the Tool Number it is not Mandatory to use the same Geometry/Wear offset Number as the Tool Number .

It is Common Practice to use Geometry/Wear Offsets that do not match the Tool Number in Certain Circumstances.

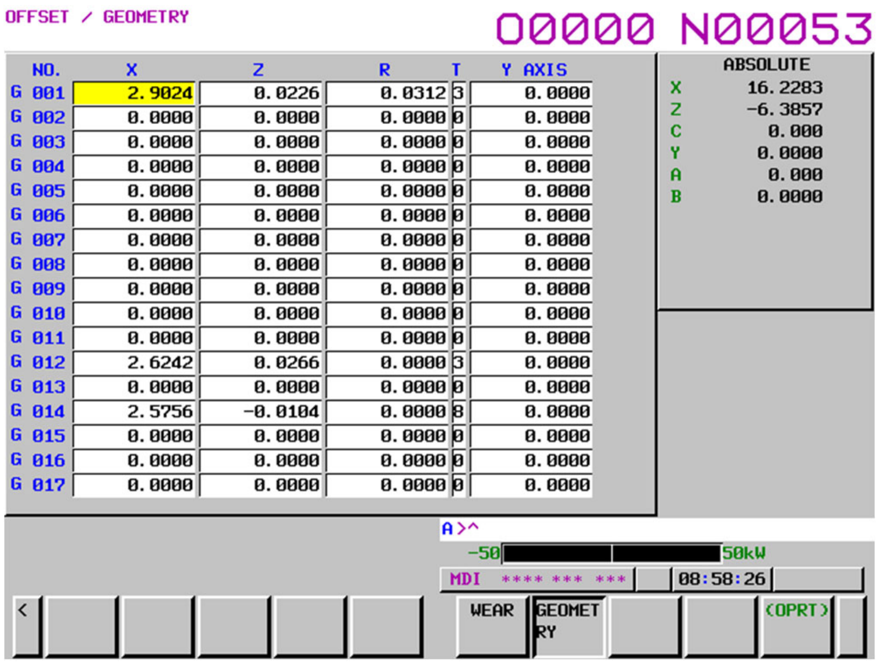
For Example one Instance would be using Quick Change Tooling.

Example:
T0323 = Tool #3, Geometry/ Wear offset #23

TOYODA

TAKISAWA
TAIWAN
Taiwan TAKISAWA Technology Co., Ltd.

Tool Geometry



OFFSET / GEOMETRY O0000 N00053

NO.	X	Z	R	T	Y AXIS
G 001	2.9824	0.0226	0.0312	3	0.0000
G 002	0.0000	0.0000	0.0000	0	0.0000
G 003	0.0000	0.0000	0.0000	0	0.0000
G 004	0.0000	0.0000	0.0000	0	0.0000
G 005	0.0000	0.0000	0.0000	0	0.0000
G 006	0.0000	0.0000	0.0000	0	0.0000
G 007	0.0000	0.0000	0.0000	0	0.0000
G 008	0.0000	0.0000	0.0000	0	0.0000
G 009	0.0000	0.0000	0.0000	0	0.0000
G 010	0.0000	0.0000	0.0000	0	0.0000
G 011	0.0000	0.0000	0.0000	0	0.0000
G 012	2.6242	0.0266	0.0000	3	0.0000
G 013	0.0000	0.0000	0.0000	0	0.0000
G 014	2.5756	-0.0104	0.0000	8	0.0000
G 015	0.0000	0.0000	0.0000	0	0.0000
G 016	0.0000	0.0000	0.0000	0	0.0000
G 017	0.0000	0.0000	0.0000	0	0.0000

ABSOLUTE

X	16.2283
Z	-6.3857
C	0.000
Y	0.0000
A	0.000
B	0.0000

A>^
-50 50kW
MDI **** *** *** 08:58:26
< WEAR GEOMETRY (OPRT)

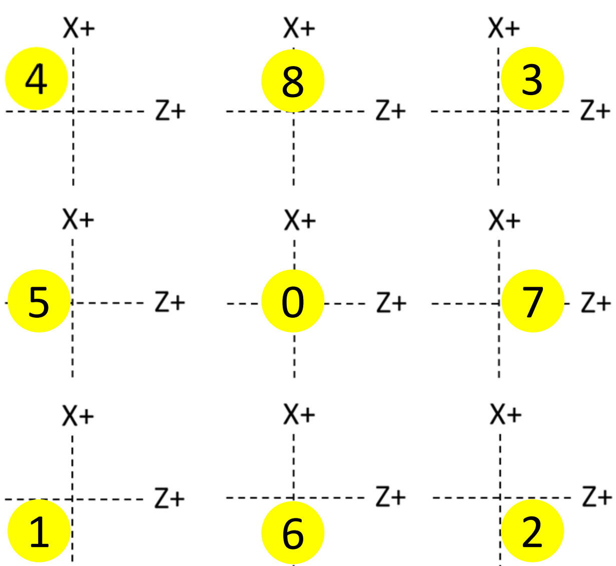
X Value is relative to spindle center Line and is Designated as a Diameter Value

Z Value is relative to Number 1 Tool if not using a Tool Setter
When using a Tool Setter Z is relative the surface the Tool Setter is calibrated to. Typically the face of the turret on 2 axis machines and center of the milling tool on machines with live tooling .

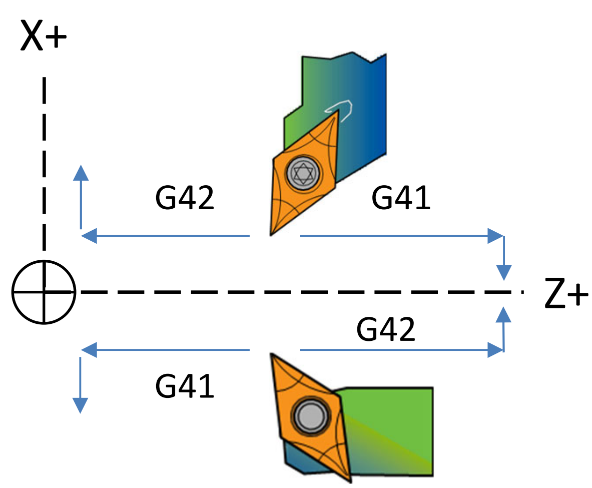
R is the cutter nose radius

T is the tool nose position

Tool Nose Position Table



Nose Radius Compensation By Cut Direction



Turning Spindle Speed Control G96/G97

G96, G97(Constant Surface Footage control ON, OFF)

G Code	Constant Surface Footage	Meaning	Unit
G 96	ON	To Control the Travelling Speed Constantly	SF/Min
G 97	OFF	Designate the rotating time of main spindle	RPM

Example:

G96 S100

Cutting Speed is 100 Surface Feet Per-Minute Based at the Current Cutting Diameter or Current X Machine Position.

G97 S100

Cutting Speed is a Fixed RPM Set at 100 Regardless of the Current Cutting Diameter or Current X Machine Position.

Feedrate Selection G98/G99

G GODE	Meaing	Unit
G 98	Feedrate Per-Minute	Inch/min
G 99	Feedrate Per-Revolution	Inch /rev

Example:

G99 G1 Z-5. F.005

The Cutter Moves Along the Z axis at .005 Per-Revolution of the Spindle

G98 G1 Z-5. F1.5

The Cutter Moves Along the Z axis at 1.5 Inches Per Minute

It is Most Common to Program Turning Centers in G99 Mode. Even when Utilizing Live Tooling for Milling and Drilling.

Most common uses of G98 In a turning center are:

Moving An Axis Without the Spindle Rotating under G1
Scrape Broaching
Bar Pulling
Live Tooling Milling and Drilling

Work Coordinate Setting Procedure

Setting Work Coordinate Offset

Before Setting ***Work Coordinate Offset*** Make Sure X and Z ***Work Shift*** are Set to 0
If there is ***No Tool Setter, T 0101 or the Master Tool's Z Wear and Geometry Offset should be set to 0***

1. In MDI
G54; (G54-59)
T0101;
G97S500M3;
Insert
Cycle Start
Do Not Hit Reset. If Reset is Pressed Redo From 1.

2. Switch into Manual Mode, Using the Manual Pulse Wheel, face .010-.020" Off the Face of the Stock or Touch Off with Feeler Gauge **Leave at that Z Position**

3. Go to Work Coordinate Offset Page and Cursor to **Z** Register for Offset **54-59(Whichever is to be Set)**

4. Type **Z0**

5. On the Soft Keys on the Bottom of the Monitor Press **Measure**
(Z coordinate should be saved in the Z register for Work offset 54)

6*.* Manually turn Spindle Off from Control Panel

Setting Tools Without Tool Setter

-Setting the Work Coordinate must be done first
-With the Turret in a Safe Position Call Up Desired Tool that Needs to be Set (Tool 1's Z Offset Remains 0)
In MDI:
T0101;
G54 S500M3;
Insert
Cycle Start

Calibrating X Geometry

-Manually Skim Cut O.D of Part Till Clean Up

-Back Off Turret in Z only, Leave X in the Position the Cut was Taken

-Measure the Cut O.D and Record Size(For example 1.100")

-Press the Offset Soft Key

-On the Soft Keys Bellow the Monitor Press the **Geometry** Key

-Highlight the X Data Box for Tool 1

-Type in Recorded Measurement as Follow's
X1.100

-Press the **Measure** Key On the Soft Keys Bellow the Monitor
Tool 1 is Now Set

Calibrate The rest of the tools on X and Z as follows

-Using a Feeler Gauge of .001 or a Piece of Paper Manually Touch Face Of Tool on the Face of the Part

-Leave Tool in that Z Position

-Highlight the Z Data Box for Tool

-Type in Z0

-Press the **Measure** Key On the Soft Keys Bellow the Monitor

-Set X of Tools as Described Above Under **Calibrating X Geometry**
For Center Cutting tools
Center Cutting Tooling like Drills, Taps, Reamers etc. Can be Set to 0 (Zero) If it is Found Necessary, Indicate Tool

-Type **X0**

-Press the **Measure** Key On the soft keys bellow the monitor

M43 Chatter Turn (Chatter Reducer)

M43 mode can be used to help reduce chatter when turning or boring.
This mode pulses the spindle based on 2 variables. Percentage of speed and time interval.
M5 or Reset will Cancel M43 Mode.

Limitations

Maximum initial Spindle speed 1000 RPM

Requires

Macro Program O9029
Parameter 6089=43

M43 V_F_;

V:Percentage of the Spindle Speed Range 10~50 %.

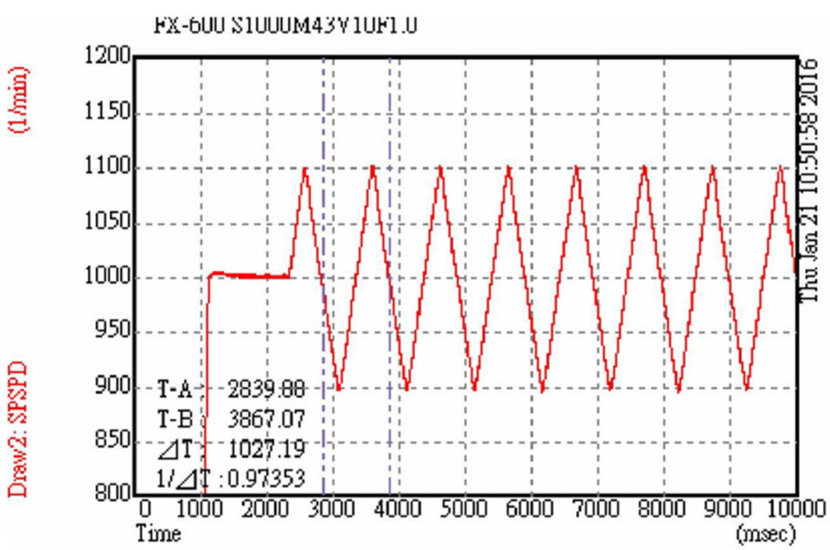
If it exceeds the range .it will alarm the "3007 IMPROPER V VALUE".

F: Time interval for Pulse. Range 0.2~2.0s(unit 0.1 second).

If it exceeds the range. it will alarm the "3008 IMPROPER F VALUE".

Example:

M3 S1000.
M43 V10. F1.0 (or You can input before M3)
When spindle speed is at arrival.
Spindle Speed pulses ± 10% (900~1100rpm)



Canned Drilling, Tapping and Boring Cycles

G code	Drilling axis	Inward motion	Action at the bottom of the hole	Outward motion	Applications
G80					Cancels drilling canned cycle
G83	Z-axis	Intermittent or continuous feed	Dwell	Rapid traverse	Front drilling
G84	Z-axis	Continuous feed	Dwell, followed by spindle CCW	Continuous feed	Front tapping
G85	Z-axis	Continuous feed	Dwell	Continuous feed (2x)	Front boring
G87	X-axis	Intermittent or continuous feed	Dwell	Rapid traverse	Side drilling
G88	X-axis	Continuous feed	Dwell, followed by spindle CCW	Continuous feed	Side tapping
G89	X-axis	Continuous feed	Dwell	Continuous feed (2x)	Side boring

Note: When using Canned Drilling and Tapping Cycles. Program C/A Axis brake ON and OFF in the cycle.

When braking is controlled by the canned Cycle there can be in-position errors that occur or out of tolerance hole positions.

See example 1:1

Face and Side Drilling Cycles

G83 X(U) C(H) Z(W) R P Q F K M

or

G87 Z(W) C(H) X(U) R P Q F K M

X,C or Z,C -Hole position data
Z ,X - The distance from point R to bottom of the Hole
R- The distance from initial level to point R
P-Dwell time at bottom of hole Example: P0100 =100 milliseconds (No decimal input)
Q- Depth of cut for each peck Example: Q1000 =.100 (No decimal input)
Note: Q must be present on all positioning lines if peck is desired for every position
K-Number of Repeats
M-Brake on if needed
Note: Retraction distance set in Parameter 5115

Example 1:1

G83 X…. Z…. C/A……Q…..R….P….K….F….M50(BRAKE ON)
M51(BRAKE OFF)
C….Q…. M50(BRAKE ON)
M51(BRAKE OFF)
C…. Q…. M50(BRAKE ON)
M51(BRAKE OFF)
C…. Q…. M50(BRAKE ON)
M51(BRAKE OFF)
G80

Face and Side Tapping Cycles

G84 X(U) C(H) Z(W) R P Q F K M

or

G88 Z(W) C(H) X(U) R P Q F K M

X,C or Z,C -Hole position data
Z ,X - The distance from point R to bottom of the Hole
R- The distance from initial level to point R
P-Dwell time at bottom of hole Example: P0100 =100 milliseconds (No decimal input)
Q- Depth of cut for each peck Example: Q1000 =.100 (No decimal input)
K-Number of Repeats
M-Brake on if needed
Note: For Peck Tapping to be activated the following parameters need to be set.
5104 bit 6 =1
5200 bit 5 =1
High speed Peck Tapping
5104 bit 6 =1
5200 bit 5 =0

Example

M29 S250 (Rigid Tapping On/ Spindle Speed 250)
G84 X…. Z…. C……Q…..F….M50(BRAKE ON)
M51(BRAKE OFF)
C…. M50(BRAKE ON)
M51(BRAKE OFF)
C…. M50(BRAKE ON)
M51(BRAKE OFF)
C…. M50(BRAKE ON)
M51(BRAKE OFF)
G80

Reverse Rigid Tapping

G97 S250 (Omit G97 in Milling Mode)
M29 (Rigid Tapping On)
M28 (Reverse Rigid Tapping On)
G84 X…. Z…. C……Q…..F….M50(BRAKE ON)
M51(BRAKE OFF)
C…. M50(BRAKE ON)
M51(BRAKE OFF)
C…. M50(BRAKE ON)
M51(BRAKE OFF)
C…. M50(BRAKE ON)
M51(BRAKE OFF)
G80

Face and Side Boring Cycles

G85 X(U) C(H) Z(W) R P F K M

or

G89 Z(W) C(H) X(U) R P F K M

X,C or Z,C -Hole position data
Z ,X - The distance from point R to bottom of the Hole
R- The distance from initial level to point R
P-Dwell time at bottom of hole Example: P0100 =100 milliseconds (No decimal input)
K-Number of Repeats
M-Brake on if needed

Example

G85 X…. Z…. C……R….P….F….M31(BRAKE ON)
M32(BRAKE OFF)
C…. M31(BRAKE ON)
M32(BRAKE OFF)
C…. M31(BRAKE ON)
M51(BRAKE OFF)
C…. M32(BRAKE ON)
M31(BRAKE OFF)
G80

G76 MULTI LINE THREADING CYCLE

T0202(5/8-11 PITCH O.D THREAD)
M1
G0 G54 X.655 Z.100 G97S850M3
M8
G76 P020055 Q0025 R0050
G76 X.522 Z-.950R0 P0515 Q0050 F.0909

FIRST LINE

G76
P=(m)(r)(a)
m(Amount of finish passes 1-99),
r(Chamfer out amount, number thread leads 0-99, 10 would equal 1 lead)
a(Angle of tool Nose Typical 60°,55°)
Note: When Tool Nose Angle is set to 55° the Cycle will run a Compound Infeed

Q :Minimum Cutting Depth 4 Places No Decimal Example: Q0025 *=.0025*

R: Finish Allowance 4 Places No Decimal Example: Q0050 *=.0050*

SECOND LINE

G76
X: Minor Diameter of Thread

Z: Length of thread

R: Taper Amount +/- Radial difference from Major to Minor Diameter From Start Position .
If 0 or Omitted, Straight thread will be cut.
Negative Value for O.D
Positive for I.D

P: Height of Thread Radial difference Between Major and Minor Diameter.
Major ø .625 minus Minor ø .522 /2= P Amount
Note: The Radial Difference from the Minor Diameter to X Clearance/Start Position Must be more than the Height Of Thread .
With Tapered threads the P value is calculated the same.

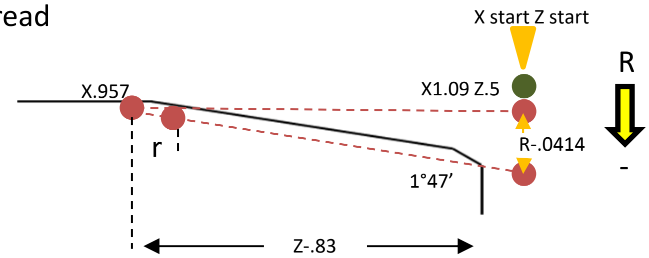
Q: Depth of First Cut 4 Places No Decimal Example: Q0050 *=.0050*

F: Lead of thread in Decimal Up to 4 places
Standard Thread Pitch= 1/Pitch
1/11=.0909
Metric Pitch x .03937
1.5mm x .03937=0.0590

G76 MULTI LINE THREADING CYCLE

Tapered Threading Example

O.D Thread

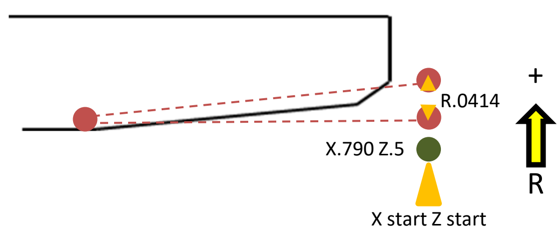


G0X1.09Z.5
G76 P021060 Q0020 R0020
P(m)(r)(a)
G76 X.957 Z-.83 R-.0414 P0570 Q0160 F.0714

Equation to find R Value
1°47” Angle for NPT
Z Initial Position + Z Depth of Cut
Multiplied by Tangent of 1°47”

*.83+.500*Tan 1°47’=R*
Note: Angle Referenced from the Machinist Hand Book

I.D Thread



G0X.790Z.5
G76 P021060 Q0020 R0020
P(m)(r)(a)
G76 X.910 Z-.83 R.0414 P0570 Q0160 F.0714

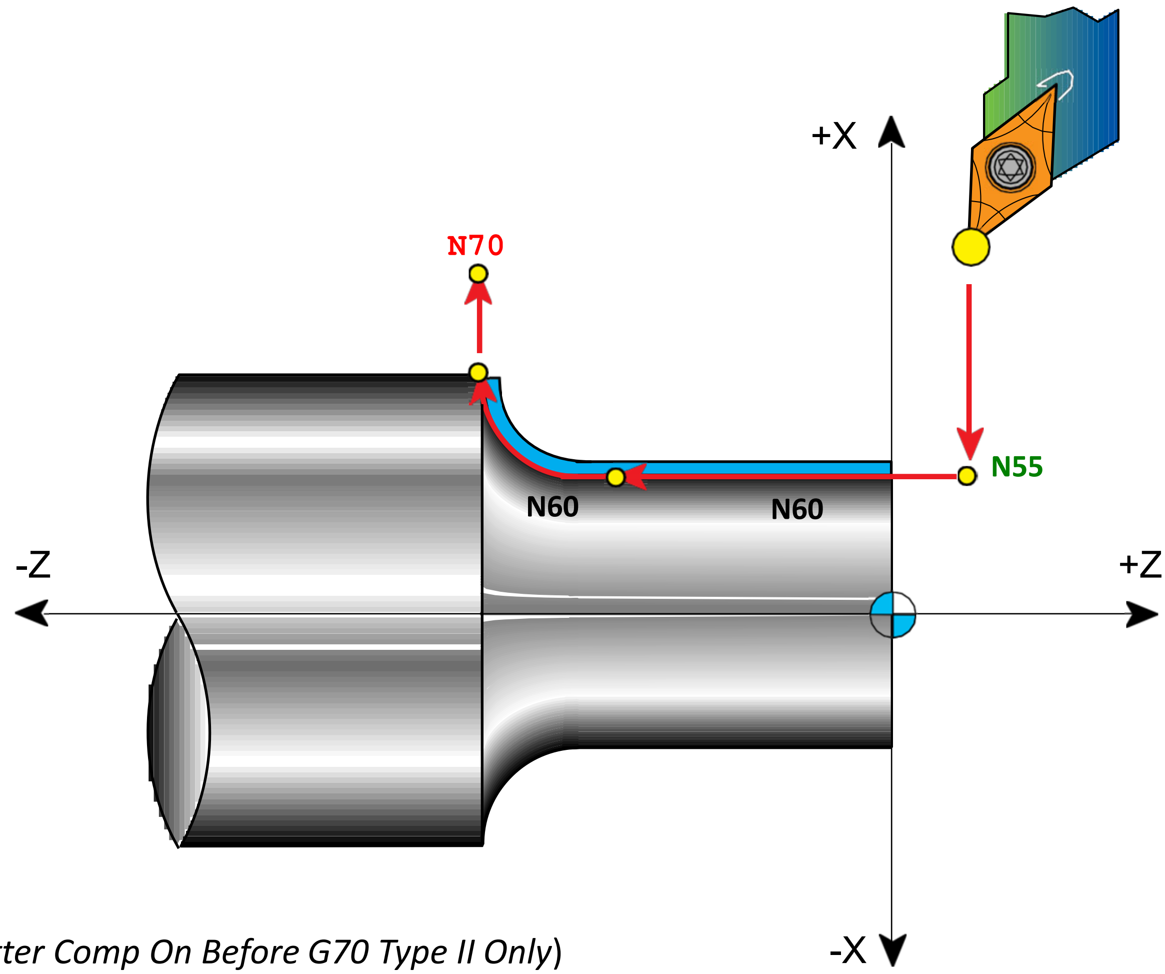
Multi-Lead Programing Example

Main Program
G00 X48   Z5
M98 P1000 L3 (P= Sub routine Number L repeat Times)

Sub-Program
O1000
G76 P020000 Q100 R0.05
G76 X42 Z-15 P0974 Q0200 F4.5(F= Pitch * Repeat times)
G00 W1.5
M99

G70 Finishing Cycle Type II

Note: *Please Refer to Fanuc Lathe Operations Manual for Detailed Specifications, Limitations and Parameters.*



G42 (*Cutter Comp On Before G70 Type II Only*)

G70 P **55** Q **70**
(P= Start Line Number)
(Q= End Line Number)

N55 G0 X.. (Z Position Or Wø Must be on this Line if Contour has Pockets)

N60 G1 Z-..

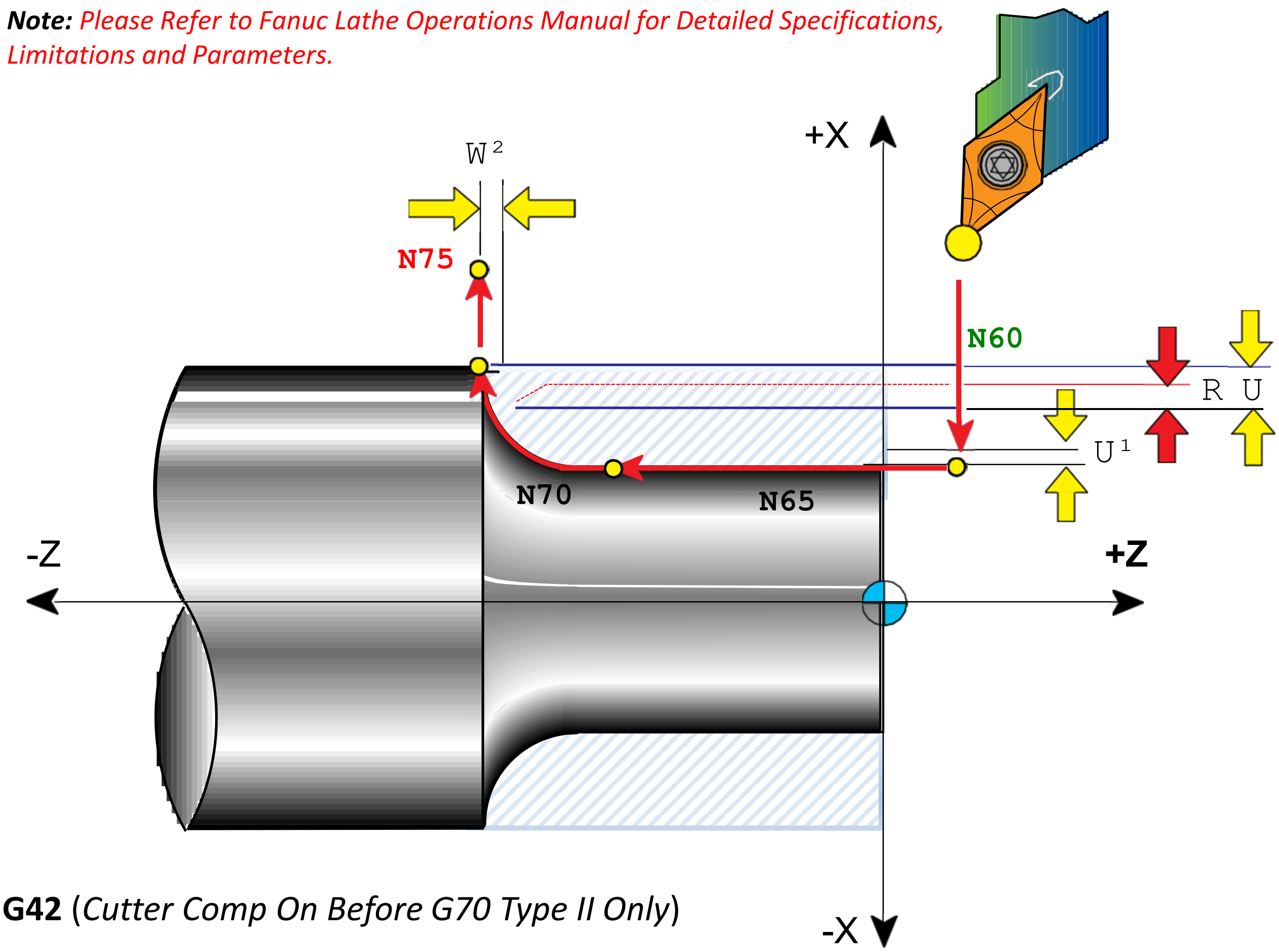
N65 G2 X.. Z-.. R..

N70 G1 X..

G40 (*Cutter Comp Off After G70 Type II Only*)

G71 Repetitive Turning Cycle Type II

Note: *Please Refer to Fanuc Lathe Operations Manual for Detailed Specifications, Limitations and Parameters.*



G42 (*Cutter Comp On Before G70 Type II Only*)

G71 U… R…

(U= *Cut Volume One Time. Radial Distance*)

(R= Escape distance, Always at 45°)

G71 P **60** Q **75** U¹…. W²… F…

(P= Start Line Number)

(Q= End Line Number)

(U¹= X Finish Allowance)

(W²= Z Finish Allowance)

(F = Roughing Feed Rate)

N60 G0 X.. (Z Position Or Wø Must be on this Line if Contour has Pockets)

N65 G1 Z-.. F..(*Finishing feed rate for G70 Cycle*)

N70 G2 X.. Z-.. R..

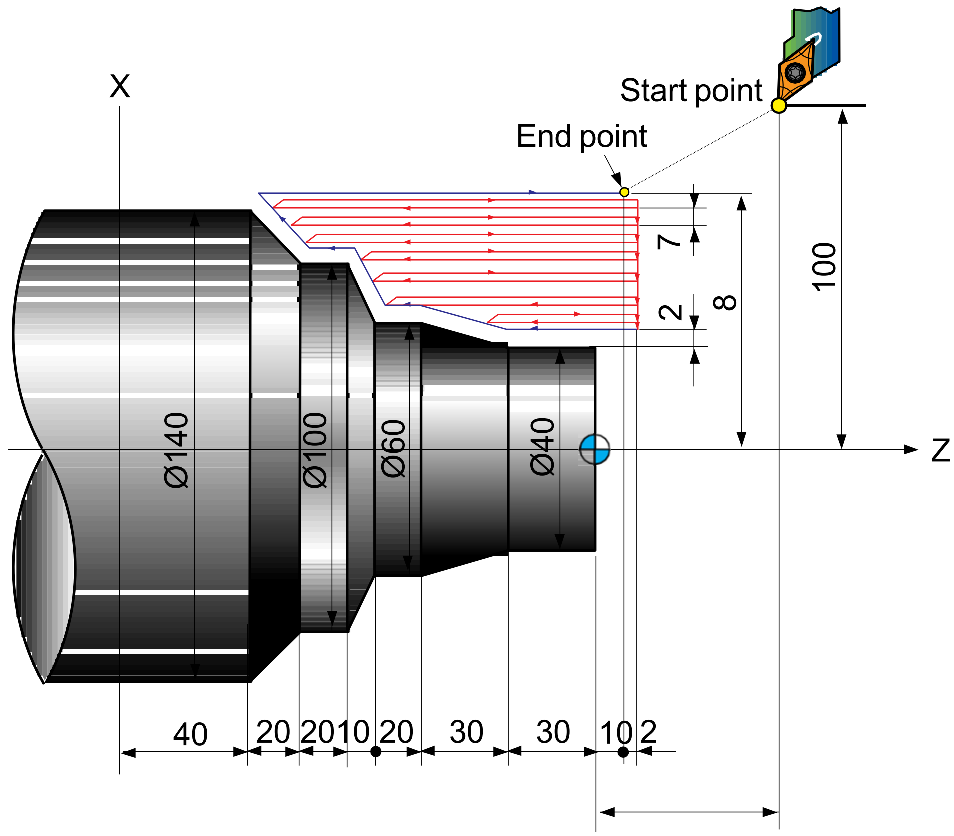
N75 G1 X..

G40 (*Cutter Comp Off After G70 Type II Only*)

Example Program:

Stock Removal in Turning G71 and Finishing G70 (Type II)

Note: *Please Refer to Fanuc Lathe Operations Manual for Detailed Specifications, Limitations and Parameters.*



T0101 (ROUGHING TOOL)
G0 G54 X200. Z100. G96 S325 M3 M8
X160. Z10.
_G42_
G71 U7. R1.
G71 P10 Q20 U4. W2. F.3
N10 G0 X40.
G01 W-40. F.15
X60. W-30.
W-20.
X140. W-20.
N20 U2.
_G40_ M9
G30 W0 U0

T0202 (FINISHING TOOL)
G0 G54 X200. Z100. G96 S525 M3 M8
X160. Z10.
_G42_
G70 P10 Q20
_G40_ M9
G30 W0 U0

G72 Stock Removal In Facing (Type II)

Note: *Please Refer to Fanuc Lathe Operations Manual for Detailed Specifications, Limitations and Parameters.*

G72 W($\underline{\Delta d}$) R(e)

W($\underline{\Delta d}$) = Cut volume of one time

R(e) = Escape volume

G72 P_Q_U(Δu) W(Δw) F....

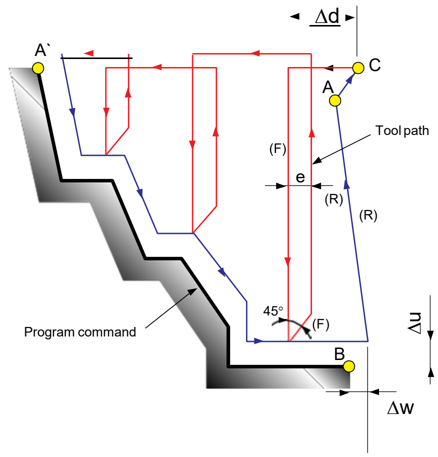
P = Start Sequence No.

Q = End Sequence No.

U(Δu) = Finishing in clearance X axis (Radial)

W(Δw) = Finishing in clearance Z axis

F = Cutting Feedrate

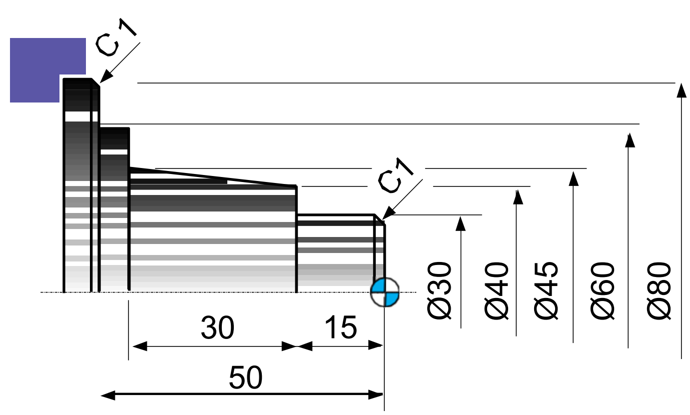


Note: (X Position Or Uø Must be on first Line of pattern, If feature has Concave feature)

Example Program:

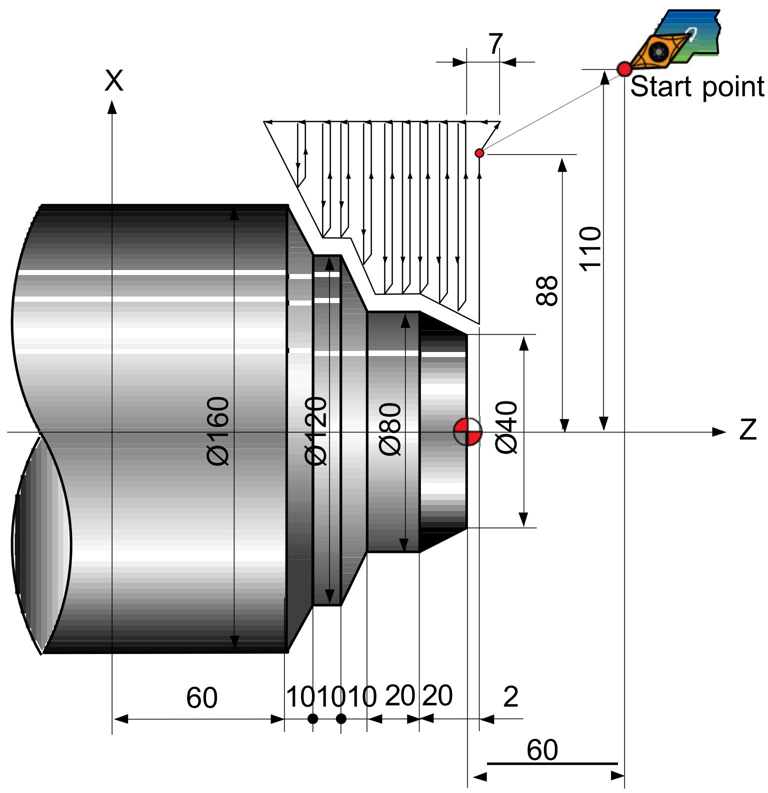
```
G50 S1200
T0101(ROUGH)
G0 G54 X85.  Z0 G96 S325 M3 M8
G1 X-1.6 F.02
G0 X85. Z1.0
G41
G72 W2.0 R1.0
G72 P12 Q14 U0.5 W0.2F0.25
N12 G0 Z-51.0 U0
G1 X80. F.02
X78. W1.
X60.
Z-45.
X40. Z-15.
X30.
Z-1.
N14 X26.0 Z1.0
G40 M9
G30 U0 W0
```

```
T0202(FINISH
G0 G54 X85.  Z0 G96 S325 M3 M8
G1 X-1.6 F.02
G0 X85. Z1.0
G41
G70 P12 Q14
G40 M9
```



G72 Stock Removal In Facing (Type II)

Note: *Please Refer to Fanuc Lathe Operations Manual for Detailed Specifications, Limitations and Parameters.*



(Diameter designation, metric input)

G00 X220.0 Z60.0

X176.0 Z2.0

G41

G72 W7.0 R1.0

G72 P14 Q21 U4.0 W2.0 F0.3 S550

N14 G00 Z-70.0 S700

X160.0

G01 X120.0 Z-60.0 F0.15

W10.0

X80.0 W10.0

W20.0

N21 X36.0 W22.0

G40

G41

N022 G70 P14 Q21

G40

G00 X220.0 Z60.0

M30

G73 Pattern Repeating (Type II)

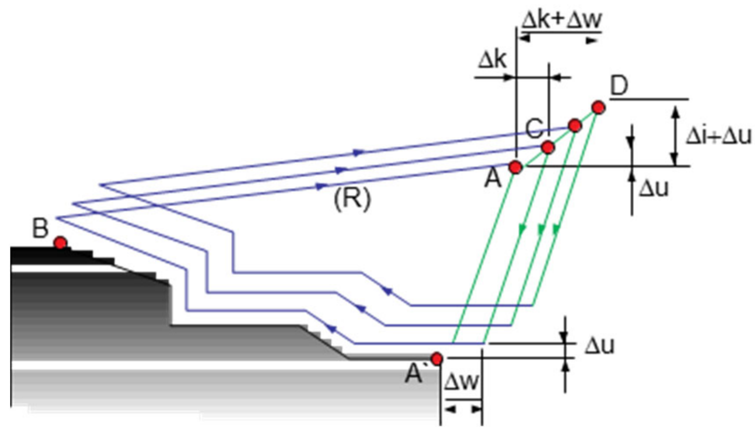
Note: *Please Refer to Fanuc Lathe Operations Manual for Detailed Specifications, Limitations and Parameters.*

G73 U(Δ i) R(d) W(Δ k)

U(Δ i) : Escape Distance and Direction in X axis (Designated By radius)

W(Δ k) : Escape Distance and Direction in Z axis

R(d) : Repeating Times (Relative to the cut volume of each time)



G73 P Q U(Δ u) W(Δ w) F

P : Start sequence No.

Q : Final sequence No.

U(Δ u) : Finishing in clearance X axis(Radius designated)

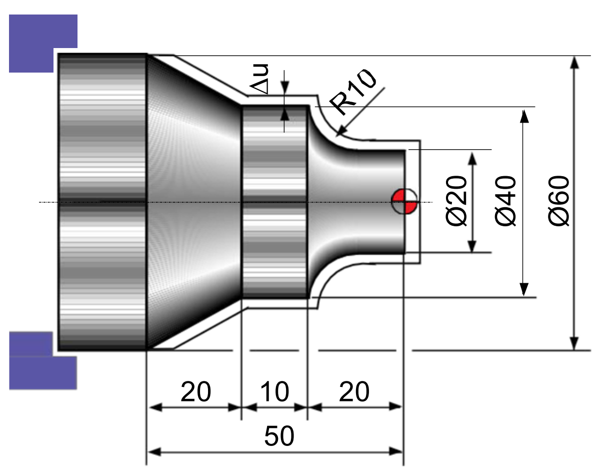
W(Δ w) : Finishing in clearance Z axis

F(f) : Cutting Feedrate

Example Program:

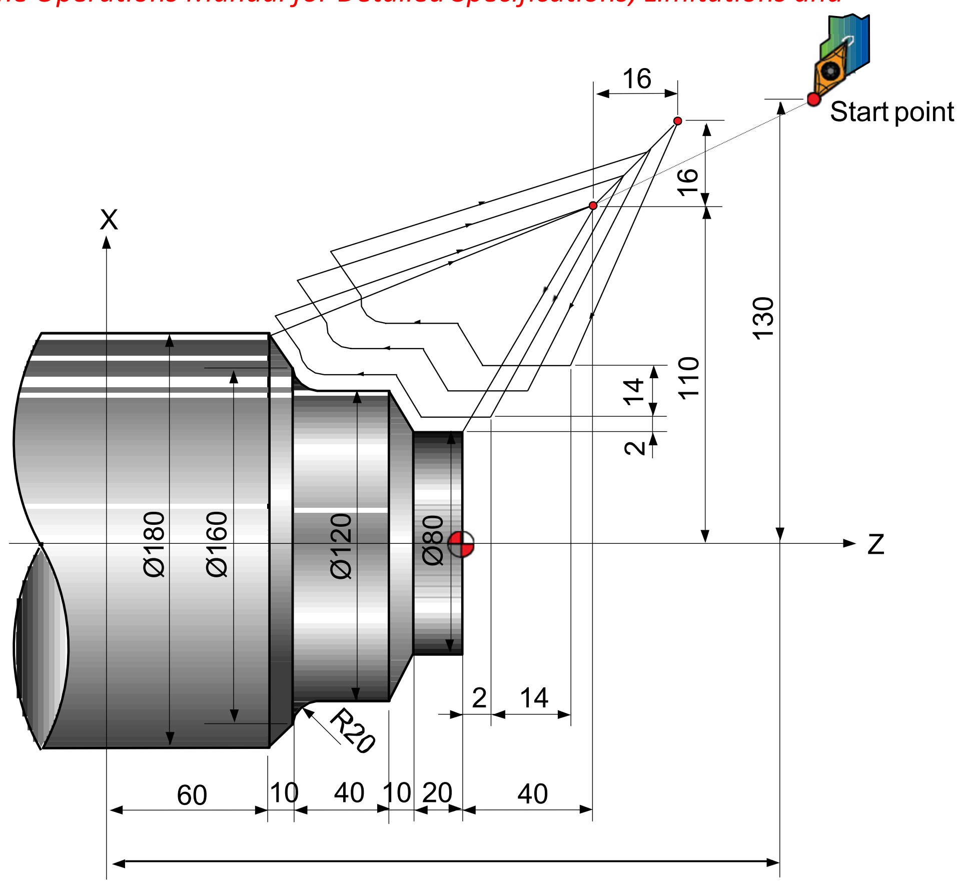
```
G50 S2000
T0303
G0 G54 X35.0 Z5.0 G96 S200 M03
 Z0
G01 X-1.6 F0.2
G0 X70.0 Z10.0
G42
G73 U3.0 W2.0 R2
G73 P12 Q16 U0.5 W0.1 F0.25
N12 G00 G42 X20.0 Z2.0
G1 Z-10.0 F0.15
G2 X40.0 Z-20.0 R10.0
G1 Z-30.0
X60.0 Z-50.0
N16 U1.0
G40
```

```
G42
G70 P12 Q16
G40
G30 W0 U0
M30 :
```



G73 Pattern Repeating (Type II)

Note: *Please Refer to Fanuc Lathe Operations Manual for Detailed Specifications, Limitations and Parameters.*



(Diameter designation, metric input)

G50S1200

T0101

G0G54 X220.0 Z40.0 G96S180 M3 M8

G42

G73 U14.0 W14.0 R3

G73 P14 Q20 U4.0 W2.0 F0.3

N14 G00 X80.0 Z2.0

G01 W-20.0 F0.15 S0600

X120.0 W-10.0

W-20.0

G02 X160.0 W-20.0 R20.0

N20 G01 X180.0 W-10.0

G40

G42

G70 P14 Q20

G40

X260.0 Z80.0

M30

Direct Drawing Dimensional Programming

Note: *Please Refer to Fanuc Lathe Operations Manual for Detailed Specifications, Limitations and Parameters.*

Direct Drawing is a programing function intended to simplify manually programing profiles with Angles, Corner Radiuses and 45° chamfers.

Format as Follows:
,C= 45° Chamfer

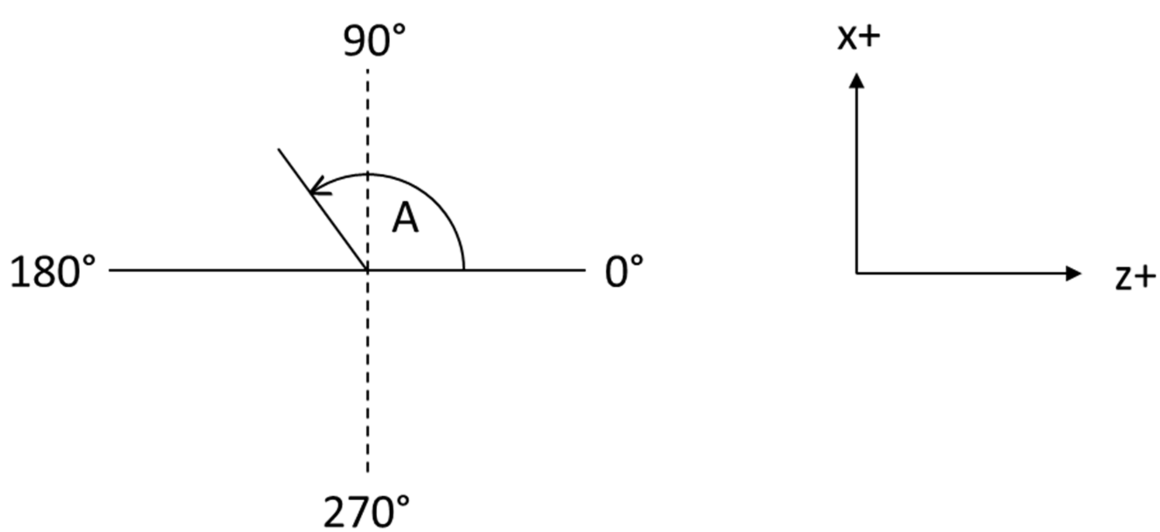
,R= Radius

,A=Angle

Note:
Parameter 3405 Bit 4 must equal 0 to use:
,C
,R
,A

When parameter 3405 bit 4 equals 1
C and R are used without Comas
A is not valid when 3405.4=1. Angles must be programed point to point between X and Z axis.

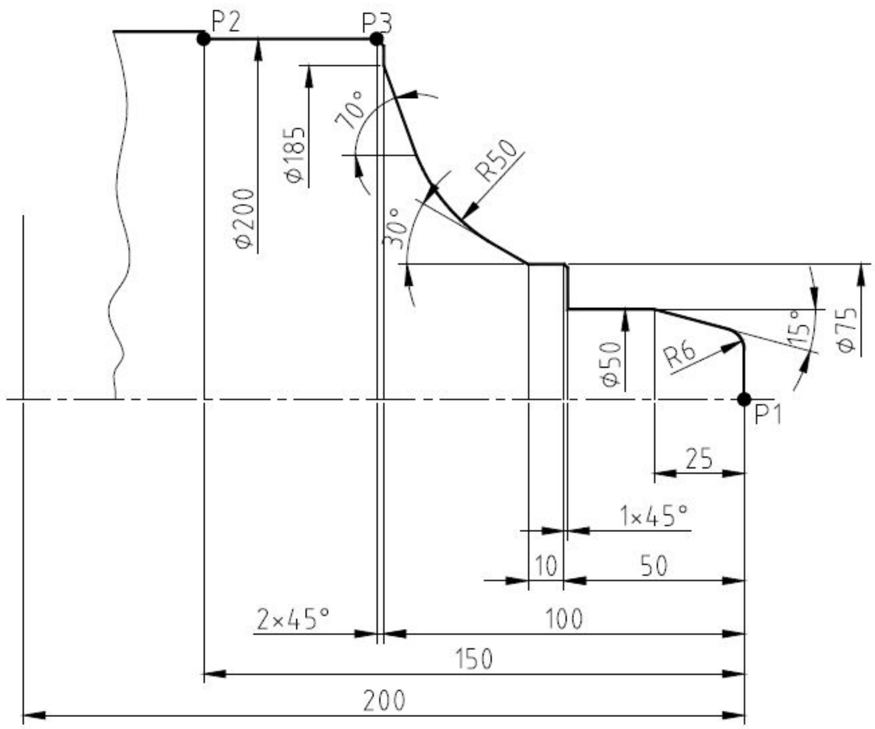
In Multiple repetitive canned cycle, in blocks with sequence numbers between those specified at P and Q, a program using direct drawing dimension programing can be used . The Block with the last sequence number specified at Q must not be an intermediate block of these specified blocks.



Direct Drawing Dimensional Programming

Note: *Please Refer to Fanuc Lathe Operations Manual for Detailed Specifications, Limitations and Parameters.*

Program Example




```
G50 S2200;
T0101 (PROFILING);
G0 G54 G96 S230 M3M8;
G0 X0 Z3;
G1 Z0 F0.12; (P1)
,A90 R6; (FIRST ANGLE)
X50 Z-25 ,A165; (SECOND ANGLE A165 comes from 180°-15°=165°)
,A180
X75 Z-49 ,A90 ,C1;
Z-60;
,A150 R50; (A150 comes from 180°-30°=150°)
X185 Z-100 ,A110;(A110 comes from 180°-70°=110°)
,A90 ,C2;
X200 Z-150 ,A180; (P2)
G0 X250 Z200;
M30;
```

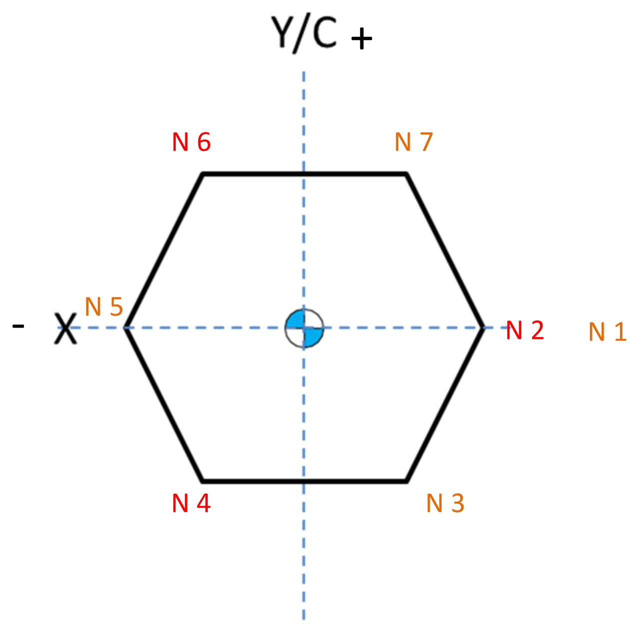
Polar Coordinate Milling G12.1

Polar coordinate milling is an effective way to mill profiles or pockets in the Y/X plane using X and C Axis
This function simplifies the programing using X and C, it allows feed rates to be programed in Inch per Minute and Inch per Rev and the use of linear coordinates.
While at first look the programing may seem difficult it is rather simple.

1. Lay out your required Feature in 2D as if it where X an Y on a Mill.
2. Double your X values to make them Diameters
3. Change your Y to C

Note If using the sub-spindle on a single path machine X and A Axis are the Axis used.

```
<POLARTEST>
(1.125 HEX)
(CUTTER RADIUS IN COMP, T=0)
G40G80G13.1
M101Z-10.*
M76
T1111(.500 MILL)
G0G55X2.6250Z.050S1000M3
N1G1G99 X1.7250 C0
G12.1
N2 G1G42 X1.2992 C0 F.005
N3 X0.6496 C-0.5625
N4 X-0.6496 C-0.5625
N5 X-1.2992 C0
N6 X-0.6496 C0.5625
N7 X0.6496 C0.5625
N2 X1.2992 C0
G40
X1.9992 F0.0300
G13.1
G0X2.7992
M201Z-10.
```



* Custom Macro program Not Supplied From Takisawa Taiwan

TOYODA

TAKISAWA TAIWAN Taiwan TAKISAWA Technology Co., Ltd.

Cylindrical Interpolation G07.1

In the Cylindrical interpolation function, the amount of movement of a rotary axis is specified by angle is converted to the amount of movement on the circumference to allow linear and circular interpolation with another axis.

Since programming is enabled with the cylinder side face expanded, programs such as a program for a cam slot can be programmed easily.
The stock G07.1 function requires programing using Rotary Angle Designation(programed in degrees).

When a feature is laid out in the 2D X and Y grid. The Y Axis becomes the Z Axis Movement and the X axis becomes the Rotary Axis movement . When manually programing some calculations need to be performed to covert the X Axis linear movements to Rotary Axis Movements (Degrees) .
Radius moves should be programed with R.

For the Oi-TF control there is an option to be able to program by Cylindrical Plane distance (linear distance). Option #A02B-0339-R578
If interested in this option, please contact your Toyoda Dealer .
See next page for example from the Fanuc lathe Operations manual .

Format

G18 Z0 C0 or A0 (SET PLAIN)
G07.1 R (Radius of Work Piece, Can be Specified with Decimal point) ;
;
;
;
G07.1 C0 OR A0(Cancel Cylindrical Interpolation)

Formulas for Converting Linear Dimensions to Rotary Degrees

The following Formulas will Aid in Manual Programing or using 2D Mill Post to Generate Code

Diameter × π = L (Linear Length of Part O.D)

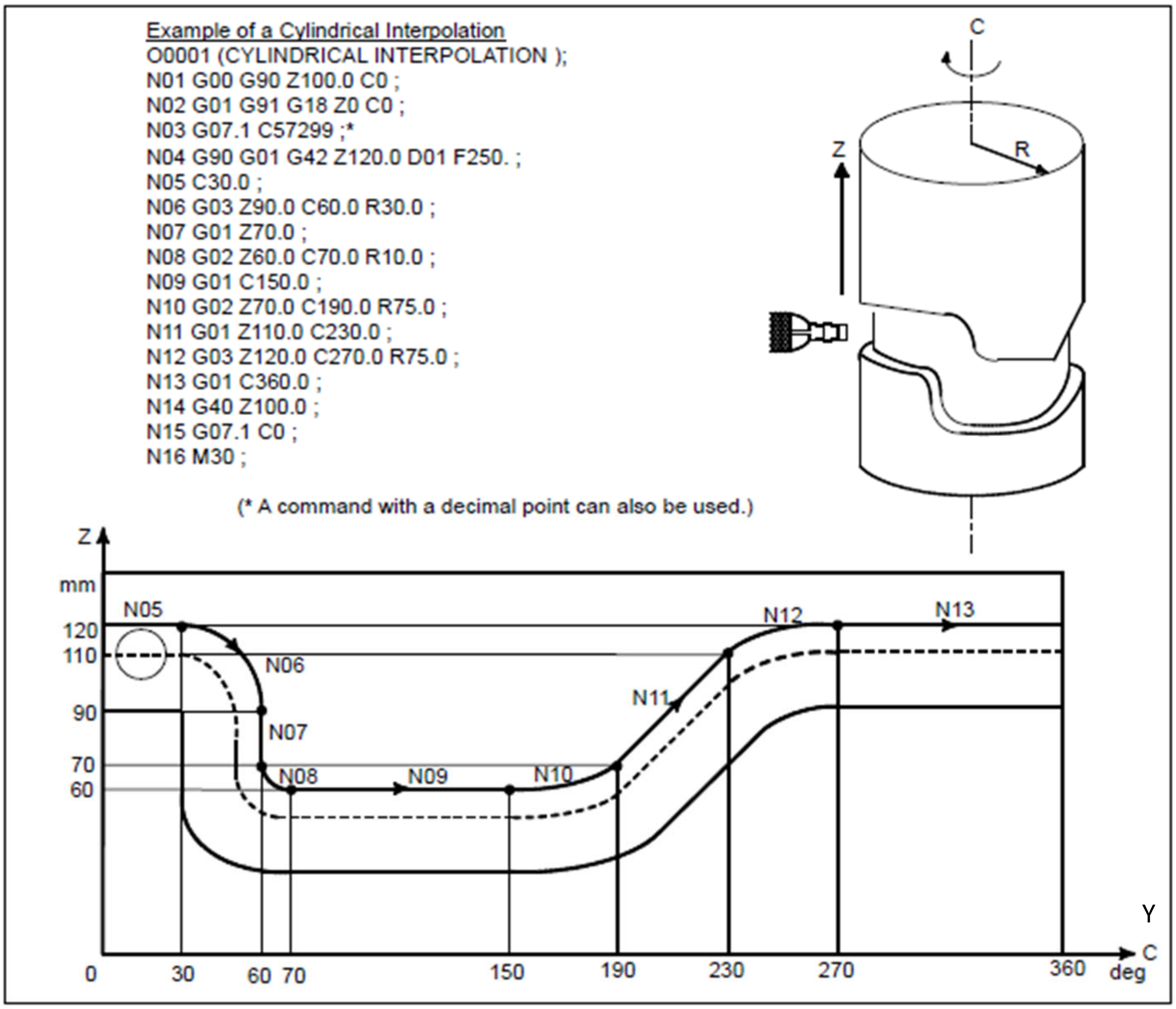
L / 360 = LD (linear Amount Per degree)

Linear Position / LD = C (Rotary Axis Degree Value at Specified Diameter)

Cylindrical Interpolation G07.1

Fanuc Manual Program Example

Note :This example was written as a generic sample.
G90,G91 and D (Tool Diameter Geometry) are only used in G code Systems B and C Which are typically associated with the Fanuc M series (Milling Control)



M98 LOOP EXAMPLE WITH SUB-ROUTINE AFTER M30 OF MAIN PROGRAM

Following this procedure will allow placing sub-routine programs under the M30. Using this technique will reduce additional program numbers in the Program Library associated with one job.
Examples of when to use, for simple loops or with nested parts. There are limitations such as you can not pass variables.
This is also a simple way to call a sub-routine by line number from a Difference Sub-Program.
In order to use this method of sub-routine parameter 6005.0 (SQC) needs to be set to 1.

In MDI Mode
From settings page turn on PWE (Parameter Write Enable)
PARAMETER 6005.0 (SQC) =1

Program Example:

```
07777(MAIN PROGRAM)
N101
T0101(TOOL)
M1
G0G99G54X45G97S5000M3
Z5.M8
M98P7777(MAIN PROGRAM #)Q1111(line Number) L3(REPEAT TIMES)
;
;
M30
N1111(SUB PROGRAM AFTER M30 OF MAIN PROGRAM)
G76 P020000 Q100 R0.05
G76 X42 Z-15 P974 Q200 F4.5
G00 W1.5
M99
```

Creating Custom G and M code Macro calls

From Setting s menu turn on parameter write

1.Selector switch or button needs to be in MDI

2.Press the offset Key and then the settings soft key

3. Highlight the parameter write enable

4. Type 1 , press input

5. Press the system key

6. Type 3202 and press search soft key

7. Move cursor to NE9 on parameter 3202 and change to 0

8.Assign Parameter that matches the Macro program number the M or G code Number you wish to use.

For example if you are writing a M code with no arguments to pass through. If your Code was M100 and your program number Is O9001 you would type 100 in Parameter 6071.

9. Highlight parameter 6071 and type 100 press input.

10. Create you program using O9001 as the program name.

Example program:

```
%
O9001(Default start M code)
G40 G80 G99
M99
%
```

Now instead of typing G40 G80 G99 at the top or at every safe start line you put *M100* and it preforms the G40 G80 G99 Functions .

11. Turn Parameter 3202 NE9 back to 0

12. Go to Setting page and put Parameter Write Back to 0

Note: If there are already program numbers or G/M codes in the control assigned or being used. Use an open program number/Parameter. Do not over write existing functions.

Fanuc Models: 16M, 18M, 16i, and 18i

Macro call using G Code

Program Number	Parameter Number
O9010	6050
O9011	6051
O9012	6052
O9013	6053
O9014	6054
O9015	6055
O9016	6056
O9017	6057
O9018	6058
O9019	6059

Macro call using M Code

Program Number	Parameter Number
O9020	6080
O9021	6081
O9022	6082
O9023	6083
O9024	6084
O9025	6085
O9026	6086
O9027	6087
O9028	6088
O9029	6089

Note:
Macro calls with G and M codes can pass arguments while SubProgram calls with M codes cannot.

SubProgram call using M Code

Program Number	Parameter Number
O9001	6071
O9002	6072
O9003	6073
O9004	6074
O9005	6075
O9006	6076
O9007	6077
O9008	6078
O9009	6079

Safe Tool Change Positions and Cancel Line

There are many ways to mange a cancelation line and safe tool change positioning. This section is going to cover simplifying the process to aid in saving time and the repetition of typing in unnecessary code by giving some Custom M code examples.

Cancel Line Macro

Set parameter 6076=100

```
O9006(M100 CLEAR LINE)
G80 G40 G20 G13.1 G99
M99
```

Safe Start Main Spindle

Set parameter 6020=101

```
O9020(M101 MAIN SPINDLE SAFE TOOL CHANGE)
(Z= MACHINE POSITION USER DEFINED ON M-CODE LINE )
(EXAMPLE M101 Z-17.)
IF[#26GE#0]THEN#3000=1(Z MISSING OR+)
G0
G28U0
G53Z#26
M99
```

Example:

```
N1
M100
M101 Z-20.(Machine Position of -20”)
M75
T0101
```