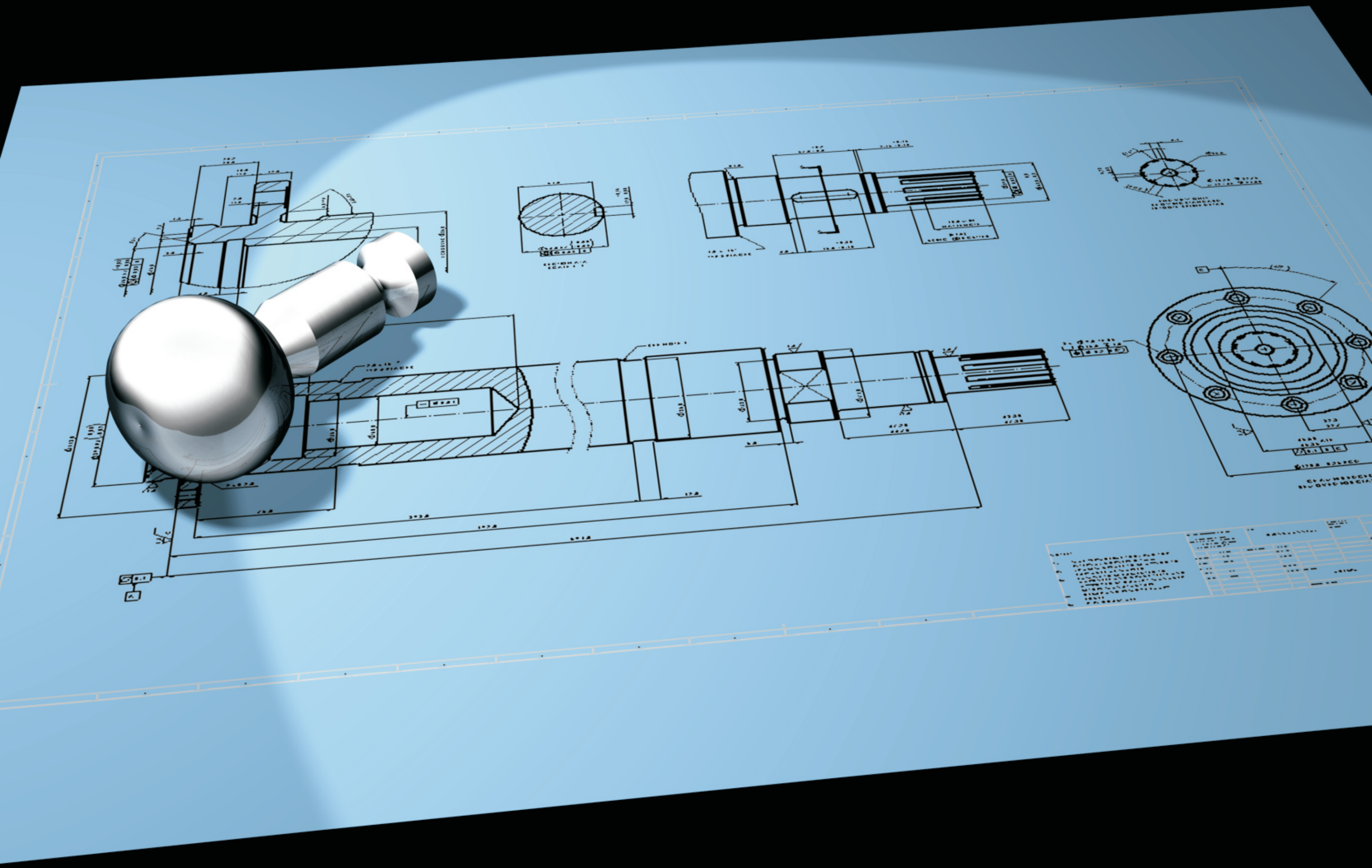
CNC Programming Guide
TURNING



CADEM

Foreword

The purpose of this guide is to help faculty teach CNC programming without tears. Most books currently available on CNC programming are not only inadequate, but also specific to certain CNC control systems and aimed at the professionals in industry. Such manuals only have very basic explanations on various codes, without adequate pictures and program examples. Therefore, they do not help in teaching the fundamentals of CNC programming.

My company Cadem strongly believes in doing its best to bridge the gap between education and industry, and consistently works on a lot of initiatives to achieve this. This handbook and the attendant material is one such initiative, to provide a clear set of training material to teach CNC programming effectively. This handbook comes free of cost to you. You can print and use it, or freely distribute soft copies.

It is part of a package of free CNC Learning Kit CD-ROM that includes the following for both CNC milling and turning:

- Programming guide CNC turning and milling.
- Set of sample CNC programs.
- Set of tool path simulations, corresponding to the sample programs.
- Demo versions of NCyclo multimedia software to teach CNC technology.
- Teacher and Student handbooks for lab. Exercises.

I hope you enjoy teaching or learning using the Free CNC Learning Kit as much as I enjoyed making it.

G.V.Dasarathi
Director, CADEM

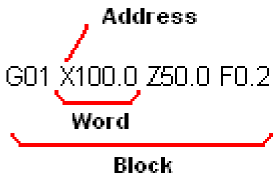
10 December 2012
Bangalore

Table of Contents

Program format

Program formats and commands explained in this chapter relate to the Fanuc 0iT controller.
A CNC program consists of a number of lines, called blocks. Each block contains a number of commands.

Block format



G01 X100.0 Z50.0 F0.2 is a block. It tells the tool to move along a straight line to X100.0 Z50.0 at a feed rate of 0.2 mm/revolution.
A block consists of a set of words. Each word is a command. E.g., X100.0 is a word.
A word consists of an alphabet called the address, followed by a number. In X100.0, X is an address.

Other than coordinates, the most commonly used words in a program are the G-codes and M-codes.

G codes mostly involve tool motion commands like rapid motion, feed motion, circular motion, dwell, and canned cycle codes.

M codes mostly involve machine actions like spindle on / off, tool change and coolant on / off.

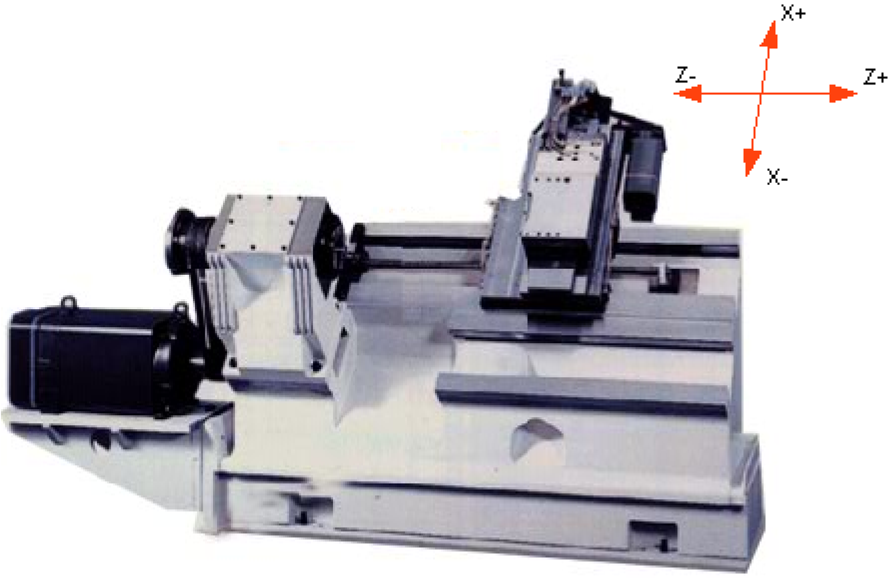
Common addresses

N	Block number - specifies the start of the block
G	Preparatory functions
M	Miscellaneous functions
X	X-axis coordinate
Z	Z-axis coordinate
I	X-axis location of arc center
K	Z-axis location of arc center
R	Radius of arc
S	Spindle speed or Cutting speed
F	Feed rate
T	Tool number

Coordinate system

Axes convention

The Z axis is along the spindle, while the X axis is perpendicular to it. The program zero is the intersection of these axes. All coordinates in a program are referenced from this point.



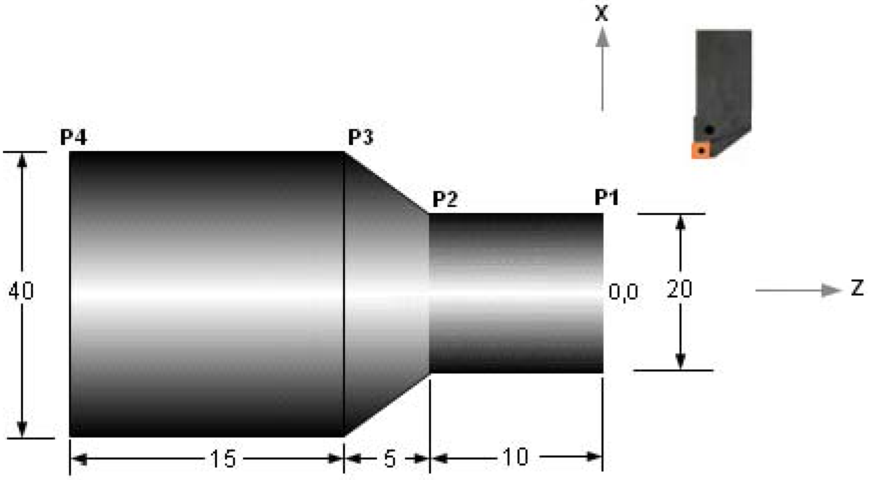
Axes on a lathe

Z is along the part axis and X is normal to it.

Absolute, incremental coordinates

In Absolute programming the end point of a motion is programmed with reference to the program zero point. Coordinates are specified as X,Z. X coordinate is the diameter of the part.
In Incremental programming the end point is specified with reference to the current tool position. Coordinates are specified as U,W. U is the incremental diameter of the part.

Example



Absolute traverse from P1 to P2, then to P3
X20.0 Z-10.0
X40.0 Z-15.0

Incremental traverse from P1 to P2, then to P3
U0.0 W-10.0
U20.0 W-5.0

Incremental mode programming is seldom used.

Note: All examples in this chapter are in absolute mode.

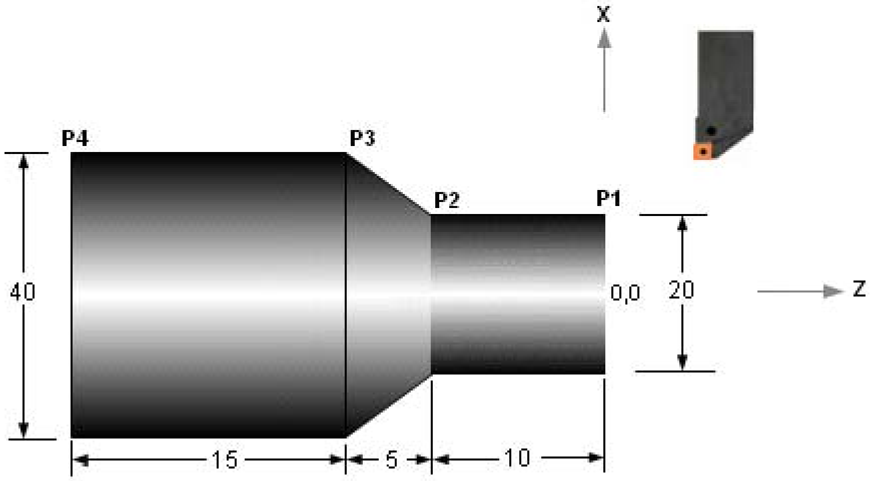
Basic motion commands

G00 - Rapid traverse

When the tool is moving to a position preparatory to executing a cutting motion or when it is moving to the tool change position, the motion is a essentially a waste of time and is executed as fast as possible. The motion is called Rapid traverse. The time taken to execute a rapid motion is also called the Air cut time.
Typical rapid traverse rates are 20 to 40 m /min., but can be as high as 80 m/min.

Format
G00 X_ Z_
X, Z are the destination coordinates

Example



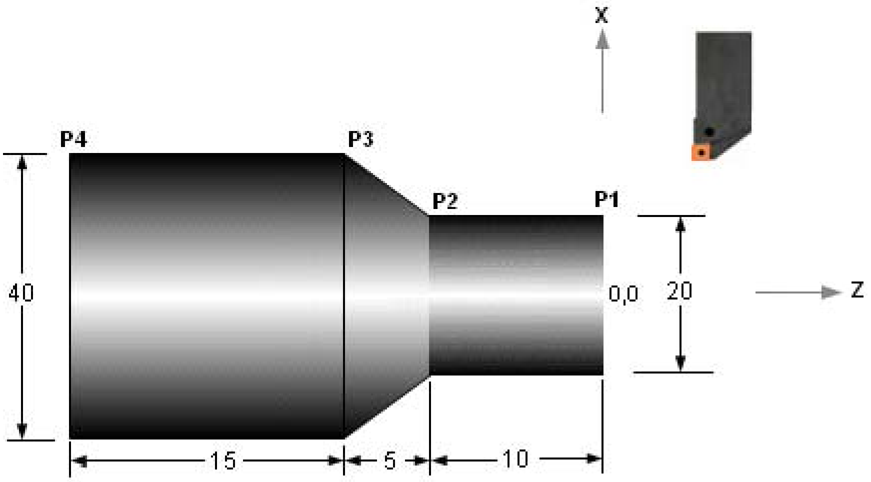
Rapid motion to P1
G00 X20.0 Z0.0

G01 - Linear interpolation

The tool moves along a straight line in one or two axis simultaneously at a programmed linear speed, the feed rate.

Format
G01 X__ Z__ F__
X, Z are the destination coordinates
F is the feed rate, the speed of the linear motion

Example



Linear motion from P1 to P2, then to P3
G01 X20.0 Z-10.0 F0.2
X40.0 Z-15.0

G01 need not be repeated in the second line because it is a 'modal command' – it stays active till it is changed by a different motion command.

G02 / G03 - Circular interpolation

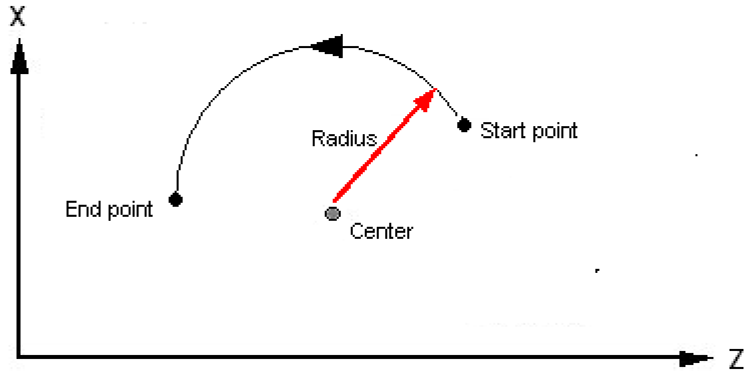
Motion along a circular arc at a programmed linear speed, the feed rate. G02 moves along a Clockwise (CW) arc, G03 moves along a Counterclockwise (CCW) arc.

An arc can be programmed using its radius or the coordinates of its center point.

Format

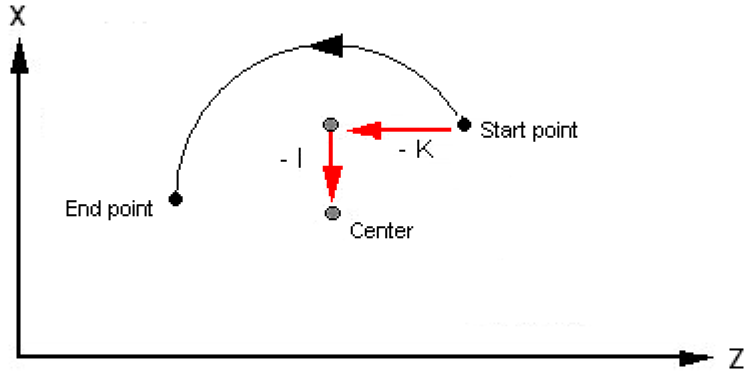
Using arc radius:

G02/03 X__ Z__ R__ F__
X, Z are the destination coordinates
R is the radius
F is the feed rate

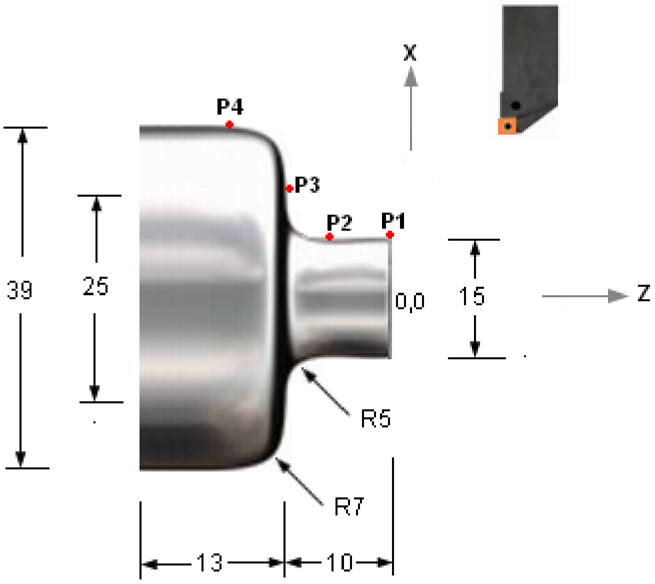


Using arc center coordinates:

G02/03 X__ Z__ I__ K__ F__
X, Z are the destination coordinates
I and K are the relative distance from the arc start point to the arc center
I = X coord. of start point - X coord. of center
K = Z coord. of start point - Z coord. of center
I and K must be written with their signs



Example



Arc radius programming:
Motion from P2 to P3, then to P4
G02 X25.0 Z-10.0 R5.0 F0.25
G03 X39.0 Z-17.0 R7.0

Arc center programming:
Motion from P2 to P3, then to P4
G02 X25.0 Z-10.0 I5.0 K0.0 F0.15
G03 X39.0 I0.0 K-7.0

G32 – Threading motion

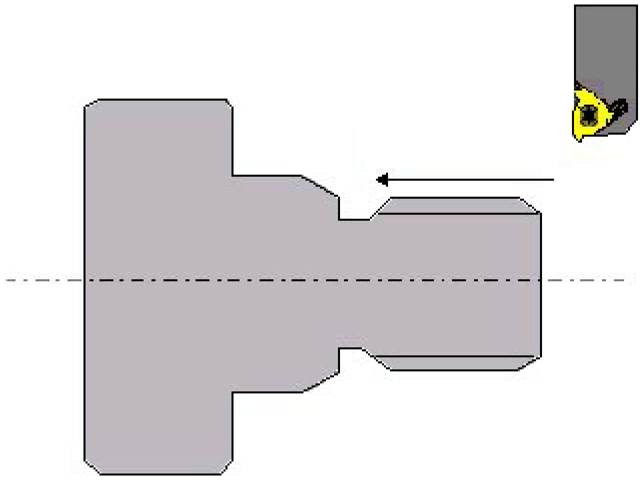
A threading motion is a motion along a straight line, but is NOT a linear interpolation motion. The tool motion does not start immediately when the command is encountered. It is coordinated with the rotation of the spindle - the tool starts moving when an *index pulse* is received from the spindle encoder. This pulse occurs at a specific angular position of the spindle, once in each spindle rotation. This ensures that each thread starts at the same angular position, and each cut follows the path of the earlier cut.

The Lead is the axial distance the nut advances in one revolution of the screw, while the pitch is the distance between adjacent threads. Lead = Pitch x No. of starts. In a single start thread the lead is equal to the pitch.

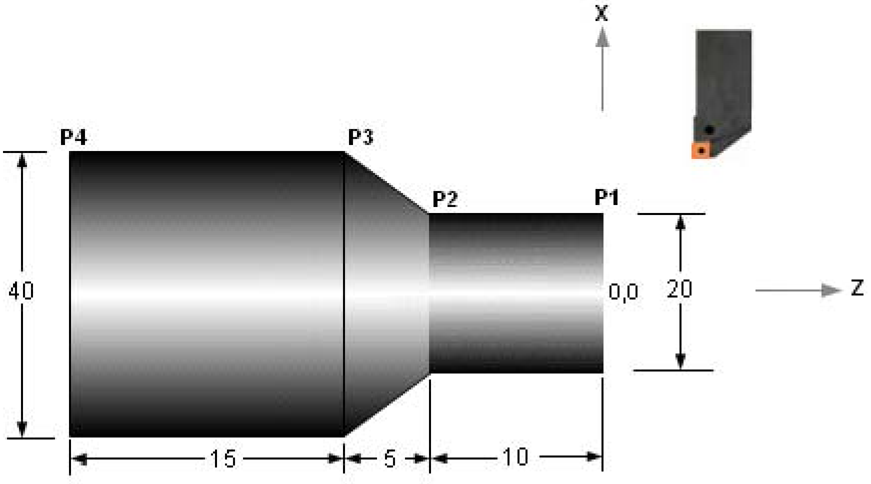
When cutting a thread, for every revolution of the part the tool moves axially by a distance equal to the Lead of the thread.

Format
G32 X__ Z__ F__

X, Z are the destination coordinates
F is the lead of the thread.



Example



The following program segment cuts a thread of 2 mm. pitch to a depth of 0.6 mm., from point P1 to 2 mm. before point P2:

```
G00 X19.6 Z2.0
G32 Z-8.0 F2.0
G00 X22.0
Z2.0

G00 X19.2 Z2.0
G32 Z-8.0 F2.0
G00 X22.0
Z2.0

G00 X18.8 Z2.0
G32 Z-8.0 F2.0
G00 X22.0
Z2.0
```

The G32 command is seldom used. The G76 canned cycle is commonly used because it can cut a thread with multiple cuts at various depths by specifying the pitch, thread depth, etc. in two lines.

G04 – Dwell

A dwell command results in a temporary stoppage of all axis motions for a specified duration. The spindle motion is not affected. It is typically used when the tool has reached the final position in an operation and needs to stay there for a few spindle rotations to obtain good dimensional accuracy or surface finish. For example, in a grooving operation when the tool reaches the bottom of the groove it needs to stay there for at least one full revolution. Without a dwell it would retract back instantaneously and result in a non-circular cross section at the groove bottom.

Format
G04 X_
X is the dwell time in seconds.

Example
G04 X1.0

This results in a dwell of 1 second.

F, S, T commands

Feedrate

The feed rate is specified in mm. per revolution.

Format
F_
F is specfied in mm. per revolution.

Example
F0.25

This means a feed rate of 0.25 mm. / rev.

Spindle rotation

Spindle rotation is started by specifying a spindle direction command and a spindle speed command.

Spindle direction:
This is specified by an M code.
M03 : Spindle clockwise (CW)
M04 : Spindle counter-clockwise (CCW)
M05 : Spindle stop

Spindle speed:
The spindle speed is specified either as a constant surface speed or as a constant spindle speed.

Constant surface speed
This is commanded by G96, and is always accompanied by a limiting spindle speed command G50.

Example:

```
G96 S225 M03
G50 S3000
```

The first line commands a constant surface speed of 225 m./ min. (meters per minute) with the spindle rotating CW.
The second one commands a limiting spindle speed of 3000 RPM.

Constant spindle speed
This is commanded by G97.

Example:

```
G97 S1350 M04
```

This results in a spindle speed of 1350 RPM, spindle rotating CCW.

Constant spindle speed is used in threading and drilling, while constant surface speed is used in all other operations.

Tool change

The tool change command includes the tool number and the tool offset number of the commanded tool. When the command is executed, the tool changer causes the commanded tool to come to the cutting position. E.g., if the tool changer is a turret, it indexes so that the commanded tool comes to the active position.

Format
Taabb
aa is the tool number
bb is the tool offset number.

The tool number and offset number must be written with leading zeros. E.g., tool number 6 is written as 06.

Example
T0303

This means tool number 3 and offset number 3.

Program structure

Start

The first line is the % character.
The second line is the program number, written as Onnnn. E.g., O2345 means program number 2345.

End

The last but one line is the program end command (M02 or M30).
The last line is the % character.

Block numbers

Block numbers add clarity to the program. They are written as N_

E.g.,

```
- -
- -
N0123 G00 G90 X100.0  Y150.0
N0124 G01 Z-10.0 F250.0
N0125 X120.0
- -
- -
```

Block numbers are optional. They can be omitted from all blocks or included in some blocks only. Quite often block numbers are used only in tool change blocks. The leading zero is optional. E.g., N0005 and N5 mean the same.

Comments

Comments can be inserted to add clarity to the program. They can be operation names, tool names, instructions to the operator, etc. Comments are inserted within brackets. Without comments a program is just a mass of alphabets and numbers and you cannot figure out what each section of the program is doing. A comment can be in a separate block by itself, or after a set of commands, as shown below.

```
(RAPID TO TOOL CHANGE POSITION)
G00 X200.0 Z150.0 M05
T0202 (GROOVING TOOL)
```

Modal commands

A Modal command is a command that remains active till it is canceled or changed by another command of the same family.
E.g.,

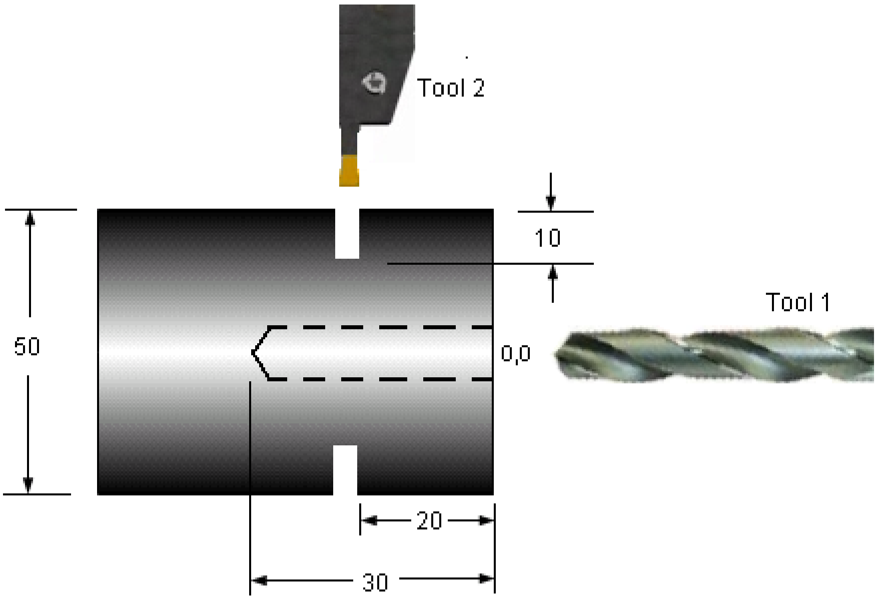
```
G01 X50.0 F0.2
G01 Z-5.0 F0.2
G01 X60.0 F0.2
G00 X100.0
G01 Z-80.0 F0.2
G01 X120.0 F0.2
```

Here G01 and F are modal, and need not be repeated in every block. G01 remains active till it is changed by G00. The block after G00 has it, but here F need not be repeated. The blocks can be written as:

```
G01 X50.0 F0.2
Z-5.0
X60.0
G00 X100.0
G01 Z-80.0
X120.0
```

Sample program

This sample program is a simple full program that does a drilling operation followed by a grooving operation.

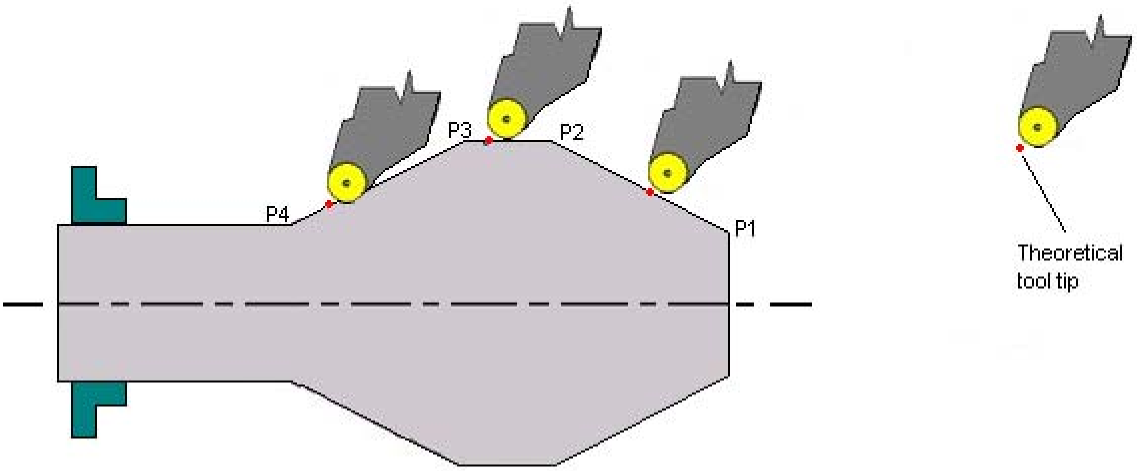


Program block	Explanation
%	Program start character
O0998	Program number 998
G00 X200.0 Z150.0	Move to position away from part for tool change
T0101	Tool change to tool number 1 (Drill)
G97 S1200 M03	Constant spindle speed of 1200 RPM, spindle CW
M08	Coolant ON
G00 X0.0 Z2.0	Move at rapid to position for drilling
G01 Z-30.0 F0.15	Drill the hole feed rate 0.15 mm/rev.

G00 Z2.0 M09	Rapid out of hole and coolant OFF
G00 X200.0 Z150.0 M05	Rapid to tool change position and spindle OFF
T0202	Tool change to tool number 2 (Grooving tool)
G96 S180 M03	Constant spindle speed 180 m/min, spindle CW
G50 S3500	Limiting spindle speed 3500 RPM
G00 X54.0 Z-20.0 M08	Rapid to position above groove, coolant ON
G01 X30.0 F0.1	Feed to bottom of groove
G04 X1.0	Dwell 1 second
G00 X 54.0	Rapid out of groove
G00 X200.0 Z150.0 M05	Rapid to tool change position and spindle OFF
M09	Coolant OFF
M02	Program end
%	End character

Tool radius compensation (TNRC)

Tool nose radius compensation, or TNRC, is required for generating accurate profiles. When you command the tool to move to a position, you are actually commanding the Theoretical Tool Tip (TTT) to move to the position. When doing an operation like contour turning, you just program the contour according to the coordinates in the part drawing. This causes the TTT point moves along the commanded path.

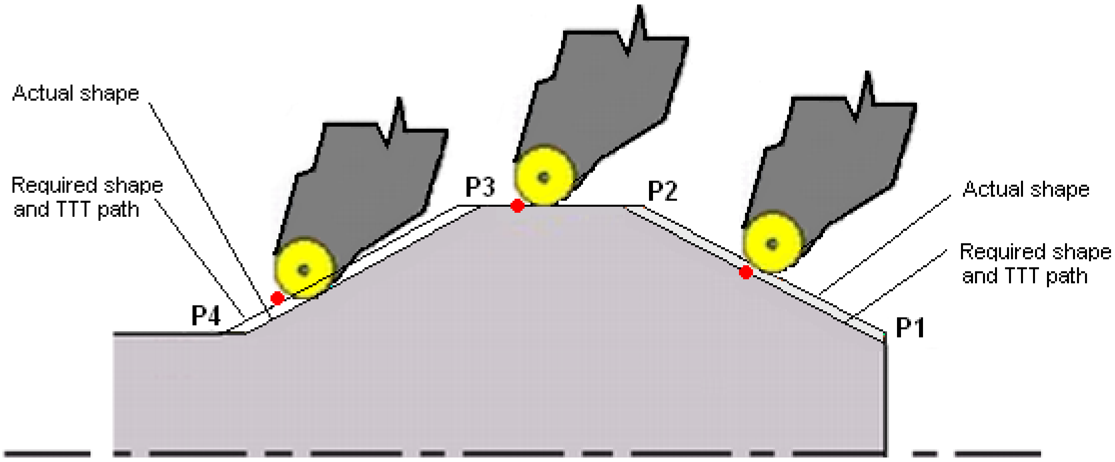


TTT moving along contour

This is the point on the tool that is used as the reference point for determining tool offsets.

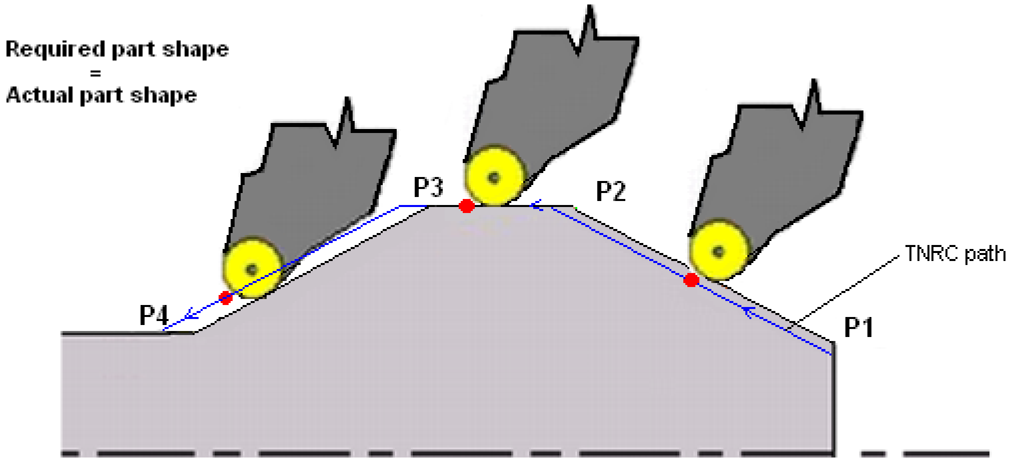
Necessity of TNRC

As the tool moves along the programmed contour, the point on the tool nose radius that is actually doing the cutting keeps changing. We actually need the nose radius to be tangential to the part contour at the point where it is cutting, but moving the Theoretical Tool Tip (TTT) along the contour does not ensure this. As a result, the tool leaves unmachined material in some areas (P1 to P2 in picture) and digs into the material in some areas (P3 to P4 in picture).



Tool path without TNRC

To get an accurate contour during machining, an alternate tool path is generated such that the nose radius is tangential to the contour. This is the path with Tool Nose Radius Compensation (TNRC).



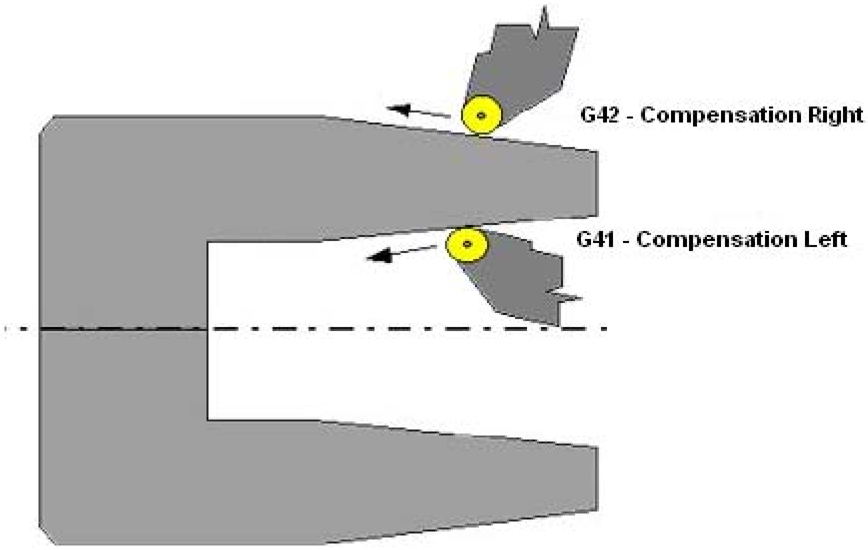
Compensated tool path

Compensation commands

The compensated tool path must be either to the left or the right of the tool path programmed with the coordinates from the part drawing. The direction of compensation depends on the direction of motion and whether the tool is cutting on the inside or outside of the part. In the program you can specify whether the compensation must be to the left or right, and the controller determines the compensated tool path. The tool nose radius too must be specified in a separate area of the memory.

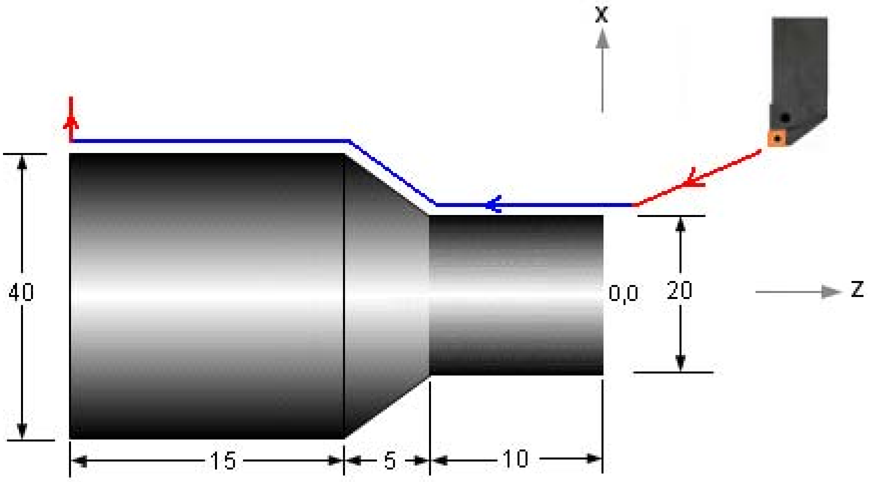
The commands are:

- G41 Tool nose radius compensation Left
- G42 Tool nose radius compensation Right
- G40 Tool nose radius compensation Cancel



TNRC Left and Right

Example



Program to move along the contour in the part (red lines in the picture indicate rapid traverse, and blue lines linear interpolation).

```
-----
-----
G00 G42 X20.0 Z2.0
G01 Z0.0
Z-10.0
X40.0 Z-15.0
Z-30.0
G00 G40 X60.0
-----
-----
```

Subprograms

A tool path pattern that is repeated can be stored as a subprogram and called multiple times. Using a subprogram reduces the program length and programming time, and makes the program more readable. A subprogram looks like a normal program, but is terminated with an M99 command at the end instead of M02 or M30. It is called from the main program by a subprogram call command.

Format
Subprogram call:
M98 Paaabbbb

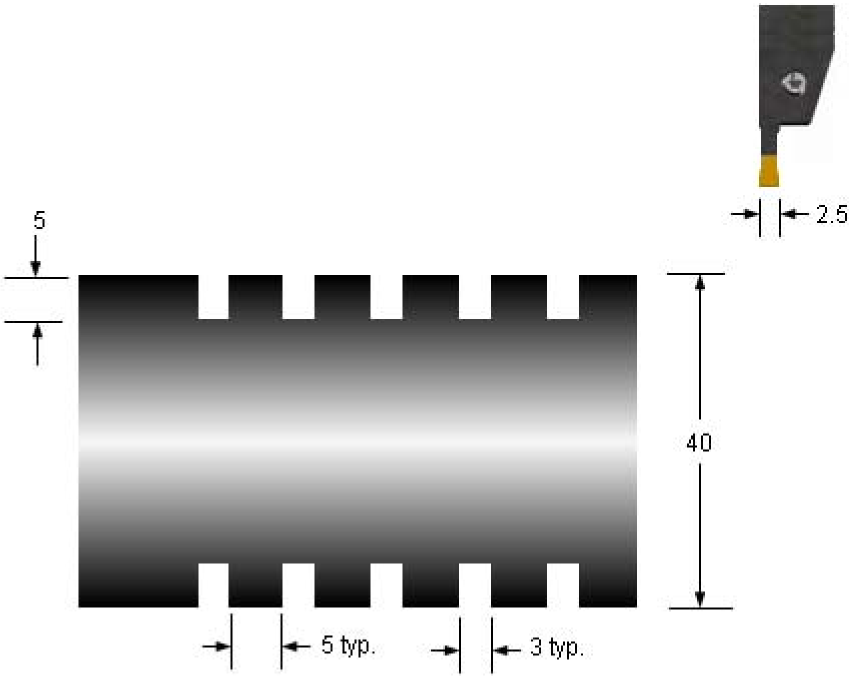
M98 = subprogram call command
aaa = number of subprogram repetitions, written as a 3 digit number
bbbb = subprogram number, written as a 4 digit number

aaa and bbbb MUST be written as 3 and 4 digit numbers respectively, if necessary by padding them with leading zeros.

E.g., M98 P0051234.
This command calls subprogram 1234, 5 times.

If a subprogram is only called once, the aaa parameter can be omitted.
E.g., M98 P1234
This calls subprogram 1234 just once.

Example



Since the tool width is 2 mm. and the groove width is 3 mm., two plunges are required at each groove. The tool path at each groove is:

1. Move at rapid to the start position of the groove in Z
2. Feed into the groove.
3. Rapid out of the groove
4. Rapid sideways to the start point of the next cut.
5. Feed into the groove.
6. Rapid out of the groove

The program segment to cut the grooves would look like this (the text in brackets is comments, and this is exactly how you can insert comments in an actual program):

```
-----
-----
G00 X44.0 Z0.0 (MOVE TO START SAFE POSITION JUST ABOVE PART)

(GROOVE 1)
W-5.0 (MOVE SIDEWAYS TO POSITION FOR FIRST CUT)
G01 X30.0 F0.1
G00 X44.0
W-0.5
G01 X30.0 F0.1
G00 X44.0

(GROOVE 2)
W-5.0
G01 X30.0 F0.1
G00 X44.0
W-0.5
G01 X30.0 F0.1
G00 X44.0

(GROOVE 3)
W-5.0
G01 X30.0 F0.1
G00 X44.0
W-0.5
G01 X30.0 F0.1
G00 X44.0

(GROOVE 4)
W-5.0
G01 X30.0 F0.1
G00 X44.0
W-0.5
G01 X30.0 F0.1
G00 X44.0

(GROOVE 5)
W-5.0
G01 X30.0 F0.1
```

```
G00 X44.0
W-0.5
G01 X30.0 F0.1
G00 X44.0
-----
-----
```

The tool path is the same for each groove. This segment can be put in a subprogram that is called 5 times from the main program. The main program and subprogram can be written like this:

Main program:

```
-----
-----
G00 X44.0 Z0.0
M98 P0052253 (CALL SUBPROGRAM 2253, 5 TIMES)
-----
-----
```

Subprogram:

```
%
O2253 (SUBPROGRAM 2253)
W-5.0
G01 X30.0 F0.1
G00 X44.0
W-0.5
G01 X30.0 F0.1
G00 X44.0
M99 (END OF SUBPROGRAM)
```

Canned Cycles – single cut

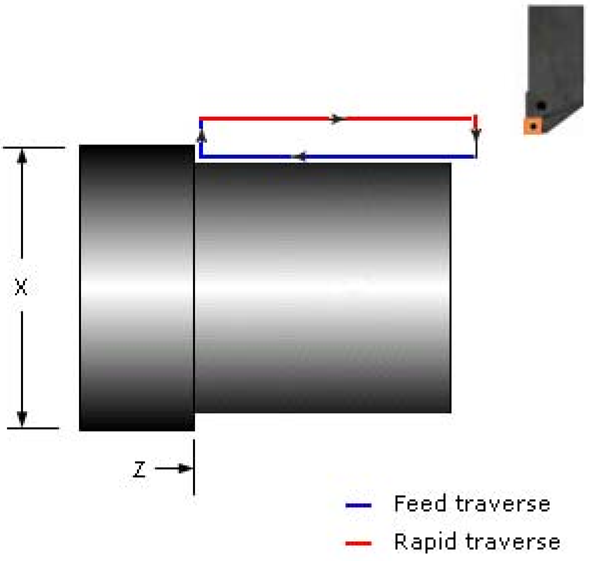
A single cut canned cycle executes a sequence of motions required to perform a cut – rapid approach to the start position, cutting motion, and rapid departure. A single block replaces 4 motions - 1 cutting and 3 rapid. Operations normally involve the removal of material in multiple cuts, so these cycles are seldom used. The multi-cut canned cycles are the ones generally used.

Turning cycle - G90

This cycle does a single turning cut (along the part axis).

Straight turning

Tool path



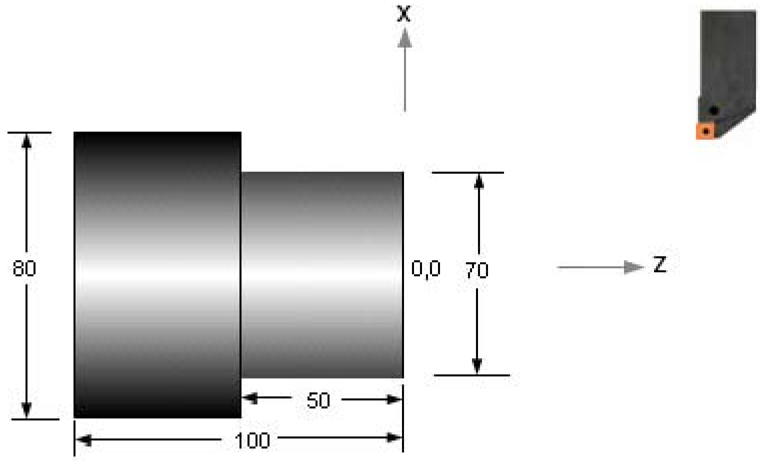
Format
G90 X_ Z_ F_

X = X coordinate of end point of cut, absolute
Z = Z coordinate of end point of cut, absolute
F = Feed rate

The end point can be specified by incremental coordinates instead of absolute coordinates. In this case:

1. Use addresses U and W instead of X and Z.
2. Use appropriate signs with the end point, since incremental coordinates are specified with reference to the start point.

Example

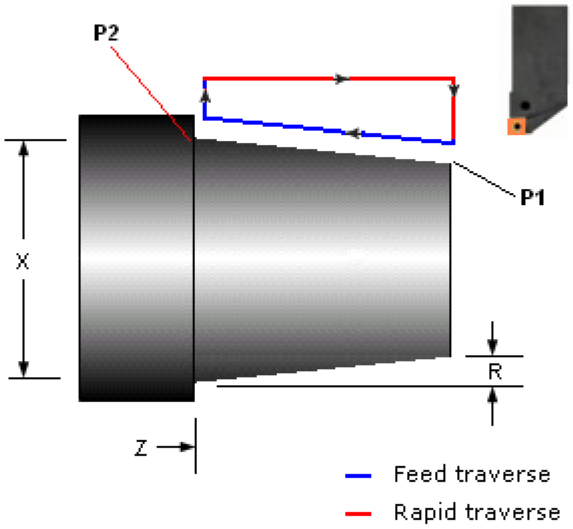


Raw material is a cylinder of 80 diameter.

```
-----
-----
G00 X82.0 Z2.0 (RAPID TO INITIAL POSITION)
G90 X75.0 Z-50.0 F0.2 (CUT TO DIAMETER 75)
X70.0                        (CUT TO DIAMETER 70)
G00 Z2
-----
-----
```

Taper turning

Tool path



Format
G90 X_ Z_ R_ F_

X = X coordinate of end point of cut, absolute
Z = Z coordinate of end point of cut, absolute
R = Taper amount, radial.
F = Feed rate

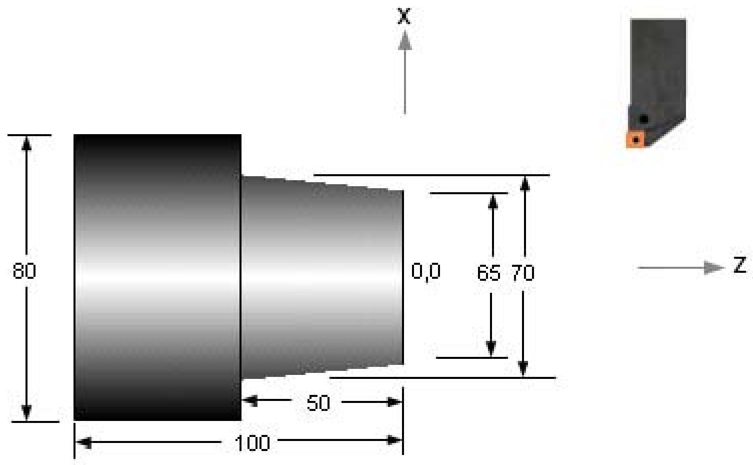
The cut starts at point P1, ends at point P2.

R = (Diameter at start of cut – Diameter at end of cut) / 2
R must be specified with the proper sign.

The end point can be specified by incremental coordinates instead of absolute coordinates. In this case:

1. Use addresses U and W instead of X and Z.
2. Use appropriate signs with the end point, since incremental coordinates are specified with reference to the start point.

Example



Raw material is a cylinder of 80 diameter.

```
-----
-----
G00 X67.0 Z1.0
G90 X65.0 Z-50.0 R-2.5 F0.25
-----
-----
```

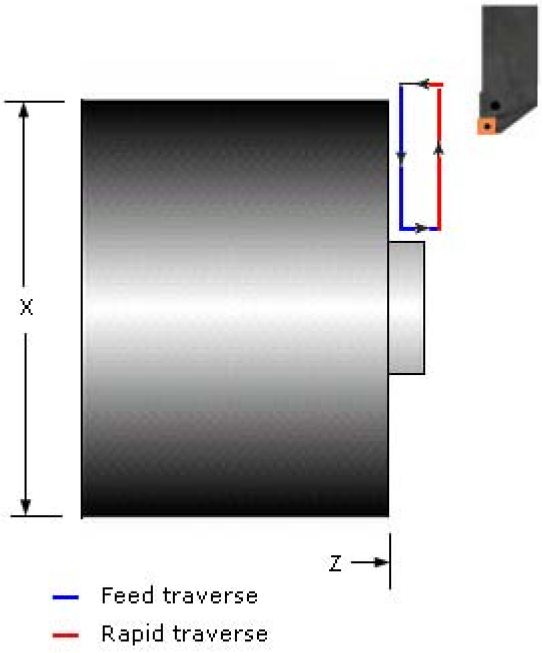
Note that the R value has a small approximation here since the cut is starting at Z1.0 instead of Z0.

Facing cycle - G94

This cycle does a single facing cut (perpendicular to the part axis).

Straight facing

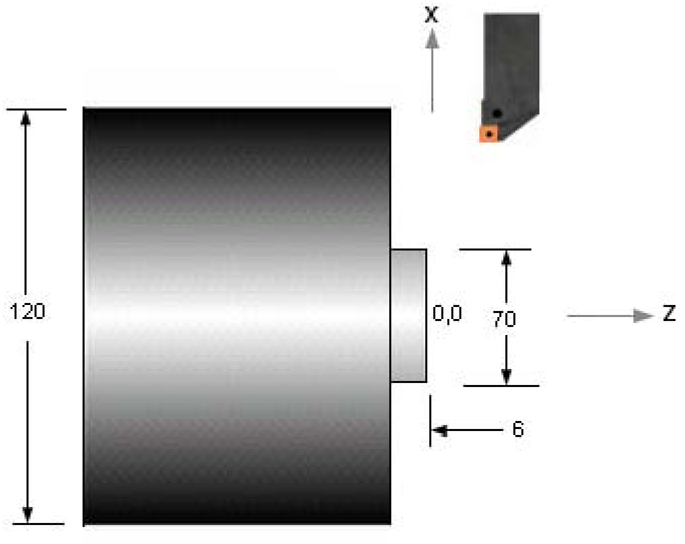
Tool path



Format
G94 X_ Z_ F_

X = X coordinate of end point of cut
Z = Z coordinate of end point of cut
F = Feed rate

Example

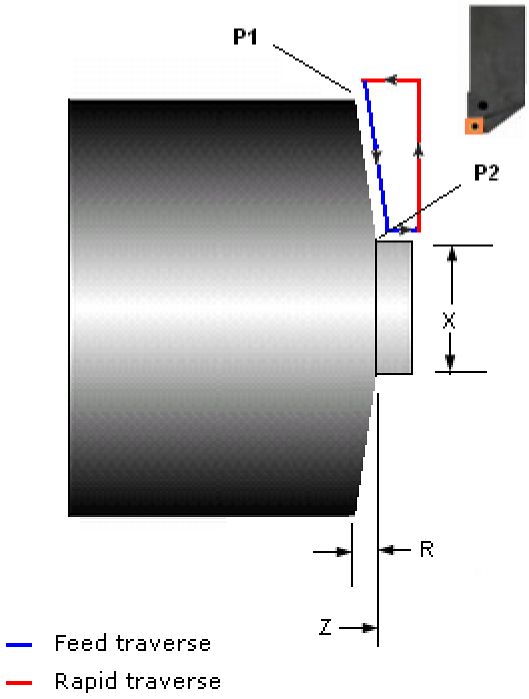


Raw material is a cylinder of 120 diameter.

```
-----
-----
G00 X122.0 Z1.0 (RAPID TO INITIAL POSITION)
G94 X70.0 Z-3.0 F0.25(FACE TO Z-3)
Z-6.0 (FACE TO Z-6)
-----
-----
```

Taper facing

Tool path



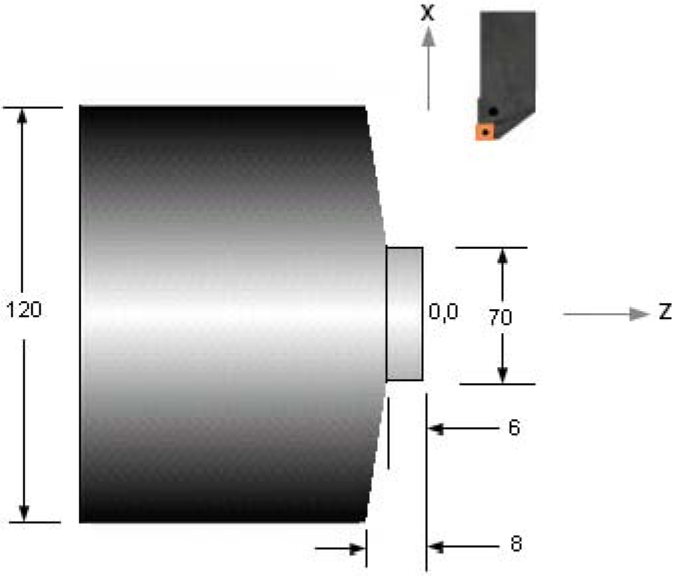
Format
G94 X_ Z_ R_ F_

X = X coordinate of end point of cut
Z = Z coordinate of end point of cut
R = Taper amount
F = Feed rate

The cut starts at point P1, ends at point P2.

R = Z coordinate of start point – Z coordinate of end point.
R must be specified with the proper sign.

Example



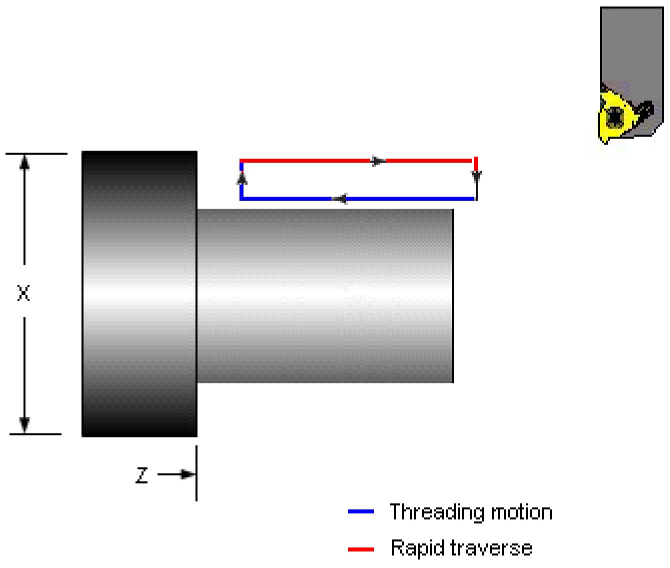
Raw material is a cylinder of 120 diameter.

```
-----
-----
G00 X122.0 Z1.0
G94 X70.0 Z-6.0 R-2.0 F0.2
-----
-----
```

Threading cycle - G92

This cycle does a single threading cut.

Tool path

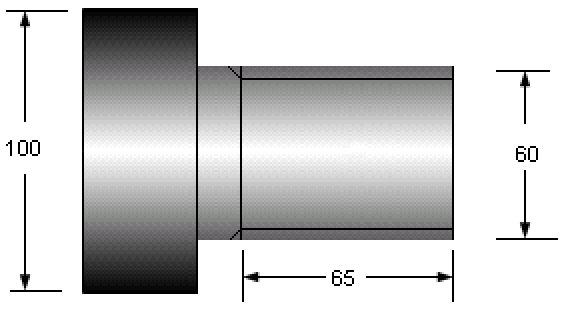


Format

G92 X_ Z_ F_

X = X coordinate of end point of thread
Z = Z coordinate of end point of thread
F = Thread lead

Example



G00 X60.0 Z2.0

```
G92 X59.0 Z-65.0 F3.0
X58.4
-----
-----
```

The G92 command, Z and F are modal values, which remain till they are changed. They are therefore omitted in the third block.

Canned Cycles – multiple cut

A canned cycle is a single command that executes a whole machining operation that requires repetitive tool motions. The cycle typically consists of a few blocks with data defining the area to be machined and some cutting parameters. The coordinates of individual tool motions are determined automatically by the machine controller and the motions are executed. An operation that may require tens or even hundreds of blocks of program can be written in just a few blocks.

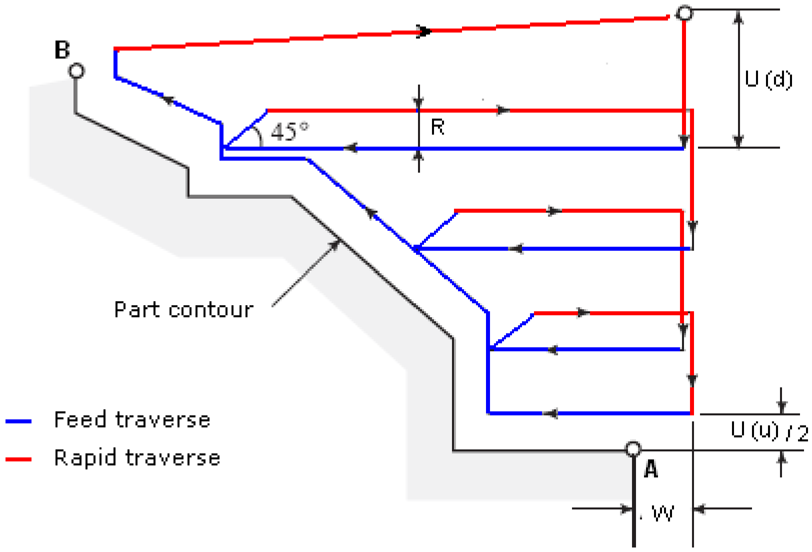
Canned cycles in Fanuc

G71	Stock removal in turning
G72	Stock removal in facing
G73	Pattern repeat
G70	Finish turning
G74	Axial drilling
G75	Radial grooving
G76	Threading

Turning cycle – G71

This cycle generates a part shape from a cylindrical raw material, with cuts along the axis. The cycle definition has the part shape, depth of cut, finish allowance and couple of other parameters.

Tool path



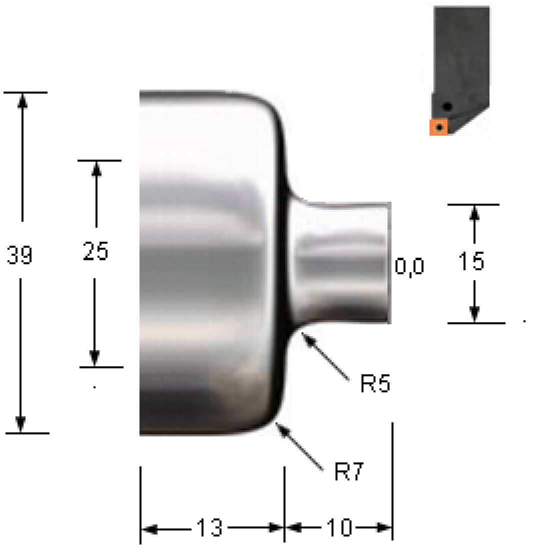
Format
G71 U(d)_ R_

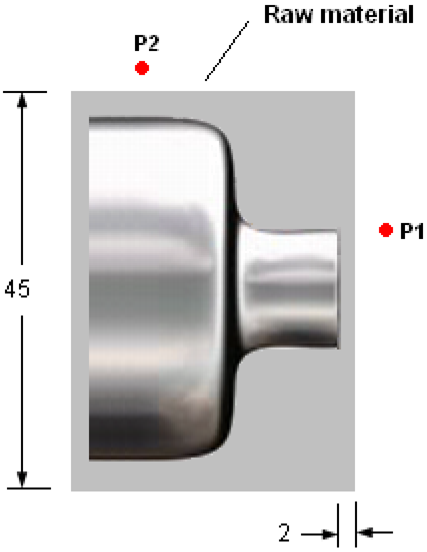
```
G71 P(s)_  Q(e)_  U(u)_  W_  F_
Ns _ _ _ _
_ _ _ _ _ _ _
_ _ _ _ _ _ _
Ne_ _ _ _
```

U(d) = Depth of cut, radius value
R = Retract amount, radius value
P = Number of the first block of the shape
Q = Number of the last block of the shape
U(u) = Finishing allowance in X, diameter value
W = Finishing allowance in Z
F = Feed rate

The blocks after the second G71 block define the part contour A to B. Parameter P has the number of the first block Ns and Q has the last block Ne.

Example

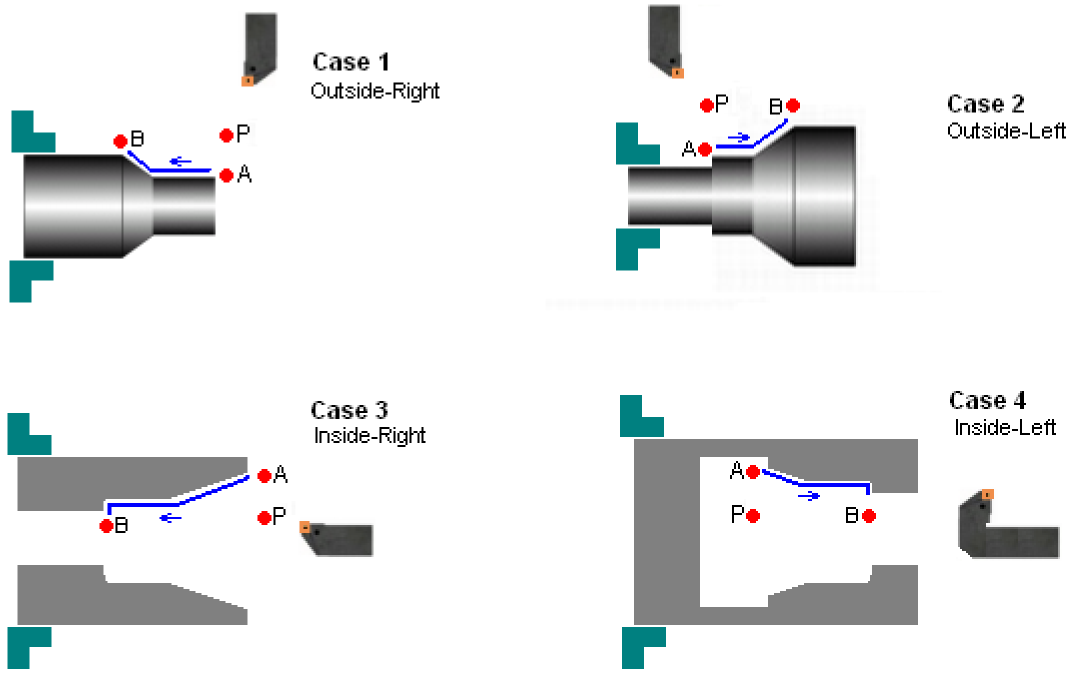




```
-----
-----
G00 X49.0 Z5.0
G71 U3.0  R0.5
G71 P10 Q20  U1.0 W0.5  F0.2
N10 G00 X15.0 Z4.0
G01 Z-5.0
G02 X25.0 Z-10.0 R5.0
G03 X39.0 Z-17.0 R7.0
G01 Z-20.0
N20 G00 X49.0
G00 Z5.0
-----
-----
```

The tool path defining the shape (between the blocks defined by P and Q) must start and end beyond the raw material. In this example the start and end points are points P1 and P2 respectively, 2 mm. away from the raw material. Note the use of block numbers in the program example. Block numbers are optional, need not be used in every block.

Contour definition and signs of finish allowances:



In the cycle, the area that is being machined decides:

1. The signs of the finishing allowances U and W, and
2. The way the part profile is defined

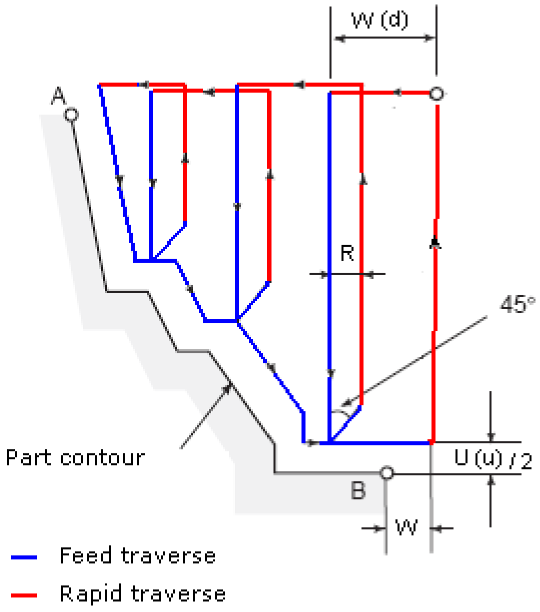
In each of the cases shown above, the tool is positioned at point P before calling the cycle and the part profile is defined from point A to B. The signs of the finish allowances U and W are as follows.

Case 1 (Outside-Right) : U +, W +
Case 2 (Outside-Left) : U +, W -
Case 3 (Inside-Right) : U -, W +
Case 4 (Inside-Left) : U -, W -

Facing cycle G72

This cycle generates a part shape from a cylindrical raw material, with cuts perpendicular to the axis. The cycle definition has the part shape, depth of cut, finish allowance and couple of other parameters.

Tool path



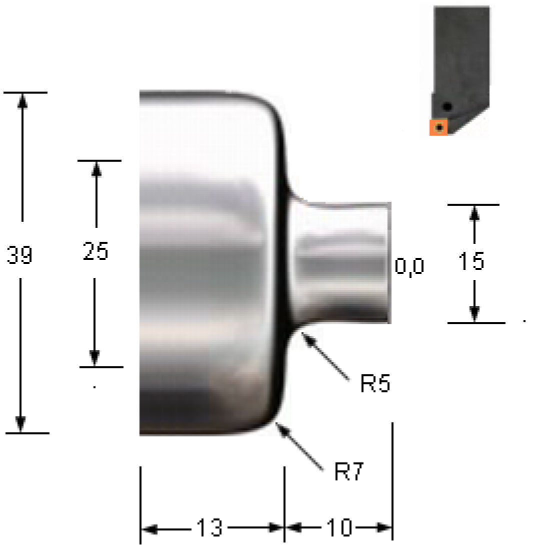
Format

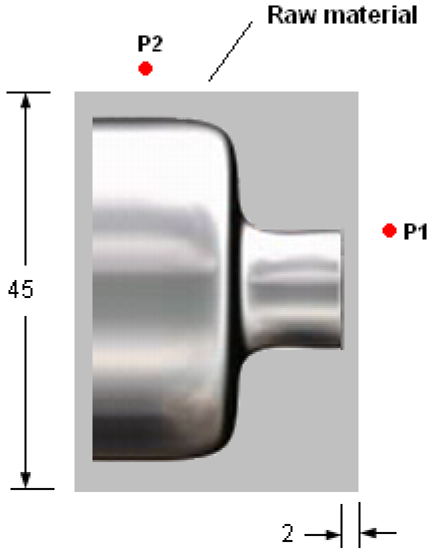
G72 W(d)_ R_
G72 P(s)_ Q(e)_ U(u)_ W_ F_
Ns _ _ _ _
_ _ _ _ _ _ _
_ _ _ _ _ _ _
Ne_ _ _ _

W(d) = Depth of cut
R = Retract amount, radius value
P = Number of the first block of the shape
Q = Number of the last block of the shape
U(u) = Finishing allowance in X, diameter value
W = Finishing allowance in Z
F = Feed rate

The blocks after the second G72 block define the part contour A to B. Parameter P has the number of the first block Ns and Q has the last block Ne.

Example

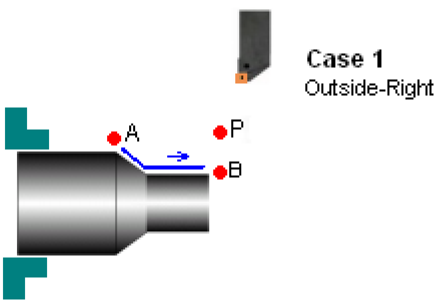


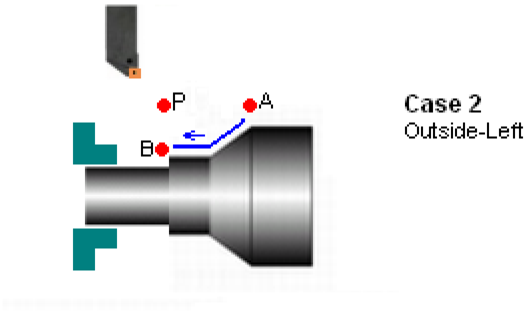


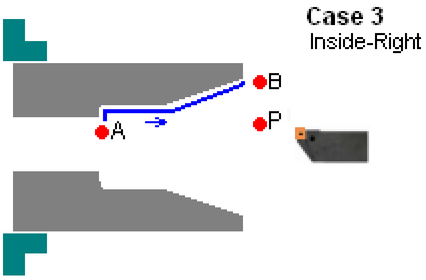
```
-----
-----
G00 X49.0 Z-20.0
G72 W3.0  R0.5
G72 P10 Q20  U1.0 W0.5  F0.2
N10 G00 X49.0 Z-20.0
G01 X39.0
Z-17.0
G02 X25.0 Z-10.0 R7.0
G03 X15.0 Z-5.0 R5.0
N20 G01 Z4.0
G00 X49.0
-----
-----
```

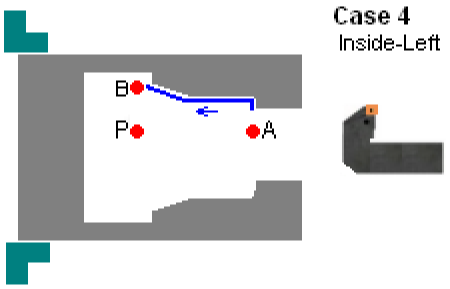
In this example the start and end points are points P2 and P1 respectively, 2 mm. away from the raw material. Note that these are the reverse of the points in the G71 turning cycle.

Contour definition and signs of finish allowances:









In the cycle, the area that is being machined decides:

3. The signs of the finishing allowances U and W, and
4. The way the part profile is defined

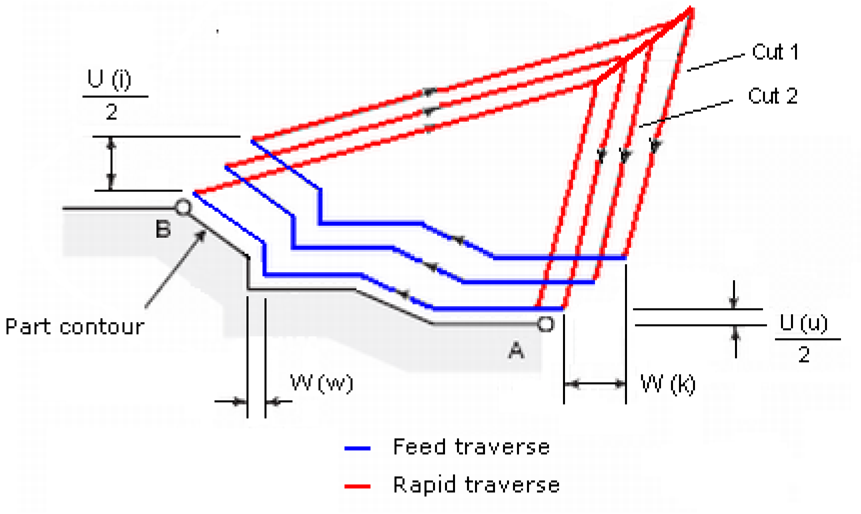
In each of the cases shown above, the tool is positioned at point P before calling the cycle and the part profile is defined from point A to B. The signs of the finish allowances U and W are as follows.

Case 1 (Outside-Right)	: U +, W +
Case 2 (Outside-Left)	: U +, W -
Case 3 (Inside-Right)	: U -, W +
Case 4 (Inside-Left)	: U -, W -

Pattern repeat cycle G73

This cycle generates a part shape from raw material that is the same shape as the final part with cuts parallel to the along the part shape. It is used when the raw material is a casting or forging. The cycle definition has the part shape, depth of material to be removed, number of cuts and finish allowance.

Tool path

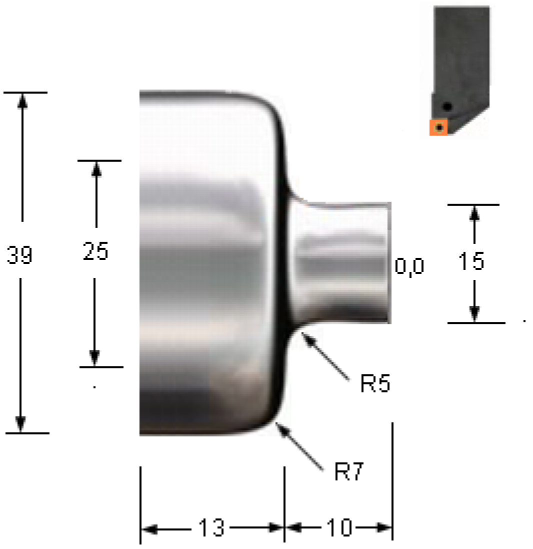


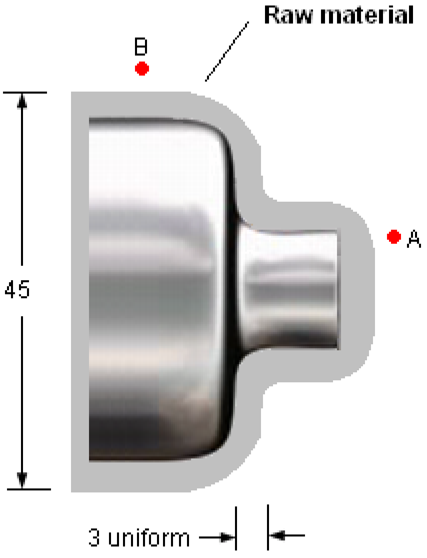
Format

G73 U(i)_ W(k)_ R_
G73 Ps_ Qe_ U(u)_ W(w)_ F_
Ns_ _ _ _ _
_ _ _ _ _ _ _ _ _
_ _ _ _ _ _ _ _ _
Ne_ _ _ _ _

U(i) = Relief in the X axes direction
W(k) = Relief in the Z axis direction
R = Number of cuts
P = Number of first block of the shape
Q = Number of the last block of the shape
U(u) = Finishing allowance in X
W(w) = Finishing allowance in Z
F = Feed rate

Example





```
-----
-----
G00 X60.0 Z10.0
G73 U5.0 W5.0 R3
G73 P10 Q20  U0.5 W0.5  F0.2
N10 G00 X15.0 Z4.0
G01 Z-5.0
G02 X25.0 Z-10.0 R5.0
G03 X39.0 Z-17.0 R7.0
G01 Z-20.0
N20 G00 X49.0
G00 Z5.0
-----
-----
```

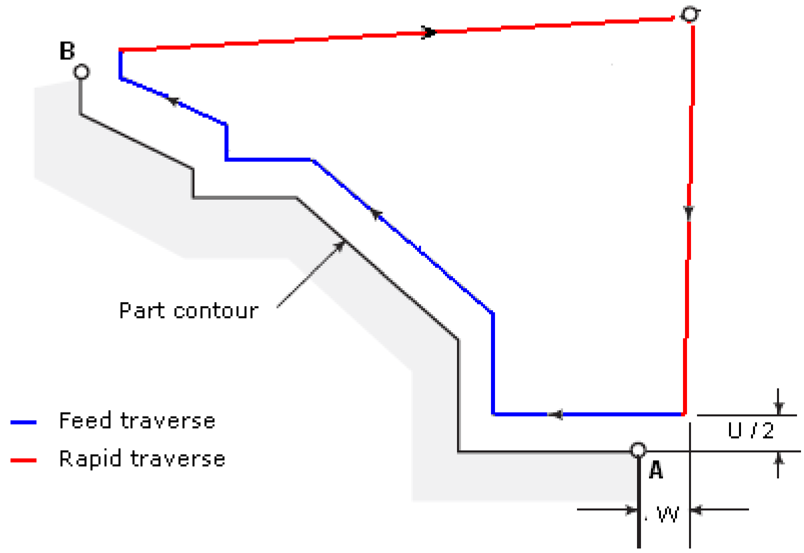
The tool path defining the shape (between the blocks defined by P and Q) must start and end beyond the raw material. In this example the start and end points are points A and B respectively, 2 mm. away from the raw material.

Contour definition and signs of finish allowances:
These are the same as in the G71 cycle.

Finish turning cycle G70

This cycle does a single finish pass along a contour that has typically already been rough turned with a G71, G72 or G73 cycle. Nose radius compensation is automatically activated in G70.

Tool path



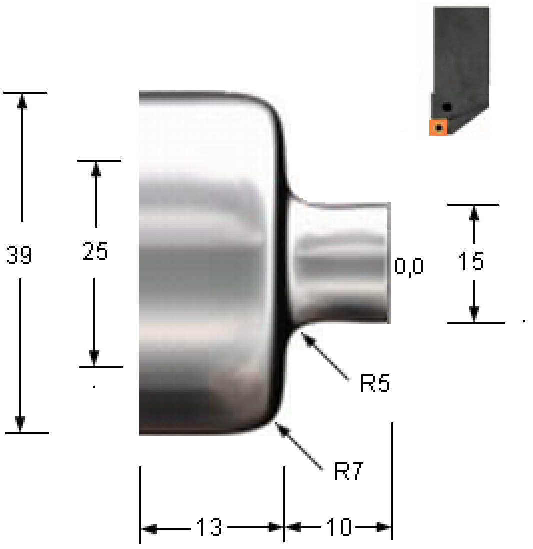
Format

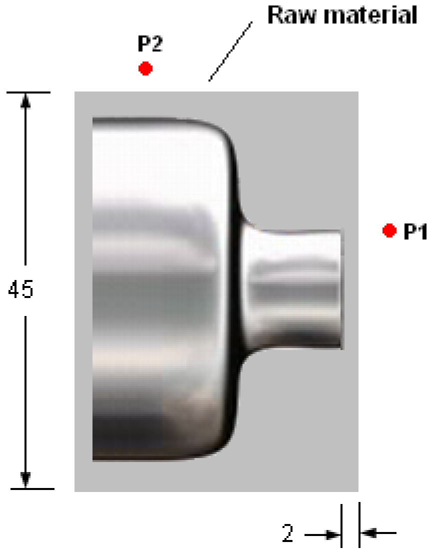
Ns_ _ _ _ _
_ _ _ _ _ _ _ _ _
_ _ _ _ _ _ _ _ _
Ne_ _ _ _ _
_ _ _ _ _
_ _ _ _ _
G70 P(s)_ Q(e)_ U_ W_

P = Number of first block of the shape
Q = Number of the last block of the shape
U = Finishing allowance in X
W = Finishing allowance in Z

If U or W are zero they can be omitted.

Example



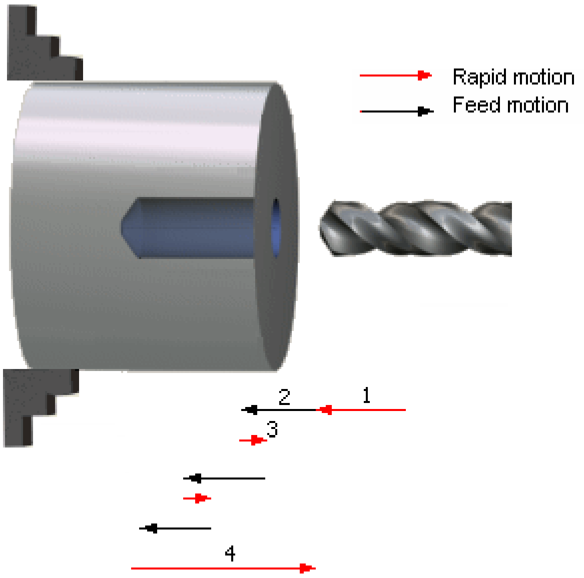


```
-----
-----
G00 X49.0 Z5.0
(ROUGH TURN CONTOUR)
G71 U3.0  R0.5
G71 P10 Q60  U1.0 W0.5  F0.2
N10 G00 X15.0 Z4.0
N20 G01 Z-5.0
N30 G02 X25.0 Z-10.0 R5.0
N40 G03 X39.0 Z-17.0 R7.0
N50 G01 Z-20.0
N60 G00 X49.0
G00 X200.0 Z150.0 M05
M09
T0202 (TOOL CHANGE)
(FINISH TURN CONTOUR)
G96 S200 M03
G50 S2500
X49.0 Z5.0 M08
G70 P20 Q60 F0.15
-----
-----
```

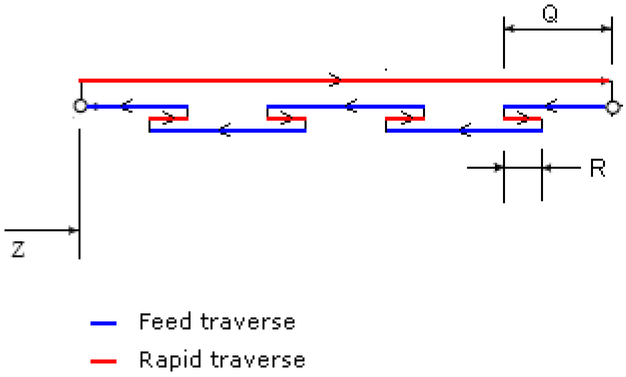
Axial drilling / grooving cycle - G74

This cycle does a peck drilling operation to drill a hole along the axis. The cycle can actually be used to drill multiple axial holes at various positions on the radius, on a machine with a C-axis and live tools. The explanation here is restricted to drilling a single axial hole.

Tool path



Format



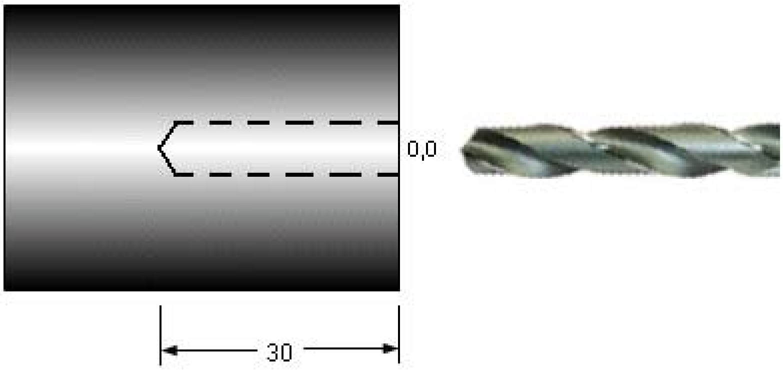
G74 R_
G74 Z_ Q_ F_

R = Retract amount at each peck
Z = Z coordinate of hole bottom

Q = Peck depth, in microns
F = Feed rate

To drill the hole in a single pass (without pecking), set Q equal to the depth of the hole.

Example

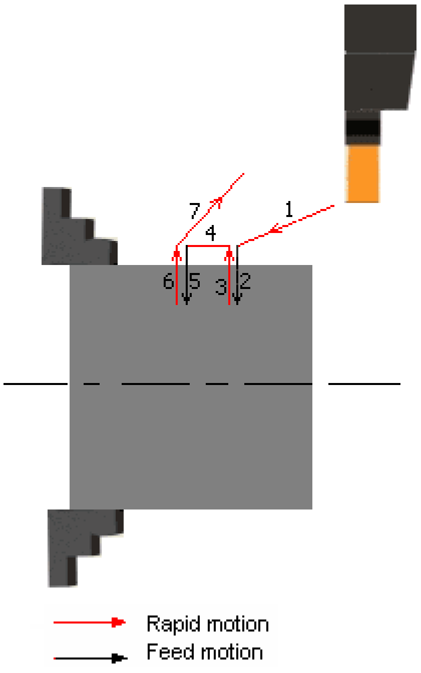


```
-----
-----
G00 X0 Z2.0
G74 R0.5
G74 Z-30.0 Q6000 F0.15
G00 X50.0
-----
-----
```

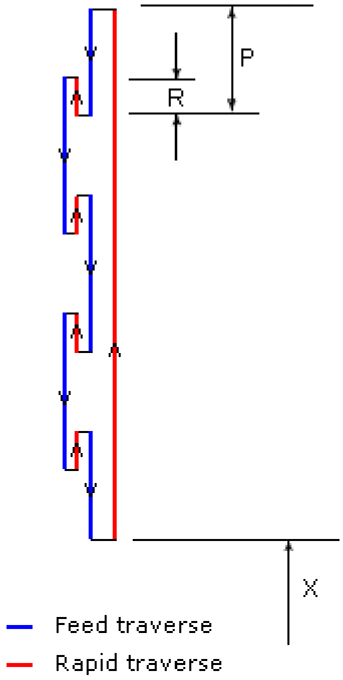
Radial drilling / grooving cycle - G75

This cycle does a peck drilling operation for grooving or drilling perpendicular to the axis. The cycle can actually be used to cut multiple grooves, or (on a machine with a C-axis and live tools) drill multiple radial holes at various positions along the length,. The explanation here is restricted to cutting a single groove.

Tool path



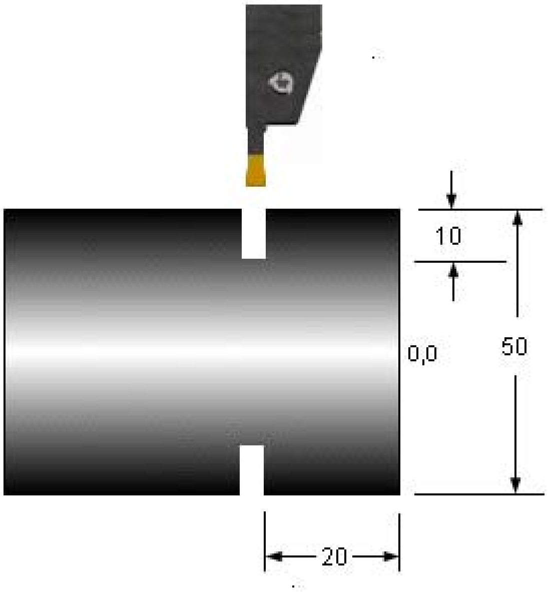
Format



G75 R_
G75 X_ P_ F_

R = Retract amount after each peck, radial distance
X = X coordinate of groove bottom
P = Peck depth, radial distance in microns
F = Feed rate

Example

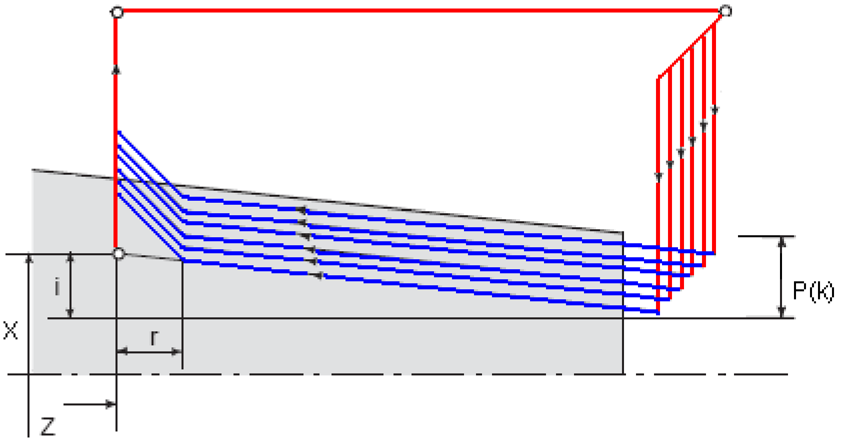


```
-----
-----
G00 X54.0 Z-20.0
G75 R0.5
G75 X30.0 P3000 F0.1
G00 X100.0 Z50.0
-----
-----
```

Threading cycle - G76

This cycle cuts a straight or taper thread with multiple cuts. The cycle definition has the thread coordinates, pitch, depth of thread, etc.

Tool path

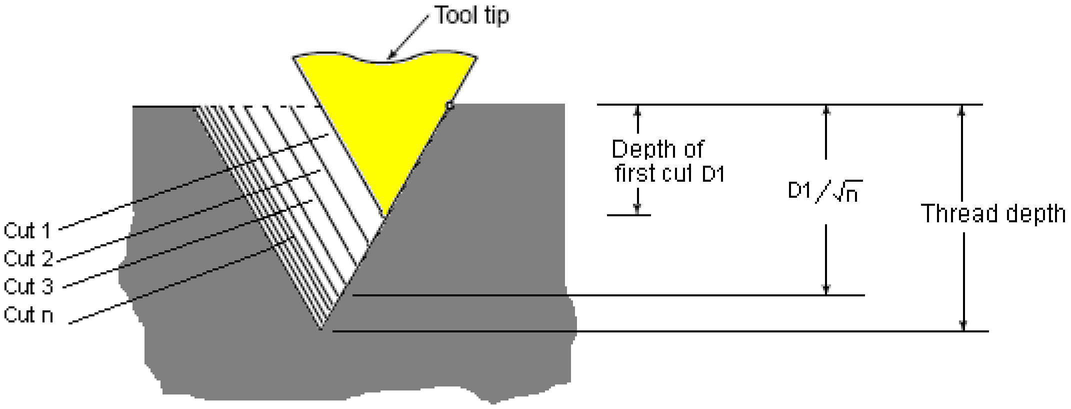


Format

G76 P(m)(r)(a) Q(d min)_ R(d)_
G76X_ Z_ R(i)_ P(k)_ Q(d)_ F_

m = No.of idle passes after the last cut. E.g., 02 would mean 2 idle passes
r = Chamfer distance at end of thread, fraction of the lead multiplied by 10. E.g., 12 would mean a chamfer distance 1.2 times the lead.
a = Angle of tool tip. E.g., 60 would mean a thread angle of 60 degrees.
Q(d min) = Minimum depth of cut, in microns. E.g., 0.1 mm. is written as 100.
R(d) = Finishing Allowance, radial value, in microns. E.g., 0.15 mm. is written as 150.
X,Z = coordinates of end point of thread.
R(i) = Taper value. Positive for external threads, negative for internal threads, 0 for straight threads.
P(k) = Thread depth, radial value, in microns. E.g., 1.2 mm. is written as 1200.
Q(d) = Depth of first cut, radial value, in microns. E.g., 0.4 mm. is written as 400.
F = Lead of thread

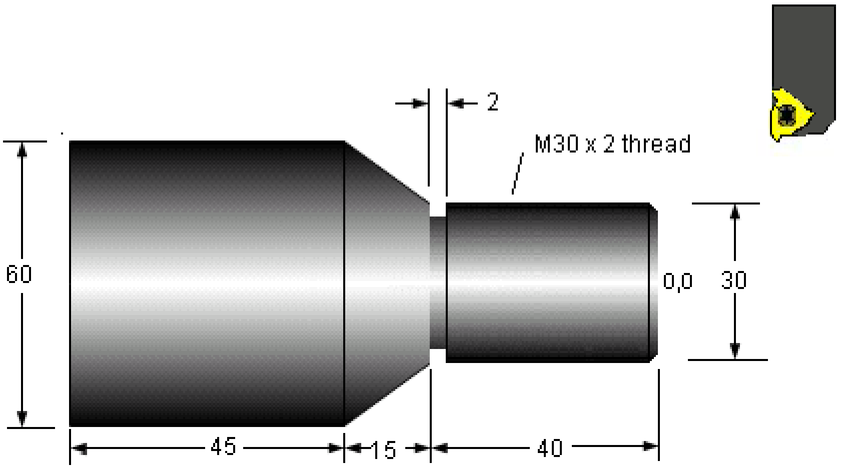
Depth of cut calculation for equal area:



Threads are always cut with multiple cuts. In a V thread if the depths of cut are equal, the cross sectional area of successive cuts increases and hence the cutting load too increases. This may result in bad thread quality or insert breakage. To ensure that this does not happen, the depths of cut are determined so that the cross sectional area is constant – the depth of cut reduces for each successive cut. The depth of each cut is determined using the total thread depth and the depth of the first cut.

If the depth of cut becomes too low the tool just 'rubs' against the part material and does not cut it. Specifying the dmin. Value prevents this from happening. If the depth of cut is smaller than the d min. value, it is clamped at this value.

Example



```
-----
-----
G00 X34.0 Z2.0
G76 P020060 Q100 R0
G76 X30.0 Z-39.0 R0 P1200 Q0.5 F2.0
G00 X100.0 Z50.0
-----
-----
```

Typical G and M codes

G codes

G codes on a machine are decided by its controller's programming format. Machines of different makes with the same controller will have the same set of G codes.

Sample list of G codes:

G00	Rapid traverse
G01	Linear interpolation (feed)
G02	Circular interpolation CW
G03	Circular interpolation CCW
G04	Dwell
G20	Inch unit
G21	Metric unit
G28	Automatic zero return (returns to a fixed position, typically for tool change)
G30	2nd reference point return
G32	Thread cutting (single motion)
G40	Tool nose radius compensation cancel
G41	Tool nose radius compensation left
G42	Tool nose radius compensation right
G50	Limiting spindle speed setting
G70	Finishing cycle
G71	Stock removal in turning
G72	Stock removal in facing
G73	Pattern repeating
G74	Peck drilling on Z axis / Face grooving
G75	Peck drilling on X axis / Grooving
G76	Threading cycle
G90	Single cut turning cycle
G92	Single cut threading cycle
G94	Single cut facing cycle
G98	Feed per minute
G99	Feed per revolution
G96	Constant surface speed
G97	Constant spindle speed

M-codes

Most M codes activate machine functions like the coolant, spindle, etc. These are decided by the machine manufacturer, and depend on the features that are available on the machine. E.g., a machine with a tailstock will have M codes for tailstock in/out. A few (like M00, M01, M02, M98, etc.in the list below) are fixed and based on the controller.

Sample list of M codes:

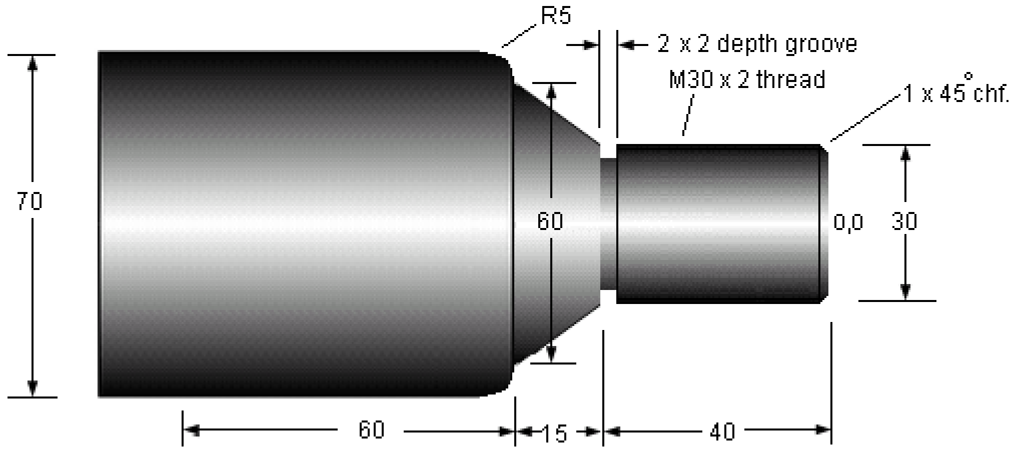
M00	Program stop
M01	Optional program stop
M02	Program end
M03	Spindle ON clock wise (CW)
M04	Spindle ON counter clock wise (CCW)
M05	Spindle stop

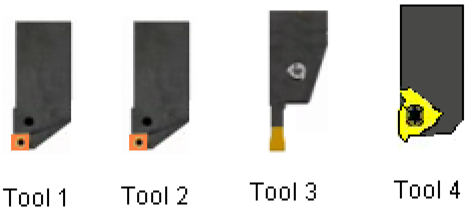
M06	Tool change
M08	Coolant ON
M09	Coolant OFF
M30	End of program and reset to start
M98	Sub program call
M99	Sub program end

Full sample program

This is a sample program for a part with multiple operations – Rough turning, Finish turning, Grooving and Threading. It shows how a full program is put together.

The blocks just before a tool change typically have a number of codes specific to a particular machine, specifically the type of its tool changer and its tool change position. They may appear odd and unfamiliar, and may be ignored for the purpose of understanding this program. The program has been generated by a CAD/CAM software that automatically considers the tool nose radius during contouring. Coordinates in finish turning are calculated with nose radius compensation, and will therefore not match the part coordinates.





Raw material : 80 dia. Bar, 2 mm. extra material for facing.

```
%
O1234
T0000
G0 X150.0 Z200.0
N1 T0101 (PCLNL 2525M12 R0.8)
G50 S3000
G96 S247 M03
(ROUGH FACE)
G0 X90. Z4. M07
X84.
G72 W3. R0.5
G72 P25 Q40 U0. W0.2 F0.3
N25 G0 Z0.
N30 G01 X80. Z0.
```

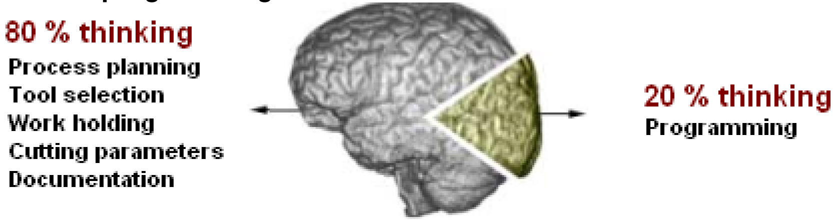
```
N35 X0.
N40 Z2.
G0 X90.
(ROUGH TURN)
Z2.2
X84.
G71 U3. R0.5
G71 P45 Q95 U0.4 W0.2 F0.3
N45 G00 X26.
N50 G01 X26. Z0.2
N55 Z0.
N60 X30. Z-2.
N65 Z-38.
N70 Z-40.
N75 X60. Z-55.
N80 G03 X70. Z-60. I0. K-5.
N85 G01 Z-115.
N90 X80.
N95 X84. Z-115.
G0 X90.
M09
M05
T0000
G0 X150.0 Z200.0
N2 T0202 (PCLNL 2525M16 R0.4)
G50 S3000
G96 S296 M03
(FINISH FACE)
G0 X90. Z0. M07
X30.566
G01 G41 X26.566 F0.24
X0.
Z2.2
G0 G40 X90.
(FINISH TURN)
Z1.883
X25.766
G42 Z2.
G01 X26. F0.24
Z0.
X30. Z-2.
Z-40.
X60. Z-55.
G03 X70. Z-60. I0. K-5.
G01 Z-115.
X74.4
G0 G40 X90.
M09
M05
T0000
G0 X150.0 Z200.0
N3 T0303 (25X25, 2.00W, 0.20R, 08DEPTH, LH)
G50 S3000
G96 S258 M03
(GROOVE)
```

```
G0 X34. Z-40. M07
G01 X26. F0.2
G04 X0.046
G0 X34.
M09
M05
T0000
G0 X150.0 Z200.0
N4 T0404 (THREAD 16 X 16, 60 DEG., DEPTH 3.0, LH)
G97 S1645 M04
(THREADING)
X90. Z-1. M08
X34.
G76 P020060 Q0 R0
G76 X27.44 Z-39. R0.  P1280 Q404 F2.
X90.
M09
M05
T0000
G0 X150.0 Z200.0
M30
%
```

Drawing to machined part - steps

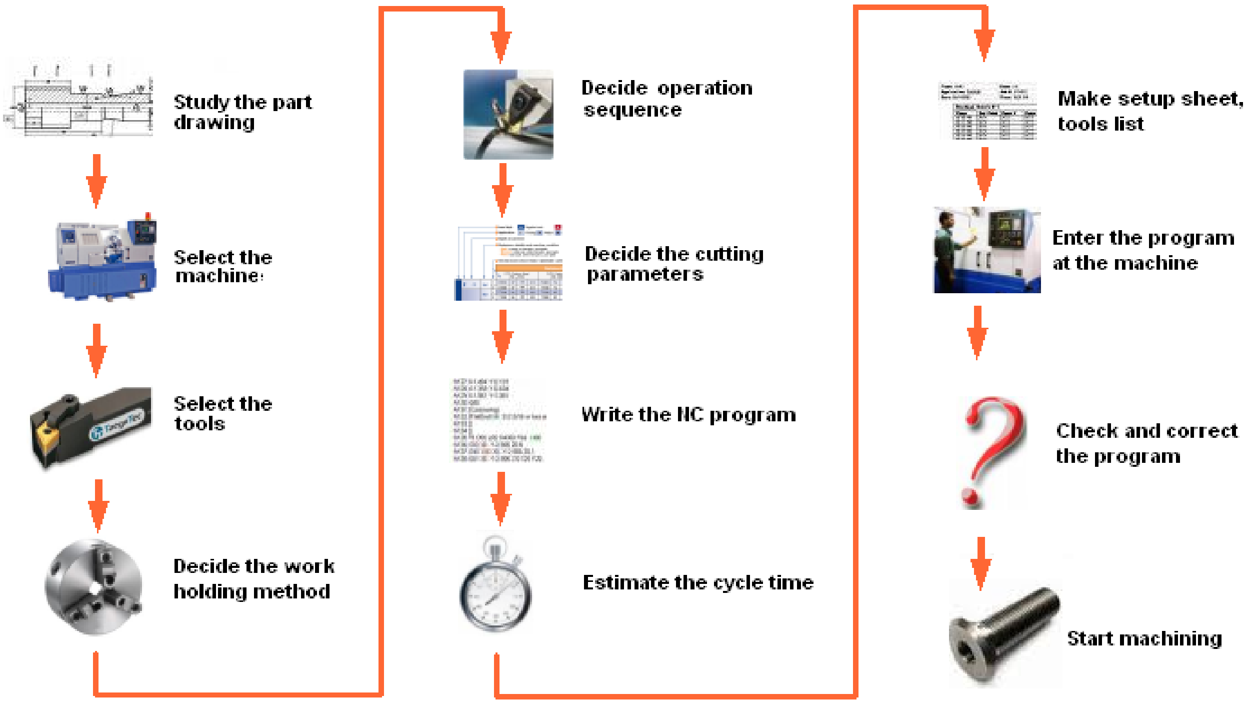
A common misconception is that writing the NC program is the main job involved in CNC machining, and that knowing how to write a program is enough to turn out parts from a CNC machine. The fact is that only a small amount of thinking is involved in actually writing the program.

Effort in programming



The complete sequence of steps involved in generating a machined part from the drawing is a complex process.

Steps in machining a part


Study the part drawing
Select the machine
Select the tools
Decide the work holding method
Decide operation sequence
Decide the cutting parameters
Write the NC program
Estimate the cycle time
Make setup sheet, tools list
Enter the program at the machine
Check and correct the program
Start machining

About Cadem

Cadem is a 20 year old company, India's leading provider of productivity solutions for the manufacturing industry. Its range of industrial products for CAD/CAM and DNC are enabling shop floors to improve the productivity of their CNC machines. Many of India's best known CNC shop floors use the products. Cadem's products are exported to 30 countries, in 4 languages. Premier CNC machine tool companies endorse and recommend Cadem's software to their customers.

Cadem's CNC educational products are the first-choice for Engineering Institutes in India. The products form the mainstay of CAD/CAM labs in most of India's best known institutes, and in in-house tech centers and training centers of industries. The Productivity Institute of the IMTMA (Indian Machine Tool Manufacturers' Association) uses Cadem software and know-how for training engineers from industry around the country.

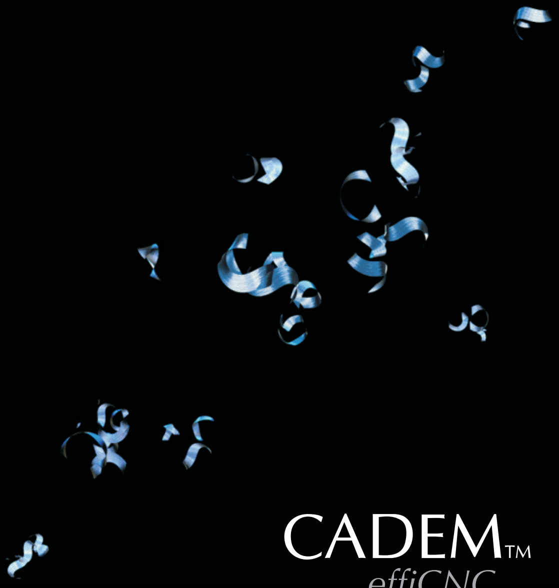
Cadem has won the coveted CMTI-PMT Trust award for Best Design Innovation Award 3 times and was named as one of the TOP 100 Innovators in 2007 by NASSCOM.

Developed in India with pride, used worldwide.

About the author

G.V. Dasarathi has a BE in Mechanical Engineering from NIT, Surat and an M.Tech. in Production Technology from IIT, Kharagpur. He has been involved with CNC machining for the past 25 years.He started his career in Bharat Earth Movers Ltd., KGF, where he was in charge of the CNC cell machining. His next stint was in the Government Tool Room and Training Centre, Bangalore and he was in charge of the CAD/CAM programming cell and CNC machining cell.

Later in 1990, he founded Cadem Technologies Pvt.Ltd. At CADEM he is responsible for the design and architecture of the award winning CNC software products for CAD/CAM, DNC and CNC education.



CADEM™
effiCNC

Cadem Technologies Pvt Ltd
264/75, 36th cross, 8th Block Jayanagar
Bangalore – 560 070, INDIA
Phone: +91 80 2663 4767 Fax: +91 80 2244 2732
Website: www.cadem.com Email: sales@cadem.com