COMPUTER AIDED MANUFACTURING LABORATORY

LAB CODE: 8ME5A

FOR IV B.TECH- VIII SEM –MECHANICAL ENGINEERING

DEPARTMENT OF MECHANICAL ENGINEERING

BHARTIYA INSTITUTE OF ENGINEERING & TECHNOLOGY

NEAR SANWALI CIRCLE, JAIPUR-BIKANER BYE-PASS, NH-11 PO-BAJAJ GRAM SANWALI, SIKAR, RAJASTHAN 332001

COMPUTER AIDED MANUFACTURING (CAM)

Computer-aided Manufacturing (CAM) is the term used to describe the use of computerized systems to control the operations at a manufacturing plant. These computerized systems assist manufacturers in various operations such as planning, transportation, management, and storage. CAM helps manufacturers improve their time to market capabilities, and create precise dimensions. In this post, you will understand how Computer-aided Manufacturing is transforming the landscape of manufacturing.

Brief Introduction to Computer-aided Manufacturing

As a process, CAM is used after Computer-aided Design (CAD) or Computer-aided Engineering (CAE). The model designed using CAD is sometimes used as the CAM input. This is why it is referred as CAD-CAM. The functions of this combination is divided into two main categories:

• Manufacturing Control: : In the process, the computer is used to manage and control the physical operations of the manufacturing plant. These may include:

Shop Floor Controlling

Process Monitoring and Controlling

Inventory Controlling

Production Delivery Controlling

ROLE OF COMPUTERS IN DESINGING

CAM is a team, which means the Computer Aided Manufacturing. CAM can be defined, as the use of the computer systems to plan, manage and control the operation of manufacturing interface with the plant is production resources.

STUDY OF CNC MACHINES AND PART PROGRAMMING

Introduction:

CAM:

CAM is a team, which means the Computer Aided Manufacturing. CAM can be defined, as the use of the computer systems to plan, manage and control the operation of manufacturing interface with the plant is production resources.

CNC machines:

CNC stands for Computer Numerical Controlled machine is on NC system that utilizes a delectated stored program computer to perform some are all the basic numerical control functions. Part programming are entered and stored in computer memory CNC offers additional flexibility and computation capacity. New system options can be incorporated into the CNC controller simple by programming the unit.

Classification of CNC machines:

1) MACHINING CENTRE

a) Horizontal spindle machining center

b) Vertical spindle machining center

c) Universal machines.

2) TURNING CENTER CNC LATHES

a) Horizontal machines

b) Chucking machines

c) Shaft machines

d) Universal machines

3) CNC DRILLING AND MILLING MACHINES

4) CNC GRINDING MACHINE

a) Surface grinding

b) Cylindrical grinding

c) Tool and cutter grinder

d) Profile grinder

5) GEAR HOBBING MACHINES

6) PUNCHING AND FORMING MACHINES

A CNC program can have two types of errors:

Syntax errors

Logical errors

Syntax errors are grammatical errors, like using improper codes or omitting codes. On a machine, these would result in a stoppage of execution because the machine is unable to understand the program. Logical errors are ones that result in an improper part shape, like using G00 (rapid) instead of G01 (feed). On a machine there would be no stoppage since it understands the program, but the part and the machine may be damaged or cause a collision. The best way to learn manual CNC programming is by actually writing programs and getting instant feedback on their correctness.

Programming is learnt by repeatedly trying, by trial and error. CNC simulation software that is designed specifically for learners will enable you to teach manual programming.

Therefore, good CNC simulation software must have these features:

1. Editor for the student to enter the program.

2. Software should support canned cycles and subprograms

3. Automatic syntax checking and display of errors, for syntax errors.

4. Graphical tool path simulation, for logical errors.

Co-ordinate system:

In order for the part programmer to plan the sequence of positions of moments of the cutting tool. Machine to the WIP, it is memory to establish a standard axis system by which the relative positions can be specified. Two axis “X & Y” are defined in the plane of the table, the „z‟ axis in perpendicular. In this plane of the table the vertical motion of the spindle controls the „z‟ direction. The positive and negative directions motion of the tool.

Programming methods:

1) Incremental method

2) Absolute method

1) Incremental Method:

In this method, every point is considered as origin from this point, the values are calculated, for example

Point A = (10, 0)

Point B = (20, 0)

Point C = (10, 0)

Point D = (20, 0)

2) Absolute method:

In this absolute system, the set point is considered as a reference point as from that point, all the values are calculated, for example

Point A = (10, 0)

Point B = (20, 0)

Point C = (30, 0)

Point D = (50, 0)

Programming methods:

In CNC machines program are programmed by two methods.

1) Manual part programming

2) Computer assisted part programming

1) Manual part programming:

To prepare a part program using the manual method, the programmer writes the machining instruction is must be hence, menu script the instruction is must be prepared in a very precise manner because the typist prepare the NC type directory from the Manu script some in various form expending on the machine tool and tape format used.

2) Computer assisted part programming:

In the more complicated point and in contour application using manual part programming because an extremely tedious basic and subject to errors. It is must more appropriate to employ the high speed digital computer to assist the part programming languages system have been developed to perform automatically most of the calculation which the programmer would otherwise be forced to do. It several time and more efficient part program.

PREPARATORY FUNCTIONS (G-CODE):

Preparatory functions are used for cutting operations like facing, turning, thread cutting, drilling, etc.,

MISCELLANEOUS FUNCTIONS (M-CODE) :

Miscellaneous functions are used for other than cutting operations like spindle ON/OFF, coolant ON/OFF, tool change, etc.,

COMMON G CODES AND M CODES FOR CNC MACHINE CONTROLS

CNC G codes

G00 - Positioning at rapid speed; Mill and Lathe

G01 - Linear interpolation (machining a straight line); Mill and Lathe

G02 - Circular interpolation clockwise (machining arcs); Mill and Lathe

G03 - Circular interpolation, counter clockwise; Mill and Lathe

G04 - Mill and Lathe, Dwell

G09 - Mill and Lathe, Exact stop

G10 - Setting offsets in the program; Mill and Lathe

G12 - Circular pocket milling, clockwise; Mill

G13 - Circular pocket milling, counterclockwise; Mill

G17 - X-Y plane for arc machining; Mill and Lathe with live tooling

G18 - Z-X plane for arc machining; Mill and Lathe with live tooling

G19 - Z-Y plane for arc machining; Mill and Lathe with live tooling

G20 - Inch units; Mill and Lathe

G21 - Metric units; Mill and Lathe

G27 - Reference return check; Mill and Lathe

G28 - Automatic return through reference point; Mill and Lathe

G29 - Move to location through reference point; Mill and Lathe

G31 - Skip function; Mill and Lathe

G32 - Thread cutting; Lathe

G33 - Thread cutting; Mill

G40 - Cancel diameter offset; Mill. Cancel tool nose offset; Lathe

G41 - Cutter compensation left; Mill. Tool nose radius compensation left; Lathe

G42 - Cutter compensation right; Mill. Tool nose radius compensation right; Lathe

G43 - Tool length compensation; Mill

G44 - Tool length compensation cancel; Mill (sometimes G49)

G50 - Set coordinate system and maximum RPM; Lathe

G52 - Local coordinate system setting; Mill and Lathe

G53 - Machine coordinate system setting; Mill and Lathe

G54~G59 - Work piece coordinate system settings #1 to #6; Mill and Lathe

G61 - Exact stop check; Mill and Lathe

G65 - Custom macro call; Mill and Lathe

G70 - Finish cycle; Lathe

G71 - Rough turning cycle; Lathe

G72 - Rough facing cycle; Lathe

G73 - Irregular rough turning cycle; Lathe

G73 - Chip break drilling cycle; Mill

G74 - Left hand tapping; Mill

G74 - Face grooving or chip break drilling; Lathe

G75 - OD groove pecking; Lathe

G76 - Fine boring cycle; Mill

G76 - Threading cycle; Lathe

G80 - Cancel cycles; Mill and Lathe

G81 - Drill cycle; Mill and Lathe

G82 - Drill cycle with dwell; Mill

G83 - Peck drilling cycle; Mill

G84 - Tapping cycle; Mill and Lathe

G85 - Bore in, bore out; Mill and Lathe

G86 - Bore in, rapid out; Mill and Lathe

G87 - Back boring cycle; Mill

G90 - Absolute programming

G91 - Incremental programming

G92 - Reposition origin point; Mill

G92 - Thread cutting cycle; Lathe

G94 - Per minute feed; Mill

G95 - Per revolution feed; Mill

G96 - Constant surface speed control; Lathe

G97 - Constant surface speed cancel

G98 - Per minute feed; Lathe

G99 - Per revolution feed; Lathe

CNC M Codes

M00 - Program stop; Mill and Lathe

M01 - Optional program stop; Lathe and Mill

M02 - Program end; Lathe and Mill

M03 - Spindle on clockwise; Lathe and Mill

M04 - Spindle on counterclockwise; Lathe and Mill

M05 - Spindle off; Lathe and Mill

M06 - Tool change; Mill

M08 - Coolant on; Lathe and Mill

M09 - Coolant off; Lathe and Mill

M30 - Program end, return to start; Lathe and Mill

M97 - Local sub-routine call; Lathe and Mill

M98 - Sub-program call; Lathe and Mill

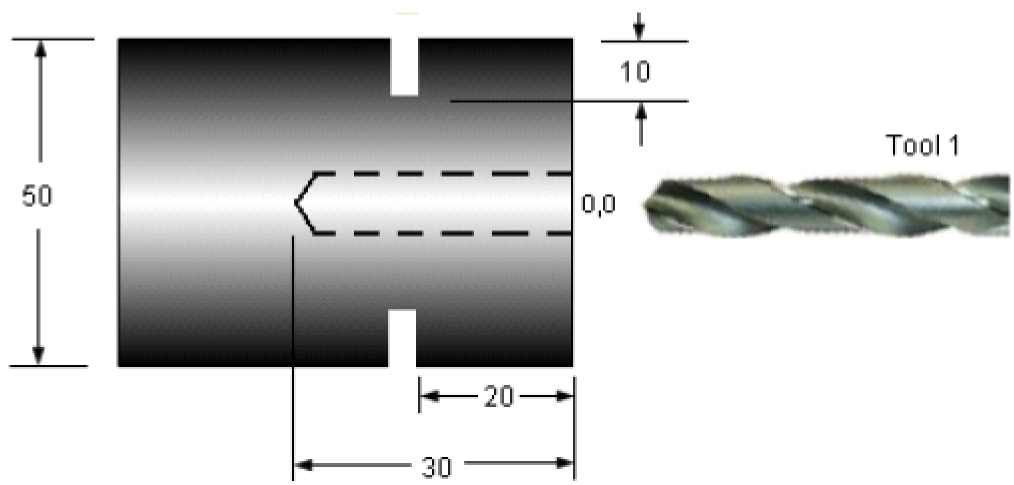
M99 - End of sub program; Lathe and Mill

DRILLING OPERATION

AIM:- To write and simulate part program of the given component for the required dimensions by using CNC trainer lathe.

MATRIAL REQUIREMENT :- Aluminum Rod of 30 mm diameter and 80 mm length.

PART PROGRAM:-



N010G28U0W0;

N020T0101;

N030G96M3S600;

N040G00X0.0Z5.0;

N040G74R2;

N040G74Q3000Z60.0F2.0;

N040G28U0W0;

SN040M30;

RESULT:

The program is written and simulated and stored in System No…and file name as…

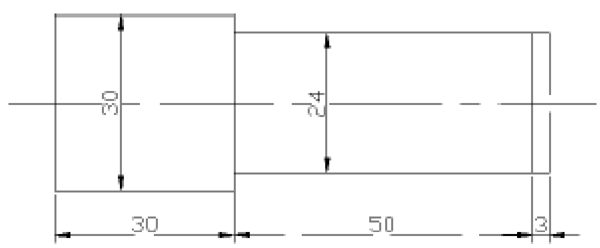
PLAIN TURNING AND FACING OPERATION

AIM: To write the part program for component shown in Fig. 01. Assuming the work piece is Aluminum and the speed is 1200 rpm, feed 20 mm/min and maximum depth of cut is 1 mm.

a. With Canned cycle
b. Without Canned cycle.

MATERIAL REQUIRED: Aluminum Rod of 30 mm diameter and 80 mm length

PART PROGRAMING:-



WITHOUT CANNED CYCLE

N010G28U0.0W0.0;

N020T0101M08;

N030G97S2000M03;

N020G00X52.0Z10.0

N020G00Z6.0;

N020G01X-1.5F0.1;

N020G520Z7.0

N020G00Z5.0;

N020G01X-1.5;

```
N020G00X52.0Z6.0;
N020G00Z4.0;
N020G01X-1.5;
N020G00X52.0Z5.0;
N020G00Z3.0;
N020G01X-1.5;
N020G00X52.0Z4.0;
N020G00Z2.0;
N020G01X-1.5;
N020G00X52.0Z3.0
N020G00Z1.0;
N020G01X-1.5;
N020G0052.0Z0.5
N020G00Z0.0;
N020G01X-1.5F0.2;
N020G28U0.0W0.0;
N020M30;
```

RESULT:

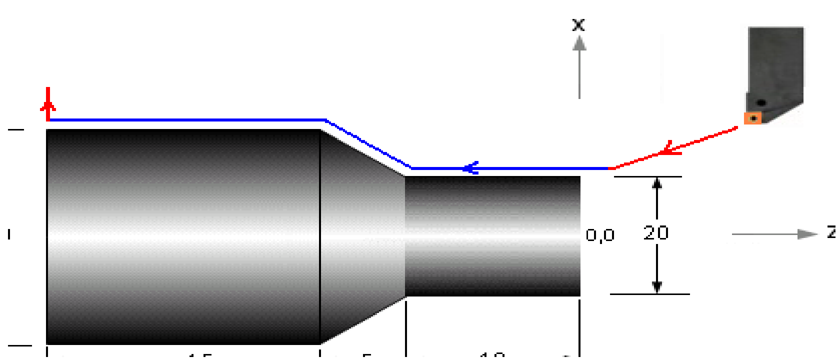
The program is written and simulated and stored in System No…and file name as…

STEP TURNING OPERTAION

AIM: - To write the part program for the component shown in fig 2. assuming work piece as AL the speed is 1200rpm, feed given is 20mm/min.

With canned cycle.

MATERIAL REQUIRED: Aluminum Rod of 30 mm diameter and 80 mm length.
PART PROGRAM:-



N010G28U0.0W0.0;
N020G92S2500;
N030T0101;
N040G96S150M04;
N050G00X57.0Z2.0;
N060G73U1.0R1.0;
N070G73P40Q50F0.2;
N080G01X25.0Z0.0;

N090Z-60.0;
N100G00X57.0Z2.0;
N110G28U0.0W0.0;
N120M30;

RESULT:-

Canned Cycles – single cut

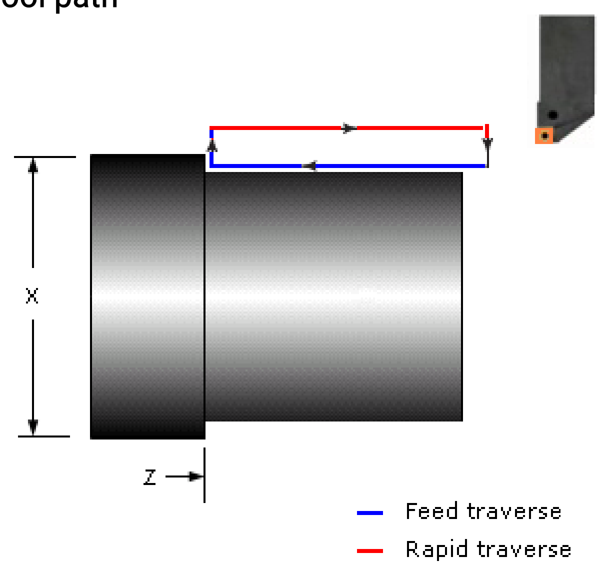
A single cut canned cycle executes a sequence of motions required to perform a cut – rapid approach to the start position, cutting motion, and rapid departure. A single block replaces 4 motions - 1 cutting and 3 rapid. Operations normally involve the removal of material in multiple cuts, so these cycles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

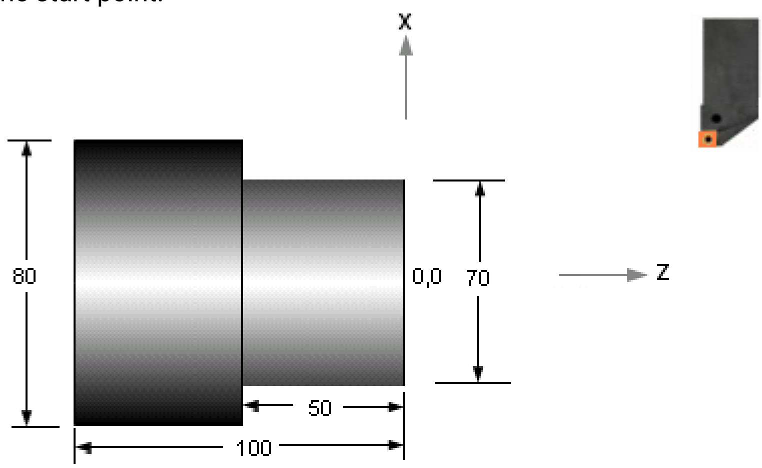


Format

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.



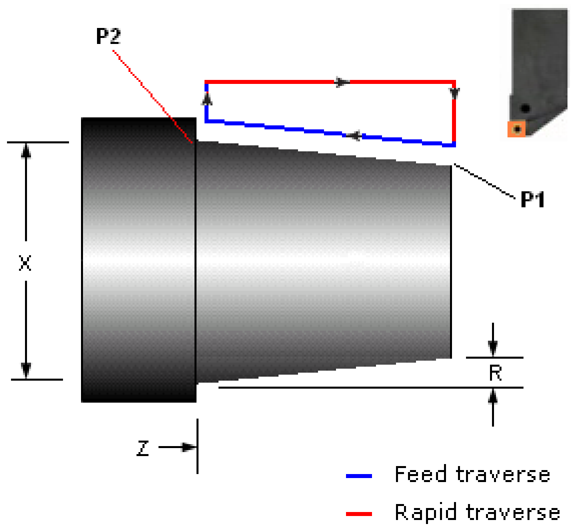
Raw material is a cylinder of 80 diameter.

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

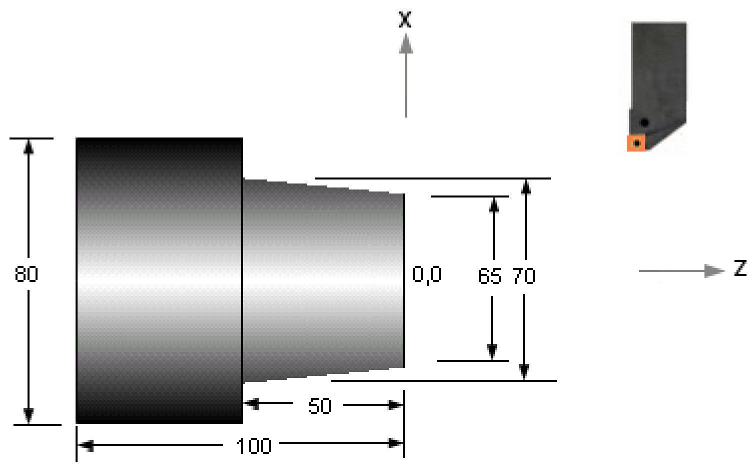
The cut starts at point P1, ends at point P2.

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 -----

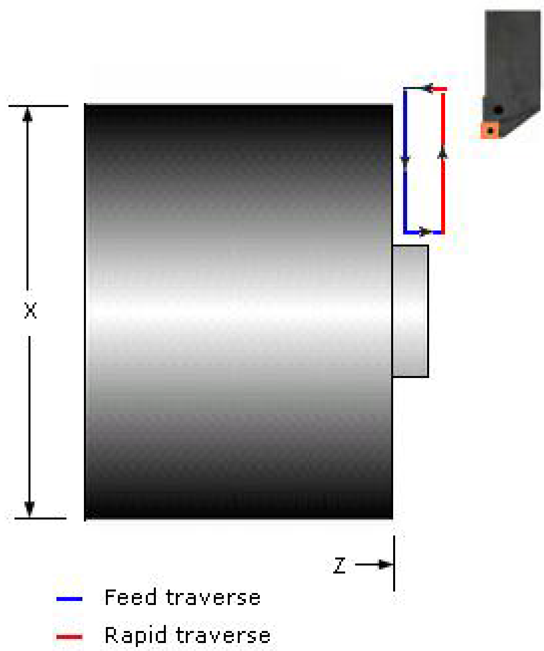
Note that the R value has a small approximation here since the cut is starting at Z1.0 instead of Z0.

Facing cycle - G94

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

Straight facing

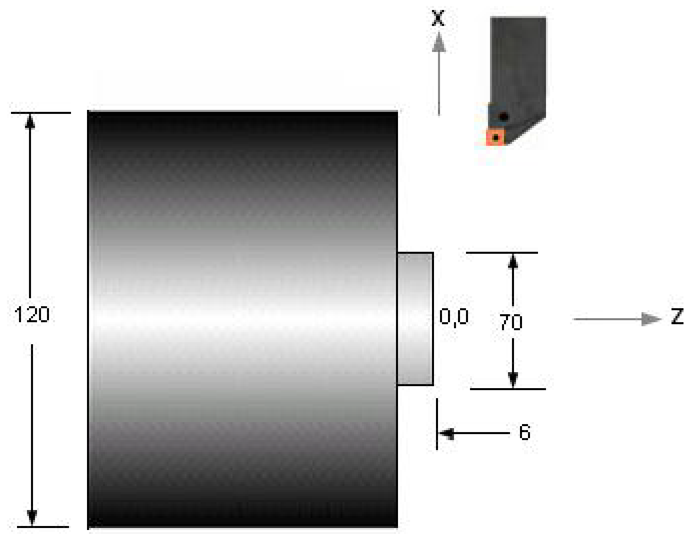
Tool path



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

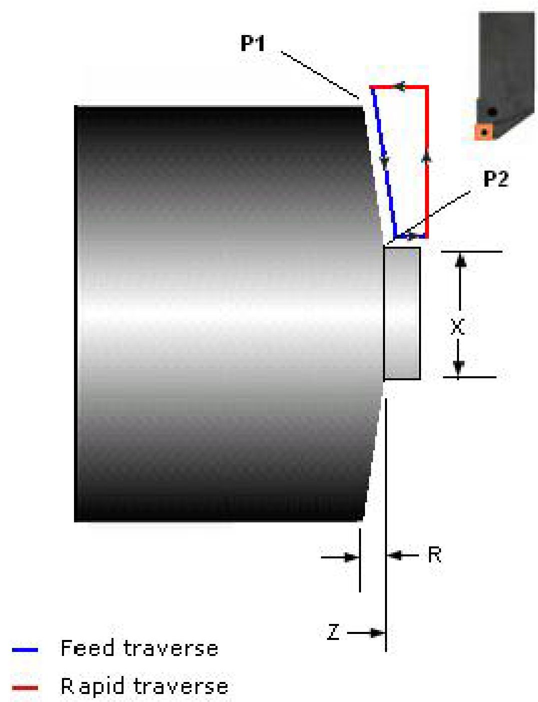


Raw material is a cylinder of 120 diameter.

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

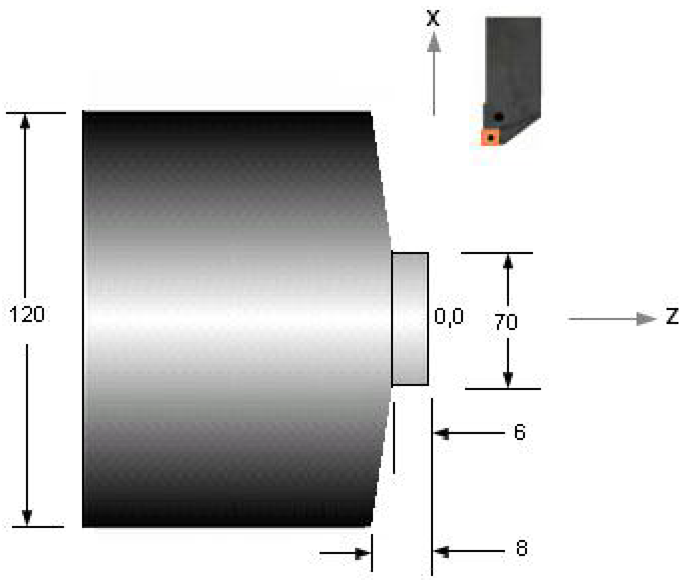


Format
G94 X_ Z_ R_ F_

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



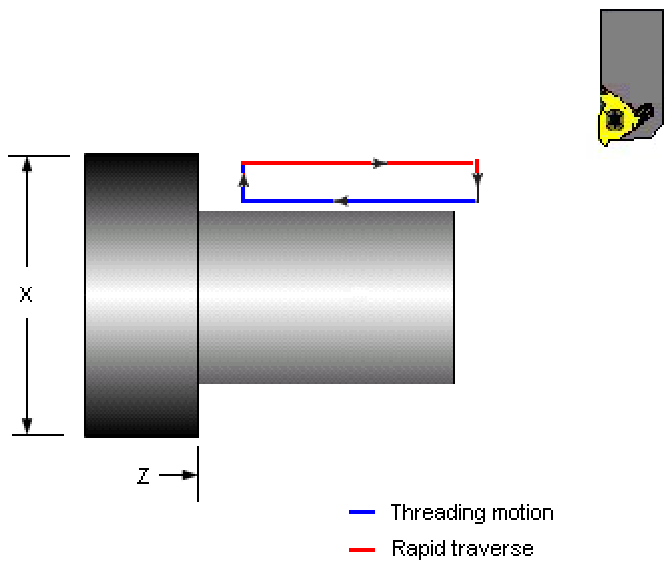
Raw material is a cylinder of 120 diameter.

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.

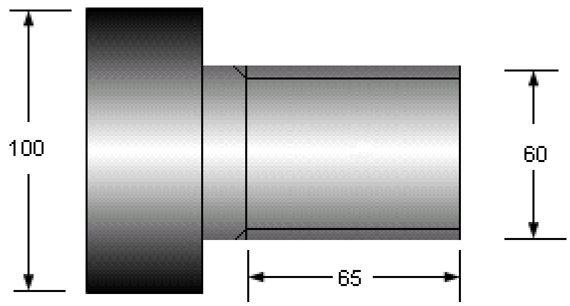
Tool path



Format

G92 X_ Z_ F_

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

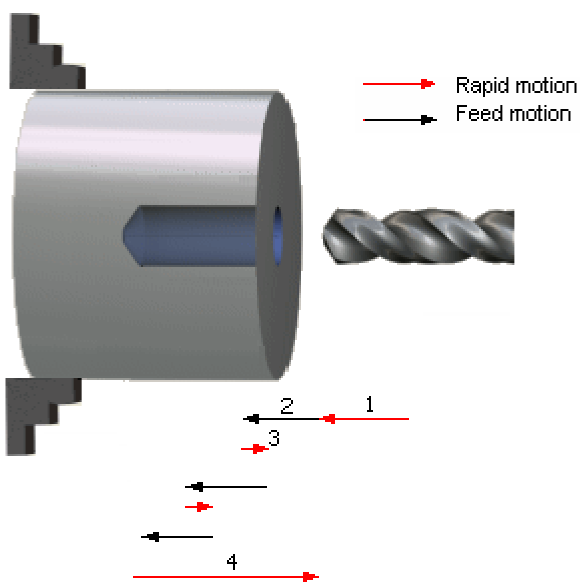


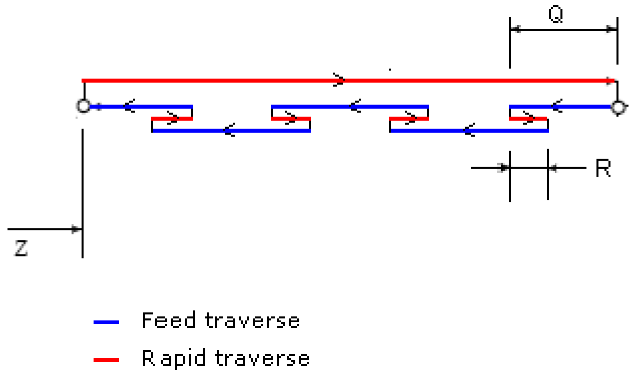
G00 X60.0 Z2.0
G92 X59.0 Z-65.0 F3.0 X58.4

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

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

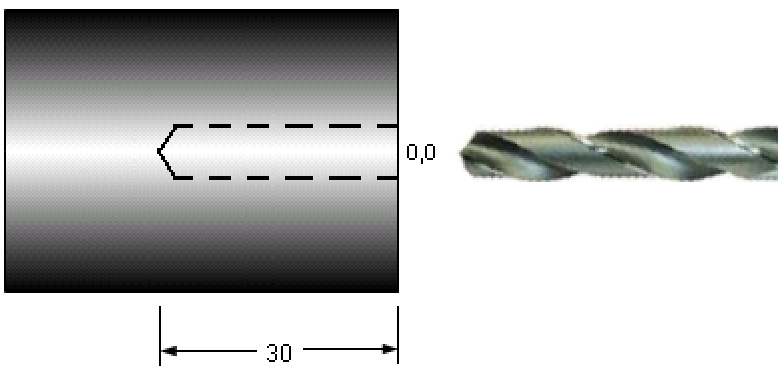




G74 R_
G74 Z_ Q_ F_
R = Retract amount at each peck

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.

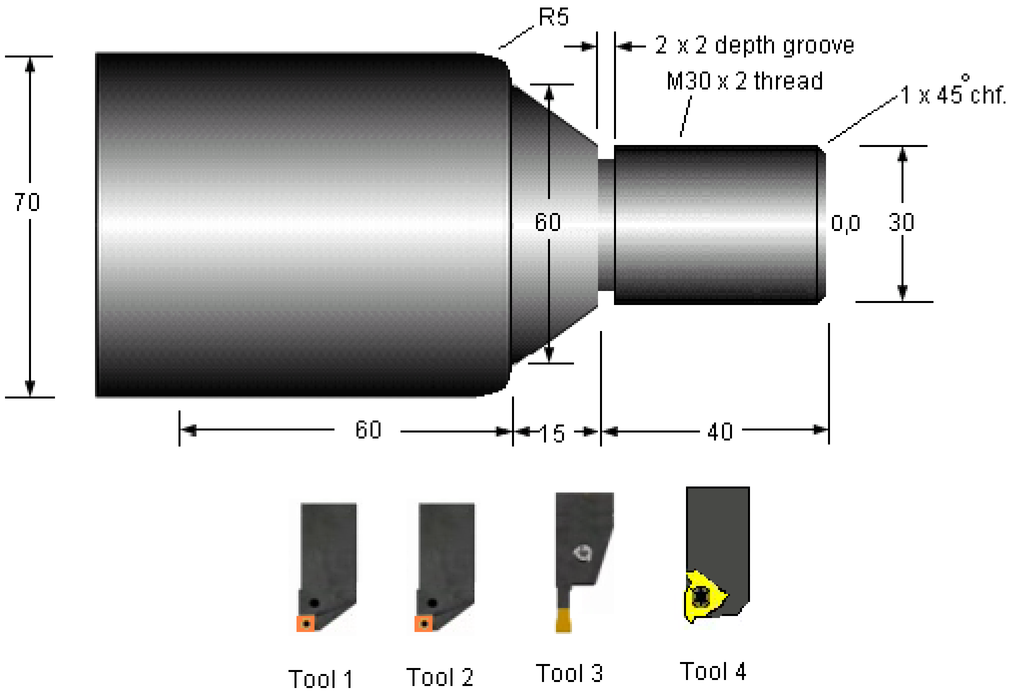
Example



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

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
%
```

RESULT:-