

## STUDY OF G CODES M CODES USED IN CNC LATHE HMT CNC- T70

### G CODES

G00	RAPID TRAVERSE POSITIONING
G01	LINEAR INTERPOLATION
G02	CIRCULAR INTERPOLATION CLOCKWISE
G03	CIRCULAR INTERPOLATION COUNTERCLOCKWISE
G04	DWELL
G25	PROGRAM TRANSFER FROM SYSTEM.
G26	PROGRAM TRANSFER TO SYSTEM
G33	THREADING
G37	SUBROUTINE CALL
G38	SUBROUTINE START
G39	SUBROUTINE END
G65	CASSETTE LOAD
G66	CASSETTE SAVE
G67	CASSETTE SEARCH
G70	INCH MODE
G71	METRIC MODE
G81	TURNING CYCLE
G84	THREADING CYCLE
G83	DRILLING WITH DWELL
G90	ABSOLUTE PROGRAMMING
G91	INCREMENTAL PROGRAMMING
G92	PROGRAM PRESET
G94	FEED PER MINUTE
G95	FEED PER REVOLUTION

### M CODES

M00	PROGRAM STOP
M02	PROGRAM END
M30	PROGRAM END AND REWIND

## VMC 200 T MACHINING CENTRE

### G CODES

G00	RAPID TRAVEERSEPOSITIONING
G01	LINEAR INTERPOLATION
G02	CIRCULAR INTERPOLATION CLOCKWISE
G03	CIRCULAR INTERPOLATION COUNTERCLOCKWISE
G04	DWELL
G17	INTERPOLATION X-YPLANE
G18	INTERPOLATION X-ZPLANE
G19	INTERPOLATION Y-ZPLANE
G25	PROGRAM TRANSFER FROMSYSTEM
G26	PROGRAM TRANSFER TO SYSTEM
G37	SUBROUTINE CALL
G38	SUBROUTINESTART
G39	SUBROUTINE END
G40	CUTTER COMPONSATIONCANCEL
G41	CUTTER COMPONSATION LEFT
G42	CUTTER COMPONSATION RIGHT
G65	CASSETTE LOAD
G66	CASSETTE SAVE
G67	CASSETTESEARCH
G70	INCH MODE
G71	METRIC MODE
G80	CANNEDCYCLECANCEL
G82	DRILLING CYCLE
G83	DRILLING WITH DWELL CYCLE
G84	PECKDRILL/WITHDRAWLCYCLE
G85	BORING CYCLE
G86	PCD DRILLING CYCLE
G88	RECTANGULAR MILLINGCYCLE
G89	CIRCULAR MILLING CYCLE
G90	ABSOLUTE PROGRAMMING
G91	INCREMENTALPROGRAMMING
G92	PROGRAM PRE SET

### M CODES

M00	PROGRAM STOP
M02	END OF PROGRAM
M03	SPINDLESTARTCW
M04	SPINDLESTARTCCW
M05	SPINDLE STOP
M06	TOOL CHANGE PROMPTING
M30	END OF PROGRAM&REWIND

## G-CODES

G00: POSITIONING  
G01: LINEAR INTERPOLATION  
G02: CIRCULAR INTERPOLATION CW  
G03: CIRCULAR INTERPOLATION CCW  
G04: DWELL  
G10: OFF SET VALUE SETTING (O)  
G20: INCH  
G21: METRIC  
G22: STORED STROKE CHECK ON (O)  
G23: STORED STROKE CHECK OFF (O)  
G25: SPINDLE SPEED DETECT OFF  
G26: SPINDLE SPEED DETECT ON  
G27: REF. POINT RETURN CHECK  
G28: REF. POINT RETURN  
G30: 2<sup>ND</sup> (3,4) REF. POINT RETURN  
G31: SKIP CUTTING  
G33: THREAD CUTTING  
G34: VARIABLE LEAD THREAD CUTTING  
G36: AUTO TOOL OFF STE X-AXIS (O)  
G37: AUTO TOOL OFF SET Z-AXIS (O)  
G40: TOOL NOSE RADIUS  
COMPENSATION CANCEL  
G41: TOOL NOSE RADIUS  
COMPENSATION LEFT (O)  
G42: TOOL NOSE RADIUS  
COMPENSATION RIGHT (O)  
G53: SUPPRESSION OF ZERO OFFSET  
G54: SETTABLE ZERO OFFSET  
G55: SETTABLE ZERO OFFSET  
G56: SETTABLE ZERO OFFSET  
G57: SETTABLE ZERO OFFSET  
G65: MACRO CALL (O)  
G68: DOUBLE TURRETS MIRROR ON (O)  
G69: DOUBLE TURRETS MIRROR OFF  
G70: FINISHING CYCLE (O)  
G71: ROUGH CUTTING (TURNING)  
(O) G72: ROUGH CUTTING  
(FACING) (O) G73: ROUGH CUTTING  
(PROFILE) (O) G74: GROOVING  
(FACING) (O)  
G75: GROOVING TURNING (O)  
G76: THREAD CUTTING CYCLE (MULTI)  
(O) G77: TURNING CYCLE  
G78: THREAD CUTTING  
CYCLE G79: FACING CYCLE  
G90: ABSOLUTE

G91: INCREMENTAL  
G92: COORDINATE SYSTEM SETTING OR  
MAX. SPINDLE SPEED SETTING  
G94: PER MINUTE FEED  
G95: PER REVOLUTION FEED  
G96: CONSTANT SURFACE SPEED (O)  
G97: REVOLUTION PER MINUTE (RPM)

## M-CODES

M00: PROGRAM STOP  
M01: OPTIONAL STOP  
M02: PROGRAM END AND RESTART  
M03: SPINDLE ROTATION CW  
M04: SPINDLE ROTATION CCW  
M05: SPINDLE STOP  
M07: COOLANT ON  
M09: COOLANT OFF  
M10: CHUCK DECLAMP  
M11: CHUCK CLAMP  
M16: CHUCK 1D SELECTION  
M18: CHUCK 0D SELECTION  
M19: SPINDLE ORIENTATION  
M20: SPINDLE ORIENTATION CANCEL  
M30: END OF MAIN PROGRAM  
M32: TAILSTOCK QUIL FORWARD  
M33: TAILSTOCK QUIL RETRACT  
M35: PARTS CATCHER RETRACT  
M46: DOOR OPEN  
M47: DOOR CLOSE  
M50: SPINDLE LOCK  
M51: SPINDLE UNLOCK  
M78: STEADY REST OPEN  
M79: STEADY REST HOLD  
M82: TAIL STOCK BODY FWD /  
UNCLAMP  
M83: TAILSTOCK BODY RET / CLAMP  
M84: TOUCH PROBE ARM FORWARD  
M85: TOUCH PROBE ARM RETRACT  
M98: SUB PROGRAM CALL  
M99: SUB PROGRAM END

## L Mill 55 VERTICAL MACHINIG

### CENTRE G CODES

G00	Rapid positioning	G74	Reverse tapping cycle
G01	Linear interpolation	G76	Fine boring cycle
G02	Circular interpolation CW	G80	Canned cycle cancel
G03	Circular interpolation CCW	G81	Drilling cycle
G04	Dwell	G82	Counter boring cycle
G09	Exact stop	G83	Peck drilling cycle
G10	Data setting	G84	Tapping cycle
G20	Inch input	G85	Boring cycle
G21	Metric input	G86	Boring cycle
G28	Zero return	G87	Back boring cycle
G30	Second Reference Point	G88	Boring cycle
G40	Tool nose radius compensation cancel	G89	Boring cycle
G41	Tool nose radius compensation left	G90	Absolute command
G42	Tool nose radius compensation right	G91	Incremental command
G43	Tool length offset	G98	Initial point level return (canned cycle)
G52	Location coordinate system	G99	Point R level return
G53	Machine coordinate system selection		
G54	Work coordinate system 1 selection		
G55	Work coordinate system 2 selection		
G56	Work coordinate system 3 selection		
G57	Work coordinate system 4 selection		
G58	Work coordinate system 5 selection		
G59	Work co ordinate system 6 selection		
G73	Peck drilling cycle		

## L Mill 55 VERTICAL MACHINIG CENTRE

### M CODES

M00	Temporary program stop
M01	Optional stop
M02	Program end
M03	CW spindle rotation
M04	CCW spindle rotation
M05	Spindle stop
M06	Tool change
M07	Secondary coolant ON
M08	Coolant pump ON
M10	4th axis clamp
M11	4th axis unclamp
M19	Spindle oriented stop
M30	Program end and rewind
M50	Oil hole coolant on
M60	Loading pallet B
M62	Loading pallet A
M63	Unloading pallet A
M73	Y - Axis mirror image off
M74	Y - Axis mirror image on
M75	X - Axis mirror image off
M76	X - Axis mirror image on
M98	Sub-program call
M99	Sub-program end

### G76 – MULTIPLE REPETITIVE PASS THREADING – CYCLE ;

G76 P m r a Q  $\Delta d$  min. R d ;

G76 X u Z w R i P k Q $\Delta d$  F l ;

G76 P \_\_\_ Q R\_ ;

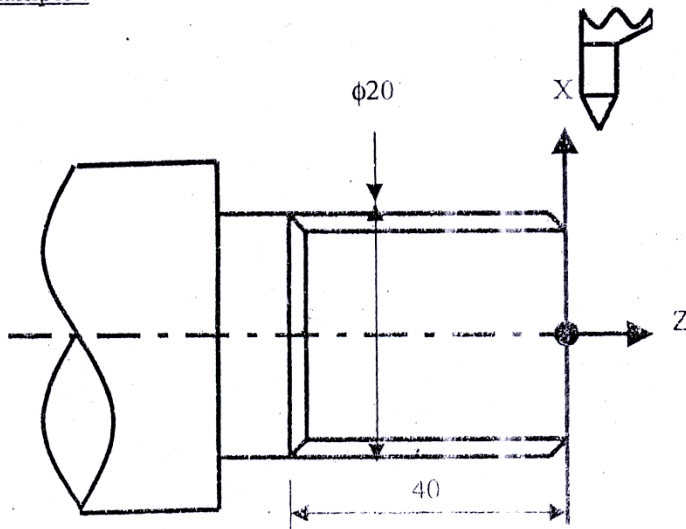
G76 X\_ Z\_ R\_ P\_ Q\_ F\_ ;

- m = Number of finish passes (1-99)  
r = Chamfering amount (01-99) 01 = 0.1, 99 = 9.9  
(r=0.1x1 to 9.9x1)  
a = Angle of tool tip / angle of thread. (0°, 60°, 55°, 30°, 29°)  
d = Finish allowance  
u = Minor diameter of the thread in case of external thread – Major diameter in case of internal.  
w = Length of thread.  
i = Difference of thread radius (for taper threads only)  
k = Height of thread / depth of thread  
l = Lead or pitch of thread

( $\Delta d$  min) = minimum depth of cut, subsequent passes depth ( $\Delta d\sqrt{n} - \Delta d\sqrt{n-1}$ ). Number of cutting passes can be controlled by varying this value.

$\Delta d$  = Depth of cut in 1<sup>st</sup> pass / 1<sup>st</sup> cut.)

Example :



PART PROGRAMMING MANUAL

CANNED CYCLE-G71-G76 (MULTIPLE REPETITIVE CYCLE) :  
G71 STOCK REMOVAL IN TURNING.

G71 U\_R\_;

G71 P\_Q\_U\_W\_F\_;

G71 U $\Delta$ d Re;

G71 pnsQnfU $\Delta$ uW $\Delta$ wFf;

$\Delta$ d = Depth of cut in radius

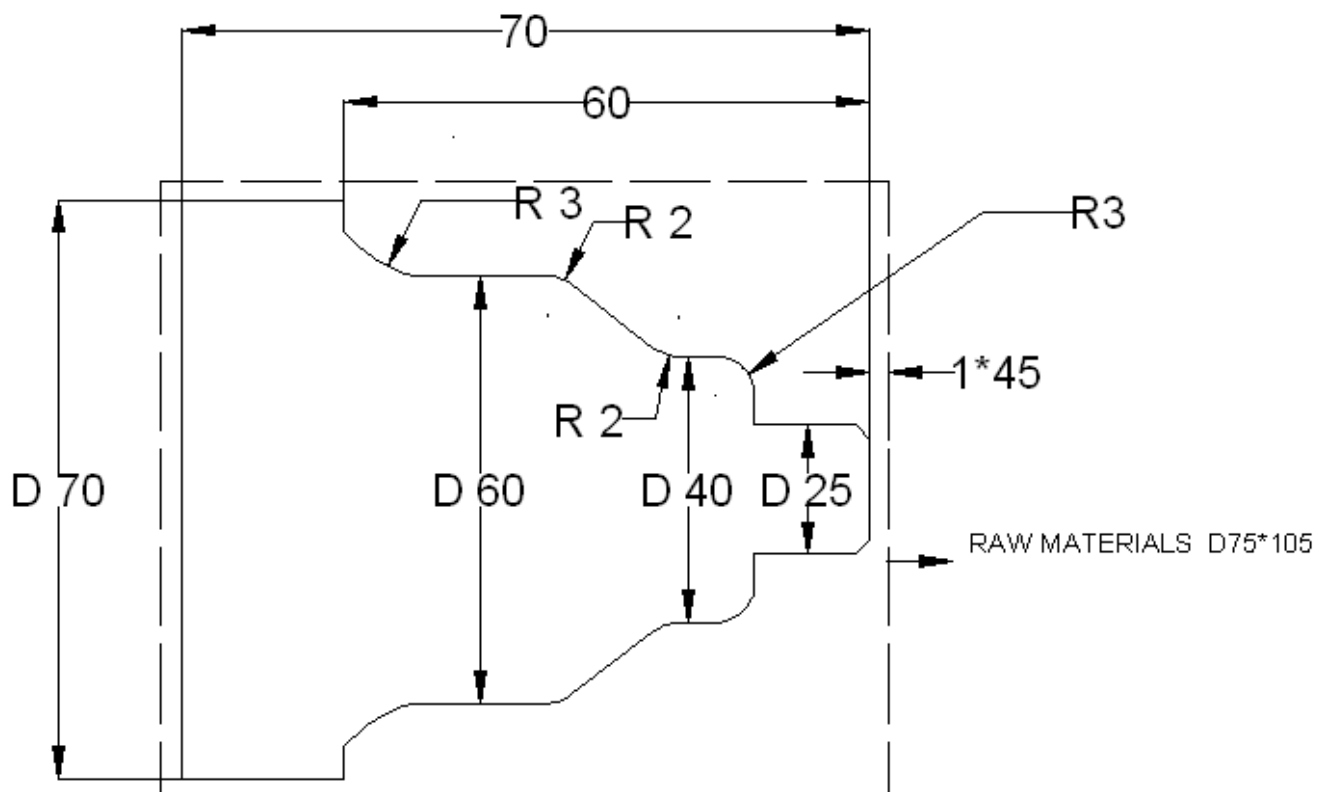
e = Tool escape / tool retraction distance

ns = Sequence number of the first block of the program which specifies the finish figure.

nf = Sequence number of the last block of the program which specifies the finish figure.

$\Delta$ u = Finish allowance on "x"axis / diameter

F = Feed.



## JOBBER XL CNC LATHE

<b>G-CODES</b>	
G00:	POSITIONING
G01:	LINEAR INTERPOLATION
G02:	CIRCULAR INTERPOLATION CW
G03:	CIRCULAR INTERPOLATION CCW
G04:	DWELL
G10:	OFF SET VALUE SETTING (O)
G20:	INCH
G21:	METRIC
G22:	STORED STROKE CHECK ON (O)
G23:	STORED STROKE CHECK OFF (O)
G25:	SPINDLE SPEED DETECT OFF
G26:	SPINDLE SPEED DETECT ON
G27:	REF. POINT RETURN CHECK
G28:	REF. POINT RETURN
G30:	2 <sup>ND</sup> (3,4) REF. POINT RETURN
G31:	SKIP CUTTING
G33:	THREAD CUTTING
G34:	VARIABLE LEAD THREAD CUTTING (O)
G36:	AUTO TOOL OFF STE X-AXIS (O)
G37:	AUTO TOOL OFF SET Z-AXIS (O)
G40:	TOOL NOSE RADIUS COMPENSATION CANCEL
G41:	TOOL NOSE RADIUS COMPENSATION LEFT (O)
G42:	TOOL NOSE RADIUS COMPENSATION RIGHT (O)
G53:	SUPPRSSION OF ZERO OFFSET
G54:	SETTABLE ZERO OFFSET
G55:	SETTABLE ZERO OFFSET
G56:	SETTABLE ZERO OFFSET
G57:	SETTABLE ZERO OFFSET
G65:	MACRO CALL (O)
G68:	DOUBLE TURRETS MIRROR ON (O)
G69:	DOUBLE TURRETS MIRROR OFF
G70:	FINISHING CYCLE (O)
G71:	ROUGH CUTTING (TURNING) (O)
G72:	ROUGH CUTTING (FACING) (O)
G73:	ROUGH CUTTING (PROFILE) (O)
G74:	GROOVING (FACING) (O)
G75:	GROOVING TURNING (O)
G76:	THREAD CUTTING CYCLE (MULTI) (O)
G77:	TURNING CYCLE
G78:	THREAD CUTTING CYCLE
G79:	FACING CYCLE
G90:	ABSOLUTE
G91:	INCREMENTAL
G92:	COORDINATE SYSTEM SETTING OR MAX. SPINDLE SPEED SETTING
G94:	PER MINUTE FEED
G95:	PER REVOLUTION FEED
G96:	CONSTANT SURFACE SPEED (O)
G97:	REVOLUTION PER MINUTE (RPM)
<b>M-CODES</b>	
M00:	PROGRAM STOP
M01:	OPTIONAL STOP
M02:	PROGRAM END AND RESTART
M03:	SPINDLE ROTATION CW
M04:	SPINDLE ROTATION CCW
M05:	SPINDLE STOP
M07:	COOLANT ON
M09:	COOLANT OFF
M10:	CHUCK DECLAMP
M11:	CHUCK CLAMP
M16:	CHUCK 1D SELECTION
M18:	CHUCK 0D SELECTION
M19:	SPINDLE ORIENTATION
M20:	SPINDLE ORIENTATION CANCEL
M30:	END OF MAIN PROGRAM
M32:	TAILSTOCK QUIL FORWARD
M33:	TAILSTOCK QUIL RETRACT
M35:	PARTS CATCHE RETRACT
M46:	DOOR OPEN
M47:	DOOR CLOSE
M50:	SPINDLE LOCK
M51:	SPINDLE UNLOCK
M78:	STEADY REST OPEN
M79:	STEADY REST HOLD
M82:	TAIL STOCK BODY FWD / UNCLAMP
M83:	TAILSTOCK BODY RET / CLAMP
M84:	TOUCH PROBE ARM FORWARD
M85:	TOUCH PROBE ARM RETRACT
M98:	SUB PROGRAM CALL
M99:	SUB PROGRAM END



EX:NO: 1  
DATE: 22.01.15

## **PROGRAMMING, SIMULATION AND MACHINING PROFILE TURNING USING CNC - T70 LATHE**

### **AIM:**

To write a programme for the given component and execute the same in T70 Trainmaster Lathe.

### **TOOLS REQUIRED:**

1. Tools
2. Aluminium shaft
3. vernier caliper

### **PROCEDURE**

1. For the given dimensions of the work piece to be machined write the program using G codes and M codes
2. Using the simulation software or by running the machine in test mode check the program and if there is any error make the correction in the program.
3. Fix the work piece on the chuck
4. Move the tool to the start point of the work piece by manual mode.
5. Reset the Machine.
6. Change the machine from manual mode to single block mode or auto mode.
7. Execute the program to get the required shape of the work piece.
8. Remove the machined work piece from the chuck

## **PROGRAM**

```
%  
N01 G90 – Absolute programming  
N02 G71 – Metric Mode  
N03 G92 X0Z0 Program pre set  
N04 G81 X-200 Z-6500 F200      (G81 – Turning cycle)  
N05 G81 X-400 Z-5500  
N06 G81 X-600 Z-4500  
N07 G81 X-800 Z-3500  
N08 G81 X-1000 Z-3400  
N09 G81 X-1200 Z-3300  
N10 G01 X-1500 Z0  
N11 G01 X-1300 Z-200  
N12 G01 X-1300 Z-3000  
N13 G02 X-900 Z-3500 I500 K0  
N14 G01 X-400 Z-5500  
N15 G01 X-400 Z-6500  
N16 G01 X0 Z-6500  
N17 G01 Z0  
N18 M30  
%
```

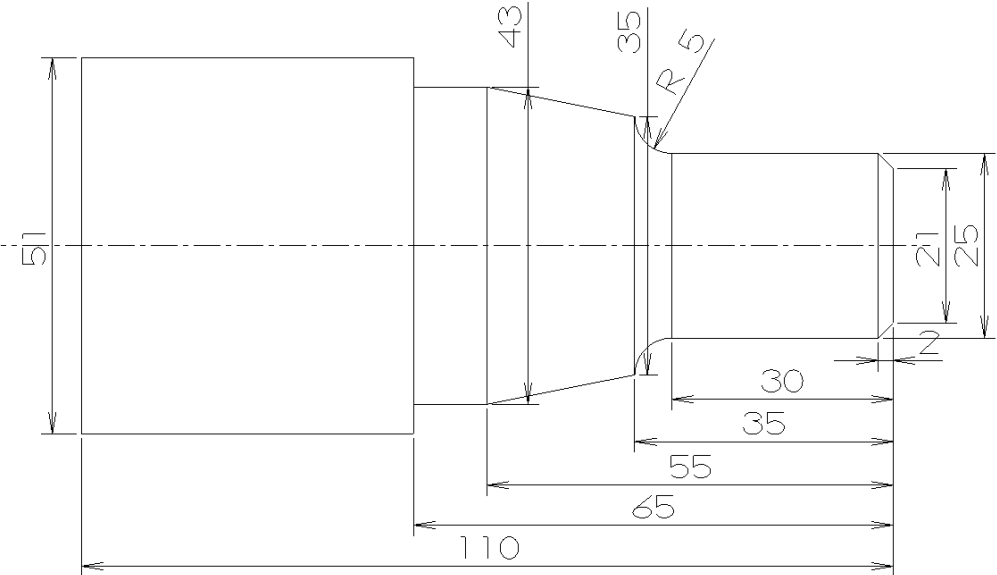
## **Result**

The part program for producing the given model is written and the given aluminium work piece is machined to the given dimension

CNC Lathe: T70

1.Profile Turning using CNC - T 70 lathe

Profile Turning 1



EX:NO: 2  
DATE: 05.02.15

## **PROGRAMMING, SIMULATION AND MACHINING PROFILE TURNING AND THREAD CUTTING USING ACE JOBBER XL CNC LATHE**

### **AIM:**

To write a programme for the given component and execute the same in ACE Jobber XL Lathe.

### **TOOLS REQUIRED:**

1. Tool
2. Mild Steel shaft
3. Micro meter
4. Vernier

### **PROCEDURE:**

1. For the given dimensions of the work piece to be machined write the program using G codes and M codes
2. Using the simulation software or by running the machine in test mode check the Program and if there is any error make the correction in the program.
3. Fix the work piece on the chucks.
4. Move the tool to the start point of the work piece by manual mode.
5. Reset the Machine.
6. Change the machine from manual mode to single block mode or auto mode.
7. Execute the program to get the required shape of the work piece.
8. Remove the machined work piece from the chuck.

**PROGRAM: (Z)**

%O0004  
T0000  
G21

(FACING)  
G0T0801  
G97S1200M04  
G0X55.0Z0M07  
G99G1X-1.0F0.2  
G0Z2.0

(OD TURNING)

G92S1250M04  
G96S210  
X51.0  
Z1.0  
G71U1.0R2.0  
G71P1Q3U0.2W0.12F0.15  
N1G0X21.0  
G01X21.0Z0  
G1X25.0Z-2.0  
Z-30.0  
G2X35.0Z-35.0R5.0  
G1X43.0Z-55.0  
Z-65.0  
N3X51.0  
G97M09  
T0000  
G00X0Z0

(FINISHING)  
T0402  
G92S1200M04  
G96S240  
X55.0Z2.0M07  
G70P1Q3F0.1  
G97M09  
T0000  
G28U0W0

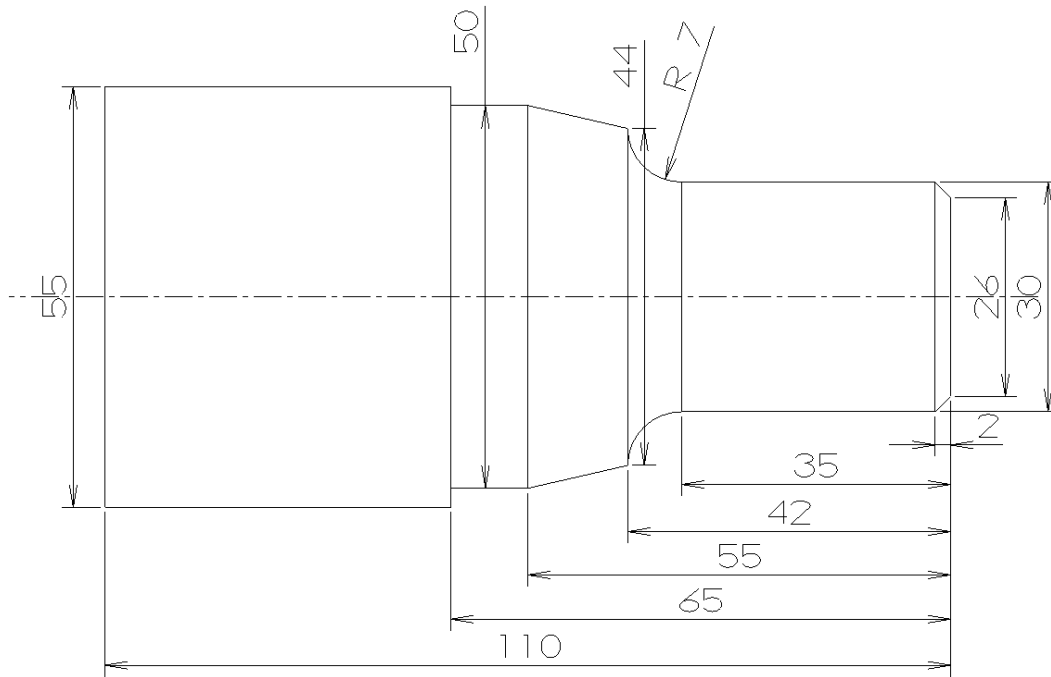
(THREADING)  
T0304;  
G00X0Z-100.0;  
G97S100M04;

G00X25.0Z5.0;  
G76P020060Q200R100;  
G76X23.268Z-25.0P866Q400F1.0;  
S0T0000;  
G0X0Z-100.0M09;  
M05  
M30

**RESULT:**

The part program for the given model is written and the given Component is machined to the given dimension.

# Profile Turning



EX:NO: 3  
DATE: 12.02.15

## **PROGRAMMING, SIMULATION AND MACHINING LETTER MILLING USING CNC VMC 200 T**

### **AIM:**

To write a programme for the given component and execute the same in VMC200T Trainmaster.

### **TOOLS REQUIRED:**

1. Tools
2. Aluminium shaft
3. vernier caliper

### **PROCEDURE:**

1. For the given dimensions of the work piece to be machined write the program using G codes and M codes
2. Using the simulation software or by running the machine in test mode check the program and if there is any error make the correction in the program.
3. Fix the work piece on the vice.
4. Move the tool to the start point of the work piece by manual mode.
5. Reset the Machine.
6. Change the machine from manual mode to single block mode or auto mode.
7. Execute the program to get the required shape of the work piece.
8. Remove the machined work piece from the vice.



**PROGRAM:**

```
%  
N01 G90  
N02 M03S200  
N03 G17  
N04 G01 X2000 Y5000 F80  
N05 G18  
N06 G01Z-500  
N07 G17  
N08 G01 Y2000  
N09 G18  
N10 G01 Z500  
N11 G17  
N12 G01 Y3500  
N13 G18  
N14 G01 Z-500  
N15 G17  
N16 G01 X5000 Y5000  
N17 G18  
N18 G01 Z500  
N19 G17  
N20 G01 X3000 Y4000  
N21 G18  
N22 G01 Z-500  
N23 G17  
N24 G01 X5000 Y2000  
N25 G18  
N26 G01 Z500  
N27 G17  
N28 G01 X9000 Y5000  
N29 G18  
N30 G01 Z-500  
N31 G17  
N32 G01 X6000 Y5000  
N33 G01 Y2000  
N34 G01 X9000  
N35 G18  
N36 G01 Z500  
N37 G17  
N38 G01 X6000 Y3500  
N39 G18  
N40 G01 Z-500  
N41 G17  
N42 G01 X8000  
N43 G18  
N44 G01 Z500  
N45 G17  
N46 G01 X12600 Y2500  
N47 G18
```

N48 G01 Z-500

N49 G17

N50 G02 X12600 Y4500 I-1100 J1000

N51 G18

N52 G01 Z500

N53 G17

N54 G01 X0Y0

N55 M30

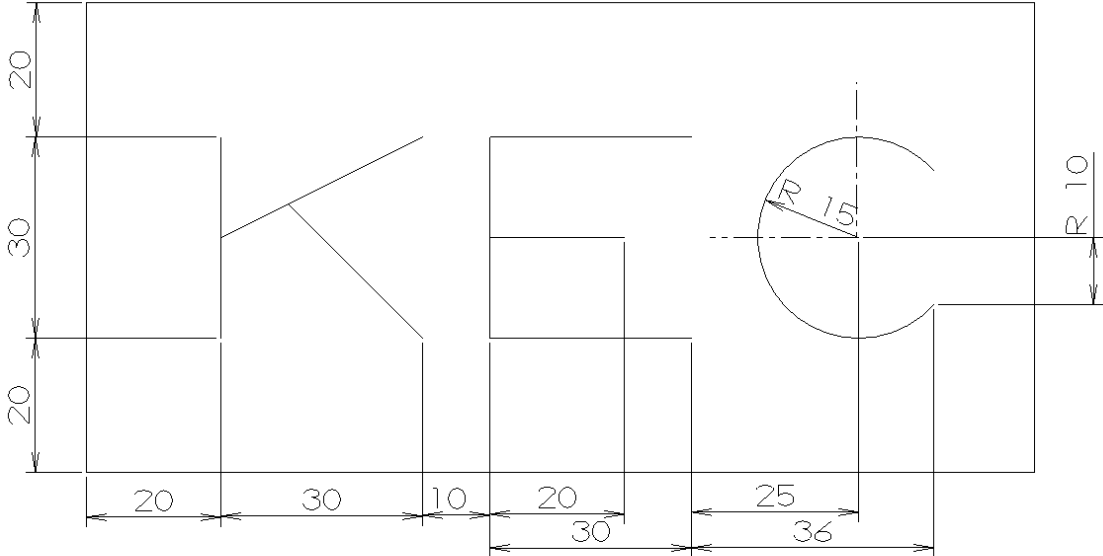
%

**RESULT:**

The part program for producing the given model is written and the given aluminium work piece is machined to the given dimension.

CNC Milling: 200T

Letter Milling-1 using VMC 200 T



**PROGRAMMING, SIMULATION AND PECK DRILLING & BORING  
USING CNC LMILL 55 VERTICAL MACHINING CENTRE**

**AIM:**

To write a programme for machining the given Component and execute the same in L MILL55 Vertical Machining Centre.

**TOOLS REQUIRED:**

1. Tool
2. Mild Steel shaft
3. Micro meter
4. Vernier

**PROCEDURE:**

1. Study the Drawing Carefully to plan for the Machining operations.
2. Use the Man Machine Interface to Programme for the given Geometry.
3. Set the job and the offset for the given Workpiece.
4. Use the Processor for the operation sequence and set the parameters of the operation.
5. Select all the operations for the Simulation purpose and execute the program and verify the same in L MILL55 Vertical Machining Centre.
6. Execute the Program and Remove the work piece from the Clamp.

**PROGRAM:**

O0020 (PECK DRILLING & BOARING);

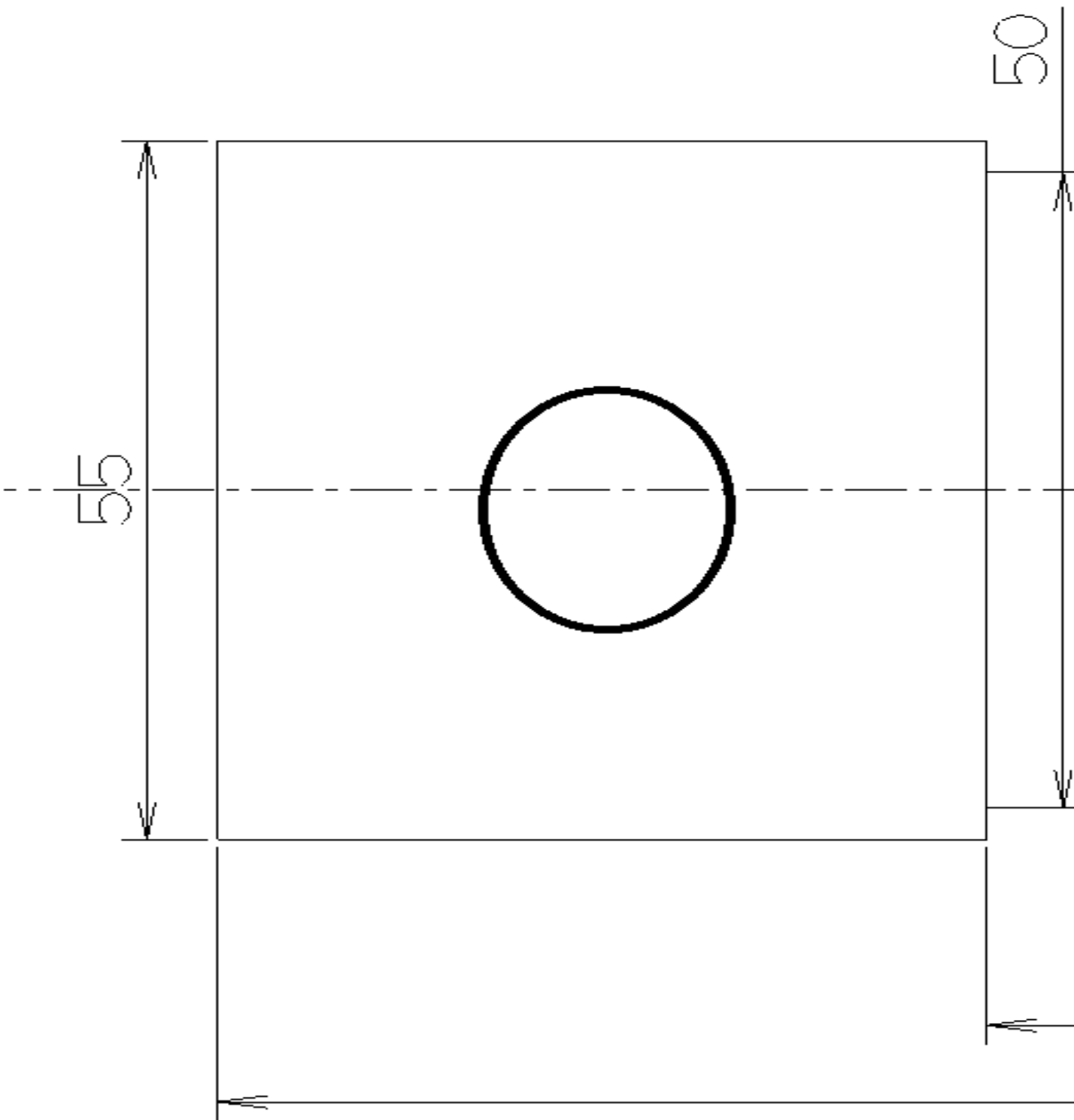
N01  
G0G91G28Z0  
G28X0Y0  
T04 (Centre drill)  
M06  
M03S800  
G0G56G40G49X0Y0Z10.0  
G01Z-10.0F5.0  
G0Z0  
M05

N02  
G0G91G28Z0  
G28X0Y0  
T05(Drill dia 10.2mm)  
M06  
M03S800

G95G98G83X0Y0Z-140.0R-100.Q5.0F5.0  
G80  
M05  
G91G28Z0

N03  
T08 (U Drill dia22mm)  
M06  
M03S600  
G0G58G40G49X0Y0Z0  
G95G98G83X0Y0Z-140.R-100.Q5.F5.0  
G80  
M05  
G91G28X0Y0

N4  
T10 (Boring bar dia23)  
M06  
M03S800  
G0G90G59G40G49X0Y0Z0  
G95G98G86X0Y0Z-140.R-100.F5.0  
M05  
G80  
G91G28Z0  
G28X0M30



EX:NO: 5  
DATE: 05.03.15

## **NC CODE GENERATION USING MASTER CAM LATHE**

### **AIM:**

To generate the NC codes for given profile operation in master CAM lathe

### **COMMANDS USED:**

1. Lines
2. Fillet
3. Tool Path
4. Operation
5. Job Setup.

### **PROCEDURE:**

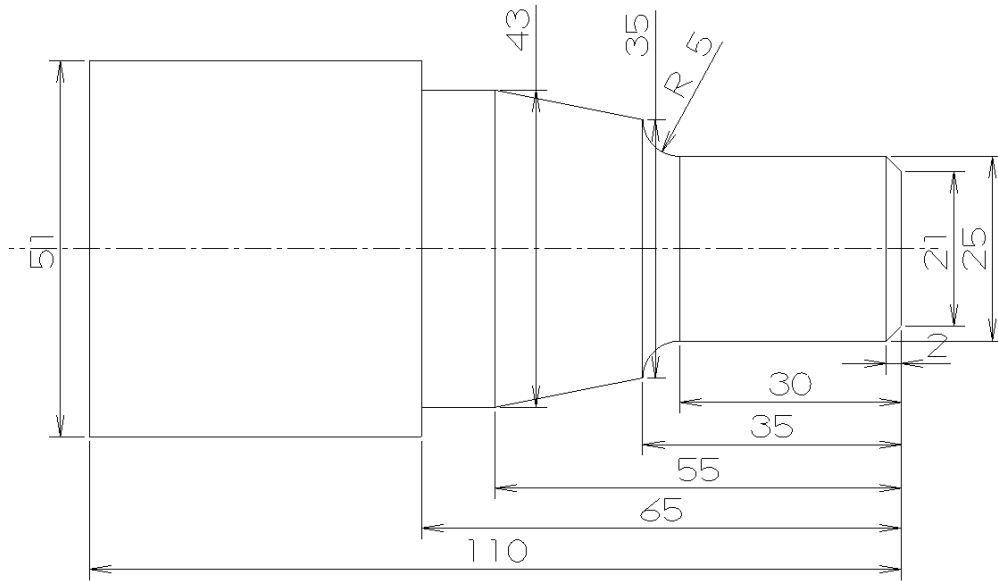
1. Using lines command draw the basic given profile in the editor by using Multi points option.
2. Use fillet wherever needed.
3. Select Tool Path -> Rough -> Chain and select the lines at each extreme.
4. Then choose Tool path -> Job setup -> Boundary -> Parameters -> OD and Length.
5. Select Done.

### **CODE GENERATION :**

1. Select Operation -> Regen Path -> Verify (iso) -> Post.
2. To get the code select Save.

### **RESULT:**

Thus for given profile the NC code has been generated using master camlathe.





EX:NO: 6  
DATE: 19.03.2015

## **NC CODE GENERATION USING MASTER CAM LATHE**

### **AIM:**

To generate the NC codes for given profile operation in master CAM lathe

### **COMMANDS USED:**

Lines, Fillet, Tool Path , operation, job Setup.

### **PROCEDURE:**

1. Select Create ->Rectangle ->I point ->Enter width and height in the dialog box which appears.
2. Select Main menu ->Tool Path ->Contour or Pocketing ->Chain ->Select the required items and click on OK.
3. select Tool Path dialog box give the tool diameter and depth.
4. Select Job Setup ->Enter x, y, z values and click on OK.

### **CODE GENERATION :**

1. Select Operation ->Regen Path -> Verify (iso) -> Post.
2. To get the code select Save.

### **RESULT:**

Thus for given profile the NC code has been generated using Master CAM Mill.

