

**ADICHUNCHANGIRI INSTITUTE OF TECHNOLOGY,  
CHIKMAGALUR**

DEPARTMENT OF MECHANICAL ENGINEERING



Computer Integrated Manufacturing Lab Manual – 18MEL76

**Part-A**

**Manual CNC part programming for 2 turning and 2 milling parts.** Selection and assignment of tools, correction of syntax and logical errors, and verification of tool path.

**CNC part programming using CAM packages.** Simulation of Turning, Drilling, Milling operations. 3 typical simulations to be carried out using simulation packages like: Cadem CAM Lab-Pro, Master- CAM.

Program generation using software. Optimize spindle power, torque utilization, and cycle time. Generation and printing of shop documents like process and cycle time sheets, tool list, and tool layouts. Enter program, take tool offsets, cut part in single block and auto mode, measure the virtual part on screen in the virtual CNC machine simulator, for standard CNC control systems FANUC, FAGOR, HAAS and SINUMERIK.

**Part B (Only for Demo/Viva voce)**

**FMS (Flexible Manufacturing System):** Programming of Automatic storage and Retrieval system (ASRS) and linear shuttle conveyor Interfacing CNC lathe, milling with loading unloading arm and ASRS to be carried out on simple components.

**(Only for Demo/Viva voce)**

**Robot programming:** Using Teach Pendant & Offline programming to perform pick and place, stacking of objects (2 programs).

**Course Outcomes:** After studying this course, students will be able to:

CLO1 Generate CNC Lathe part program for Turning, Facing, Chamfering, Grooving, Step turning, Taper turning, Circular interpolation etc.

CLO2 Generate CNC Mill Part programming for Point to point motions, Line motions, Circular interpolation, Contour motion, Pocket milling- circular, rectangular, Mirror commands etc.

CLO3 Use Canned Cycles for Drilling, Peck drilling, Boring, Tapping, Turning, Facing, Taper turning Thread cutting etc.

CLO4 Simulate Tool Path for different Machining operations of small components using CNC Lathe & CNC Milling Machine.

CLO5 Use high end CAM packages for machining complex parts; use state of art cutting tools and related cutting parameters; optimize cycle time; set up and cut part on.

CLO6 Understand & write programs for Robot control; understand the operating principles of hydraulics, pneumatics and electro pneumatic systems.

**Scheme for Examination:**

Two Questions from Part A - 60 Marks (30 +30)

Viva-Voce - 20 Marks

Total: 80 Marks

## CIM LABORATORY.

**Lab Coordinators:** Dr. Abhijith Vaidya S.

**Technical Supporting Staff:** Mr. Suresh H N.

**Supporting Staff:** Mr. Hariharan

### DO'S AND DON'TS

#### **Do's**

1. Enter the laboratory with Lab uniform and keep the bags in the racks properly.
2. Keep the surrounding clean and tidy.
3. Any computer working improperly, take the help from the supporting staff.
4. Report fires or any accidents to your lecturer/laboratory technician immediately.
5. If any broken plugs or electrical wires found, inform to your lecturer/supporting staff immediately.

#### **Don'ts**

1. After the lab class, switch of the computers properly.
2. Do not misbehave in the computer laboratory.
3. Avoid stepping on electrical wires or any other computer cables.
4. Do not remove anything from the computer or fix any portable drives without permission the supporting staff.
5. Do not touch, connect or disconnect any plug or cable without your lecturer/laboratory technician's permission.

## **LIST OF SOFTWARE AVILABLE**

1. Autodesk AutoCAD 2014
2. Autodesk AutoCAD 2014 Upgraded to Inventor 8.0 education version
3. EMRC NISA – II + DISPLAY – III, DYMES
4. Denford Offline Programming Milling & Turning Network Based 20 Users
5. Six Sigma suite of Product (SPC-IV)
6. CATIA V5 education version
7. Ansys 11 25 users
8. Robocell 3D Simulation Software Version 4.7
9. AMESim ID software for CIM & Automation Lab

## TURNING PROGRAMMING

DENFORD FANUC

### MISCELLANEOUS FUNCTION ( M codes )

#### **M00 PROGRAM STOP**

By inserting M00 in a program, the cutting cycle is stopped after the block containing M00 code. This facility is useful if an inspection check is necessary during an operation.

**M01 OPTIONAL STOP** Cycle operation is stopped after a block containing M01 is executed.

#### **M02 PROGRAM END**

This code is inserted at the end of the program, when encountered the cycle will end.

#### **M03 SPINDLE FORWARD**

Starts the spindle spinning forward, clockwise or negative direction at the last specified spindle speed. EXAMPLE: M03 S1200

#### **M04 SPINDLE REVERSE**

Starts the spindle spinning reverse, counter clockwise or positive direction at the last specified spindle speed. EX. M04 S1000

#### **M05 STOP SPINDLE**

Stops the spindle without changing the spindle speed.

#### **M06 TOOL CHANGE**

The M06 in conjunction with **T** word is used to call up the required tool on an automatic indexing turret machine, and to activate its tool offsets. EX. M06 T04

#### **M09 COOLANT OFF**

#### **M10 / M11 CHUCK OPEN/CHUCK CLOSE**

#### **M13 SPINDLE FORWARD, COOLANT ON**

Sets spindle rotation forward and coolant on. EX. M13 S1200

#### **M14 SPINDLE REVERSE, COOLANT ON**

Sets spindle rotation reverse and coolant on. EX. M14 S1200

**M30 PROGRAM END** Stops the spindle. Turns the coolant off. Terminates and resets the CNC program

#### **M38 / M39 DOOR OPEN / DOOR CLOSE**

Opens or closes the door, waiting until the door is open or closed.

#### **M98 SUBPROGRAM CALL**

#### **M99 SUBPROGRAM EXIT**

## **PREPARATORY FUNCTION ( G codes )**

### **G00 FAST OR RAPID TRAVERSE**

This instruction traverses the tool to the target point at maximum traverse rate. The rapid traverse is used for movements where no tool is in engagement. EX. G00 X30 Z-10

### **G01 LINEAR MOTION**

The instruction " straight-line at feed rate " requires the program word G01. The following supplementary functions are also needed: Target point co-ordinates, feed rate, spindle speed or cutting speed. EX. G01 X30 Z10 F100 S1000

### **G02 CIRCULAR INTERPOLATION (Clockwise)**

EX. G02 X20 Z-20 R10 F45 .....where R10 is the radius F45 feed rate mm/min.

### **G03 CIRCULAR INTERPOLATION (Counter Clockwise)**

EX. G03 X20 Z-25 R10 F50 .....where X20 Z-25 is a target point R- radius F-Fed.rate.

### **G04 DWELL**

It causes the program to wait for a specified amount of time.The time can be specified in seconds with the X or U prefixes or in milliseconds with the P prefix.

EX. G04 X1.5 or G04 U1.5 or G04 P1500

### **G20 INCH DATA INPUT (IMPERIAL UNITS )**

### **G21 METRIC DATA INPUT**

### **G28 GO TO REFERENCE POINT**

It causes a fast traverse to the specified position and then to the machine datum.

EX G28 X35 Z5 or G28 U0 W0

### **G94 FACING CYCLE**

G94 is a box type cutting cycle. This cycle is used for stock removal either parallel or at an angle to work piece face.

EX. a) **G94 X(U) Z(W) F** b) **G94 X(U) Z(W) R- F** c) **G94 X(U) Z(W) R+ F**

.....where, X = Diameter to which the movement is being made

U = The incremental distance from the current tool position to the required final diameter.

Z = The Z axis Co-ordinate to which the movement is being made

W = The incremental distance from the current tool position to the required Z axis position.

R = The difference in incremental of the cut start radius value and the cut finish radius

value

EX. G94 X10 Z-0.5 F35  
Z-1  
Z-2  
Z-3

#### G94 TAPER FACING CYCLE

EX. G94 X22 Z-10 R-10 F30  
X21  
X20  
.....  
.....  
X10

#### **G90 SINGLE TURNING CYCLE**

This cycle can be used to produce either a parallel or tapered tool path. This cycle performs four distinct moves with one line of information and it is the equivalent of

\* Rapid to X position \* feed to Z position \* feed to start X position \* rapid to start Z position

EX. G90 X (U) Z (W) F

..... where, X = Diameter to which the movement is being made

U = The incremental distance from the current tool position to the required final diameter.

Z = The Z axis coordinate to which the movement is being made.

W = The incremental distance from the current tool position to the required Z axis position.

F = feed

**EX: G90 X30 Z-25 F0.4**

**X25**

**X20**

## **G90 TAPER TURNING CYCLE**

EX. G00 X44 Z2  
G90 X36 Z-20 R-2 F3  
X30 R-.5  
X30 R-1  
.....  
.....  
X30 R-4.5

## **G98 FEED PER MINUTE**

This command coupled with the F word is used to specify feedrate per minute. This can be mm/min or inch/min. This is the default.

## **G99 FEED PER REVOLUTION**

This command coupled with the F word is used to specify feedrate per revolution. This can be mm/rev or inch/rev. This is the default.

## **G81 GROOVING CYCLE**

Ex. G81 X18.5 F15 after calling 2 mm GROOVING TOOL  
X18  
.....  
.....  
X14 grooving upto 14 mm diameter

## **G92 SINGLE THREADING CYCLE**

This is a 'BOX TYPE' cycle producing a single pass of the threading tool. The position specified is that of the end of the thread. The F value specifies the pitch, NOT the feed.

EXAMPLE: G00 X32 Z5  
G92 X20.977 Z-20 F2.5 ... X = depth of cut (abs) z=length of thread  
X19.955 ... F = lead or pitch of thread  
X18.932



## **G76 MULTIPLE THREADING CYCLE**

This is a 'BOX TYPE' cycle that is repeated a given number of times. After the first pass subsequent passes cut with one edge of the threading tool only to reduce the load at the tool tip. this cycle requires two distinct blocks of data. When the cutting depth of one cycle becomes smaller than the limit, the actual amount of cut is clamped at the minimum cut depth

**G76 P(m)(r)(a) Q(\*q1) R(\*r1)**

**G76 X(\*x) Z(\*z) P(\*p2) Q(\*q2) F(\*f)**

where m = Repetative count in finishing (1 to 99)

r = Chamfaring amount(0.01 to 9.91, where 1 the thread's lead)

a = angle of tool tip (80°, 60°, 55°, 30°, 29° and 0° )

\*x = finished depth of Thread

\*z = End position of thread

\*p2 = Height of the thread as a radius value of x 1000, as the controller accepts this vale in microns. e.g. 1.02 mm becomes P1020.

\*q2 = Depth of the first cut as a radius value x1000, value in microns  
e.g. 0.25 mm becomes Q250.

\*f = Lead or pitch of thread. e.g. lead 1.5 = F1.5

\*q1 = Min cutting depth

\*r1 = Finishing allowance

EXAMPLE: G76 P031560 Q250 R0.15

G76 X10.466 Z-50 P767 Q250 F1.25

..where Core dia= Nominal dia - 2 x 0.61343 x p where p = pitch of threads.

Height of thread = 0.61343 x p

## **G74 END FACE PECK DRILLING**

This cycle is designed for deep hole drilling, the drill entering the workpiece by a predetermined amount then backing off by another set amount to provide breaking and allowing swarf to clear the drill flutes. The cycle is commanded by two distinct lines of data.

**G74 R(\*r1)**

**G74 Z(W) Q(\*q) r(\*r2) F(\*f)**

where, \*r1 = Return amount

Z = Total depth (absolute )

W = Total depth (incremental)

\*q = Depth of cut (incremental, unsigned ) IN MICRONS for each pass.

\*r2 = Relief amount of tool at the bottom of the hole produced, for drilling this value is ZERO.

\*f = feed rate

EXAMPLE G74 R1.0

G74 Z-40 Q5000 R0.5 F100

### TURNING CYCLE G90

EXAMPLE :

O1234

[ BILLET X50 Z75 .....Size of work piece dia 50 length 75

G21 G98..... Metric , Feed per minute

G28 U0 V0 ..... Send to reference position

M06 T01 ..... Tool Change, Tool 1

M03 S1200..... spindle ON Forward, speed 1200 rpm

G00 X48 Z5..... Rapid Traverse Position the tool

G90 X48Z-50 F30 .....Turning cycle dia 48 for 50 mm length

X46

X44

X42

X40

G00 X38 Z5

G90 X38 Z-25

X36

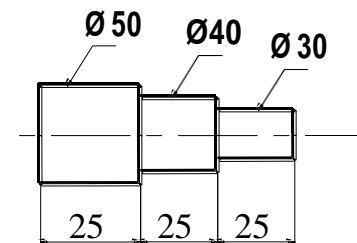
X34

X32

X30

G28 U0 V0 ..... Send the tool to reference position

M05 M30 ..... Stop spindle, end program



**FACING CYCLE G90**

EXAMPLE:

G00 X0 Z5  
G01 Z-1 F20.....Linear Traverse  
90 X22 Z-1 F20  
Z-2  
Z-3  
Z-4  
Z-5 ..... facing for 5 mm length

OR

**FACING CYCLE G94**

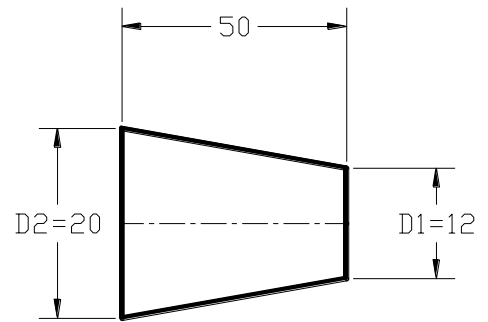
EXAMPLE:

G00 X22 Z1  
G94 X22 Z-1 F20  
Z-2  
Z-3  
Z-4  
Z-5.....facing for 5 mm length

**TAPER TURNING CYCLE G90**

EXAMPLE : For the taper shown in FIG.2

G00 X20 Z0  
G90 X20 Z-50 R0  
X20 R-0.5  
X20 R-1.0  
X20 R-1.5  
.....  
.....  
X20 R-4

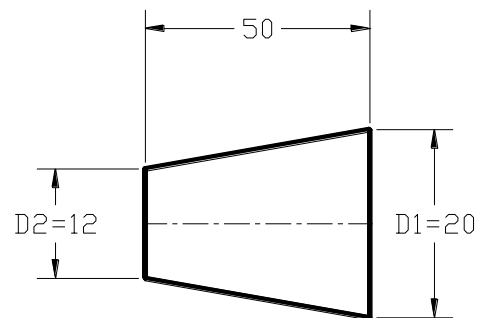


$R = (D1 - D2) / 2 = (12 - 20) / 2 = -4$

**TAPER TURNING CYCLE G90**

EXAMPLE : For the taper shown in FIG.3

G00 X20 Z0  
G90 X20 Z-50 R0  
X19 R0.5  
X18 R1.0  
X17 R1.5  
.....  
.....  
X11 R3.5  
X12 R4.0



$R = (D1 - D2) / 2 = (20 - 12) / 2 = +4$

## GROOVING CYCLE G81

EXAMPLE : For the PART shown in FIG.4

G28 U0 V0

M06 T04 select 6mm grooving tool

G00 X20 Z-65

G81 X18 F15

X16

X14

X12

X10

G00 X20 Z-59

G81 X18 F15

X16

X14

X12

X10

G00 X20 Z-56

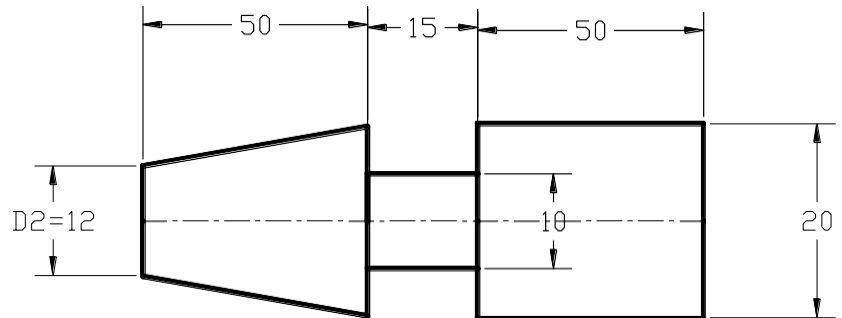
G81 X18 F15

X16

X14

X12

X10



## END FACE PECK DRILLING G74

EXAMPLE : For thePART shown in FIG.5

G28 U0 V0

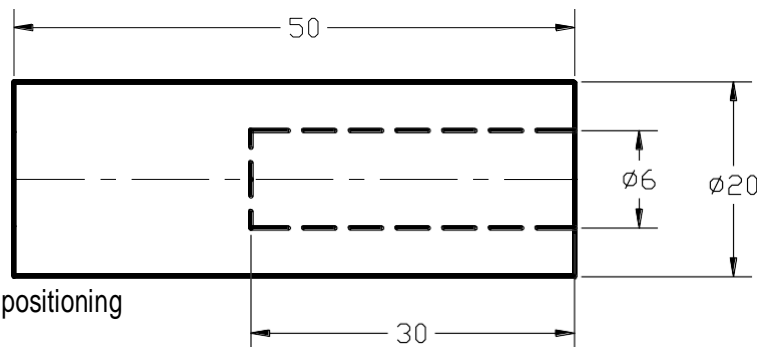
M06 T04 .....select 6 mm drill bit

G00 X0 Z4.....Rapid traverse, initial positioning

G74 R1.....Return amount 1mm

G74 Z-30 Q5000 R0 F30..... for a depth of 30 mm , depth of cut for each pass = 5000 microns

Relief amount R0 for peck drilling, feed 30 mm per minute



### SINGLE THREADING CYCLE G92

EXAMPLE : For the PART shown in FIG.6

G28 U0 W0

M06 T04 .....select threading tool

G99 .....Feed per Revolution

G00 X16 Z4 .....Initial positioning

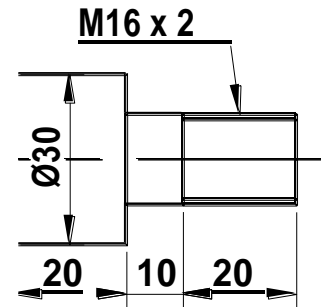
G92 X15 Z-20 F2 .....Threading cycle

X14

X13.546 ..... up to core diameter

For Nominal dia 16 mm and pitch 2 mm, the Core diameter is 13.546 mm

(From Data Hand Book )



### MULTIPLE THREADING G76

EXAMPLE: For the PART shown in FIG.6

G28 U0 W0

M06 T04 .....select threding tool

G99 .....Feed per Revolution

G00 X16 Z4 .....Initial positioning

G76 P031560 Q250 R0.25 ( 03- Number of passes, 15-Chamfer amount in microns,  
60- thread angle , R- minimum cutting depth= 0.25 mm)

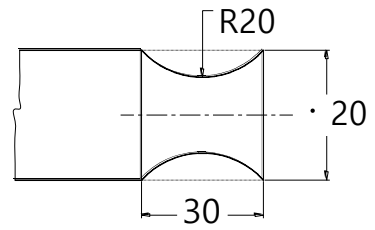
G76 X13.546 Z-20 P1383 Q300 F2.0 ( X- core diameter (0.846D),mm. Z- Length of the thread,  
P -Height of the thread(0.613F), Q- Depth of cut =0.3  
expressed as 1000, F- Pitch of the thread = 2.0)

**CIRCULAR INTERPOLATION G02 (CW)**

EXAMPLE: For the PART shown in FIG. 7

G00 X21 Z0

G02 X20 Z-30 R 20 F35

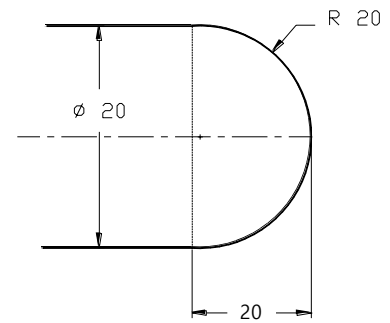


**CIRCULAR INTERPOLATION G03 (CCW)**

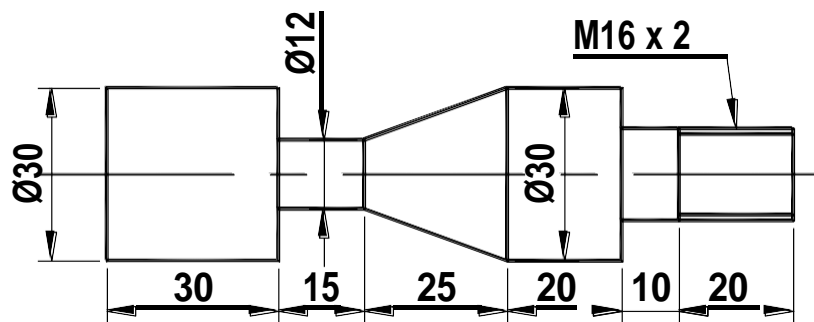
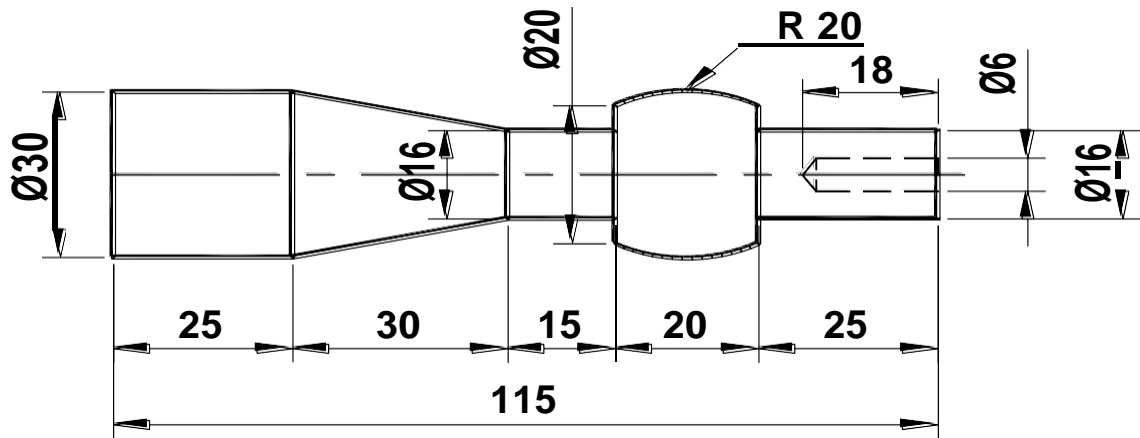
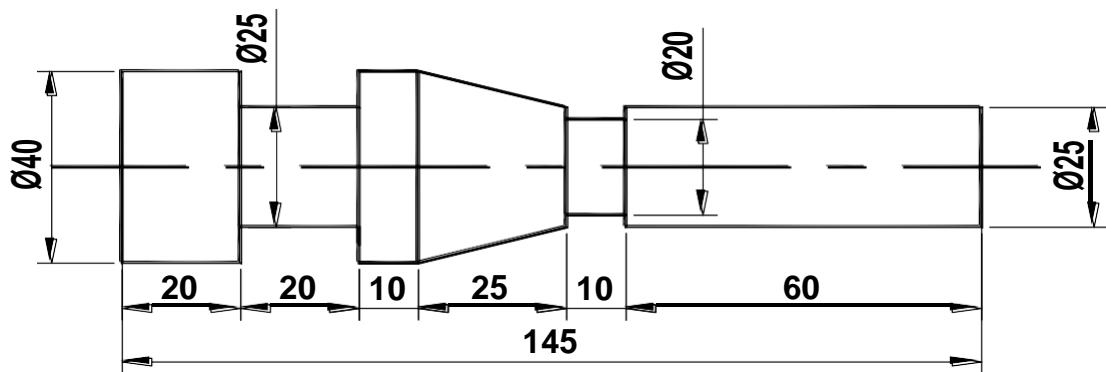
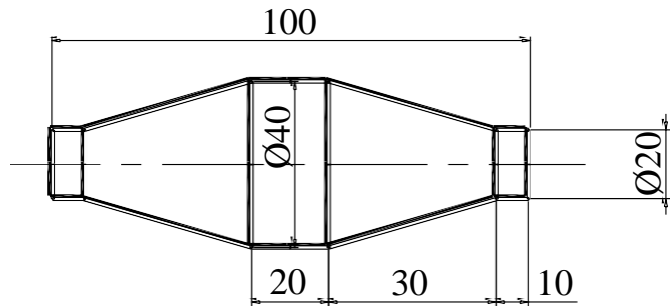
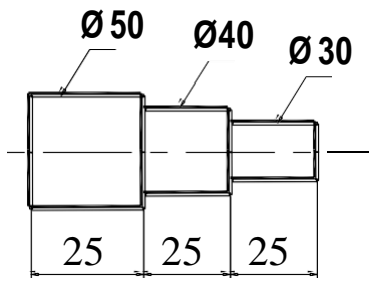
EXAMPLE: For the PART shown in FIG. 8

G00 X0 Z1

G03 X20 Z-20 R 20 F35



# CNC TURNING EXERCISES



# CNC MILLING EXERCISES

