

Computer Aided Manufacturing



**Dr.S.RAMABALAN,
PRINCIPAL,
E.G.S. PILLAY ENGINEERING COLLEGE,
NAGAPATTINAM.**



Cleansing through Breathing

Nadi Suddhi
(Alternate
Nostril breathing)



1 min

Recap and review of previous class

Let's
Recap



5 mins

Unit III

PROGRAMMING OF CNC LATHE

Coordinate system - structure of a part program -G & M Codes - Programming for FANUC and SIEMENS controller -Single pass and canned cycle -Turning, facing and threading -**Multi-pass canned cycle** -**Rough and Finish turning, facing, pattern repeating, grooving, threading, drilling, boring, peck drilling, high speed drilling cycle** -Subprogram and Macro programming -Tool length and nose radius compensation - offset -Tool, work and coordinate -Insert -Materials, Classification, Nomenclature and Selection -Tool and Work holding devices -Automatic tool changer -Turret and drum type -Tool holder nomenclature and selection -CNC part programming using CAD/CAM software and interfacing with CNC machines

Prerequisite Knowledge

- Metal cutting processes

3.EVOCATION



4. GENERAL OBJECTIVE (GO)

- Students will be able to apply the knowledge to create CNC part program for circular components with canned cycles.

5. SPECIFIC OBJECTIVE (SO) MAPPED WITH STEM

The students will be able to

1. Label the canned cycles used in CNC lathe programming with their syntax. (R-F) (E)
2. Explain the function and application areas of canned cycles (G70-G76). (U-C) (E)
3. Select the suitable canned cycles for CNC part programming of given component and construct part program for same component. (Ap-C) (E)

MULTIPLE REPETITIVE CANNED CYCLE (G70-G76)

1. Roughing cycle (G71)
2. Facing cycle (G72)
3. Pattern repeating cycle (G73)
4. Finishing cycle (G70)
5. Peck drilling cycle (G74)
6. Grooving cycle (G75)
7. Threading cycle (G76)

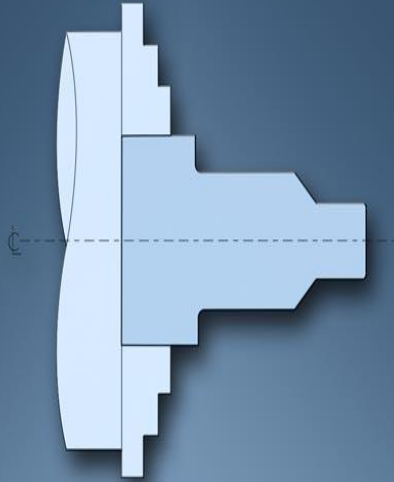
STOCK REMOVAL CYCLE (G71)

- Also called as roughing cycle
- The material will be removed when the tool moves parallel to the spindle axis
- Since it is a roughing cycle, the material removal rate will be more
- In this cycle, small amount of material is retained on the work piece which will be removed later by finishing cycle

FINISHING CYCLE (G70)

- In this cycle, the excess material is removed which was retained during previous operations
- This cycle will be used at the end of roughing, facing and pattern repeating cycles
- since it is a finishing operation, the material removal rate will be less

Calling a Subroutine



G70 P Q

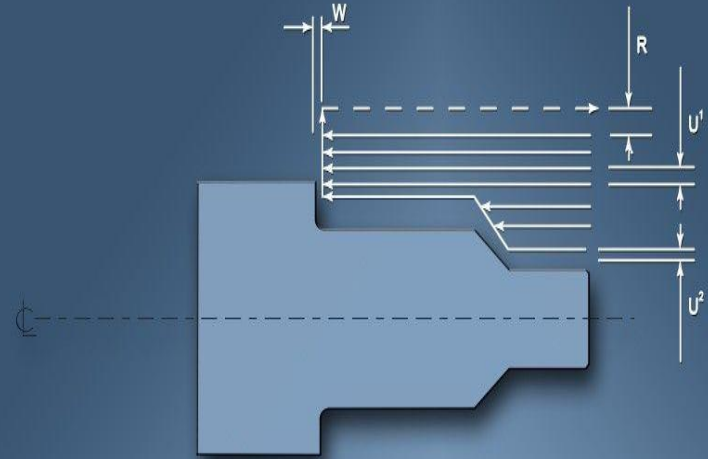
LATHE EXAMPLE

G70 - FINISH TURN CYCLE

P - LINE NUMBER AT START OF SUBROUTINE

Q - LINE NUMBER AT END OF SUBROUTINE

G71 Roughing Cycle



G71 U1.0 R1.0;

G71 P100 Q200 U0.2 W.05 F0.2

G71 - ROUGHING CYCLE

U1.0 (U) - DEPTH OF CUT

R1.0 - RETRACT VALUE

P100 - FIRST LINE OF SUBROUTINE

Q200 - LAST LINE OF SUBROUTINE

U0.2 (U) - AMOUNT LEFT ON FOR FINISHING IN X

W0.05 - AMOUNT LEFT ON FOR FINISHING IN Z

F0.2 - FEED RATE

Roughing Cycle G71 & Finishing Cycle G70

Stock Removal (Roughing Cycle)

G71 U__ R__

U= DEPTH OF CUT PER PASS IN X AXIS

R= RELIEF AMOUNT AFTER EACH PASS

G71 P__ Q__ U__

W__ F__

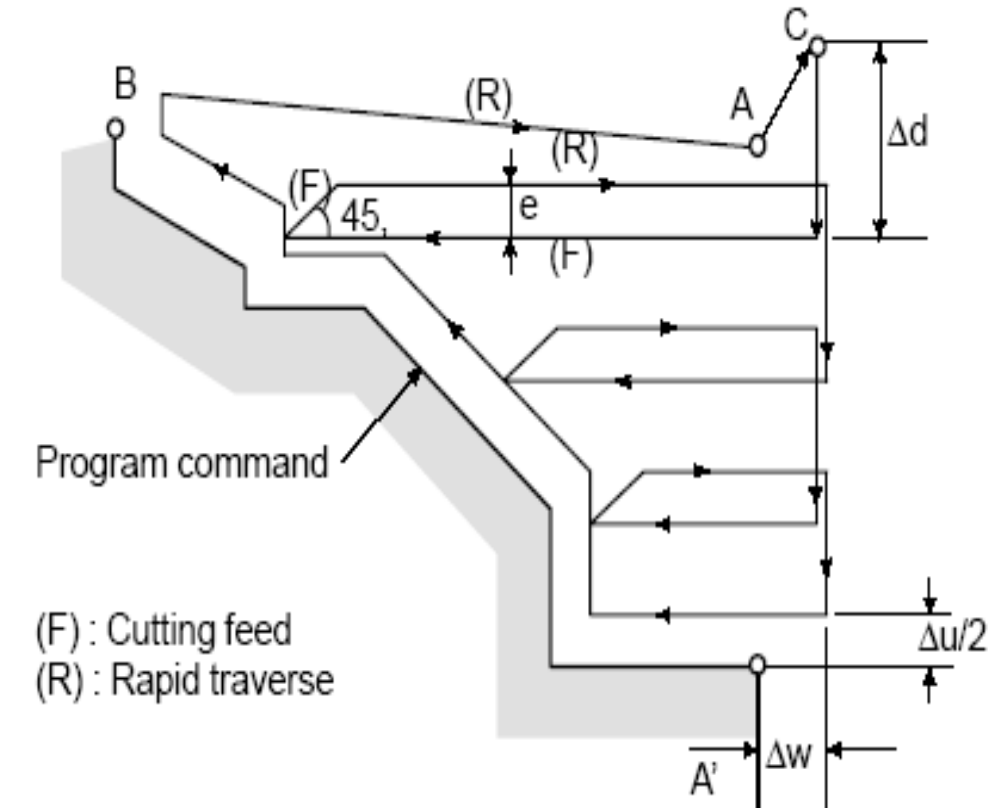
P= PROFILE STARTING BLOCK NUMBER

Q= PROFILE END BLOCK NUMBER

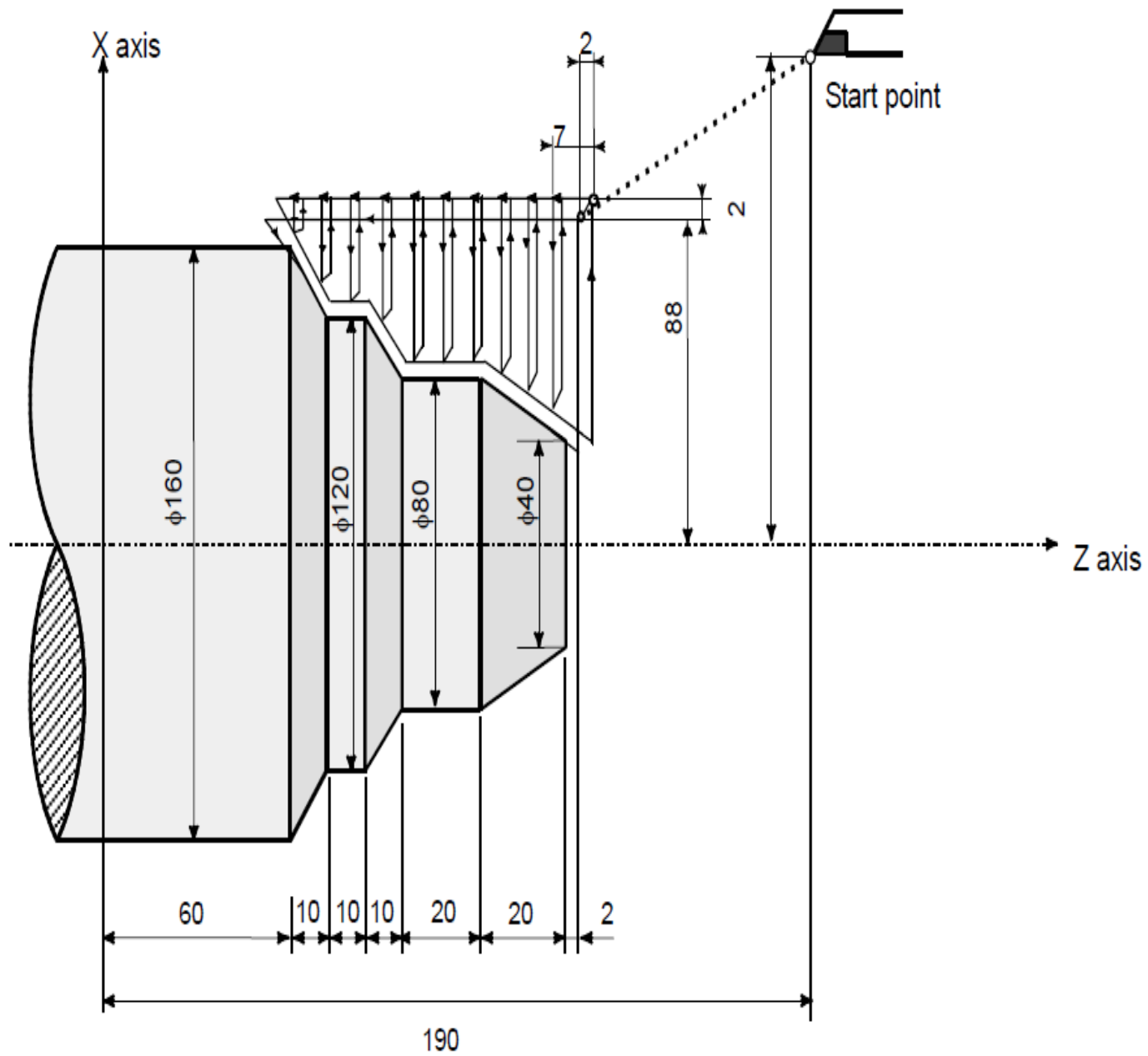
U= FINISHING ALLOWANCE IN X AXIS

W= FINISHING ALLOWANCE IN Z AXIS

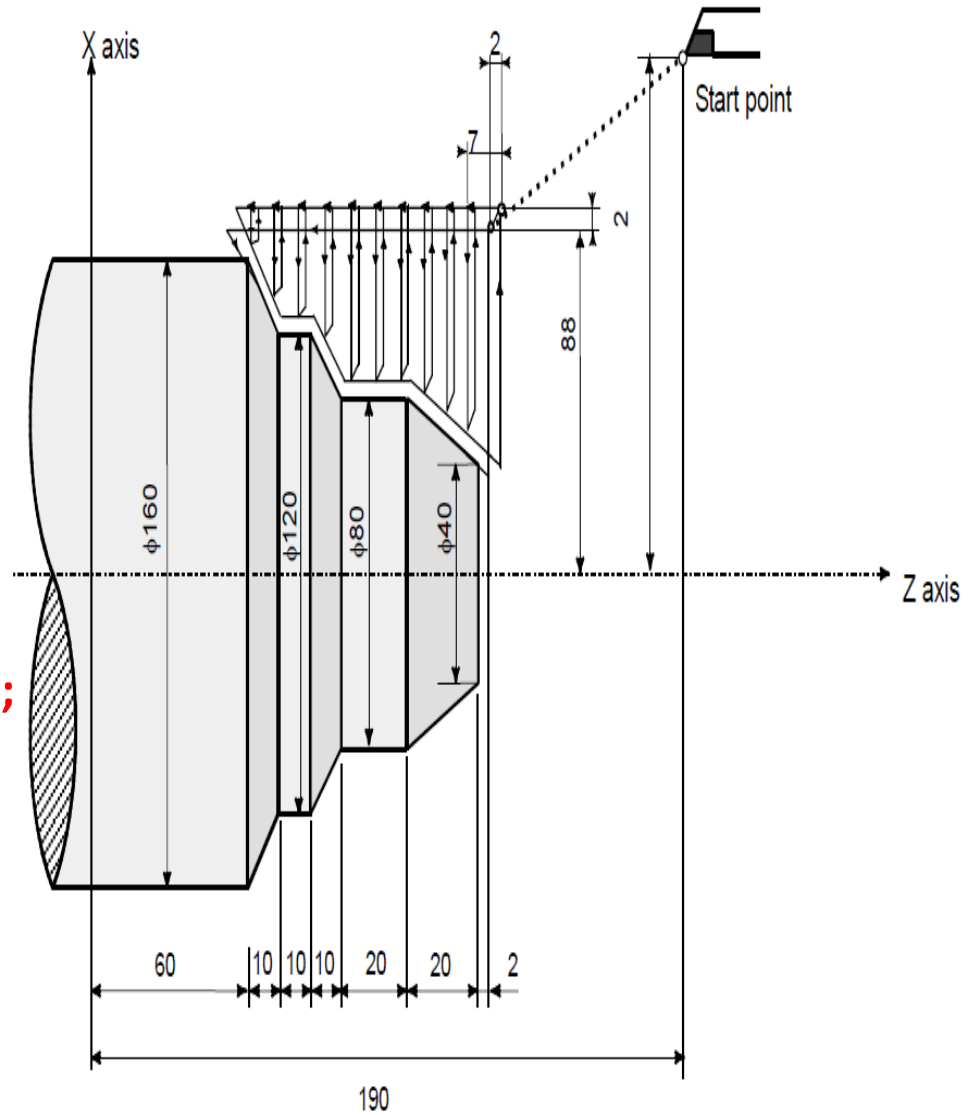
F= FEED RATE



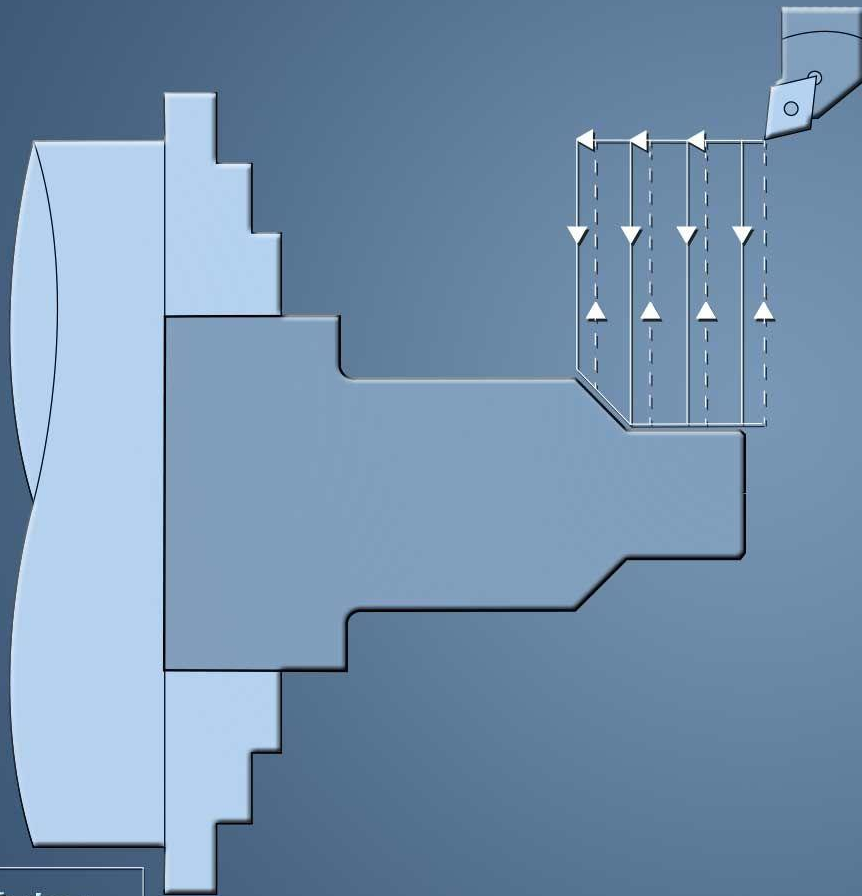
Example



- O0006;
- N10 G21 G99;
- N20 G28 U0 W0;
- N30 T0101;
- N40 S1250 M03;
- N50 G00 X161.0;
- N60 G00 Z0.0;
- N70 G01 X-1.0F0.3;
- N80 G00 Z2.0;
- N90 G00 X161.0;
- N100 G00 Z0.0;
- **N110 G71 U2.0 R1.0;**
- **N120 G71 P130 Q180 U0.25 W0.2 F0.3;**
- N130 G01 X40 F0.3;
- N140 G01 X80 Z-20 F0.5;
- N150 Z-40;
- N160 X120 Z-50;
- N170 Z-60;
- N180 X160 Z-70;
- **N190 G70 P130 Q180;**
- N200 G28 U0 W0 ;
- N210 M05;
- N220 M30;



G72 Facing Cycle



```
G72 W0.8 R2.0;  
G72 P200 Q300 U1.0 W0.5 F0.2;
```

G72 - Facing cycle
W0.8 - Depth of cut
R2.0 - Retract value
P200 - Start of subroutine
Q300 - End of subroutine
U1.0 - Finish allowance in X
W0.5 - Finish allowance in Z
F0.2 - Feedrate

Lathe – Facing Cycle G72

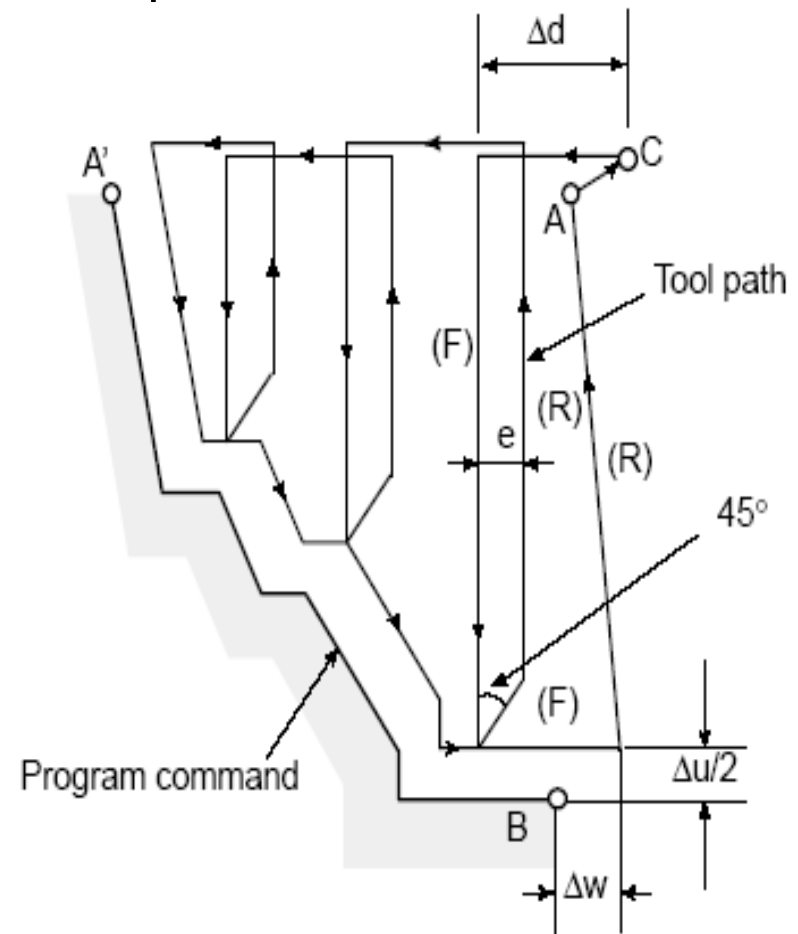
- In this cycle, material will be removed when the tool moves perpendicular to spindle axis over
- Here, the material removal rate is more
- Small amount of material is retained in the work piece which will be removed later during finishing cycle

Stock removal (Facing cycle)

**G72 W__ R__
G72 P__ Q__ U__ W__ F__ S__ T__**

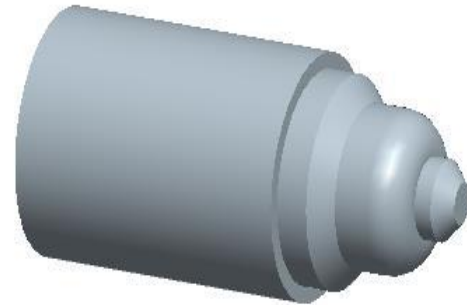
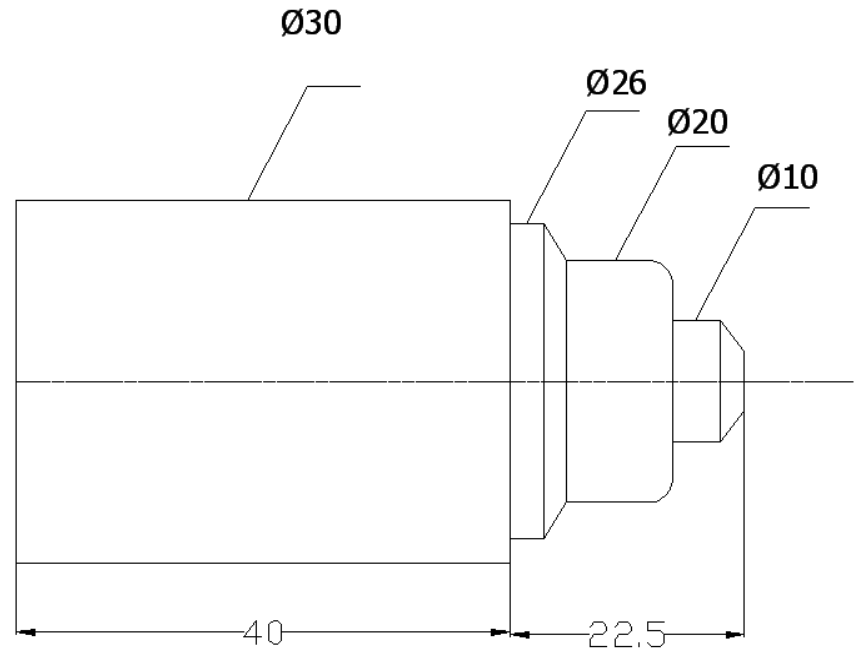
W = DEPTH OF CUT PER PASS IN Z AXIS
R = RELIEF AMOUNT AFTER EACH PASS

P = PROFILE STARTING BLOCK NUMBER
Q = PROFILE ENDING BLOCK
U = FINISHING ALLOWANCE IN X AXIS
W = FINISHING ALLOWANCE IN Z AXIS
F = FEED RATE
S = Spindle Speed
T = Tool SELECTION



Canned Cycle – G72 & G70 (Facing)

- O0003;
- N010 G21 G40 G99;
- N020 G28 U0 W0;
- N030 T0101;
- N040 M03 S1200;
- N050 G00 X31 Z5;
- N060 G01 Z0;
- **N070 G72 U0.5 R1 ; // * Multiple facing * //**
- **N080 G72 P90 Q180 U0.2 W0.2 F30;**
- N090 G01 Z-22.5 F40;
- N100 X30;
- N110 X26;
- N120 Z-17.5;
- N130 X20 Z-15;
- N140 Z-10;
- N150 G02 X10 Z-5 R5 F40;
- N160 G01 Z-2.5 F50;
- N170 X5 Z0;
- N180 X0;
- N190 G70 P90 Q180;
- N200 G28 U0 W0;
- N210 M05;
- N220 M30;



// * Finishing cycle * //

PATTERN REPEATING CYCLE (G73)

- In this cycle, the tool traces the pattern repeatedly and removes a part of material during each repeat
- The number of traces (repeats) are programmed in the cycle.
- Small amount of material is retained in the work piece which will be removed later during finishing cycle

G73 Pattern Repeating Cycle

```
G73 U1.0 W1.0 R3.  
G73 P100 Q200 U0.2 W.05 F0.2
```

```
G73  
U1.0  
W1.0  
R3.  
P100  
Q200  
U0.2  
W0.05  
F0.2
```

- PATTERN REPEATING CYCLE
- DEPTH OF CUT IN X
- DEPTH OF CUT IN Z
- AMOUNT OF ROUGHING PASSES
- FIRST LINE OF SUBROUTINE
- LAST LINE OF SUBROUTINE
- AMOUNT LEFT ON FOR FINISHING IN X
- AMOUNT LEFT ON FOR FINISHING IN Z
- FEED RATE

Format of Pattern Repeating Cycle - G73

G73 U__ W__ R__

G73 P__ Q__ U__ W__ F__

U = Distance and Direction of Relief in X Direction

W = Distance and Direction of Relief in Z Direction

R = The Number of Division (Number Of Passes)

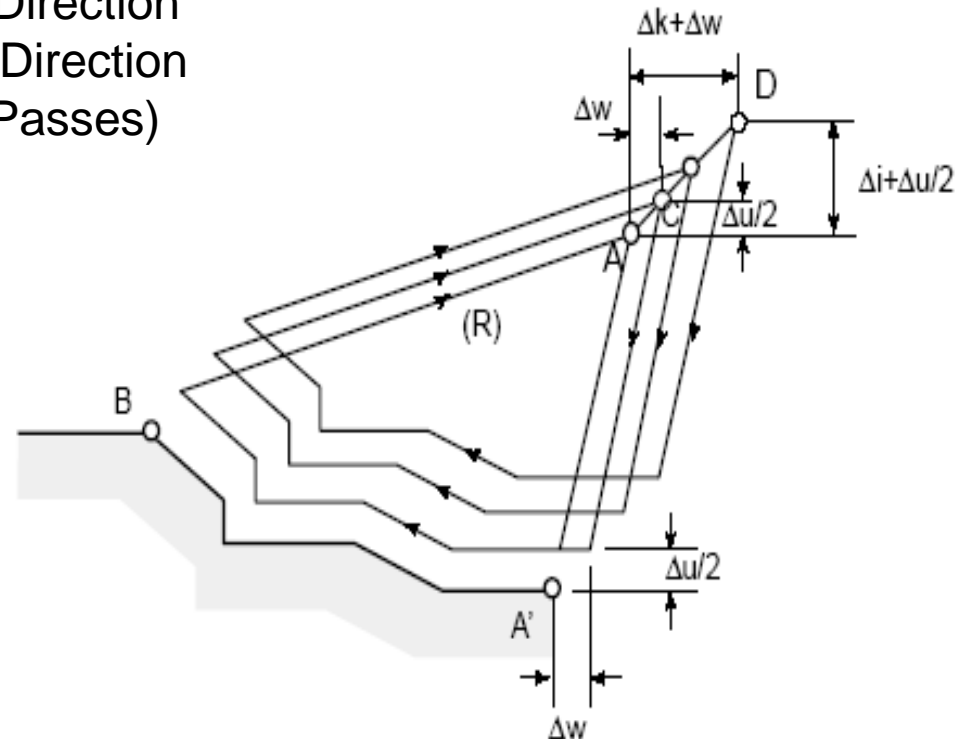
P = Profile Starting Block Number

Q = Profile Ending Block Number

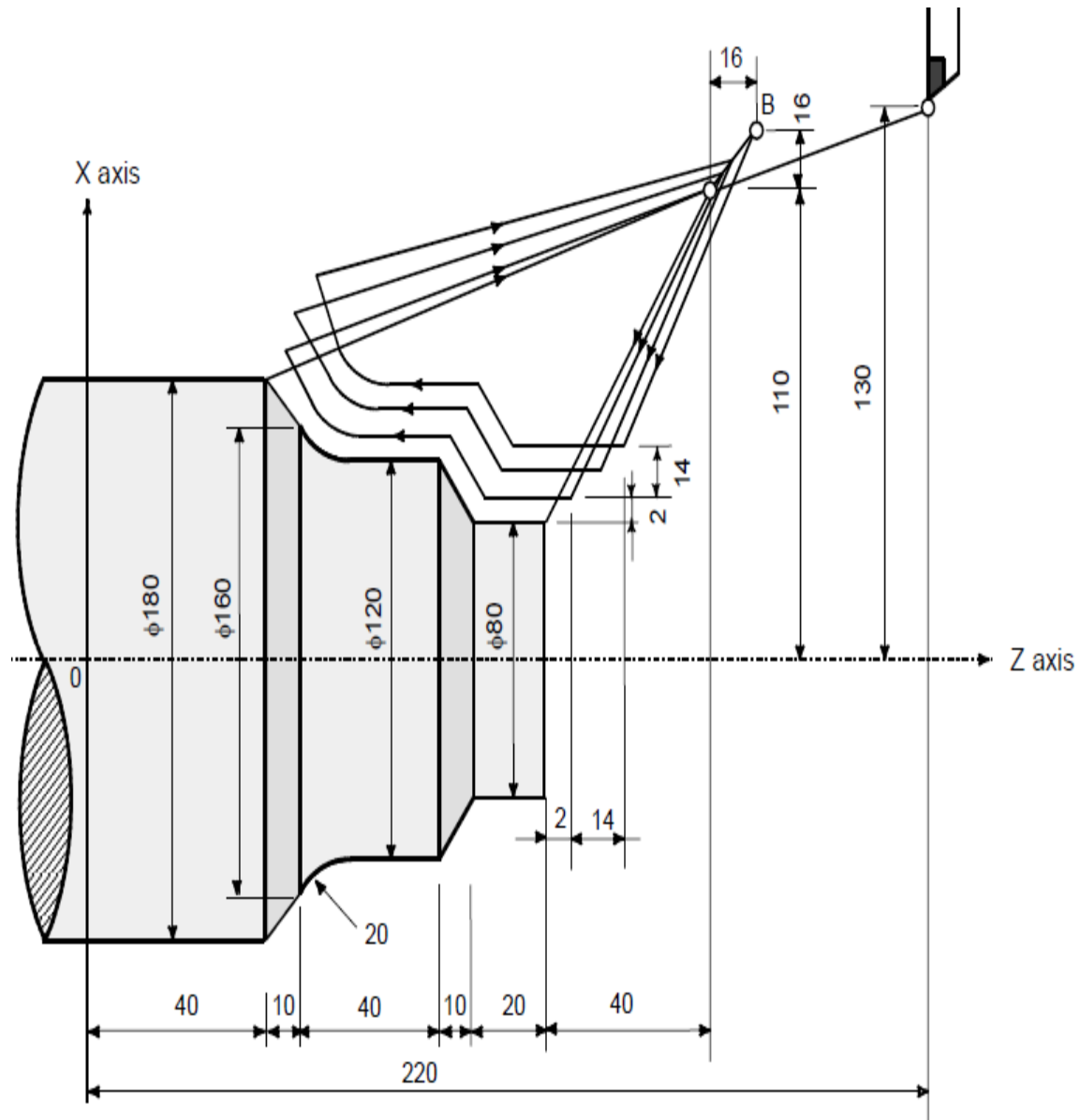
U = Finishing Allowance in X Axis

W = Finishing Allowance in Z Axis

F = Feed Rate



Example



(Diameter designation, metric input)

PECK DRILLING CYCLE (G74)

FORMAT OF PECK DRILLING CYCLE

G74 R__;

G74 Z__ Q__ F__;

R = RELIEF AMOUNT AFTER EACH PASS

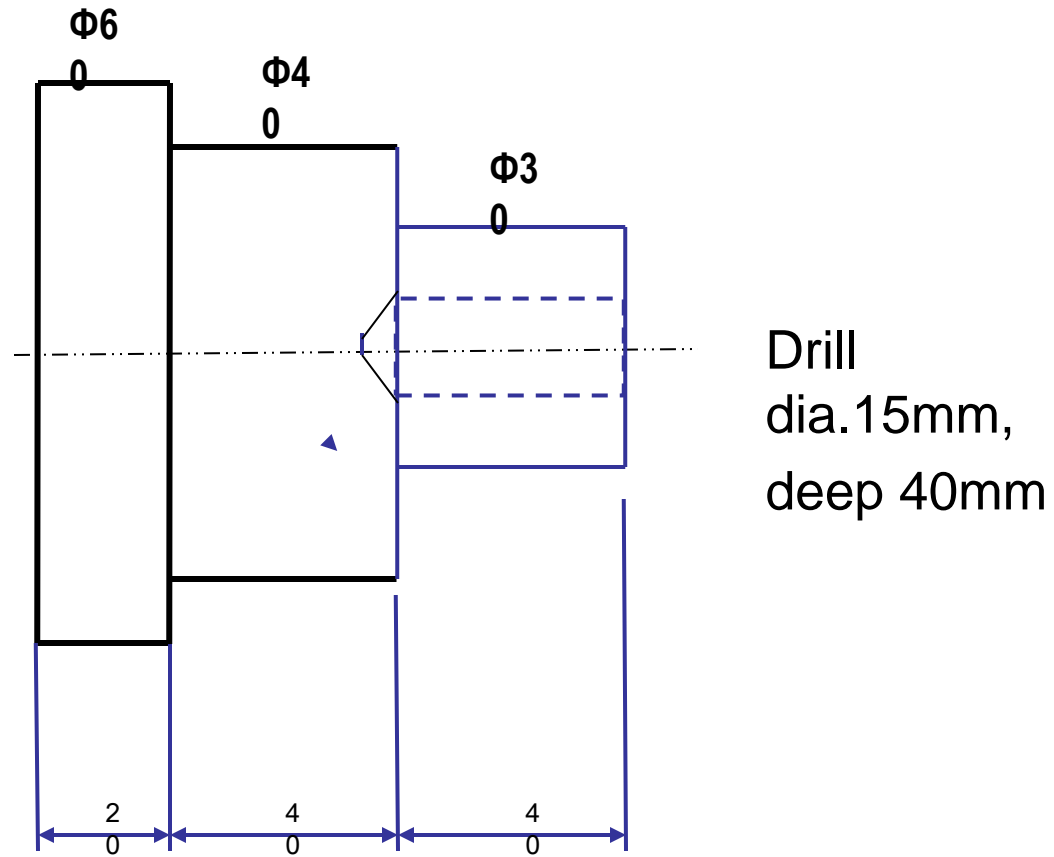
Z = FINAL DRILLING DEPTH

Q = DEPTH OF CUT PER EACH PASS IN MICRON

F = FEED RATE

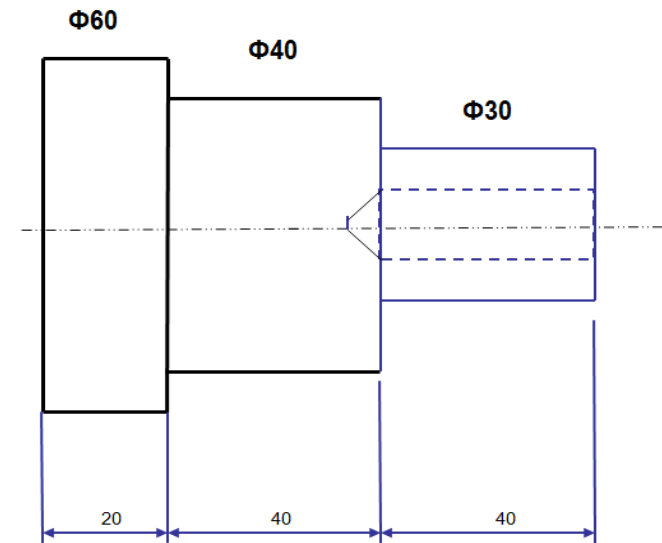
In this cycle, the drilling of total depth will be done by repeated steps. During each step, a part of the total depth (Q) will be drilled

EXAMPLE FOR DRILLING CYCLE



Canned Cycle – G74 (Peck Drilling)

- O0010:
- N10 G21 G99;
- N20 G28 U0 W0;
- N30 T0303; (15mm Ødrill)
- N40 S1250 M03;
- N50 G00 X0.0;
- N60 Z2.0;
- **N70 G74 R2.0;**
- **N80 G74 Z-40.0 Q8000 F0.3;**
- N90 G28 U0 W0;
- N100 M05;
- N110 M30;



GROOVING CYCLE (G75)

FORMAT OF GROOVING CYCLE

```
G75 R__;  
G75 X__ Z__ P__ Q__ F__;
```

R = RELIEF AMOUNT

X = FINAL VALUE OF X AT THE END OF GROOVING

Z = FINAL VALUE OF Z AT THE END OF GROOVING

P = DEPTH OF CUT PER EACH PASS (MICRON)

Q = SHIFTING DISTANCE IN Z AXIS (MICRON)

F = FEED RATE

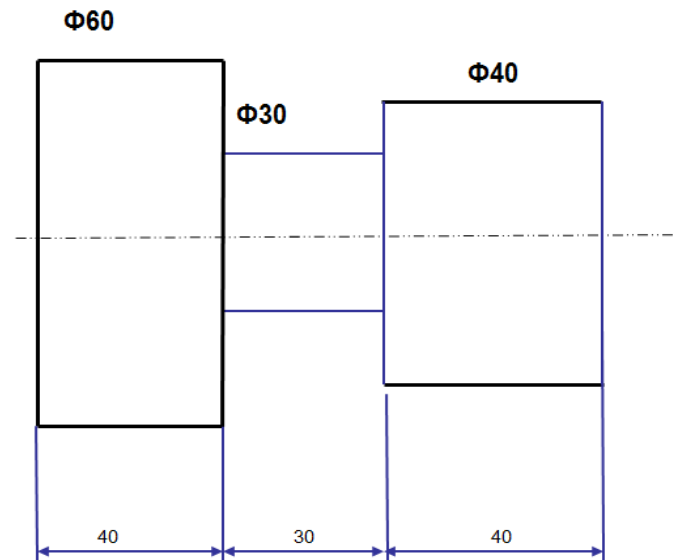
In this cycle,

1. The material of total depth will be removed by repeated steps in X axis.
2. Then the tool is shifted in Z axis for a specified distance

The steps 1 and 2 will be repeated until the groove completion

GROOVING CYCLE (G75)

- N10 G21 G99;
- N20 G28 U0 W0;
- N30 T0404; (Grooving Tool)
- N40 S1250 M03;
- N50 G00 X41.0;
- N60 Z-40.0;
- **N70 G75 R1.0;**
- **N80 G75 X30.0 Z-70.0 P1000 Q5000 F0.3;**
- N90 G28 U0 W0;
- N100 M05;
- N110 M30;



Format of Threading Canned Cycle - G76

G76 P 00 00 00 Q__ R__

G76 X__ Z__ P__ Q__ F__

P 00 = Number of finishing passes

00 = Chamfer distance multiplication factor

00 = Included angle

Q = Minimum depth of cut (Micron)

R = Finishing allowance

X = Minor diameter of the thread

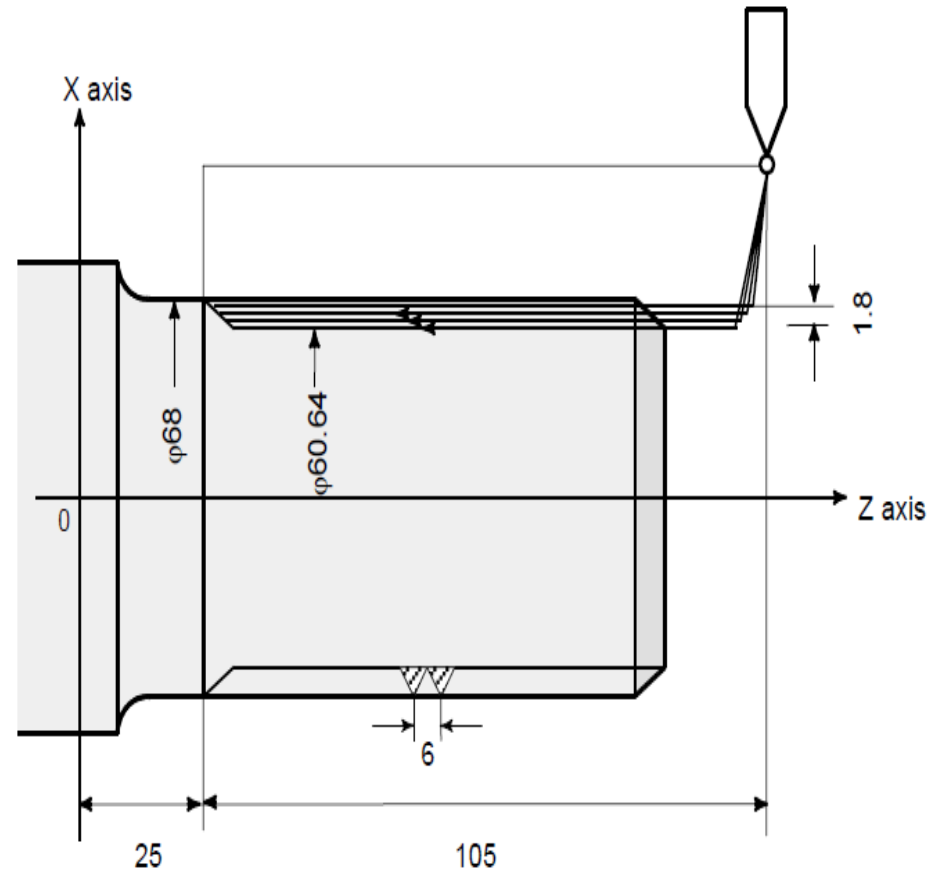
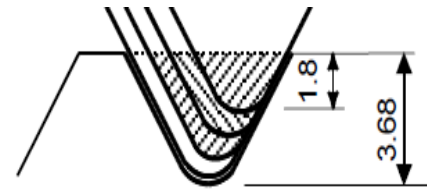
Z = The value of Z at the end of the thread

P = Height of thread (Micron)

Q = First depth of cut (Micron)

F = Pitch

Example



```
G80 X80.0 Z130.0;  
G76 P011060 Q100 R200 ;  
G76 X60.64 Z25.0 P3680 Q1800 F6.0 ;
```

THREADING CYCLE (G76)

Thread specification :

M40x2P

→ Major Dia = 40 mm

→ Pitch = 2 mm

→ Minor Dia = Major Dia – 2 x depth of thread

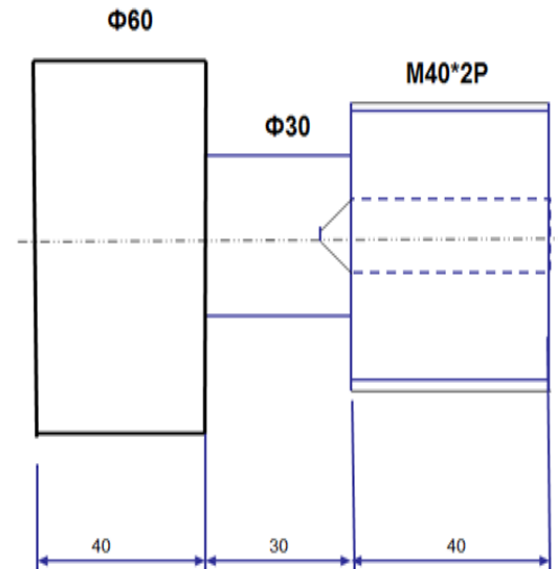
→ Dept of thread = $0.6315 \times \text{pitch} = 1.2630\text{mm}$

Hence Minor Dia = 37.474

(Included angle for metric thread is 60 Deg)

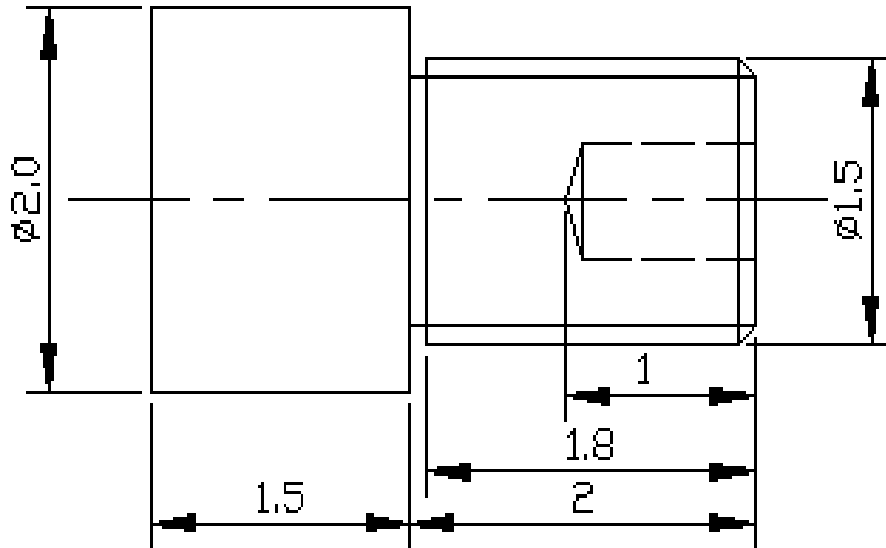
- O0014
- N10 G21 G99;
- N20 G28 U0 W0;
- N30 T0101;(Turning Tool)
- N40 S1500 M0;
- **N50 G00 X81.0; (Facing)**
- **N60 Z0.0;**
- **N70 G01 X-1.0 F0.5;**
- **N80 Z2.0;**
- **N90 G00 X80;**
- **N100 Z0.0;**
- N110 G71 U2.0 R1.0; (Rough turning Cycle)
- N120 P130 Q170 U0.3 W0.2 F0.3;
- N130 G01 X40.0 F0.3;
- N140 Z-70.0;
- N150 X60.0;
- N160 Z-110.0;
- N170 X80.0;
- **N180 G70 P130 Q170; (Finishing Cycle)**
- N190 G28 U0W0;
- N200 M05;

- **N210 T0202 ;(Grooving Tool)**
- **N220 S1500 M03;**
- **N230 G00 X41.0;**
- **N240 Z-40.0;**
- **N250 G75 R1.0;**
- **N260 G75 X30.0 Z-70.0 P1000 Q1000 F0.3;**
- **N270 G28 U0 W0 ;**
- **N280 M05;**
- **N290 T0303; (Drilling Cycle)**
- **N300 S1500 M03;**
- **N310 G00 X0.0;**
- **N320 Z2-0;**
- **N330 G74 R2.0;**
- **N340 G74 Z-50.0 Q8000 F0.3;**
- **N350 G28 U0 W0;**
- **N360 M05;**
- **N370 T0303;(Threading Cycle)**
- **N380 S1200 M03;**
- **N390 G00 X41.0;**
- **N400 Z-40.0;**
- **N410 G01 X40.0 F0.3;**
- **N420 G76 P020860 Q100 R0.01;**
- **N430 G76 X37.474 Z0.0 P1263 Q200 F2.0;**
- **N440 G00 X42.0;**
- **N450 Z2.0;**
- **N460 G28 U0 W0;**
- N470 M05;
- N480 M30;



CNCEZ TURNING - IV

CNCEZ THREAD CUTTING SPECIMEN DRAWING



ALL DIMENSIONS ARE
IN Inches

Program:

:%

: 1004

N05 G90 G20

N10 M03 S1500

N15 M06 T01

N20 G00 X0 Z0

N25 G01 X1.5 Z0 F10

N30 G01 X1.5 Z-2

N35 G01 X2.0 Z-2

N40 G00 X1.5 Z2

N45 M06 T03

N50 G00 X1.5 Z0

N55 G76 X1.5 Z-1.8 D625 K.125 A45 F0.1

N60 G00 X2 Z2

N65 M06 T12

N70 G00 X0 Z0

N75 G74 X0 Z-1 F1 D0 I0.25 K.125

N80 G00 Z1

N85 M05

N90 M30

Common G-Code for Lathe

G77 X__ Z__ Q__ R__ F__ ;

Canned cycles (side turning) , (X,Z) – end point coordinate, Q – depth of each cut, R – depth of taper, F – feedrate

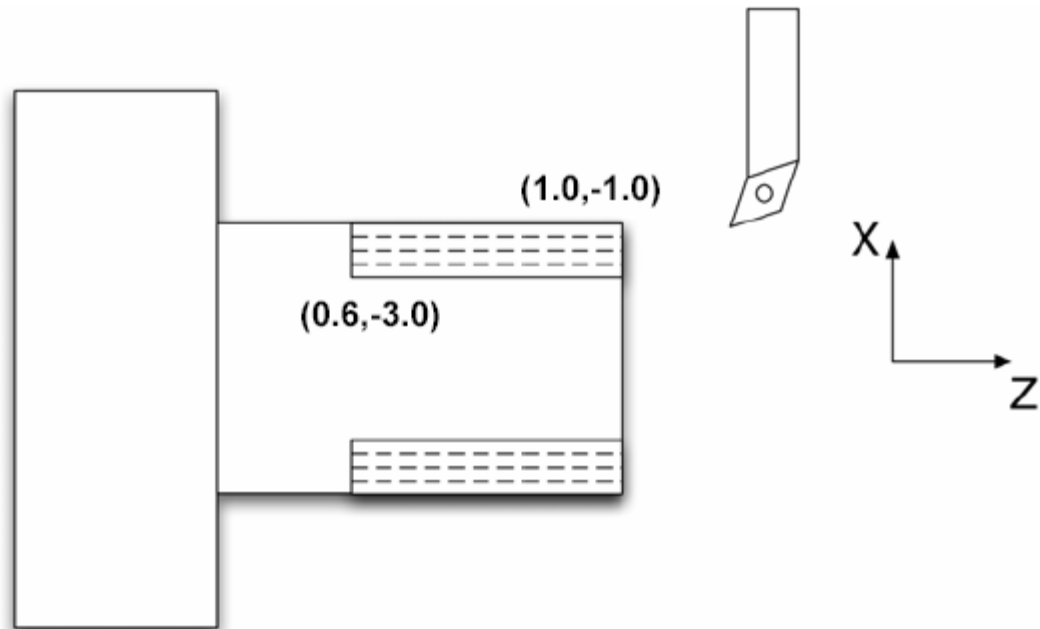
straight side turning: X, Z, F

roughing cuts: X, Z, Q, F

making tapers: X, Z, Q, R, F

G80 ;

Cancel a canned cycle, and resume normal operation (G00 or G01 can also cancel canned cycles)



Canned cycle side turning:

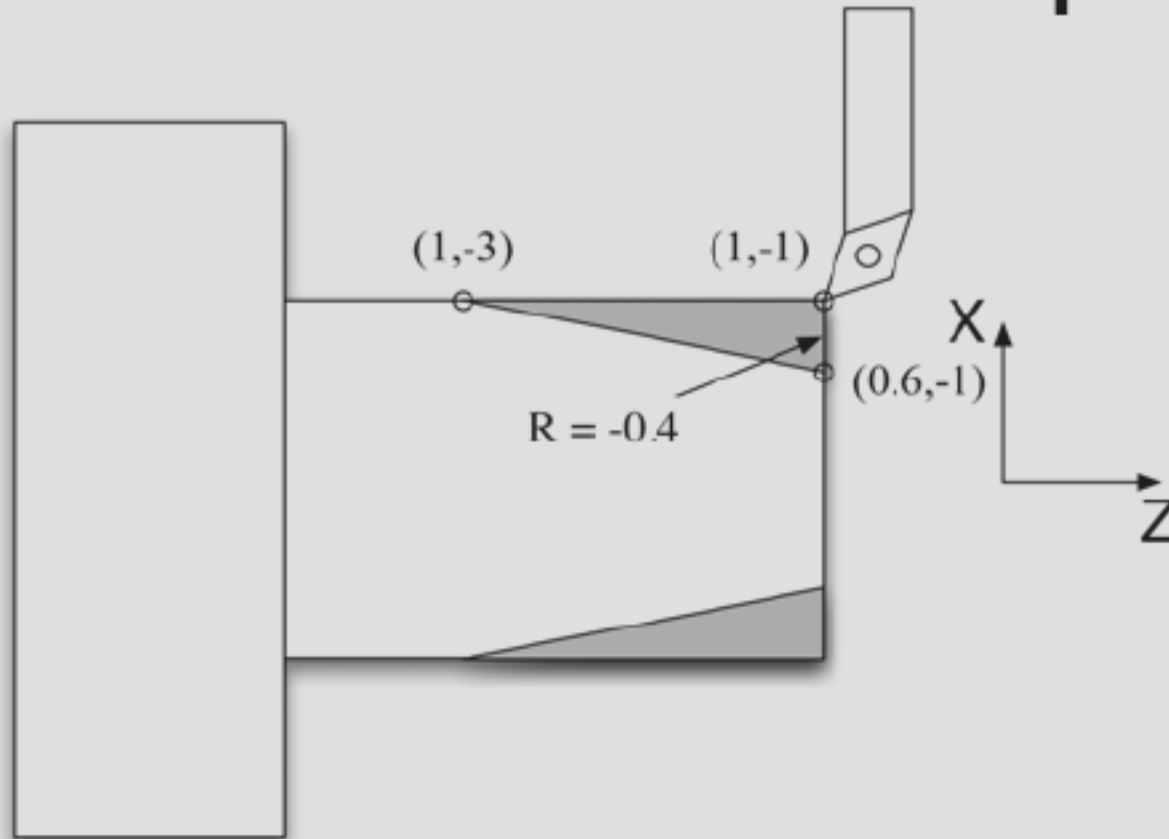
N001 G01 X1.0 Z-1.0 F10

N002 G77 X0.6 Z-3.0 Q0.1 F10

N003 G80

X,Z are end point coordinates, Q is depth of each cut

Canned Full Positive Taper



Canned cycle side turning with taper:

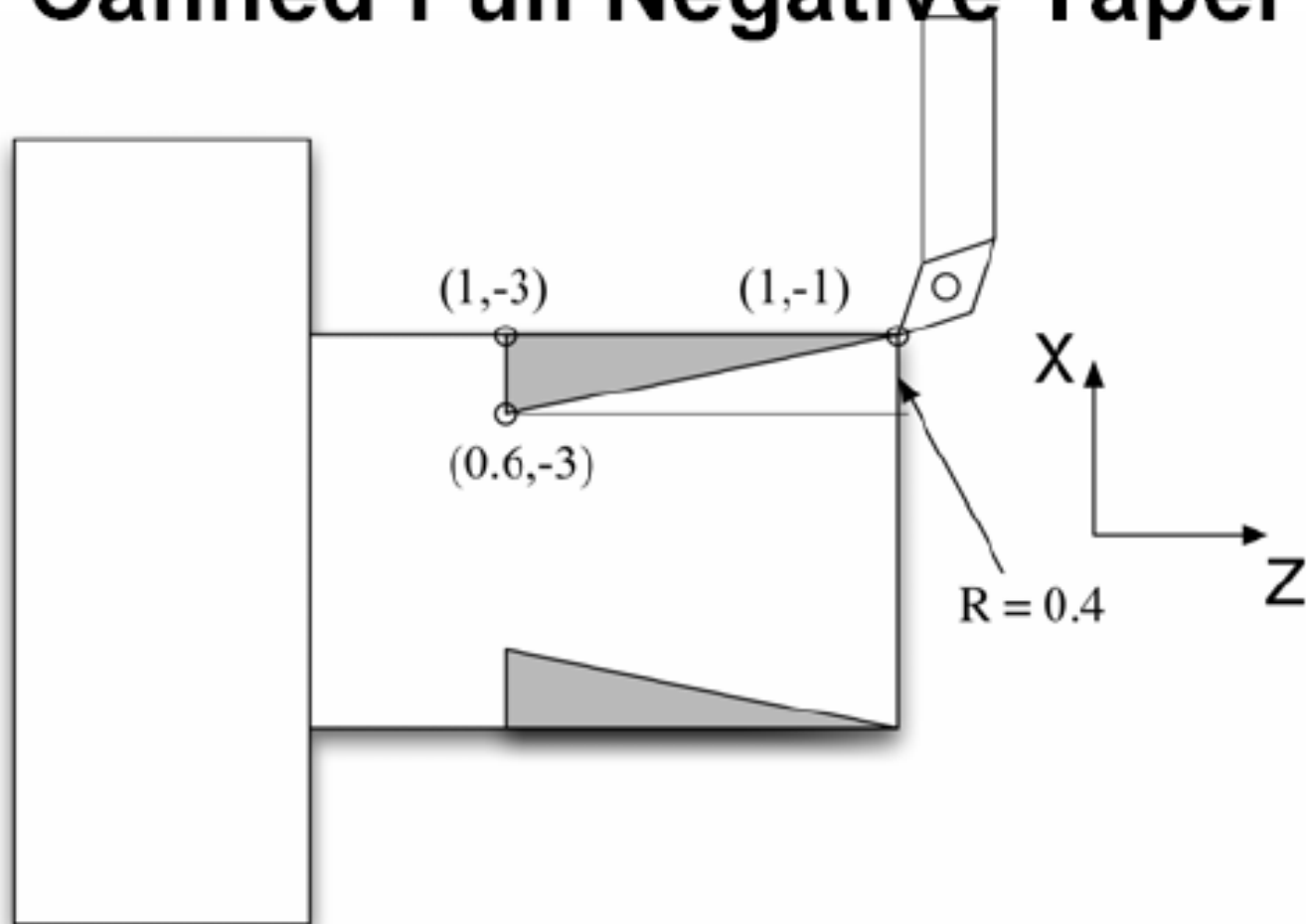
N011 G01 X1.0 Z-1.0 F10; Taper start point

N012 G77 X1.0 Z-3.0 Q0.1 R-0.4 F10; Taper end point

N013 G80

;R is depth of taper, negative values for positive taper

Canned Full Negative Taper



Canned cycle side turning with taper:

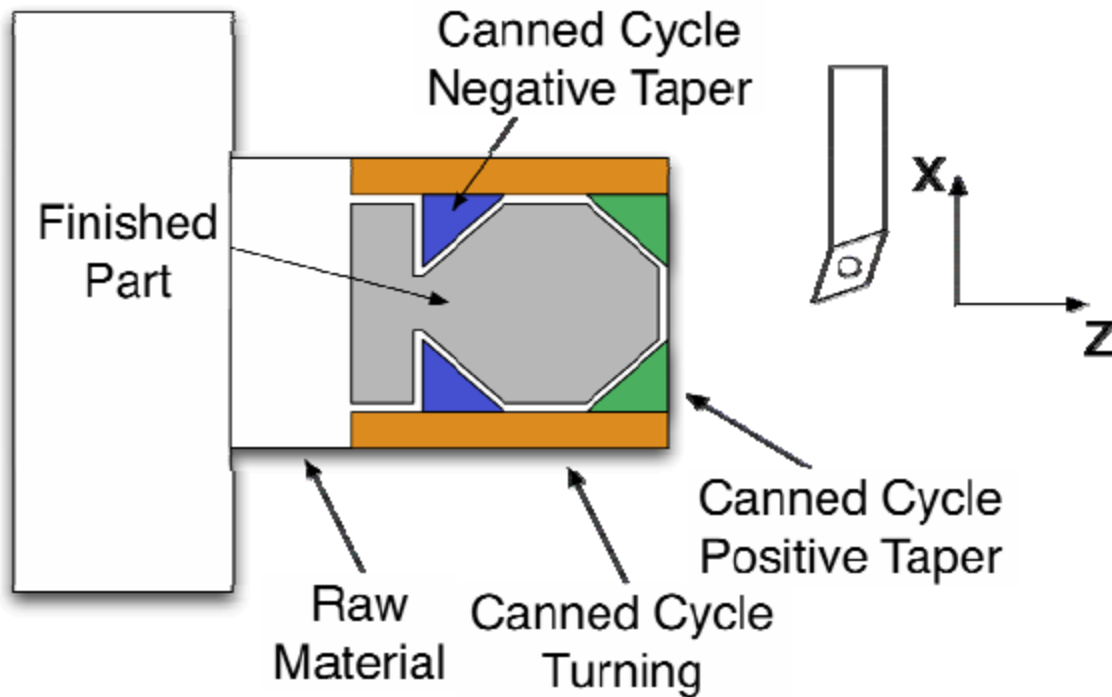
N011 G01 X1.0 Z-1.0 F10; Taper start point

N012 G77 X0.6 Z-3.0 Q0.1 R0.4 F10; Taper end point

N013 G80

;R is depth of taper, positive values for negative taper

Machining Sequence



- Rough cut using canned cycles
- Finish cut along part profile
- Cut part off

Rough Cut

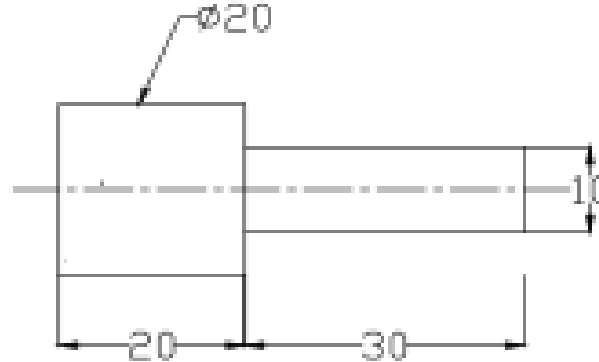
Roughing – primary considerations:

- Just removing metal, surface finish does not matter.
 - Requires a strong cutter.
 - Generally have deep depth of cuts and fast feed rates.
 - The cutting speed is generally adjusted slower to keep heat down.
- Finishing – primary considerations:
 - Must meet required surface finish and size specifications.
 - Requires a hard cutter to hold its shape well.
 - Generally have small depth of cuts and slow feed rates.
 - The cutting speed is generally adjusted upward to give a better surface finish.

Program 2 – Step Turning using G90 cycle

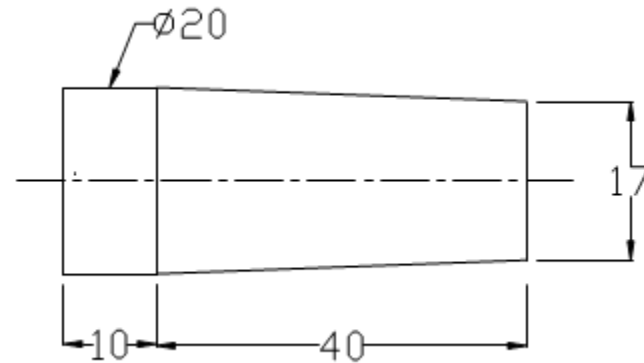
- O0002;
- N010 G21 G40 G98;
- N020 G28 U0 W0;
- N030 M06 T01;
- N040 M03 S1200;
- N050 G00 X21 Z2;
- **N060 G90 X20 Z-30 F60;**
- N070 X19;
- N080 X18;
- N090 X17;
- N100 X16;
- N110 X15;
- N120 X14;
- N130 X13;
- N140 X12;
- N150 X11;
- N160 X10;
- N170 G28 U0 W0;
- N180 M05;
- N190 M30;

/** Box turning cycle */



Program 3 - TAPER TURNING

- O0011
- N010 G21 G40 G98
- N020 G28 U0 W0
- N030 M06 T01
- N040 M03 S1200
- N050 G00 X21 Z2
- G90 X20 Z-40 R0 F60
- N070 X20 R-0.5
- N080 X20 R-1
- N090 X20 R-1.5
- N100 G28 U0 W0
- N110 M05
- N120 M30

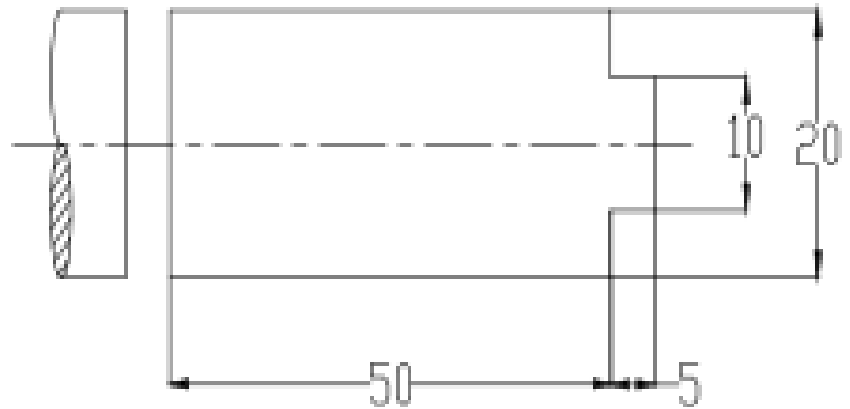


*/** Box turning cycle */*

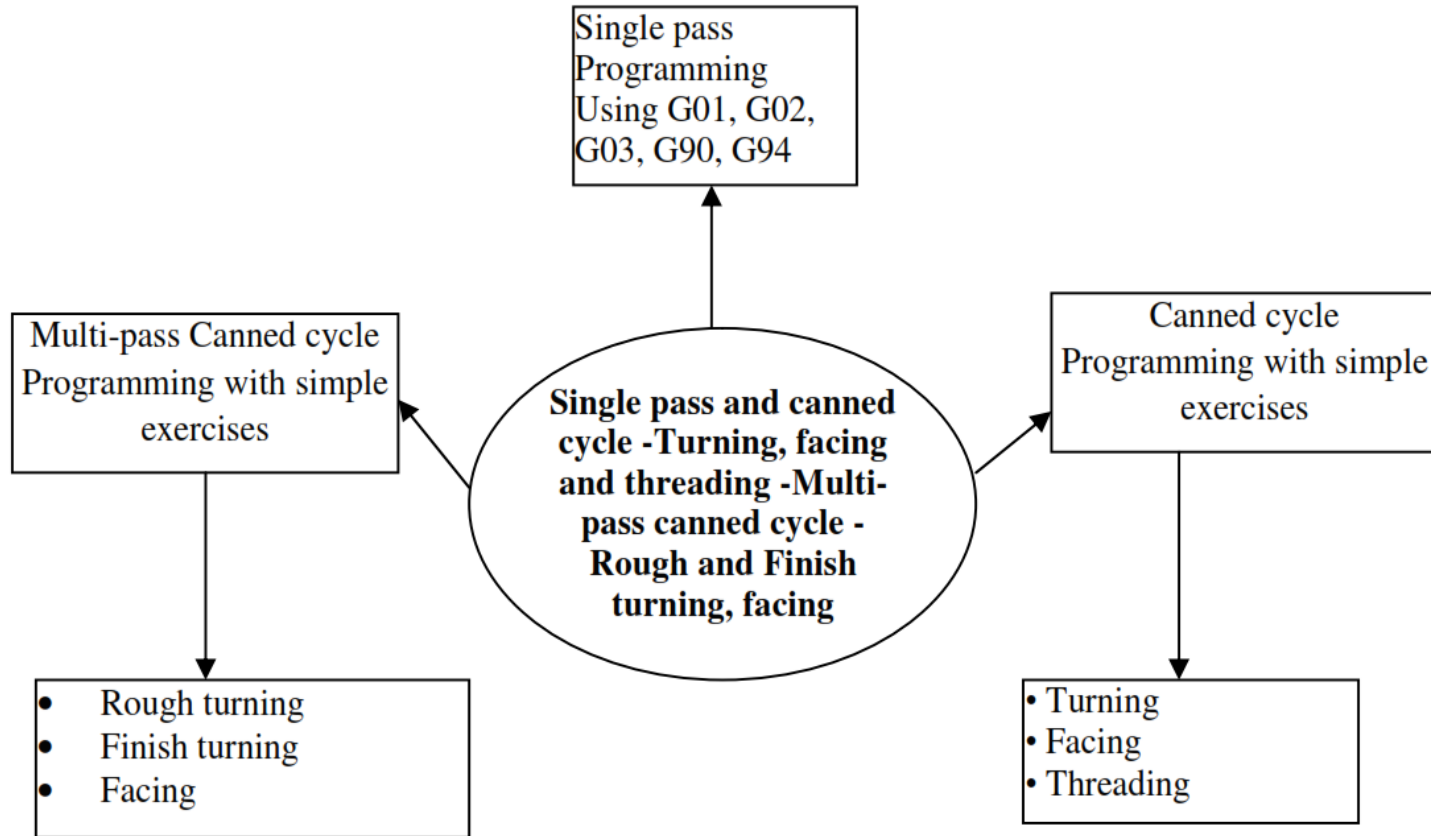
$$\begin{aligned} R &= (\text{Minor dia}) / 2 \\ &= (17 - 20) / 2 \\ &= -3/2 = -1.5 \end{aligned}$$

Program 4 - Step Facing Cycle (G94)

- O0015
- N010 G21 G40 G98
- N020 G28 U0 W0
- N030 M06 T05
- N040 M03 S1200
- N050 G00 X21 Z2
- N060 G94 X10 Z-1 F60
- N070 Z-2
- N080 Z-3
- N090 Z-4
- N100 Z-5
- N110 G28 U0 W0
- N120 M05
- N130 M30



Concept Map



Discussion



10 mins