



Basic ISO Programming Exercise 3

Turning



This document is made available as a preliminary version (draft).
Questions and feedback should be sent to support@cimco.com

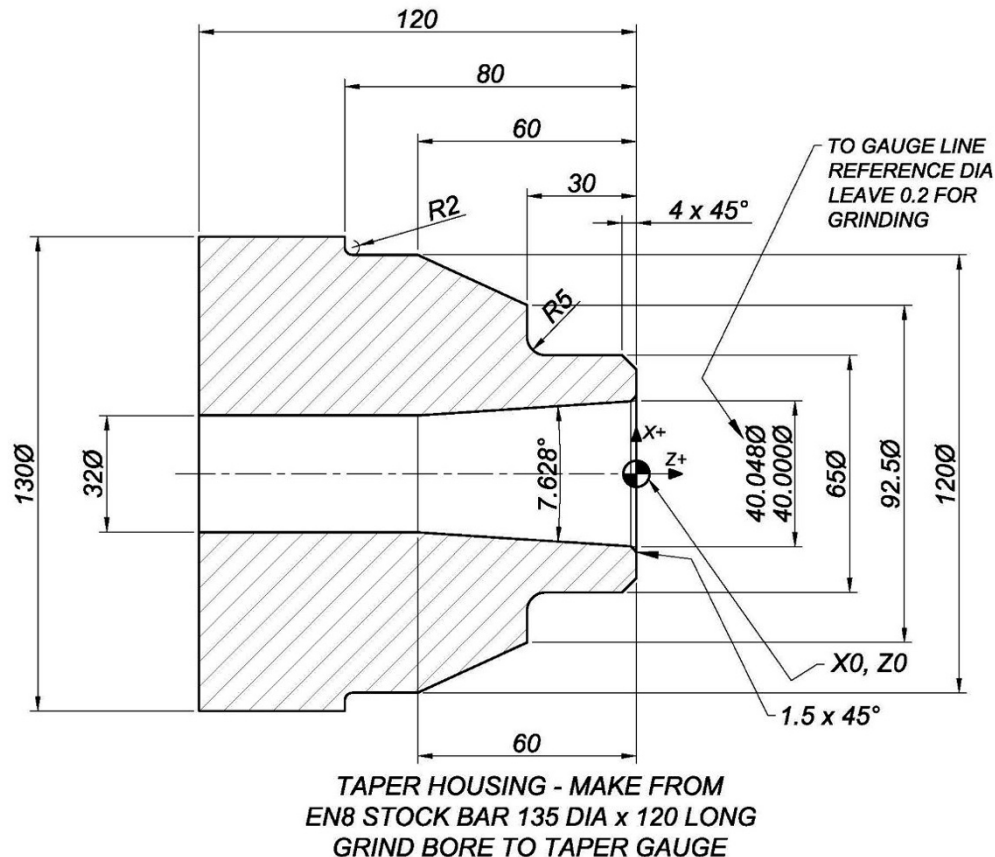
Table of Contents

1	Introduction	3
2	Defining the contours	5
2.1	Linear interpolation.....	5
2.2	Circular interpolation	6
3	Tool radius compensation	8
3.1	Roughing cycles.....	11
3.2	Header information.....	12
4	Roughing cycle.....	15
5	Set up stock dimensions and tools.....	28
5.1	Stock setup	28
5.2	Tool setup	30

1 Introduction

See below a drawing of the Housing with a taper bore. We will work through the programming using ISO G code to prepare a program to machine the outside profile, drill and bore the taper inside profile.

ISO G code is used by many CNC control manufactures and the main groups of G codes for move commands, unit designation, orientation of axis, spindle speeds, rates of feed are generally the same. Some other G codes may differ from one CNC control to another. The G codes and programming principles used here will be generally in line with Fanuc, Siemens, Haas, Fagor and other CNC controls.



We are using this part as drawn above to create a CNC programming tutorial for turning and boring the part only. The programming will be broken down into sections and then we will add the stock blank size and the individual tools to achieve a graphic Backplot that will give a true indication of the operation on the machine.

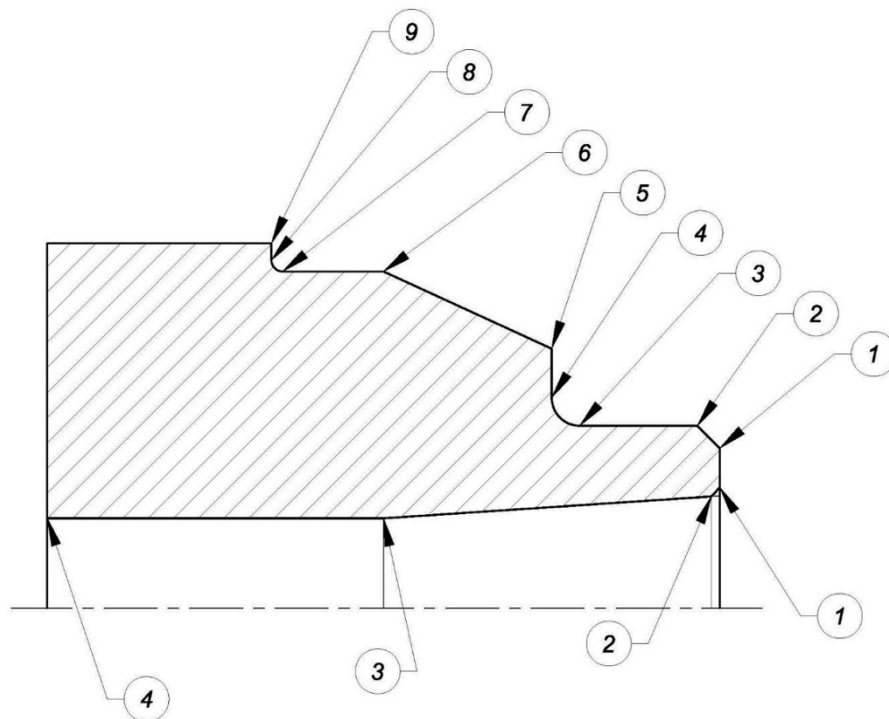
CNC PROGRAMMING SECTIONS

1. Rough turn the outside profile
2. UDrill the bore
3. Rough bore the inside profile
4. Finish turn the outside profile
5. Finish bore the inside profile.

We will consider programming the outside profile first. When programming the part, we will use the drawing dimensions to describe the tool path. The program will be made up of line by line (block by block is the term used) information that will comprise straight lines and arcs. The basic move commands that make up Group 1, ISO programming instructions are modal commands. This means that after a block with one of these commands, following blocks with X, Y, coordinates positioning moves will be carried out in the same mode. The commands are as follows:

1. G00 – Straight line moves at rapid speed (On some machines this move is made in a vector line and on others a 45-degree move is followed by a single axis move to achieve the final programmed position). This is a modal command.
2. G01 – Linear interpolation blocks will be carried out in linear vectored moves at the feed rate programmed. This is a modal command.
3. G02 – Circular interpolation clockwise moves at the feed rate programmed. This is a modal command.
4. G03 - Circular interpolation counterclockwise moves at the feed rate programmed. This is a modal command.

See the diagram below that has the external and internal profile broken down to the points on the profile where elements start and finish. The programming X and Z zero point is shown by the checkered circle in the original drawing so the dimensions for the external and internal profile will be in the X+, Z- quadrant.

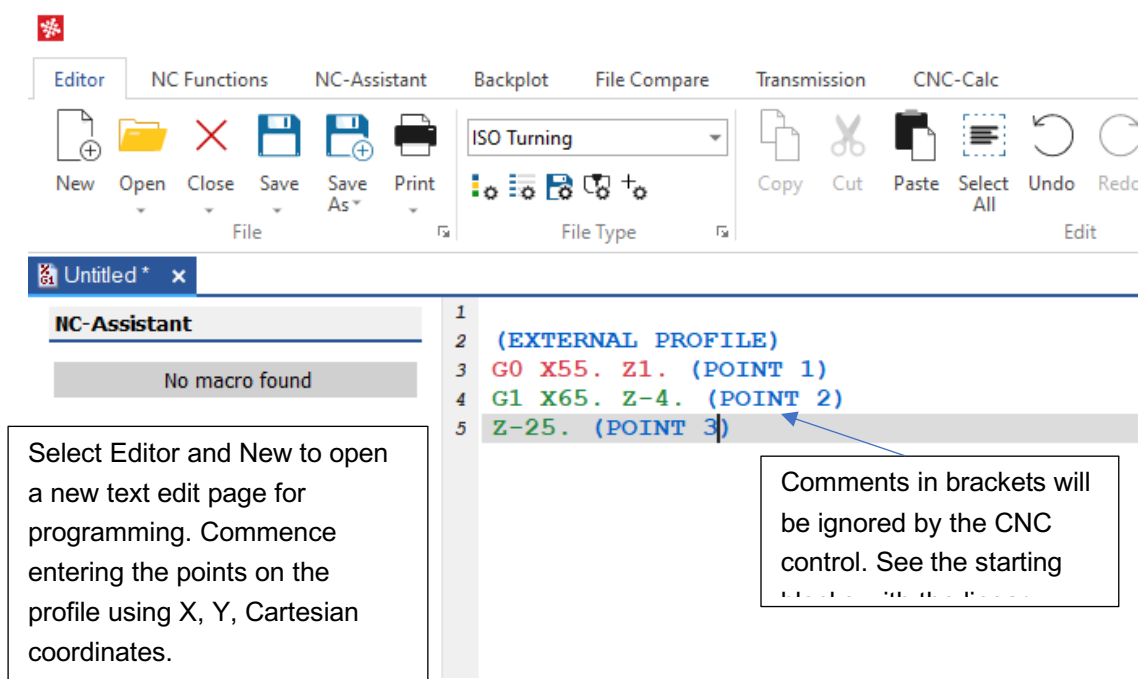


2 Defining the contours

So let us begin entering points on this profile directly into the Editor.

2.1 Linear interpolation

The linear interpolation points are very easy to enter. If we have only one axis command X or Z on a line, then a move in a straight line in that axis will take place. If we have an X and a Z on the same line with a G1 or G01 prefix, (G1 or G01 is the same command, the CNC will interpret either) then a linear interpolation will take place with both axis moving in a direct accurate vector to finish together at the commanded positions at a feed rate as designated in the block or in an earlier block as feed is also modal. Enter the first few lines of linear moves, see below:



The screenshot shows the CIMCO Editor software interface. The top menu bar includes Editor, NC Functions, NC-Assistant, Backplot, File Compare, Transmission, and CNC-Calc. Below the menu is a toolbar with icons for New, Open, Close, Save, Save As, Print, File Type, Copy, Cut, Paste, Select All, Undo, and Redo. The main window displays a CNC program with the following lines:

```
1  
2 (EXTERNAL PROFILE)  
3 G0 X55. Z1. (POINT 1)  
4 G1 X65. Z-4. (POINT 2)  
5 Z-25. (POINT 3)
```

Two callout boxes provide additional information:

- Left box: "Select Editor and New to open a new text edit page for programming. Commence entering the points on the profile using X, Y, Cartesian coordinates."
- Right box: "Comments in brackets will be ignored by the CNC control. See the starting" (with an arrow pointing to the comment in line 5).

2.2 Circular interpolation

After point 3 comes a circular interpolation move and more information is required for the CNC control to carry out this move. We have already entered the start point 3 and we are going to point 4 in a clockwise direction the block will start

G2 X75. Z-30.

The CNC control must have the arc centre fixed to be able to interpolate this move. The arc centre is fixed generally by its coordinates from the start point. The arc centre coordinates have the designation I for X and K for Z. So, from the start point the I and K are entered as incremental coordinates as below.

G2 X75. Z-30. I5. K0

Note that the I dimension is +5. and remember that arc centers are incremental from the start position and as $I = X$ arc centre then $I = 5$. (a positive sign is not needed) Note that $K = 0$. which is the position of the arc centre on the Z axis from the start position of the move. The CNC Control now has all the information to make this circular move.

Note!! It is also possible to use radius designation instead of I & K but then the maximum arc is 180 degrees. Although there are some CNC controls that prefer only I & K arc centre designations. Some CNC controls permit the use of arc centre designation from the absolute zero position rather than incrementally from the start point but here we will use I & K from start point as this method is generally acceptable and will concentrate the student's attention on the structure of the profile. When programming a specific CNC machine, the CNC control programming manual may need to be referred to.

Carry on with the coordinate entry in the editor. After three more simple straight-line moves, we come to another circular interpolation between point 7 & 8. It is a clockwise move again so a G2 move, and we will be at the start position going to the end position at point 8.

G2 X124. Z-80.

From the part drawing and from the start position looking at the drawing above:

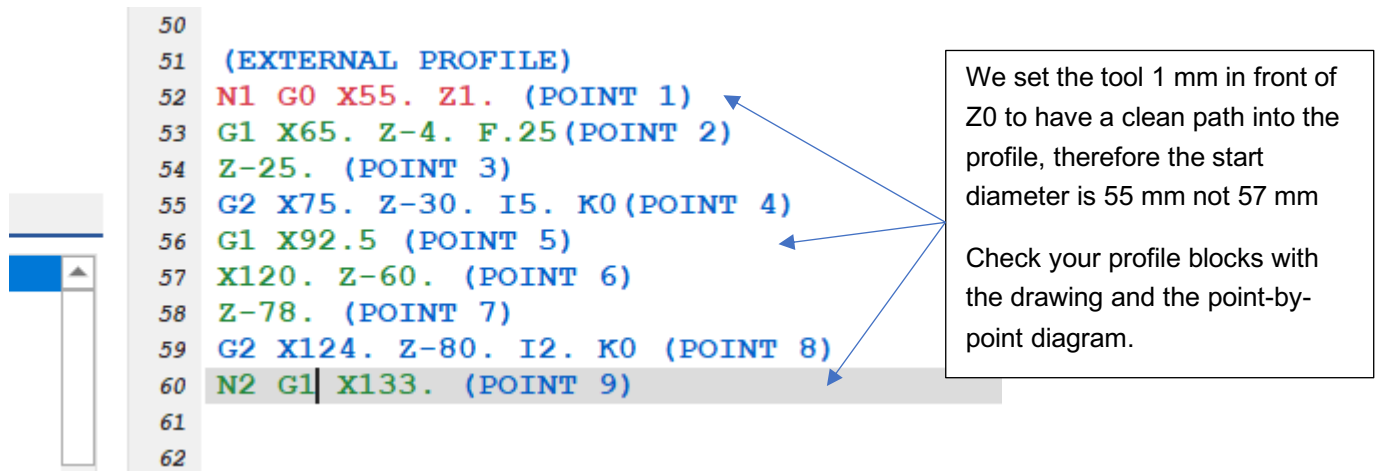
$I = 2$. (The arc centre is a + figure from the start point)

$K = 0$ (The arc center lays at 0 (zero) on the Z axis)

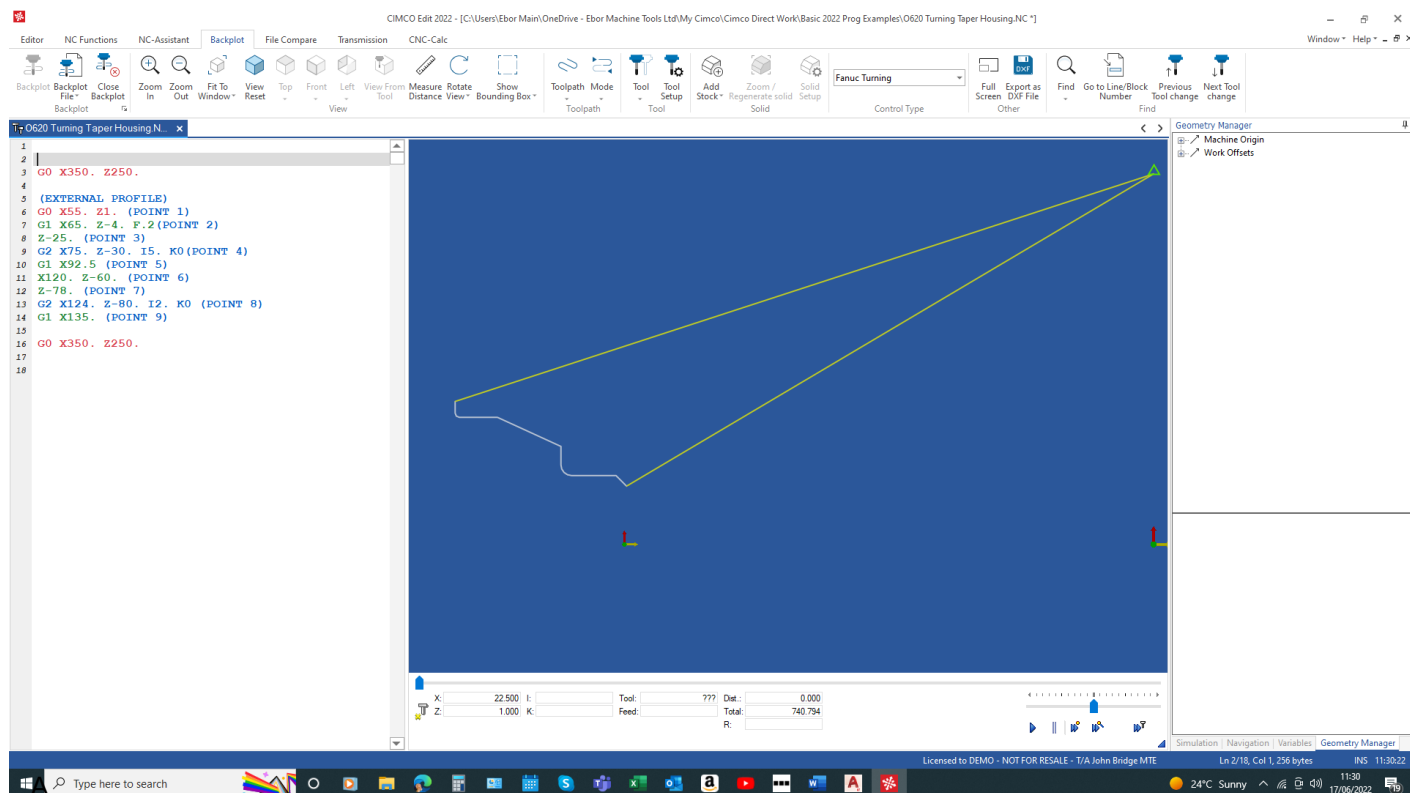
The completed block will read.

G2 X124. Z-80. I2. K0

Don't forget that the move commands are modal. Therefore, we must go back to linear interpolation (G1) to continue with the profile if the next block is a linear move. See below how your profile should look.



We can now test the profile with Backplot and if it is correct then it will look like this



Further reminder - do not forget to change from circular to linear from element to element as the profile changes.

All group 1, move commands are modal e.g., after a G2 block if you were to enter X Z coordinate the CNC control will still try to move circular unless you place a G01 at the start of the block to define that this move is now linear.

3 Tool radius compensation

Now we have a profile that is working we should apply tool radius compensation to the profile but before doing this let's look at Radius compensation on CNC Lathes and why we need it. The diagrams below illustrate the problem.

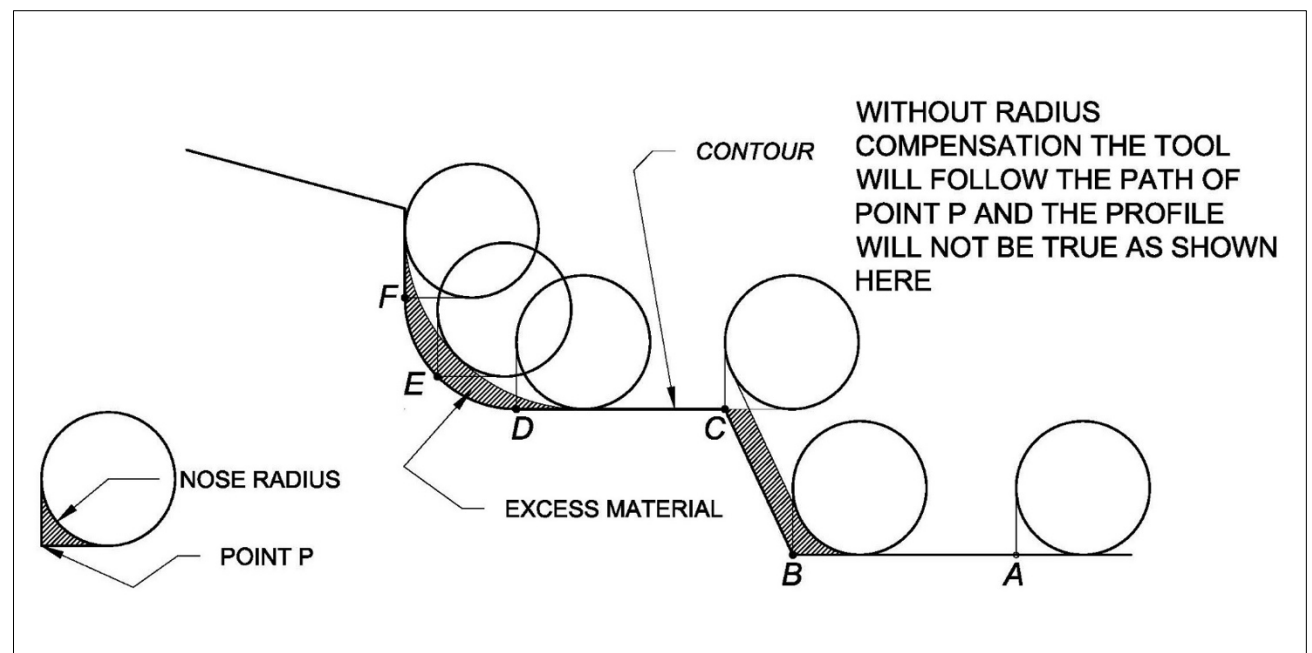
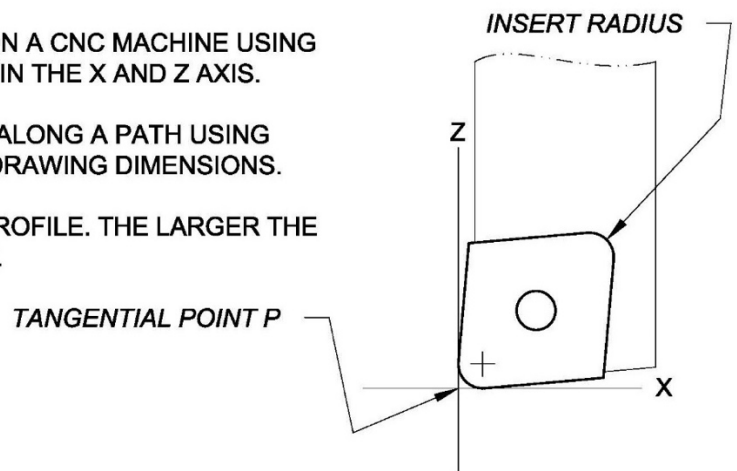
NOTE!

WHEN WE CREATE A CNC PROGRAM WE USE THE DRAWING DIMENSIONS AS IT IS CONVENIENT AND MAKES THE PROGRAM EASIER TO FOLLOW.

TOOL OFFSETS ARE SET & MEASURED ON A CNC MACHINE USING THE TANGENTIAL POINT OF THE INSERT IN THE X AND Z AXIS.

THEREFORE THE TOOL WOULD TRAVEL ALONG A PATH USING POINT P (SEE BELOW) TO FOLLOW THE DRAWING DIMENSIONS.

THIS WILL RESULT IN AN INACCURATE PROFILE. THE LARGER THE INSERT RADIUS THE MORE INACCURATE.



Having established the need for radius compensation we will look at how to introduce it so the CNC control can look ahead and correct the path to achieve an accurate profile.

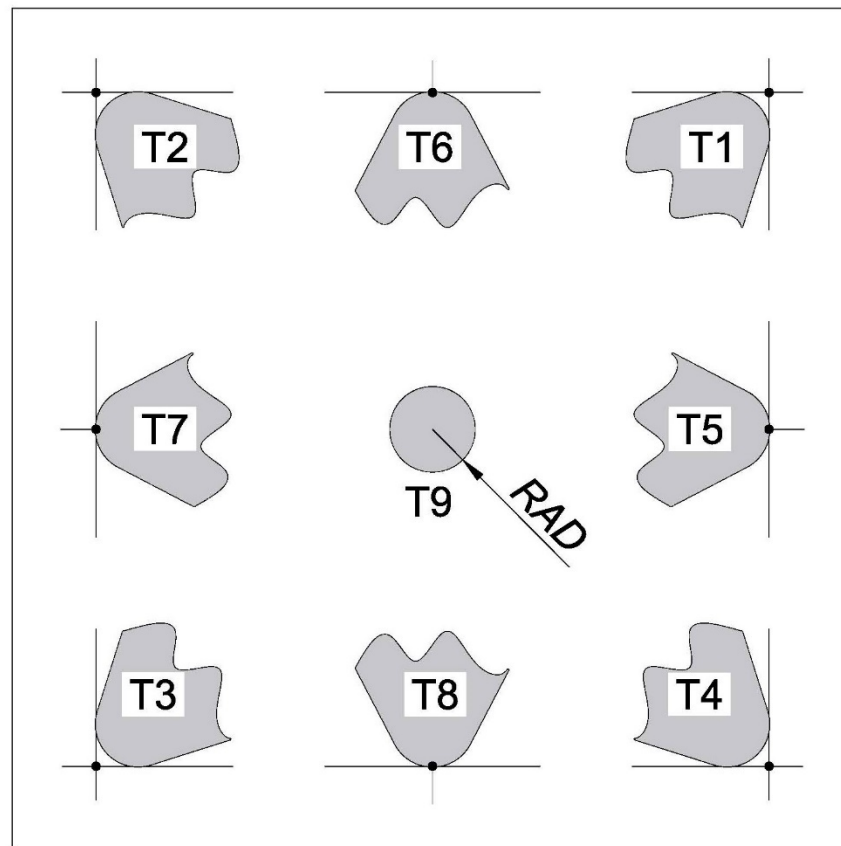
There are G commands that will activate an offset path as follows:

G40 - Cancel Radius Offset Compensation

G41 - Activate a Radius Compensation to the left in the direction of travel.

G42 - Activate a Radius Compensation to the right in the direction of travel.

We now need to apply these G codes, but the CNC control needs more information about the size of offset path. In the Tool Offset table, the insert radius is entered along with the Tool location code. See below



From the drawing we can see the direction of travel along the outside profile is toward the chuck so the path will be compensated to the right in the direction of travel therefore we will edit G42, into the CNC program profile at the correct block, and the tool insert radius is acting in location T3 see diagram above. Both insert radius and T location code will be entered into the Tool offset table.

From the drawing we can see the direction of travel along the inside of the bore profile is toward the chuck so the path will be compensated to the left in the direction of travel therefore we will edit G41, into the CNC program profile at the correct block and the tool insert radius is acting in location T2 see diagram above. Both insert radius and T location code will be entered into the Tool offset table.

Continue

(EXTERNAL PROFILE)

N1 G0 G42 X55. Z1. (POINT 1)

G1 X65. Z-4. F.25(PPOINT 2)

Z-25. (POINT 3)

G2 X75. Z-30. I5. K0(POINT 4)

G1 X92.5 (POINT 5)

X120. Z-60. (POINT 6)

Z-78. (POINT 7)

G2 X124. Z-80. I2. K0 (POINT 8)

N2 G1 G40 X133. (POINT 9)

See G42 inserted in the profile to activate Radius compensation.

See G40 to cancel Radius

At this stage we do not have an internal profile but use the same principles to activate radius compensation on the bore profile.

Programming good practice is to layout a block in order. This will help you see the problems when snagging a program for bugs. e.g.

N155 G01 X150. Y210.25 F200 M03 S1000

If you use block numbers put them first, G code command next, then X,Y,Z, coordinates in order, Feed rate commands next then M,S,T commands.

It is possible to have all these commands in one block or as few as a single command in a block and some machine tool builders permit more than one M code in a block, some others do not.

We now have a working profile, and we can insert lines to create space above the profile as we will be using this profile programming later.

The screenshot displays the CIMCO Edit 2022 software interface. The top menu bar includes Editor, NC Functions, NC-Assistant, Backplot, File Compare, Transmission, and CNC-Calc. The toolbar contains icons for New, Open, Close, Save, Save As, Print, Copy, Cut, Paste, Select All, Undo, Redo, Mark/Delete Range, and Append File. The main window shows a CNC program for "T0620 Turning Taper Housing.N...". The program code is as follows:

```

1
2
3 G99
4
5 G0 X135. Z3.
6 G1 Z.5 F1.
7 X-1.6 F.2
8 G0 Z2.
9 X135.
10 Z0
11 G1 X-1.6
12 G0 Z2.
13 X135.
14
15
16
17
18

```

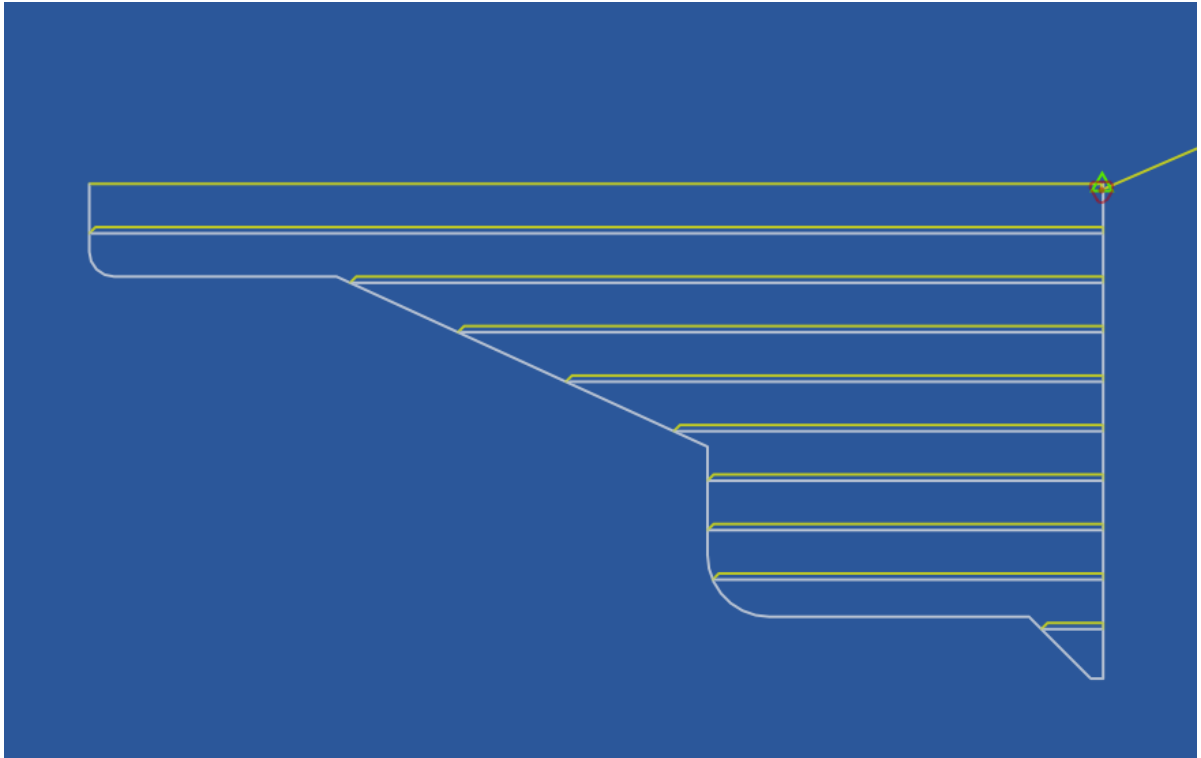
Annotations provide additional information:

- G99 code:** This G99 code is for Feed in mm/rev. The Backplot graphics uses this code to control the speed of the graphics. (With Fanuc G Code, A or Haas, G99 = Feed in mm/rev) (With Fanuc G Code, B or C, G95 = Feed in mm/rev)
- Roughing and finishing cuts:** See simple programming of a roughing and finishing facing cut. Note that the tool is programmed to an X position below the centre to ensure that the insert radius does not leave a small point at the central position.
- Backplot of facing cuts:** See Backplot of facing cuts

3.1 Roughing cycles

11 / 43

diagram below. In Fanuc CNC controls the Cycle for this type of operation is G71 for removal of material by traversing in the Z axis with incremental cuts in the X axis and is supported in the NC-Assistant macro in the Cimco Editor and Backplot, see below.

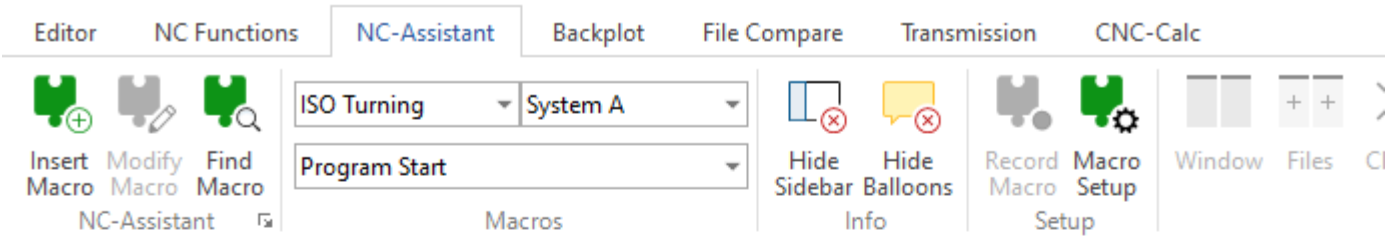


But before we can program the cycle, we need to look at other instructions the CNC control will need to carry out this operation.

Nearly all CNC programs are made up of several operations using a variety of tools to complete the machining work to the drawing of the part. Each tool should be treated as an individual program and will comprise a header, the machining details, and a trailer. The header will contain the information to introduce the tool, to activate the spindle and the coolant, activate the tool offset and move the tool into position for machining. The machining details will be made up of the coordinate moves or canned cycles set at the correct feed rate to remove the material. The trailer will move the tool to a safe position and de-activate the spindle and coolant etc.

3.2 Header information

Now, we must introduce a tool and enter blocks that will put the tool in the spindle and position the tool to commence the machining operation. Every time we start with a new tool, we need to create the header blocks prior to the machining blocks. It is possible to create a header macro for a specific machine and we will cover that in a later tutorial but here we have developed a general header macro using Fanuc G code A for a general CNC Lathe. See explanations below:

**NC-Assistant**

No macro found

Header, see explanations
belowFacing operation
previously programmed.

```
1 %  
2 O0620 (TURN & BORE TAPER HOUSING)  
3 G00 G18 G21 G99 G40  
4 G50 S2000  
5 G28 U0 W0  
6 G54  
7 G97 S300 M03  
8 (FACE & TURN CNMG)  
9 T0101  
10 G0 X137. Z3.  
11 G96 S160  
12  
13 G0 X137. Z3.  
14 G1 Z.5 F1.  
15 X-1.6 F.2  
16 G0 Z2.  
17 X137.  
18 Z0  
19 G1 X-1.6  
20 G0 Z2.  
21 X135.  
22  
23  
24  
25
```

%

;Program Start

O0620 (TURN & BORE TAPER HOUSING) ;Program number & Description comment

G00 G18 G21 G99 G40 ;Set up safe default G codes

G00	Move at rapid
G18	Main interpolation axes X, Z.
G21	Metric coordinates
G99	Feed in mm/rev
G40	Cancel tool radius compensation
G50 S2000	;Set max spindle speed in CSS (Constant Surface Speed)
G28 U0 W0	; Move to machine home for safety
G54	;Activate the main Work Offset to establish Z0
G97 S300 M03	;Start the spindle forward at fixed 300 rpm
(FACE & TURN CNMG)	;Tool description comment
T0101	;Tool Call T01 will index the turret, 01 will activate the offset
G0 X137. Z3.	; Move to the start position at rapid
G96 S160	; Engage CSS at 160 meters/min

To insert a Header, see below:

Set the cursor in the correct position.

From the NC-Assistant menu select
General Programming

Click Program Start

The screenshot shows the NC-Assistant software interface. On the left, the 'Cycles / Macros' menu is open, with 'General Programming' selected. Below it, a list of macros is visible, including 'Program Start', 'Tool Change', 'Program End', and various G-code macros. On the right, the CNC program code is displayed, starting with line 10 and ending with line 46. The code includes comments and G-code commands such as (FACING), G0 X137. Z3., G1 Z.5 F1., X-1.6 F.2, G0 Z2., X133., Z0, G1 X-1.6, G0 Z2., X130., M30, (EXTERNAL PROFILE), N1 G0 G42 X55. Z1. (POINT 1), G1 X65. Z-4. F.25 (POINT 2), Z-25. (POINT 3), G2 X75. Z-30. I5. K0 (POINT 4), G1 X92.5 (POINT 5), X120. Z-60. (POINT 6), Z-78. (POINT 7), G2 X124. Z-80. I2. K0 (POINT 8), and N2 G1 G40 X133. (POINT 9).

4 Roughing cycle

Now we have a profile. A header and facing blocks and we can proceed to include the roughing cycle. You will notice that we left 0.75 mm on the X diameter and 0.15 on the Z face so a finishing operation.

The screenshot displays the CIMCO Edit 2022 software interface. The main window shows a CNC program with the following code:

```
1 O0620 (TURN & BORE TAPER HOUSING)
2
3
4
5 G00 G18 G21 G99 G40
6 G50 S2000
7 G28 U0. W0.
8 G54
9 G97 S300 M03
10 (FACE & TURN CNMG)
11 T0101
12 G0 X137. Z3.
13 G96 S160
14
15 (FACING)
16 G0 X137. Z3.
17 G1 Z.5 F1.
18 X-1.6 F.2
19 G0 Z2.
20 X137.
21 Z0
22 G1 X-1.6
23 G0 Z2.
24 X130.
25
```

Annotations and callouts:

- Select the General Programming Menu and find the G71 Stock removal, click to open the window:** Points to the 'Cycles / Macros' menu where 'G71 OD/ID Stock Removal Cycle' is selected.
- G71 Entry window:** Points to the 'Insert: G71 OD/ID Stock Removal Cycle' dialog box.
- Set the cursor in the correct position:** Points to line 25 of the CNC program.
- Asterisk * Indicates optional entry, quite a few of the entries have already been established in the Header. See the entries to suit our part.** Points to the asterisk in the 'Depth of cut' parameter in the G71 dialog.
- G71 comes in two forms, we have used the 2-line form.** Points to the 'G71 OD/ID Stock Removal Cycle (One block notation)' option in the 'Cycles / Macros' menu.

Insert: G71 OD/ID Stock Removal Cycle

Parameters for 'G71 OD/ID Stock Removal Cycle'

- * Depth of cut for each pass of stock removal, positive
- * Retract height for each pass of stock removal
- * Feedrate [>= 0.0001]
- * X-axis size and direction of G71 rough pass allowance
- * Z-axis size and direction of G71 rough pass allowance
- Starting block number
- Ending block number
- * Spindle speed [>= 1]
- * Tool and offset
- * X-axis size and direction of G71 finish allowance
- * Z-axis size and direction of G71 finish allowance

* = Optional parameter

Default Cancel OK

wance:
 wance:
 S:

```

17 G0 Z2.
18 X137.
19 Z0
20 G1 X-1.6
21 G0 Z2.
22 X130.
23 (ROUGH TURN PROFILE)
24 G71 U2. R.5
25 G71 F.2 P1 Q2 U1. W.15
26
27 M30
28
29
30 (EXTERNAL PROFILE)
31 N1 G0 G42 X55. Z1. (POINT 1)
32 G1 X65. Z-4. F.25 (POINT 2)
33 Z-25. (POINT 3)
34 G2 X75. Z-30. I5. K0 (POINT 4)
35 G1 X92.5 (POINT 5)
36 X120. Z-60. (POINT 6)
37 Z-78. (POINT 7)
38 G2 X124. Z-80. I2. K0 (POINT 8)
39 N2 G1 G40 X135. (POINT 9)
40

```

G71 Cycle format.

M30 is the ISO code for end of programs, reset and rewind to the start

 See the profile is set outside the main program and will be called into use from the cycle by the start and finish block numbers

If we now add the block to return the tool to the home position (G28 U0 W0) and add program end M30! Then test with Backplot!

Editor NC Functions NC-Assistant Backplot File Compare Transmission CNC-Calc
 Insert Modify Find Macro Macro Macro NC-Assistant
 Program Start
 Macros
 Hide Sidebars
 Record Macro Setup
 Window Files Close Previous Next Synchronize Scrolling Setup Multi Channel
 Multi Channel

1 O620 TURNING TAPER HOUSING A...
 2 O620 (TURN & BORE TAPER HOUSING)
 3
 4 G00 G18 G21 G99 G40
 5 G50 S2000
 6 G28 U0. W0.
 7 G54
 8 G97 S300 M03
 9 (FACE & TURN CNMG)
 10 T0101
 11 G0 X133. Z3.
 12 G96 S160
 13 (FACING)
 14 G0 X137. Z3.
 15 G1 Z.5 F1.
 16 X-1.6 F.2
 17 G0 Z2.
 18 X133.
 19 Z0
 20 G1 X-1.6
 21 G0 Z2.
 22 X130.
 23 (ROUGH TURN PROFILE)
 24 G71 U2. R.5
 25 G71 F.2 P1 Q2 U1. W.15
 26 G28 U0 W0
 27 M01
 28
 29 M30
 30
 31
 32
 33 (EXTERNAL PROFILE)
 34 N1 G0 G42 X55. Z1. (POINT 1)
 35 G1 X65. Z-4. F.25 (POINT 2)
 36 Z-25. (POINT 3)
 37 G2 X75. Z-30. I5. K0 (POINT 4)
 38 G1 X92.5 (POINT 5)
 39 X120. Z-60. (POINT 6)
 40 Z-78. (POINT 7)
 41 G2 X124. Z-80. I2. K0 (POINT 8)
 42 N2 G1 G40 X135. (POINT 9)
 43
 44
 45

X: 500.000 I: ??? Dst: 0.000
 Z: 500.000 K: Feed: Rapid Total: 3528.399 R:

Simulation | Navigation | Variables | Geometry Manager
 In 1/45, Col 15, 540 bytes
 11:09 25/06/2022
 Rain off and on
 info@cimco.com

Licensed to DEMO - NOT FOR RESALE - T/A John Bridge MTE
 Copenhagen, Denmark Fax: +45 45 85 60 53

WHEN RELIABILITY MATTERS

So, we now need the next operation to UDrill a hole through the Stock blank. We need a new operation and will use the Tool Change macro in the General Programming. This will terminate to the first operation and create a header for the next operation.

NC-Assistant

ISO Turning

File: New, Open, Close, Save, Save As, Print, File Type, Edit: Copy, Cut, Paste, Select All, Undo, Redo, Mark/Delete Range, Append File, Insert File, Find, Find Previous, Find Next, Replace, Go to Line/Block Number, To

0620 Turning Taper Housing A.... | Untitled * x

NC-Assistant

1 %
 2 O0620 (TURN & BORE TAPER HOUSING)
 3 (OSO TURNING FANUC CODE A (HAAS))
 4
 5 G00 G18 G21 G99 G40
 6 G50 S2000
 7 G28 U0. W0.
 8 G5
 9 G97 S300 M03
 10 (FACE & TURN CNMG)
 11 T0101
 12 G0 X137. Z3.
 13 G96 S160
 14 (FACING)
 15 G0 X137. Z3.
 16 G1 Z-1.6 F1.
 17 X-1.6 F.2
 18 G0 Z2.
 19 X137.
 20 Z0
 21 G1 X-1.6
 22 G0 Z2.
 23 X130.
 24 (ROUGH TURN PROFILE)
 25 G71 U2.5 R.5
 26 G71 F.2 P1 Q2 U.6 W.15
 27 G28 U0. W0.
 28 M01
 29
 30 (UDRILL 30 MM DIA)
 31 G00 G18 G21 G99 G40
 32 G50 2000
 33 G28 U0. W0.
 34 G54
 35 G97 S1350 M03
 36 (30 MM UDRILL)
 37 T0202
 38 G0 X0 Z2.
 39 G1 Z-122. F.12
 40 G0 Z2.
 41 G28 U0. W0.
 42 M01

Tool Change Window

Insert: Tool Change

Parameters for 'Tool Change'

Operation
 2000 Maximum Speed in CSS
 54 Work Offset
 400 Fixed Speed Start in RPM
 3 Spindle Direction [3 - 4]
 Tool Description
 2 Tool Number
 2 Tool Offset
 65. X Start Position
 3. Z Start Position
 96 Speed Mode G96=CSS - G97=RPM
 180 Speed CSS in Meters/Min or RPM

* = Optional parameter

Default Cancel OK

From General Programming menu in NC-Assistant select Tool Change

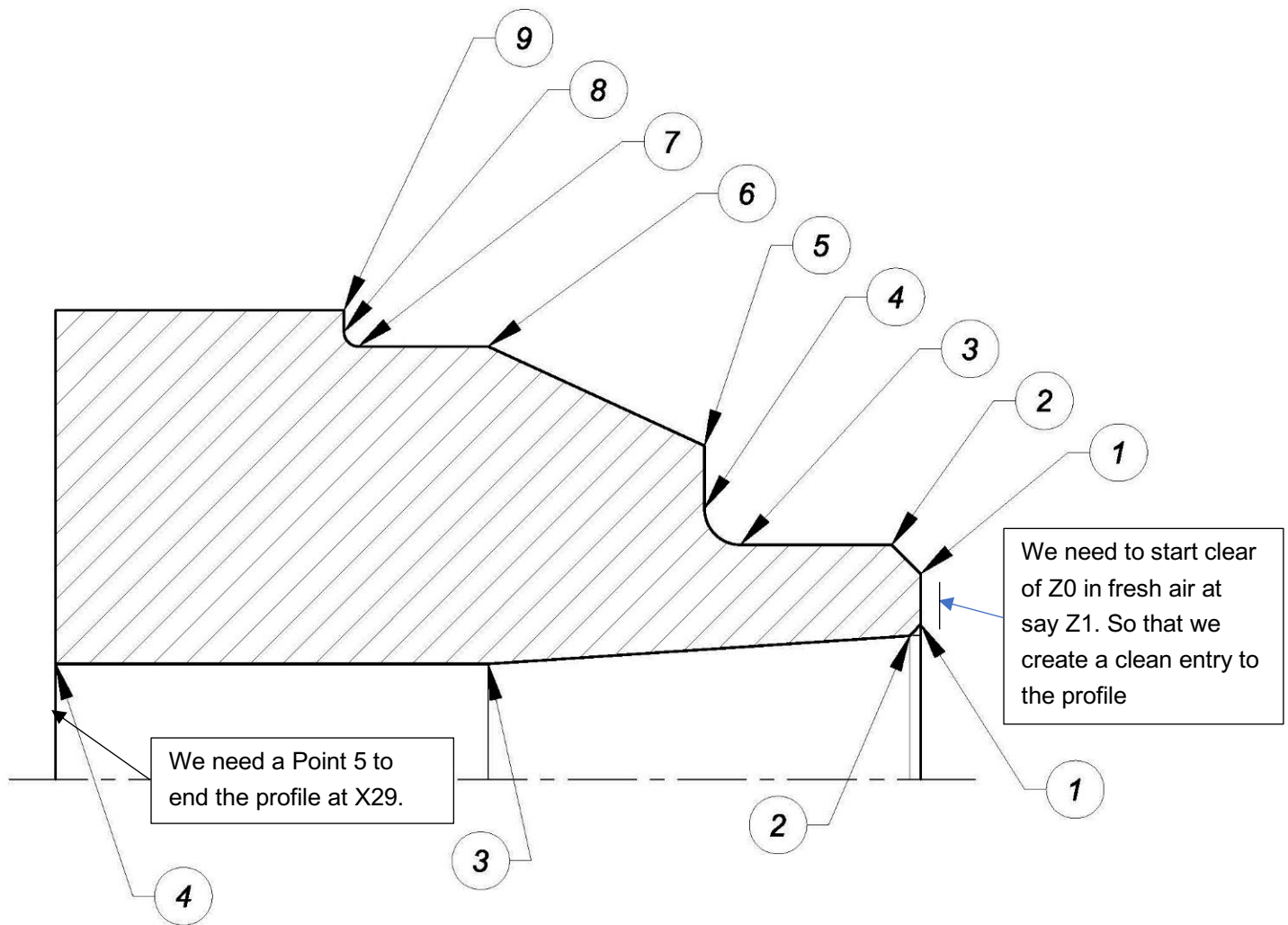
Cycles / Macros

Program Start
 Tool Change
 Program End
 G00 Rapid Motion Positioning (XZ)
 G01 Linear Interpolation Motion (XZ)
 G02 CW Circular Interpolation Motion (XZ-IK)
 G02 CW Circular Interpolation Motion (XZ-R)
 G03 CW Circular Interpolation Motion (XZ-IK)
 G03 CW Circular Interpolation Motion (XZ-R)
 G04 Dwell
 G20 Select Inches
 G21 Select Metric
 G28 Return To Machine Zero Point (UW)
 G40 Tool Nose Compensation Cancel (XY)
 G41 Tool Nose Compensation (TNC) Left
 G42 Tool Nose Compensation (TNC) Right

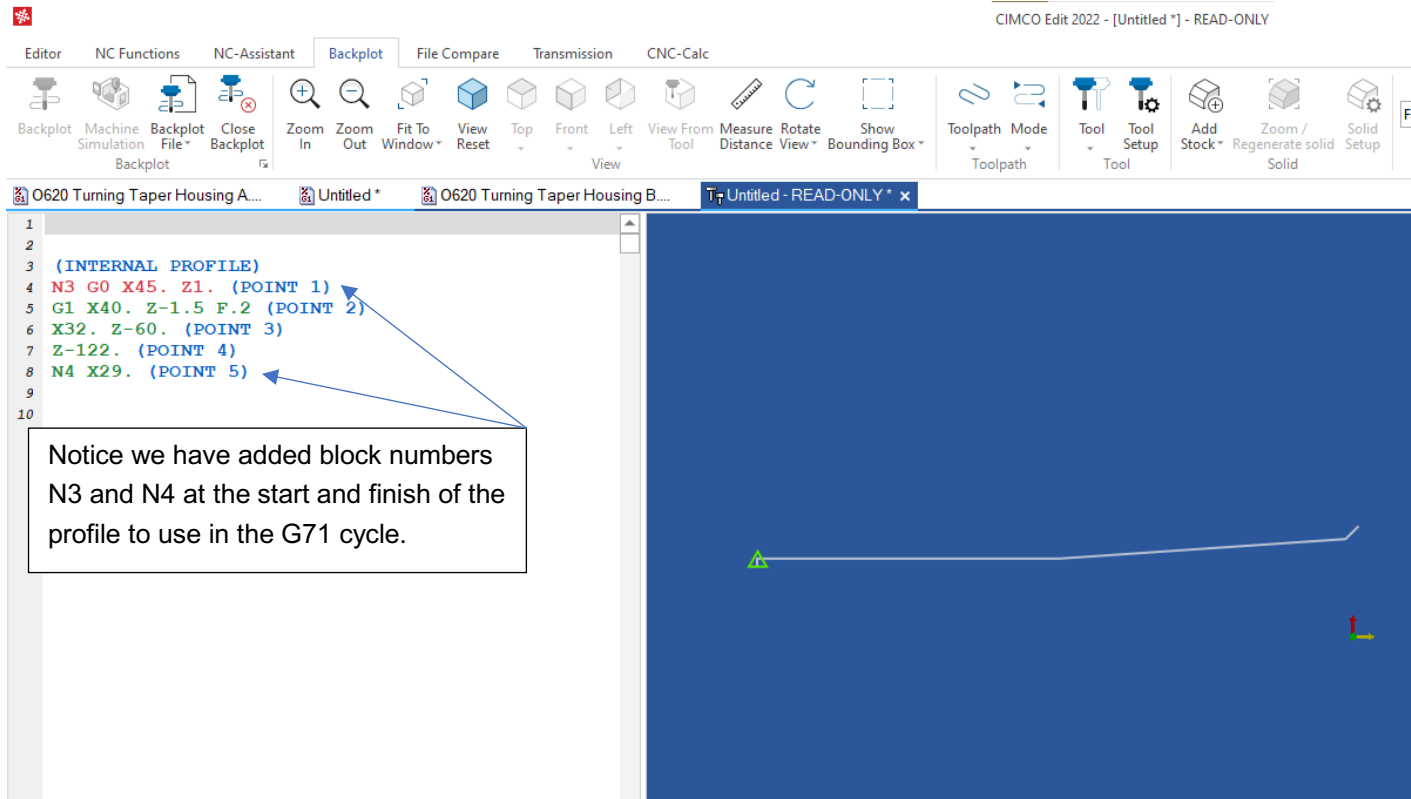
Complete the entries click OK and the result is shown here.

The last two blocks have been added to program the UDrill to penetrate the Stock and to return to the start.

The next section of programming is to rough out the bore profiles and this is achieved using the same cycle but first we need an internal profile. See the diagram below and the internal points 1 to 4



You may want to develop the profile on a separate text page in the editor and test with Backplot then cut and paste back to your main program page. The main program should be saved at this point or even earlier. See profile and Backplot test below for the internal profile of the taper bore.



We can make a small adjustment to the profile to leave the grinding allowance on the taper section of the profile. The corrected lengths should leave a sufficient on the taper section.

(INTERNAL PROFILE)

N3 G0 G41 X45. Z1. ; we have added the G41 to activate radius compensation

G1 X39.8 Z-1.5 F.2 ; reduced the finish start diameter by 40-0.2 mm = 39.8

X32. Z-58.2 ; reduced the Z end point by 60-1.8 = 58.2

Z-122.

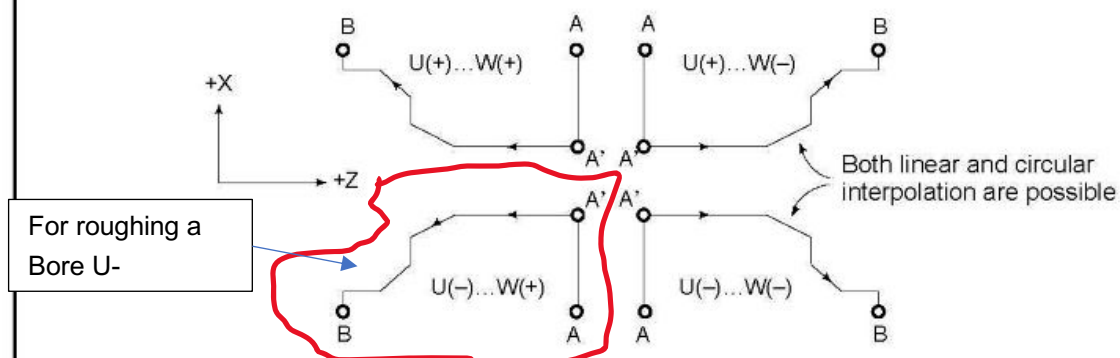
N4 G40 X29 ; we have added G40 to deactivate radius compensation.

The internal profile has been set out from largest diameter to smallest, so it is the opposite to the external profile.

The G71 Cycle needs further information to work on an internal profile see the chart below showing the designation that will control the cycle in this respect.

NOTE

- 1 While both Δd and Δu , are specified by address U, the meanings of them are determined by the presence of addresses P and Q.
- 2 The cycle machining is performed by G71 command with P and Q specification. F, S, and T functions which are specified in the move command between points A and B are ineffective and those specified in G71 block or the previous block are effective.
When an option of constant surface speed control is selected, G96 or G97 command specified in the move command between points A and B are ineffective, and that specified in G71 block or the previous block is effective.
The following four cutting patterns are considered. All of these cutting cycles are made paralleled to Z axis and the sign of Δu and Δw are as follows:



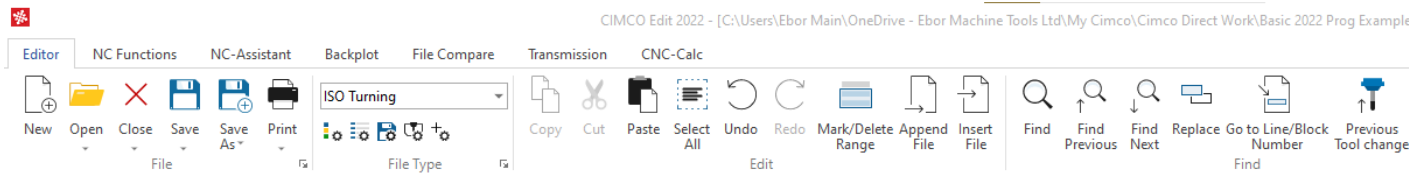
The tool path between A and A' is specified in the block with sequence number "ns" including G00 or G01, and in this block, a move command in the Z axis cannot be specified. The tool path between A' and B must be steadily increasing or decreasing pattern in both X and Z axis. When the tool path between A and A' is programmed by G00/G01, cutting along AA' is performed in G00/G01 mode respectively.

- 3 The subprogram cannot be called from the block between sequence number "ns" and "nf".
- 4 Nose radius compensation is disabled during cycle operation. When the imaginary tool nose number is 0 or 9, however, a nose radius compensation value is added to U and W.

You will notice that the top left diagram shows that U and W are positive for the direction of the profile we programmed and that is the designation we used for roughing the outside profile.

Now we are going to rough the internal profile which is shown bottom left diagram and the U value should be negative (U-xx)

We can now program roughing of the internal profile with G71 Cycle as we know that for it to work, we need to have a negative U value. But first a tool change, see below:



NC-Assistant

No macro found

Cycles / Macros

Program Start
Tool Change
 Program End
 G00 Rapid Motion Positioning (XZ)
 G01 Linear Interpolation Motion (XZ)
 G02 CW Circular Interpolation Motion (XZ-IR)
 G02 CW Circular Interpolation Motion (XZ-R)
 G03 CW Circular Interpolation Motion (XZ-IR)

```

36 (30 MM UDRILL)
37 T0202
38 G0 X0 Z2.
39 G1 Z-122. F.12
40 G0 Z2.
41 G28 U0. W0.
42 M01
43
44 (ROUGH PROFILE BORE)
45 G00 G18 G21 G99 G40
46 G50 2000
47 G28 U0. W0.
48 G54
49 G97 S300 M03
50 (25 MM BORING BAR CNMG)
51 T0303
52 G0 X30. Z02
53 G96 S150
54 G71 U1.5 R.5
55 G71 F.2 P3 Q4 U-.2 W.1
56 G28 U0 W0
57 M01
58
59 (ROUGH PROFILE THE TAPER BORE)
60 G00 G18 G21 G99 G40
61 G50 2000
62 G28 U0 W0
63 G54
64 G97 S400 M03
65 (25 MM BORING BAR 80 DEG INSERT)
66 T0404
67 G0 X30. Z1.
68 G96 S180
69
70
71

```

Insert: Tool Change

Parameters for 'Tool Change'

THE TAPER BORE

Operation

2000

Maximum Speed in CSS

54

Work Offset

400

Fixed Speed Start in RPM

3

Spindle Direction [3 - 4]

AR 80 DEG INSERT

Tool Description

4

Tool Number

4

Tool Offset

30.

X Start Position

1.

Z Start Position

96

Speed Mode G96=CSS - G97=RPM

180

Speed CSS in Meters/Min or RPM

* = Optional parameter

Default

Cancel

OK

Complete the Tool Change entries in the window and click OK

Add the G71 Roughing Cycle

CIMCO Edit 2022 - [C:\Users\Ebor Main\OneDrive - Ebor Machine Tools Ltd\My Cimco\Cimco Direct Work\Basic 2022 Prog Examples\]

Editor NC Functions NC-Assistant Backplot File Compare Transmission CNC-Calc

New Open Close Save Save As Print File File Type Edit Find Find Previous Find Next Replace Go to Line/Block Number Previous Tool change

O620 Turning Taper Housing A...

NC-Assistant

No macro found

52 G0 X30. Z02
53 G96 S150
54 G71 U1.5 R.5
55 G71 F.2 P3 Q4 U-.2 W.1
56 G28 U0 W0
57 M01
58
59 (ROUGH PROFILE THE TAPER BORE)
60 G00 G18 G21 G99 G40
61 G50 2000
62 G28 U0 W0
63 G54
64 G97 S400 M03
65 (25 MM BORING BAR 80 DEG INSERT)
66 T0404
67 G0 X30. Z1.
68 G96 S180
69 G71 U1.5 R.5
70 G71 F.2 P3 Q4 U.5 W.15
71
72
73
74
75
76
77 M30
78
79
80
81
82
83
84
85

Cycles / Macros

- G70 Finishing Cycle
- G71 OD/ID Stock Removal Cycle**
- G71 OD/ID Stock Removal Cycle (One block notation)
- G72 End Face Stock Removal Cycle
- G72 End Face Stock Removal Cycle (One block notation)
- G75 OD/ID Grooving Cycle

Insert: G71 OD/ID Stock Removal Cycle

Parameters for 'G71 OD/ID Stock Removal Cycle'

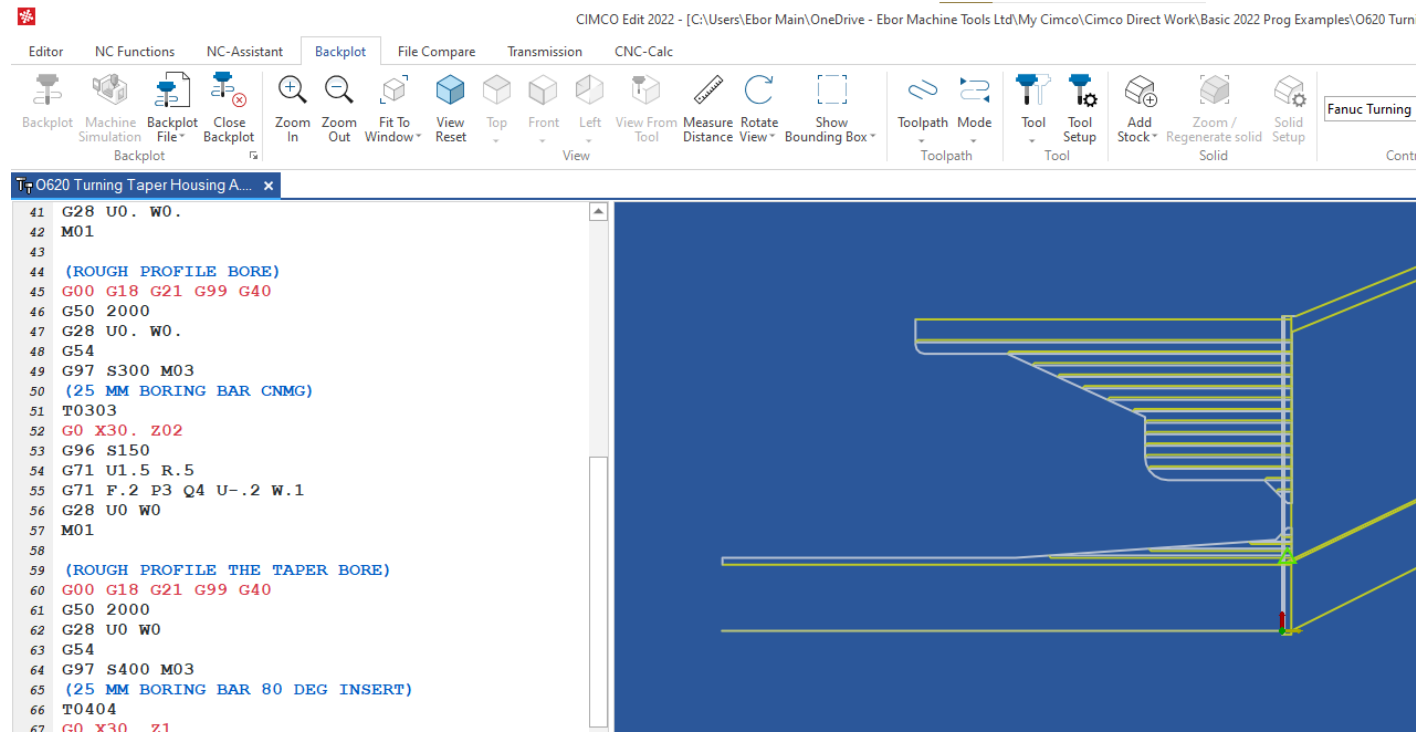
- * 1.5 Depth of cut for each pass of stock removal, positive
- * .5 Retract height for each pass of stock removal
- * .2 Feedrate [>= 0.0001]
- * X-axis size and direction of G71 rough pass allowance
- * Z-axis size and direction of G71 rough pass allowance
- 3 Starting block number
- 4 Ending block number
- * Spindle speed [>= 1]
- * Tool and offset
- * .5 X-axis size and direction of G71 finish allowance
- * .15 Z-axis size and direction of G71 finish allowance

* = Optional parameter

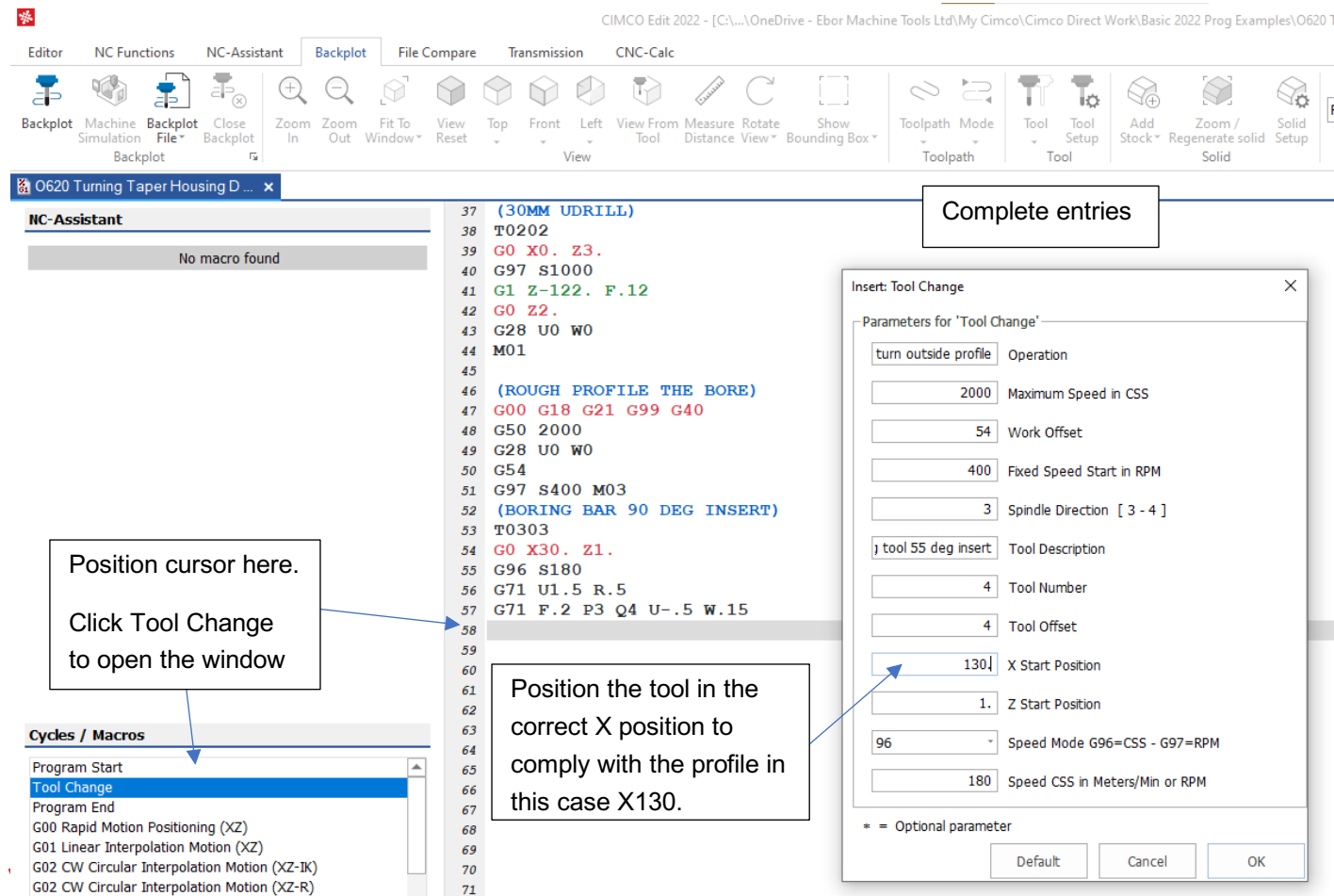
Default Cancel OK

Select G71 from the General Machining group. Complete the details as shown here and click OK to enter

See Backplot results



We can now add the finishing cut for the external profile. Use Tool Change macro see below



<Z-IK)
 <Z-R)
 <Z-IK)
 <Z-R)

71

Open the Finishing macro window see below

The screenshot shows the CIMCO Edit 2022 software interface. The top menu bar includes Editor, NC Functions, NC-Assistant, Backplot, File Compare, Transmission, and CNC-Calc. The NC-Assistant window is open, showing a list of cycles/macros. The 'G70 Finishing Cycle' is selected. A dialog box titled 'Insert: G70 Finishing Cycle' is open, showing parameters for the cycle. The 'Starting block number' is set to 1 and the 'Ending block number' is set to 2. The 'G70 Finishing Cycle' is highlighted in the list of cycles/macros. A text box indicates that the cycle is complete by adding the profile block numbers and clicking OK. Another text box points to the 'G70 Finishing Cycle' in the list, indicating where to position the cursor.

NC-Assistant

No macro found

Select G70 macro to open window

Position cursor here

Complete by adding the profile block numbers, click OK

Insert: G70 Finishing Cycle

Parameters for 'G70 Finishing Cycle'

1 Starting block number

2 Ending block number

* = Optional parameter

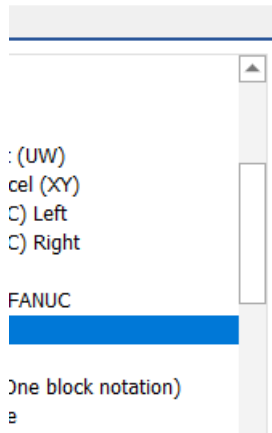
Default Cancel OK

Cycles / Macros

- G04 Dwell
- G20 Select Inches
- G21 Select Metric
- G28 Return To Machine Zero Point (UW)
- G40 Tool Nose Compensation Cancel (XY)
- G41 Tool Nose Compensation (TNC) Left
- G42 Tool Nose Compensation (TNC) Right
- G50 Spindle Speed Limit
- G52 Set Local Coordinate System FANUC
- G70 Finishing Cycle**
- G71 OD/ID Stock Removal Cycle
- G71 OD/ID Stock Removal Cycle (One block notation)
- G72 End Face Stock Removal Cycle

```
45
46 (ROUGH PROFILE THE BORE)
47 G00 G18 G21 G99 G40
48 G50 2000
49 G28 U0 W0
50 G54
51 G97 S400 M03
52 (BORING BAR 90 DEG INSERT)
53 T0303
54 G0 X30. Z1.
55 G96 S180
56 G71 U1.5 R.5
57 G71 F.2 P3 Q4 U-.5 W.15
58 G28 U0 W0
59 M01
60
61 (FINISH TURN OUTSIDE PROFILE)
62 G00 G18 G21 G99 G40
63 G50 2000
64 G28 U0 W0
65 G54
66 G97 S400 M03
67 (PROFILE TURNING TOOL 55 DEG INS
68 T0404
69 G0 X130. Z1.
70 G96 S180
71
72
73
74
75
76
77
78
79
80
81
82
83
84
85
```

See the resulting Backplot below



```

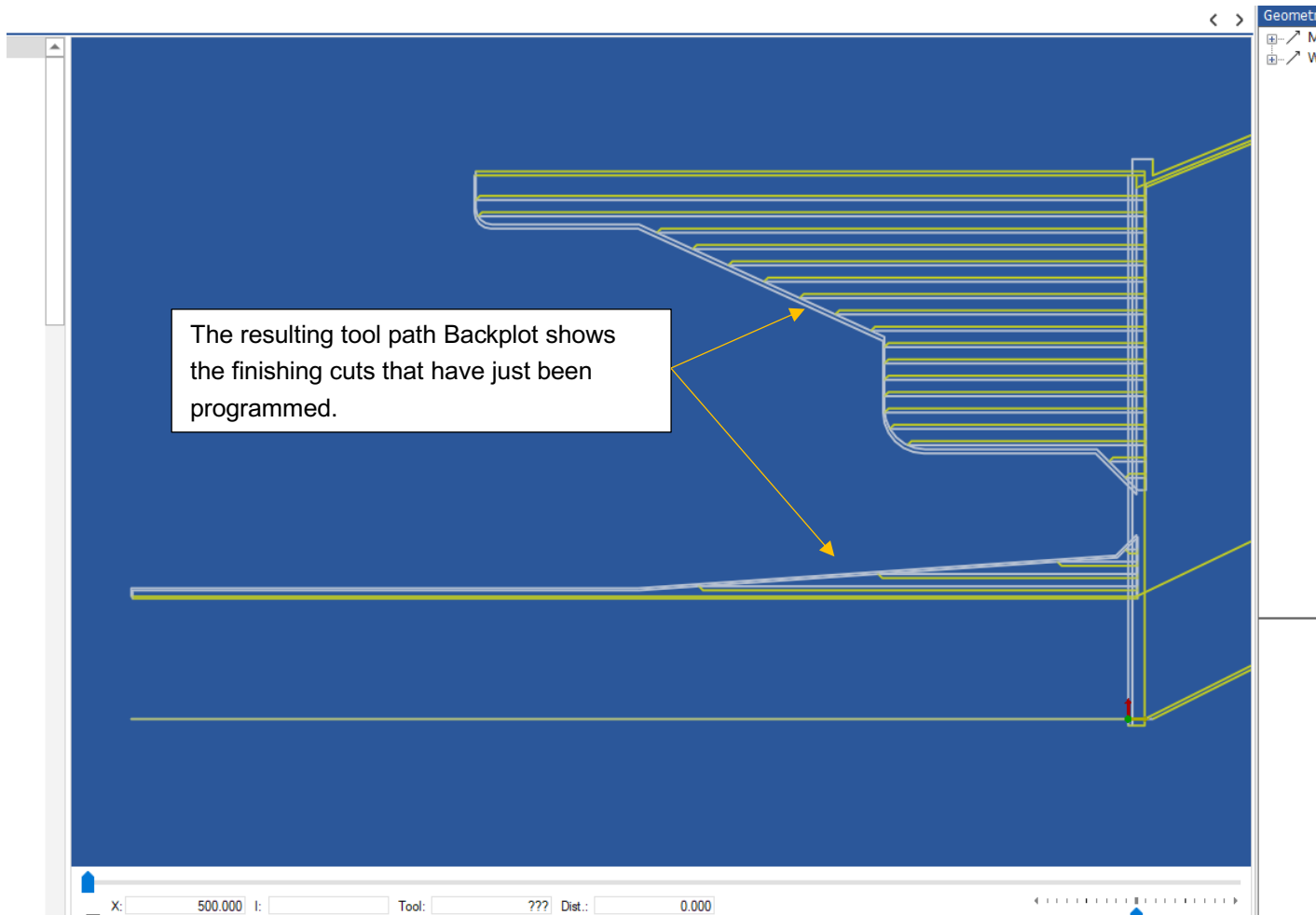
57 G11 F.2 F3 Q4 U-.3 W.13
58 G28 U0 W0
59 M01
60
61 (FINISH TURN OUTSIDE PROFILE)
62 G00 G18 G21 G99 G40
63 G50 2000
64 G28 U0 W0
65 G54
66 G97 S400 M03
67 (PROFILE TURNING TOOL 55 DEG INSERT)
68 T0404
69 G0 X130. Z1.
70 G96 S180
71 G70 P1 Q2
72 G28 U0 W0
73 M01
74
75 (FINISH BORE INTERNAL PROFILE)
76 G00 G18 G21 G99 G40
77 G50 2000
78 G28 U0 W0
79 G54
80 G97 S400 M03
81 (FINISH BORING BAR 55 DEG INSERT)
82 T0505
83 G0 X30. Z1.
84 G96 S180
85 G70 P3 Q4
86 G28 U0 W0
87 M30
88
89 (EXTERNAL PROFILE)
90 N1 G0 X55. Z1. (POINT 1)
91 G1 X65. Z-4. F.25 (POINT 2)
92 Z-25. (POINT 3)
93 G2 X75. Z-30. I5. K0 (POINT 4)

```

See internal section using a finishing boring bar. Note the start position is X30.

End of program and reset.
See profile descriptions are outside the main program.

Repeat the process for the finishing cut to the internal profile



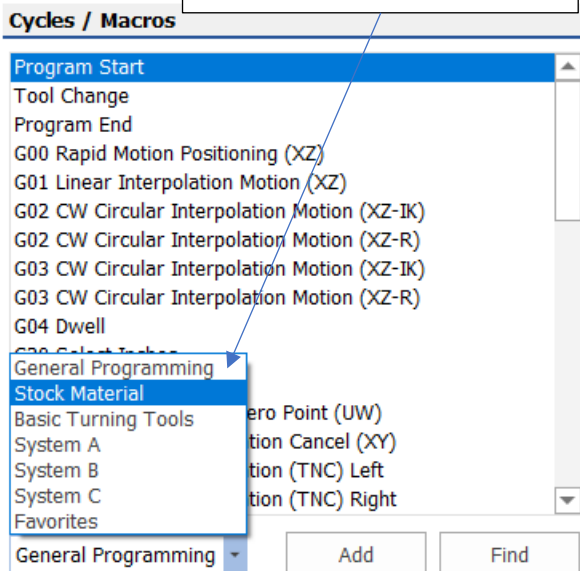
So far, we have been using the tool path Backplot to test out program. We can now add the details of Stock and Tools to achieve a 3D solid Backplot to give a true representation of the material being removed as it will happen on the CNC machine.

5 Set up stock dimensions and tools

To enable the Backplot to show the cutter path accurately and how the material removal will look on the CNC machine, we need to set up the stock material size and the tools. Then Backplot shows the metal removal to our program commands.

5.1 Stock setup

From the macro group selection click on Stock Material



```
22 G1 X-1.6
23 G0 Z2.
24 X130.
25 (ROUGH TURN PROFILE)
26 G71 U2. R.5
27 G71 F.2 P1 Q2 U1. W.15
28 G28 U0 W0
29 M01
30
31 (UDRILL 30 MM THROUGH)
32 G00 G18 G21 G99 G40
33 G50 2000
34 G28 U0 W0
35 G54
36 G97 S400 M03
37 (30MM UDRILL)
38 T0202
39 G0 X0. Z3.
40 G97 S1000
41 G1 Z-122. F.12
42 G0 Z2.
43 G28 U0 W0
44 M01
45
46 (ROUGH PROFILE THE BORE)
47 G00 G18 G21 G99 G40
```

Continued below

See Here above the Menu of NC-Assistant Groups.

General Programming - for all the regular basic programming features.

Stock Material – to establish the stock size of the part being machined

Basic Turning Tools- for the list of tools to be defined in your program.

System A, B, C, - for a comprehensive list the G and M codes used in Fanuc and Haas CNC controls.



0620 Turning Taper Housing D ... x

NC-Assistant

Position the cursor here

Click the Cylindrical Stock
to open the window

Cycles / Macros

Cylindrical Stock

1 %
2 O0620 (TURN & BORE TAPER HOUSING)
3
4
5
6 G00 G18 G21 G99
7 G50 S2000
8 G28 U0. W0.
9 G54
10 G97 S300 M03
11 (FACE & TURN CN
12 T0101
13 G0 X133. Z3.
14 G96 S160
15 (FACING)
16 G0 X137. Z3.
17 G1 Z.5 F1.
18 X-1.6 F.2
19 G0 Z2.
20 X133.
21 Z0
22 G1 X-1.6
23 G0 Z2.
24 X130.
25 (ROUGH TURN PRO
26 G71 U2. R.5
27 G71 F.2 P1 Q2 U
28 G28 U0 W0
29 M01
30
31 (UDRILL 30 MM T
32 G00 G18 G21 G99
33 G50 2000
34 G28 U0 W0
35 G54
36 G97 S300 M03

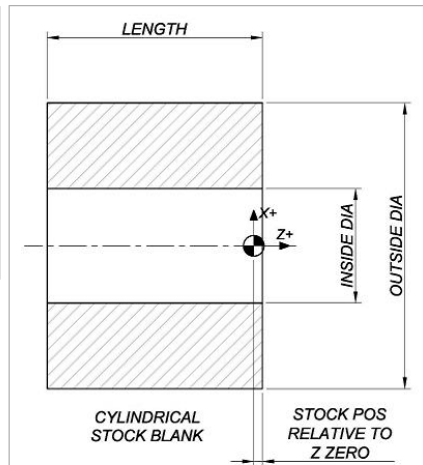
Insert: Cylindrical Stock

Parameters for "Cylindrical Stock"

132 Outside Diameter
0 Inside Diameter
123 Length
3 Stock Position Relative to Z
0 X Work Shift
0 Y Work Shift
0 Z Work Shift

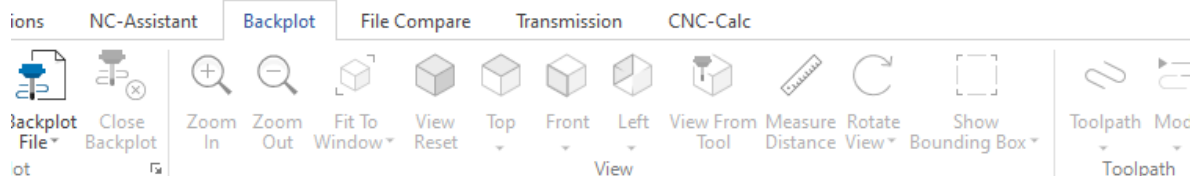
* = Optional parameter

Complete the
entries and click



Default Cancel OK

CIMCO Edit 2022 - [C:\...OneDrive - Ebor Machine Tools Ltd\My C



er Housing D ... x

Resulting entry

1 %
2 O0620 (TURN & BORE TAPER HOUSING)
3 (STOCK TURN OD132 ID0 L123 PZ3)
4 (WCS ID1 X0 Y0 Z0)
5
6
7
8 G00 G18 G21 G99 G40
9 G50 S2000
10 G28 U0. W0.
11 G54
12 G97 S300 M03

5.2 Tool setup

To enable the Backplotting of our program we must select the correct tool as we would on the CNC machine. There is a simple tool list setup in NC-Assistant.

The screenshot shows the NC-Assistant software interface. The top window is titled "O620 Turning Taper Housing D ...". The main area displays a CNC program with line numbers 1 through 47. A callout box with the text "Set cursor here" points to line 5. Below the program, there is a "Cycles / Macros" section. A callout box with the text "Select Basic Turning Tools from macro group list" points to the "Basic Turning Tools" option in the dropdown menu. The dropdown menu is open, showing options: General Programming, Stock Material, Basic Turning Tools (selected), System A, System B, System C, Favorites, and Basic Turning Tools. The "Add" and "Find" buttons are visible at the bottom of the dropdown menu. The CNC program text is as follows:

```
1 %  
2 O0620 (TURN & BORE TAPER HOUSING)  
3 (STOCK TURN OD132 ID0 L123 PZ3)  
4 (WCS ID1 X0 Y0 Z0)  
5  
6  
7  
8 G00 G18 G21 G99 G40  
9 G50 S2000  
10 G28 U0. W0.  
11 G54  
12 G97 S300 M03  
13 (FACE & TURN CNMG 80 DEG)  
14 T0101  
15 G0 X133. Z3.  
16 G96 S160  
17 (FACING)  
18 G0 X137. Z3.  
19 G1 Z.5 F1.  
20 X-1.6 F.2  
21 G0 Z2.  
22 X133.  
23 Z0  
24 G1 X-1.6  
25 G0 Z2.  
26 X130.  
27 (ROUGH TURN PROFILE)  
28 G71 U2. R.5  
29 G71 F.2 P1 Q2 U1. W.15  
30 G28 U0 W0  
31 M01  
32  
33 (UDRILL 30 MM THROUGH)  
34 G00 G18 G21 G99 G40  
35 G50 2000  
36 G28 U0 W0  
37 G54  
38 G97 S400 M03  
39 (30MM UDRILL)  
40 T0202  
41 G0 X0. Z3.  
42 G97 S1000  
43 G1 Z-122. F.12  
44 G0 Z2.  
45 G28 U0 W0  
46 M01  
47
```

Tool number 1 is to turn and face the bar and an 80 degree rhomboid shaped insert will do both jobs

CIMCO Edit 2022 - [C:\...OneDrive - Ebor Machine Tools Ltd\My Cimco\Cimco Direct Work\Basic 2022 Prog Examples\O620 Turning Taper Housing A

Editor NC Functions NC-Assistant Backplot File Compare Transmission CNC-Calc

ISO Turning Basic Turning Tool Turn & Face 80 Deg

Insert Macro Modify Macro Find Macro NC-Assistant Macros

Hide Sidebar Hide Balloons Info Record Macro Setup Window Files Close Previous Next Synchronize Scrolling Setup Multi Channel

O620 Turning Taper Housing A...

NC-Assistant

No macro found

Set the cursor here

Click the turn & Face tool to open the window,

Enter the tool number and use the default radius but you have the option to change it. Click OK

Insert: Turn & Face 80 Deg

Parameters for 'Turn Face 80 Deg'

Tool Number

Insert Radius

Optional parameter

TURNING TOOL HOLDER L USING INSERT C

CUTTER POINT = T3 ORIENTATION = 0

UNITS UM=METRIC UI=IMPERIAL

NOTE! SEE TOOL SETUP FOR MORE DETAILS

TURNING TOOL

INSERT C

Default Cancel OK

```

1 %
2 O0620 (TURN & BORE TAPER HOUSING)
3
4 (STOCK TURN OD135 ID0 L123 PZ2)
5 (WCS ID1 X0 Y0 Z0)
6
7
8
9 G00 G18 G21 G99 G40
10 G50 S2000
11 G28 U0. W0.
12 G54
13 G97 S300 M03
14 (FACE & TURN CNMG 6
15 T0101
16 G0 X133. Z3.
17 G96 S160
18 (FACING)
19 G0 X137. Z3.
20 G1 Z.5 F1.
21 X-1.6 F.2
22 G0 Z2.
23 X133.
24 Z0
25 G1 X-1.6
26 G0 Z2.
27 X130.
28 (ROUGH TURN PROFIL
29 G71 U2. R.5
30 G71 F.2 P1 Q2 U1.
31 G28 U0 W0
32 M01
33
34 (FINISH TURN)
35 G00 G18 G21 G99 G40
36 G50 2000
37 G28 U0 W0
38 G54
39 G97 S400 M03
40 (FINISHING TOOL)
41 T0202
42 G0 X65. Z3.
43 G96 S180
44 G70 P1 Q2
45 G28 U0 W0

```

Resulting entry

IC-Assistant Backplot File Compare Transmission CNC-Calc

Close Backplot Zoom In Zoom Out Fit To Window View Reset Top Front Left View From Tool Measure Rotate Show Bounding Box Toolpath Mode Tool

gD ... x

```

1 %
2 O0620 (TURN & BORE TAPER HOUSING)
3 (STOCK TURN OD132 ID0 L123 PZ3)
4 (WCS ID1 X0 Y0 Z0)
5 ( LTOOL 1 "ISO_TURNING" INSERT=C STYLE=L AO0 CR0.8 C
6
7
8

```

Tool 2 to UDrill an 30 mm hole through the stock

CIMCO Edit 2022 - [C:\...\OneDrive - Ebor Machine Tools Ltd\My Cimco\Cimco Direct Work\Basic 2022 Prog Examples\O620 Turning Taper Housing D Latest.NC *]

Editor NC Functions NC-Assistent Backplot File Compare Transmission CNC-Calc

Backplot Machine Simulation Backplot Close Backplot Backplot

Zoom In Zoom Out Fit To Window View Reset Top Front Left View From Measure Rotate Show Bounding Box Toolpath Mode Tool Tool Setup Add Stock Zoom / Regenerate solid Solid Setup Fanuc Turning Control Type

O620 Turning Taper Housing D ...

NC-Assistent

No macro found

Cycles / Macros

- Turn & Face 80 Deg
- Finish Turn & Face 55 Deg
- Bore & Face 80 Deg
- Finish Bore & Face 55 Deg
- Groove OD
- Groove ID
- Lathe Drill

```

1 %
2 O0620 (TURN & BORE TAPER HOUSING)
3 (STOCK TURN OD132 ID0 L123 PZ3)
4 (WCS ID1 X0 Y0 Z0)
5 ( LTOOL 1 "ISO_TURNING" INSERT=C STYLE=L AO0 CR0.8 CPI=T3 FHD4.5 FW32 HL35 HAT80 ICD11.818 US=UM
6
7
8
9
10
11
12 G00 G18 G21
13 G50 S2000
14 G28 U0. W0.
15 G54
16 G97 S300 M03
17 (FACE & TURN
18 T0101
19 G0 X133. Z3.
20 G96 S160
21 (FACING)
22 G0 X137. Z3.
23 G1 Z.5 F1.
24 X-1.6 F.2
25 G0 Z2.
26 X133.
27 Z0
28 G1 X-1.6
29 G0 Z2.
30 X130.
31 (ROUGH TURN PROFILE)
32 G71 U2. R.5
33 G71 F.2 P1 Q2 U1. W.15
34 G28 U0 W0
35 M01
36
37 (UPDATE 20 MM THROUGH)

```

Insert: Lathe Drill

Parameters for 'Lathe Drill'

Tool Number: 1

Diameter: 30

Point Angle: 179

Body Length: 150

Shank Length: 80

* = Optional parameter

HOLDER SQUARE SHANKS COMMON DRILL

UNITS
UM-METRIC
IM-IMPERIAL

NOTE!
SEE TOOL SETUP FOR MORE DETAILS

Diagram showing dimensions: 60, 50, 50, 25, DIAMETER, POINT ANGLE.

Default Cancel OK

Resulting insertion

plot Machine Simulation Backplot Close Backplot Backplot

Zoom In Zoom Out Fit To Window View Reset Top Front Left View From Measure Rotate Show Bounding Box Toolpath Mode Tool Tool Setup S

O620 Turning Taper Housing D ...

NC-Assistent

No macro found

```

1 %
2 O0620 (TURN & BORE TAPER HOUSING)
3 (STOCK TURN OD132 ID0 L123 PZ3)
4 (WCS ID1 X0 Y0 Z0)
5 ( LTOOL 1 "ISO_TURNING" INSERT=C STYLE=L AO0 CR0.8 CPI=T3
6 ( LTOOL 2 "ISO_LATHE_DRILLING" INSERT=COMMON STYLE=S AO90
7
8

```


Tool 3 Rough out the internal taper profile

Backplot Machine Simulation Backplot Close Backplot Zoom In Zoom Out Fit To Window View Reset View Top Front Left View From Measure Rotate Show Bounding Box Toolpath Mode Tool Tool Setup Add Stock Zoom / Regenerate Solid Solid Setup Control Type

0620 Turning Taper Housing D... x

NC-Assistant

No macro found

```

1 %
2 O0620 (TURN & BORE TAPER HOUSING)
3 (STOCK TURN OD132 ID0 L123 PZ3)
4 (WCS ID1 X0 Y0 Z0)
5 ( LTOOL 1 "ISO_TURNING" INSERT=C STYLE=L AO0 CR0.8 CPI=T3 FHD4.5 FW32 HL35 HAT80 ICD11.818 US=UM
6 ( LTOOL 2 "ISO_LATHE_DRILLING" INSERT=COMMON STYLE=S AO90 BL150 CPI=T8 D30 FL20 US=UM AD30 LS80
7
8
9
10
11
12 G00 G18 G21 G
13 G50 S2000
14 G28 U0. W0.
15 G54
16 G97 S300 M03
17 (FACE & TURN
18 T0101
19 G0 X133. Z3.
20 G96 S160
21 (FACING)
22 G0 X137. Z3.
23 G1 Z.5 F1.
24 X-1.6 F.2
25 G0 Z2.
26 X133.
27 Z0
28 G1 X-1.6
29 G0 Z2.
30 X130.
31 (ROUGH TURN P...
32 G71 U2. R.5
33 G71 P.2 P1 O2 U1. W.15

```

Insert: Bore & Face 80 Deg

Parameters for 'Bore Face 80 Deg'

Tool Number: 3

Insert Radius: 0.8

* = Optional parameter

BORING BAR STYLE L USING INSERT C
CUTTER POINT=T2 ORIENTATION = 90
UNITS UM=METRIC UI=IMPERIAL
NOTE! SEE TOOL SETUP FOR MORE DETAILS

INSERT C

24.92 124.92 27.044 25

Default Cancel OK

Cycles / Macros

- Turn & Face 80 Deg
- Finish Turn & Face 55 Deg
- Bore & Face 80 Deg
- Finish Bore & Face 55 Deg
- Groove OD

Resulting insertion

NC-Assistant Backplot File Compare Transmission CNC-Calc

Plot Close Backplot Zoom In Zoom Out Fit To Window View Reset View Top Front Left View From Measure Rotate Show Bounding Box Toolpath Mode Tool Tool Setup Add Stock Zoom / Regenerate Solid

ousing D... x

No macro found

```

1 %
2 O0620 (TURN & BORE TAPER HOUSING)
3 (STOCK TURN OD132 ID0 L123 PZ3)
4 (WCS ID1 X0 Y0 Z0)
5 ( LTOOL 1 "ISO_TURNING" INSERT=C STYLE=L AO0 CR0.8 CPI=T3 FHD4.5 FW32 HL35 HAT80 ICD11.818 US=UM
6 ( LTOOL 2 "ISO_LATHE_DRILLING" INSERT=COMMON STYLE=S AO90 BL150 CPI=T8 D30 FL20 US=UM AD30 LS80
7 ( LTOOL 3 "ISO_BORING" INSERT=C STYLE=L AO90 CR0.8 CPI=T2 FHD4.4 FW32 HL35 HAT80 ICD11.818 US=UM
8
9
10
11

```

Tool 4 Finish turn the outside profile

CIMCO Edit 2022 - [C:\...OneDrive - Ebor Machine Tools Ltd\My Cimco\Cimco Direct Work\Basic 2022 Prog Examples\O620 Turning Taper Housing D Lathe

Editor NC Functions NC-Assistant **Backplot** File Compare Transmission CNC-Calc

Backplot Machine Simulation Backplot Close Backplot Zoom In Zoom Out Fit To Window View Reset Top Front Left View From Measure Rotate Show Bounding Box Toolpath Mode Toolpath Tool Tool Setup Add Stock Zoom / Regenerate solid Solid Setup Fanuc Turning Control Type

O620 Turning Taper Housing D ... x

NC-Assistant

Bore_Face 80 Deg

Tool Number: 3

Insert Radius: 0.8

Modify

Cycles / Macros

- Turn & Face 80 Deg
- Finish Turn & Face 55 Deg**
- Bore & Face 80 Deg
- Finish Bore & Face 55 Deg
- Groove OD
- Groove ID
- Lathe Drill

```

1 %
2 O0620 (TURN & BORE TAPER HOUSING)
3 (STOCK TURN OD132 ID0 L123 PZ3)
4 (WCS ID1 X0 Y0 Z0)
5 ( LTOOL 1 "ISO_TURNING" INSERT=C STYLE=L AO0 CR0.8 CPI=T
6 ( LTOOL 2 "ISO_LATHE DRILLING" INSERT=COMMON STYLE=S AO0
7 ( LTOOL 3 "ISO_BORING" INSERT=C STYLE=L AO90 CR0.8 CPI=T
8 ( LTOOL 4 "ISO_TURNING" INSERT=D STYLE=H1 AO0 CR0.8 CPI=
9
10
11
12
13
14 G00 G18 G21 G99 G
15 G50 S2000
16 G28 U0. W0.
17 G54
18 G97 S300 M03
19 (FACE & TURN CNMG
20 T0101
21 G0 X133. Z3.
22 G96 S160
23 (FACING)
24 G0 X137. Z3.
25 G1 Z.5 F1.
26 X-1.6 F.2
27 G0 Z2.
28 X133.
29 Z0
30 G1 X-1.6
31 G0 Z2.
32 X130.
33 (ROUGH TURN PROFI
34 G71 U2. R.5
35 G71 F.2 P1 Q2 U1.
36 G28 U0 W0
37 M01
38
39 (UDRILL 30 MM THROUGH)
40 G00 G18 G21 G99 G40

```

Insert: Finish Turn & Face 55 Deg

Parameters for 'Finish Turn Face 55 Deg'

Tool Number: 4

Insert Radius: 0.8

* = Optional parameter

TURNING TOOL HOLDER H USING INSERT D

CUTTER POINT =T3 ORIENTATION = 0

UNITS UM=METRIC UI=IMPERIAL

NOTE! SEE TOOL SETUP FOR MORE DETAILS

TURNING TOOL

INSERT D

25

100

33

24

17.5

55

R.8

Default Cancel OK

Resulting insertion

CIMCO Edit 2022 - [C:\...OneDrive - Ebor Machine Tools Ltd\My Cimco\Cimco Direct Work\Basic 2022 Prog Examples\O620 Turning Taper Housing D Lathe

Editor NC Functions NC-Assistant **Backplot** File Compare Transmission CNC-Calc

Backplot Machine Simulation Backplot Close Backplot Zoom In Zoom Out Fit To Window View Reset Top Front Left View From Measure Rotate Show Bounding Box Toolpath Mode Toolpath Tool Tool Setup Add Stock Zoom / Regenerate solid Solid Setup Fanuc Turning Control Type

O620 Turning Taper Housing D ... x

C-Assistant

No macro found

```

1 %
2 O0620 (TURN & BORE TAPER HOUSING)
3 (STOCK TURN OD132 ID0 L123 PZ3)
4 (WCS ID1 X0 Y0 Z0)
5 ( LTOOL 1 "ISO_TURNING" INSERT=C STYLE=L AO0 CR0.8 CPI=T
6 ( LTOOL 2 "ISO_LATHE DRILLING" INSERT=COMMON STYLE=S AO0
7 ( LTOOL 3 "ISO_BORING" INSERT=C STYLE=L AO90 CR0.8 CPI=T
8 ( LTOOL 4 "ISO_TURNING" INSERT=D STYLE=H1 AO0 CR0.8 CPI=
9
10
11
12
13

```

Tool 5 Finish bore the internal taper profile

CIMCO Edit 2022 - [C:\...OneDrive - Ebor Machine Tools Ltd\My Cimco\Cimco Direct Work\Basic 2022 Prog Examples\0620 Turning Taper Housing D Latest.NC *]

Editor NC Functions NC-Assistant Backplot File Compare Transmission CNC-Calc

Backplot Machine Simulation Backplot File* Close Backplot

Zoom In Zoom Out Fit To Window View Reset Top Front Left View From Tool Measure Rotate Distance View* Show Bounding Box*

Toolpath Mode Tool Setup Add Stock* Zoom / Regenerate solid Solid Setup Control Type Fanuc Turning

0620 Turning Taper Housing D ... x

NC-Assistant

No macro found

Cycles / Macros

- Turn & Face 80 Deg
- Finish Turn & Face 55 Deg
- Bore & Face 80 Deg
- Finish Bore & Face 55 Deg
- Groove OD

```

1 %
2 O0620 (TURN & BORE TAPER HOUSING)
3 (STOCK TURN OD132 ID0 L123 PZ3)
4 (WCS ID1 X0 Y0 Z0)
5 ( LTOOL 1 "ISO_TURNING" INSERT=C STYLE=L AO0 CR0.8 CPI=T3 FHD4.5 FW32 HL35 HAT80 ICD11.818 US=UM
6 ( LTOOL 2 "ISO_LATHE_DRILLING" INSERT=COMMON STYLE=S AO90 BL150 CPI=T8 D30 FL20 US=UM AD30 LS80
7 ( LTOOL 3 "ISO_BORING" INSERT=C STYLE=L AO90 CR0.8 CPI=T2 FHD4.4 FW22 HL32 HAT80 ICD9.53 US=UM
8 ( LTOOL 4 "ISO_TURNING" INSERT=C STYLE=L AO90 CR0.8 CPI=T2 FHD4.4 FW22 HL32 HAT80 ICD9.53 US=UM
9
10
11
12
13
14
15 G00 G18 G21 G
16 G50 S2000
17 G28 U0. W0.
18 G54
19 G97 S300 M03
20 (FACE & TURN
21 T0101
22 G0 X133. Z3.
23 G96 S160
24 (FACING)
25 G0 X137. Z3.
26 G1 Z.5 F1.
27 X-1.6 F.2
28 G0 Z2.
29 X133.
30 Z0
31 G1 X-1.6
32 G0 Z2.
33 V1 30

```

Insert: Finish Bore & Face 55 Deg

Parameters for 'Finish Bore Face 55 Deg'

Tool Number: 5

Insert Radius: 0.8

* = Optional parameter

BORING BAR STYLE Q USING INSERT D

CUTTER POINT = T2 ORIENTATION = 90

UNITS UM=METRIC UI=IMPERIAL

NOTE! SEE TOOL SETUP FOR MORE DETAILS

INSERT D RAD

25 125 30 25 17.5

Default Cancel OK

Resulting insertion

CIMCO Edit 2022 - [C:\...OneDrive - Ebor Machine Tools Ltd\My Cimco\Cimco Direct Work\Basic 2022 Prog Examples\0620 Turning Taper Housing D Latest.NC *]

Editor NC Functions NC-Assistant Backplot File Compare Transmission CNC-Calc

Backplot Machine Simulation Backplot File* Close Backplot

Zoom In Zoom Out Fit To Window View Reset Top Front Left View From Tool Measure Rotate Distance View* Show Bounding Box*

Toolpath Mode Tool Setup Add Stock* Zoom / Regenerate solid Solid Setup Control Type Fanuc Turning

0620 Turning Taper Housing D ... x

NC-Assistant

No macro found

```

1 %
2 O0620 (TURN & BORE TAPER HOUSING)
3 (STOCK TURN OD132 ID0 L123 PZ3)
4 (WCS ID1 X0 Y0 Z0)
5 ( LTOOL 1 "ISO_TURNING" INSERT=C STYLE=L AO0 CR0.8 CPI=T3 FHD4.5 FW32 HL35 HAT80 ICD11.818 US=UM
6 ( LTOOL 2 "ISO_LATHE_DRILLING" INSERT=COMMON STYLE=S AO90 BL150 CPI=T8 D30 FL20 US=UM AD30 LS80
7 ( LTOOL 3 "ISO_BORING" INSERT=C STYLE=L AO90 CR0.8 CPI=T2 FHD4.4 FW22 HL32 HAT80 ICD9.53 US=UM
8 ( LTOOL 4 "ISO_TURNING" INSERT=D STYLE=H1 AO0 CR0.8 CPI=T2 FHD4.4 FW22 HL32 HAT80 ICD9.53 US=UM
9 ( LTOOL 5 "ISO_BORING" INSERT=D STYLE=Q AO90 CR0.8 CPI=T1 FHD4.4 FW22 HL32 HAT80 ICD9.53 US=UM
10
11
12
13
14
15

```

We can now test the whole program with Backplot but first tidy up the program to remove unwanted empty lines but we suggest the an empty line is left after every M01 so that is easy to see the end of each section.

We have one more important item to check!!

IMPORTANT

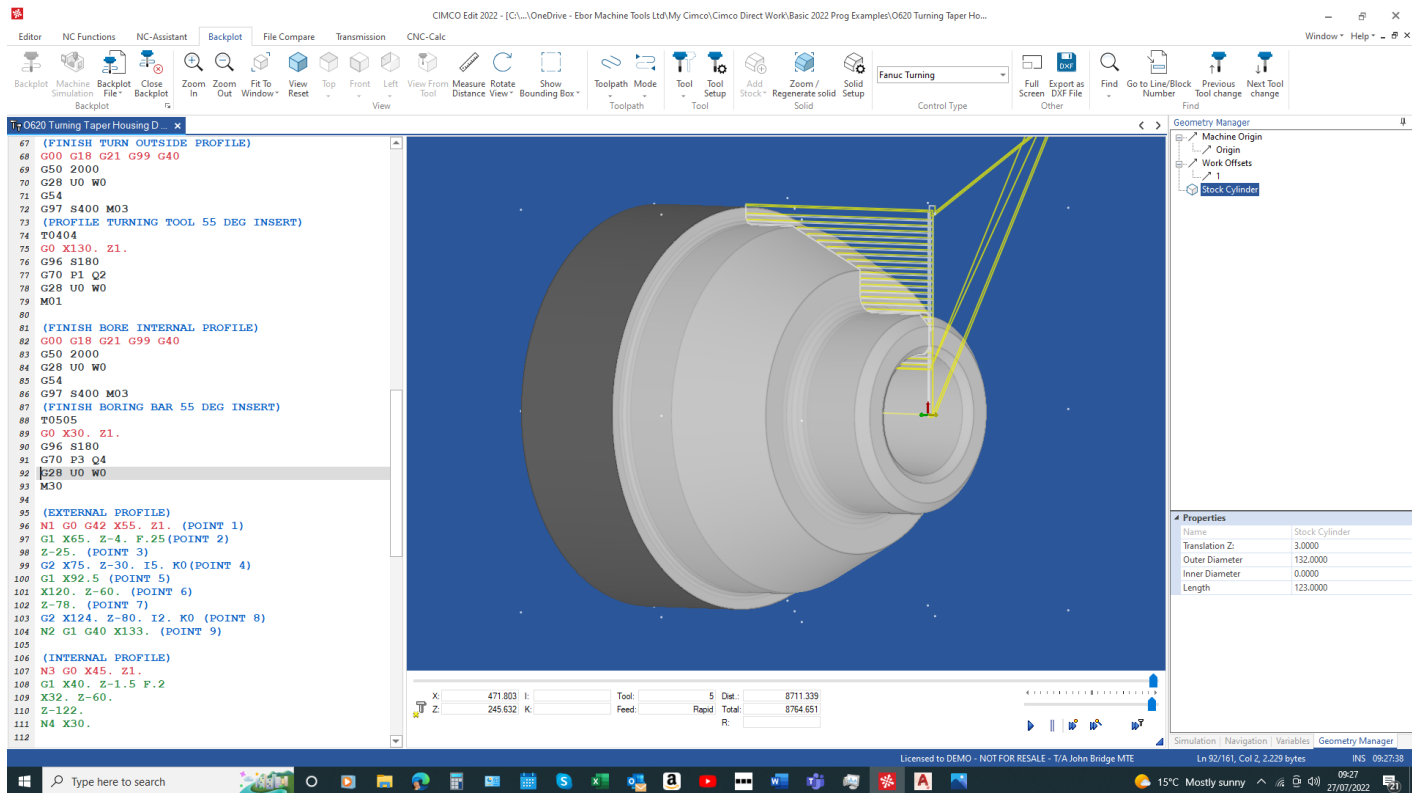
Simulation Settings	
Maximum Loop Iteration Limit	100000
Behavior When Limit Is Reached	Ignore the limit
Radius compensation	Disabled
Arc type	Enabled
Output errors	Disabled

At the right-hand side of the Backplot window select Simulation and make sure that radius compensation is enabled

Simulation | Navigation | Variables | Geometry Manager

OR RESALE - T/A John Bridge MTE Ln 1/166, Col 10, 2.176 bytes INS 08:08:42

Backplot is now showing the whole program. If to wish to manipulate the image see the notes below. If you want to run the program, click the triangular button under the Backplot window. If you would like to run it again then place the cursor at the top of the program and press Zoom / Regenerate solid button in the main ribbon.



To control the image in the Backplot window note the following:

Left Mouse Push button to Rotate the image

Right Mouse Push button to Pan the image

Mouse Wheel to Zoom the image

Here is a copy of the main program complete with a few tidy up comments:

%

O0620 (TURN & BORE TAPER HOUSING)

(STOCK TURN OD132 ID0 L123 PZ3)

(WCS ID1 X0 Y0 Z0)

(LTOOL 1 "ISO_TURNING" INSERT=C STYLE=L AO0 CR0.8 CPI=T3 FHD4.5 FW32 HL35 HAT80
ICD11.818 US=UM O=OL LS100 SW25 EL12 UHW8.66)

(LTOOL 2 "ISO_LATHE_DRILLING" INSERT=COMMON STYLE=S AO90 BL150 CPI=T8 D30 FL20
US=UM AD30 LS80 SW25 SD30 SL25 TL0 AT179)

(LTOOL 3 "ISO_BORING" INSERT=C STYLE=L AO90 CR0.8 CPI=T2 FHD4.4 FW22 HL32 HAT80 ICD9.53
US=UM O=OL DMM19 LS168 EL9.677 UHW8.66)

(LTOOL 4 "ISO_TURNING" INSERT=D STYLE=H1 AO0 CR0.8 CPI=T3 FHD4.5 FW32 HL35 HW27
ICD9.83 US=UM O=OL LS100 SW25 EL12 UHW16)

(LTOOL 5 "ISO_BORING" INSERT=D STYLE=Q AO90 CR0.8 CPI=T1 DO5 FHD4.5 FW18 HL30 HAT55
ICD9.53 US=UM O=OL DMM25 LS200 EL11.634 UHW30)

(-----)

(FACE AND TURN OUTSIDE PROFILE)

G00 G18 G21 G99 G40

G50 S2000

G28 U0. W0.

G54

G97 S300 M03

(FACE & TURN CNMG 80 DEG)

T0101

G0 X133. Z3.

G96 S160

(FACING)

G0 X137. Z3.

G1 Z.5 F1.

X-1.6 F.2

G0 Z2.

X133.

Z0

G1 X-1.6

G0 Z2.

X130.

(ROUGH TURN PROFILE)

G71 U2. R.5

G71 F.2 P1 Q2 U1. W.15

G28 U0 W0

M01

(UDRILL 30 MM THROUGH)

G00 G18 G21 G99 G40

G50 2000

G28 U0 W0

G54

G97 S400 M03

(30MM UDRILL)

T0202

G0 X0. Z3.

G97 S1000

G1 Z-122. F.12

G0 Z2.

G28 U0 W0

M01

(ROUGH PROFILE THE BORE)

G00 G18 G21 G99 G40

G50 2000

G28 U0 W0

G54

G97 S400 M03

(BORING BAR 90 DEG INSERT)

T0303

G0 X30. Z1.

G96 S180

G71 U2. R.5

G71 F.2 P3 Q4 U-.5 W.15

G28 U0 W0

M01

(FINISH TURN OUTSIDE PROFILE)

G00 G18 G21 G99 G40

G50 2000

G28 U0 W0

G54

G97 S400 M03

(PROFILE TURNING TOOL 55 DEG INSERT)

T0404

G0 X130. Z1.

G96 S180

G70 P1 Q2

G28 U0 W0

M01

(FINISH BORE INTERNAL PROFILE)

G00 G18 G21 G99 G40

G50 2000

G28 U0 W0

G54

G97 S400 M03

(FINISH BORING BAR 55 DEG INSERT)

T0505

G0 X30. Z1.

G96 S180

G70 P3 Q4

G28 U0 W0

M30

(EXTERNAL PROFILE)

N1 G0 G42 X55. Z1. (POINT 1)

G1 X65. Z-4. F.25(PPOINT 2)

Z-25. (POINT 3)

G2 X75. Z-30. I5. K0(PPOINT 4)

G1 X92.5 (POINT 5)

X120. Z-60. (POINT 6)

Z-78. (POINT 7)

G2 X124. Z-80. I2. K0 (POINT 8)

N2 G1 G40 X133. (POINT 9)

(INTERNAL PROFILE)

N3 G0 X45. Z1.

G1 X40. Z-1.5 F.2

X32. Z-60.

Z-122.

N4 X30.

Table 3 G code list (1/2)

G code			Group	Function
A	B	C		
G00	G00	G00	01	Positioning (Rapid traverse)
G01	G01	G01		Linear interpolation (Cutting feed)
G02	G02	G02		Circular interpolation/Helical interpolation CW
G03	G03	G03		Circular interpolation/Helical interpolation CCW
G04	G04	G04	00	Dwell
G10	G10	G10		Data setting
G11	G11	G11		Data setting made cancel
G17	G17	G17	16	XpYp plane selection
G18	G18	G18		ZpXp plane selection
G19	G19	G19		YpZp plane selection
G20	G20	G70	06	Input in inch
G21	G21	G71		Input in mm
G27	G27	G27	00	Reference position return check
G28	G28	G28		Return to reference position
G30	G30	G30		2nd reference position return
G31	G31	G31		Skip function
G32	G33	G33	01	Thread cutting
G40	G40	G40	07	Tool nose radius compensation cancel
G41	G41	G41		Tool nose radius compensation left
G42	G42	G42		Tool nose radius compensation right
G50	G92	G92	00	Coordinate system setting, max. spindle speed setting
G52	G52	G52	00	Local coordinate system setting
G53	G53	G53		Machine coordinate system setting
G54	G54	G54	14	Workpiece coordinate system 1 selection
G55	G55	G55		Workpiece coordinate system 2 selection
G56	G56	G56		Workpiece coordinate system 3 selection
G57	G57	G57		Workpiece coordinate system 4 selection
G58	G58	G58		Workpiece coordinate system 5 selection
G59	G59	G59		Workpiece coordinate system 6 selection
G65	G65	G65	00	Macro command
G70	G70	G72	00	Finishing cycle (Other than 0–GCD)
G71	G71	G73		Stock removal in turning (Other than 0–GCD)
G72	G72	G74		Stock removal in facing (Other than 0–GCD)
G73	G73	G75		Pattern repeating (Other than 0–GCD)
G74	G74	G76		End face peck drilling (Other than 0–GCD)
G75	G75	G77		Outer diameter/internal diameter drilling (Other than 0–GCD)
G76	G76	G78		Multiple threading cycle (Other than 0–GCD)

Table 3 G code list (2/2)

G code			Group	Function
A	B	C		
G71	G71	G72	01	Traverse grinding cycle (For 0–GCD)
G72	G72	G73		Traverse direct constant–dimension grinding cycle (For 0–GCD)
G73	G73	G74		Oscillation grinding cycle (For 0–GCD)
G74	G74	G75		Oscillation direct constant–dimension grinding cycle (For 0–GCD)
G90	G77	G20	01	Outer diameter/internal diameter cutting cycle
G92	G78	G21		Thread cutting cycle
G94	G79	G24		End face turning cycle
G96	G96	G96	02	Constant surface speed programming
G97	G97	G97		Direct rpm speed programming
G98	G94	G94	05	Per minute feed
G99	G95	G95		Per revolution feed
—	G90	G90	03	Absolute programming
—	G91	G91		Incremental programming
—	G98	G98	11	Return to initial level
—	G99	G99		Return to R point level