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

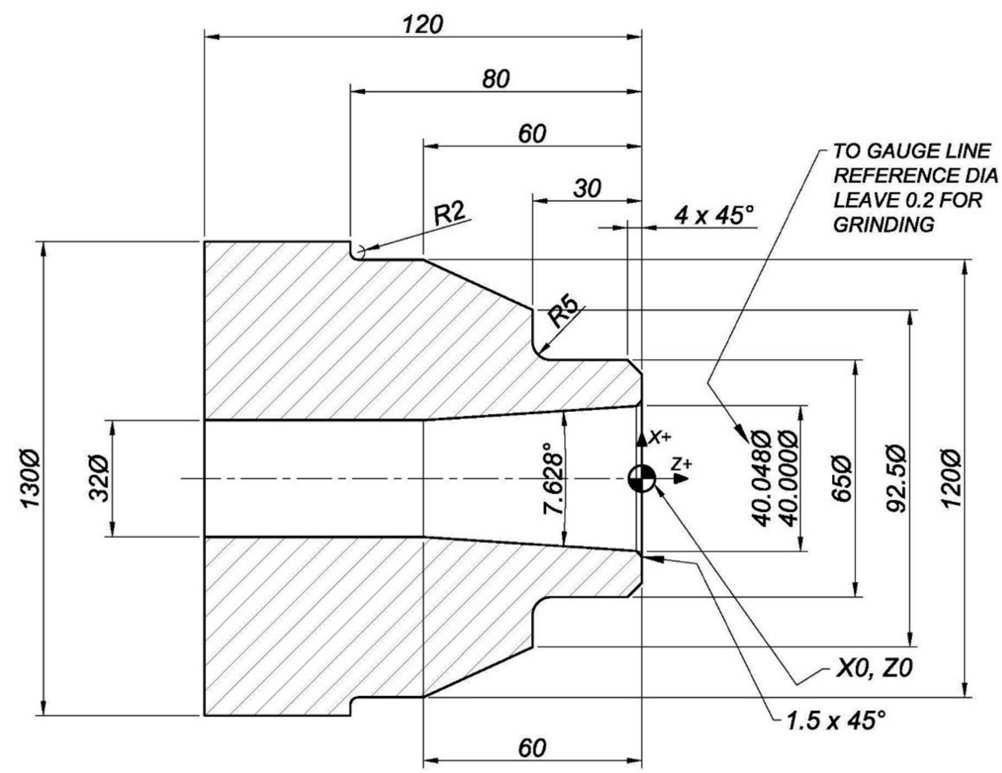
Table of Contents

CIMCO A/S
Copenhagen, Denmark
Tel: +45 45 85 60 50
Fax: +45 45 85 60 53
www.cimco.com
info@cimco.com

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:

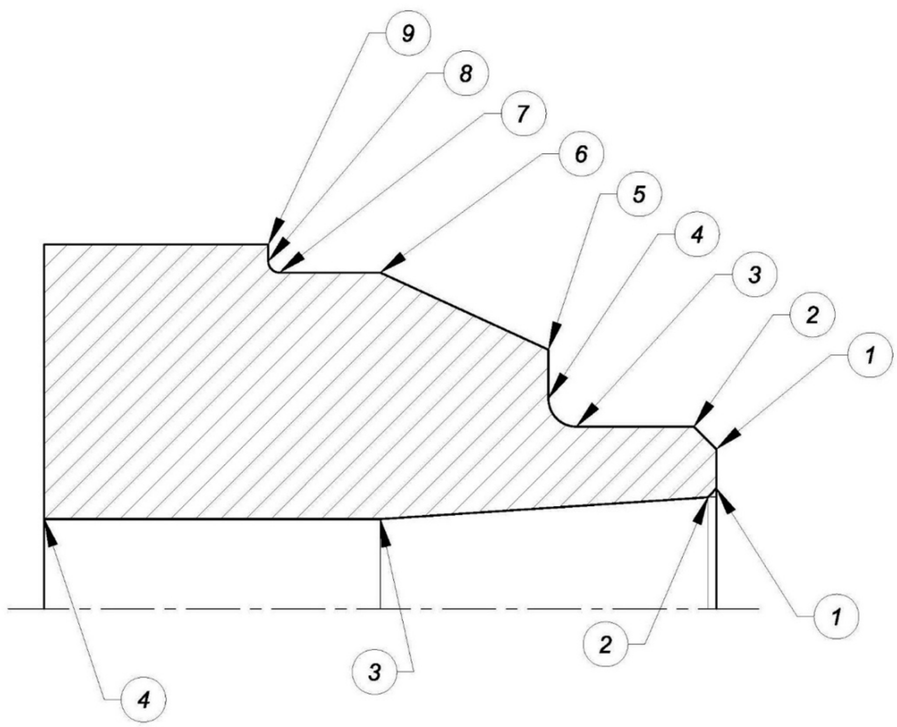
1. G00 – Straight line moves at rapid speed (On some machines this move is made in a vector line and on others a 45-degree move is followed by a single axis move to achieve the final programmed position). This is a modal command.

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.



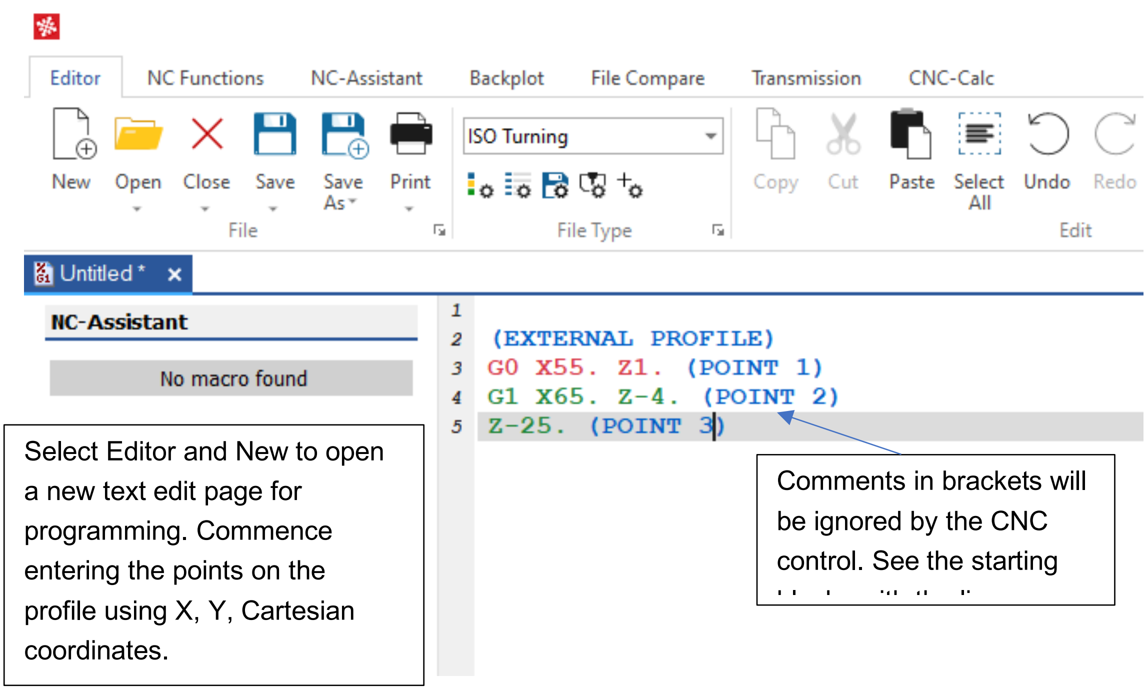
CIMCO A/S
Copenhagen, Denmark
Tel: +45 45 85 60 50
Fax: +45 45 85 60 53
www.cimco.com
info@cimco.com

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 and 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:



CIMCO A/S
Copenhagen, Denmark
Tel: +45 45 85 60 50
Fax: +45 45 85 60 53
www.cimco.com
info@cimco.com

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 maximin 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.

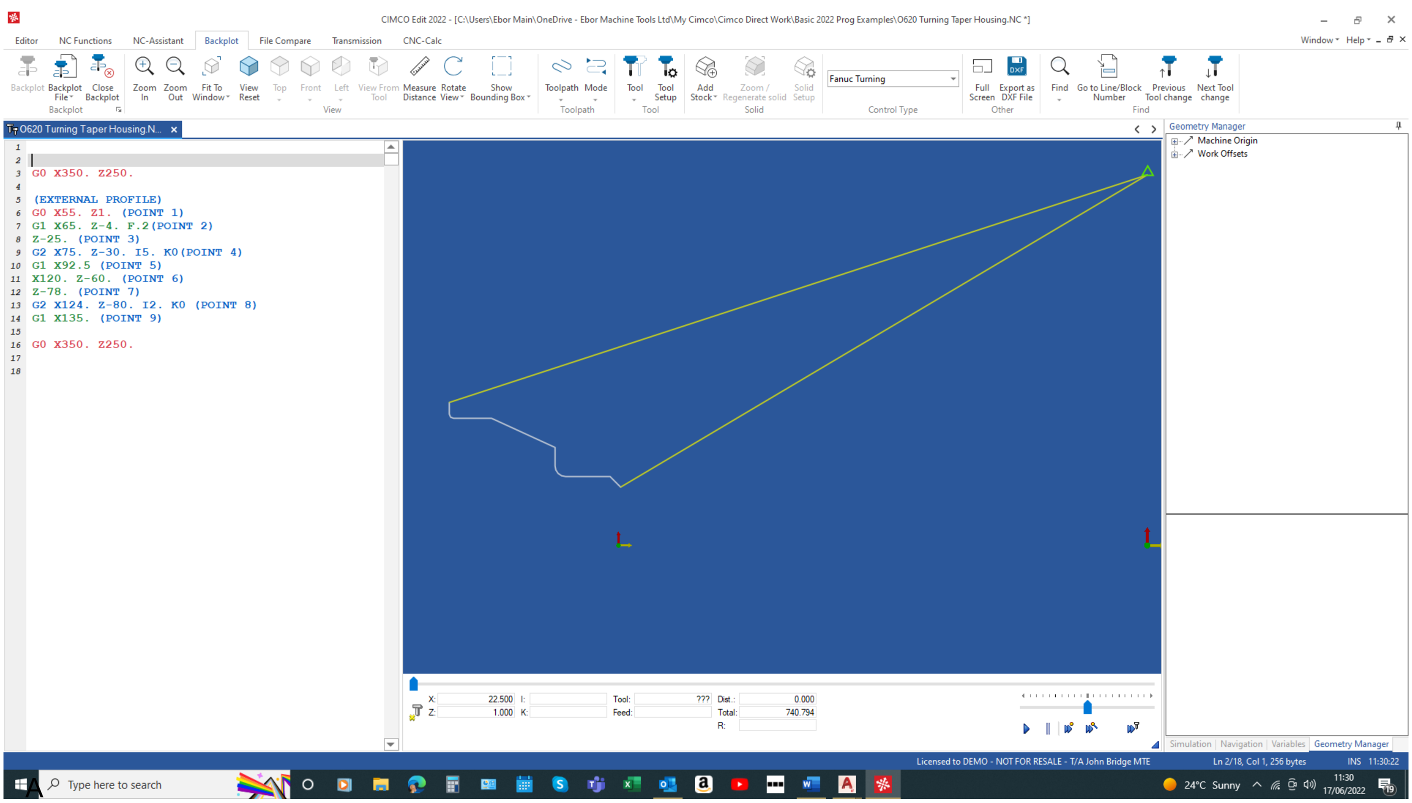
CIMCO A/S
Copenhagen, Denmark
Tel: +45 45 85 60 50
Fax: +45 45 85 60 53
www.cimco.com
info@cimco.com

```
(EXTERNAL PROFILE)
N1 G0 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)
N2 G1 X133. (POINT 9)
```

We set the tool 1 mm in front of Z0 to have a clean path into the profile, therefore the start diameter is 55 mm not 57 mm

Check your profile blocks with the drawing and the point-by-point diagram.

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



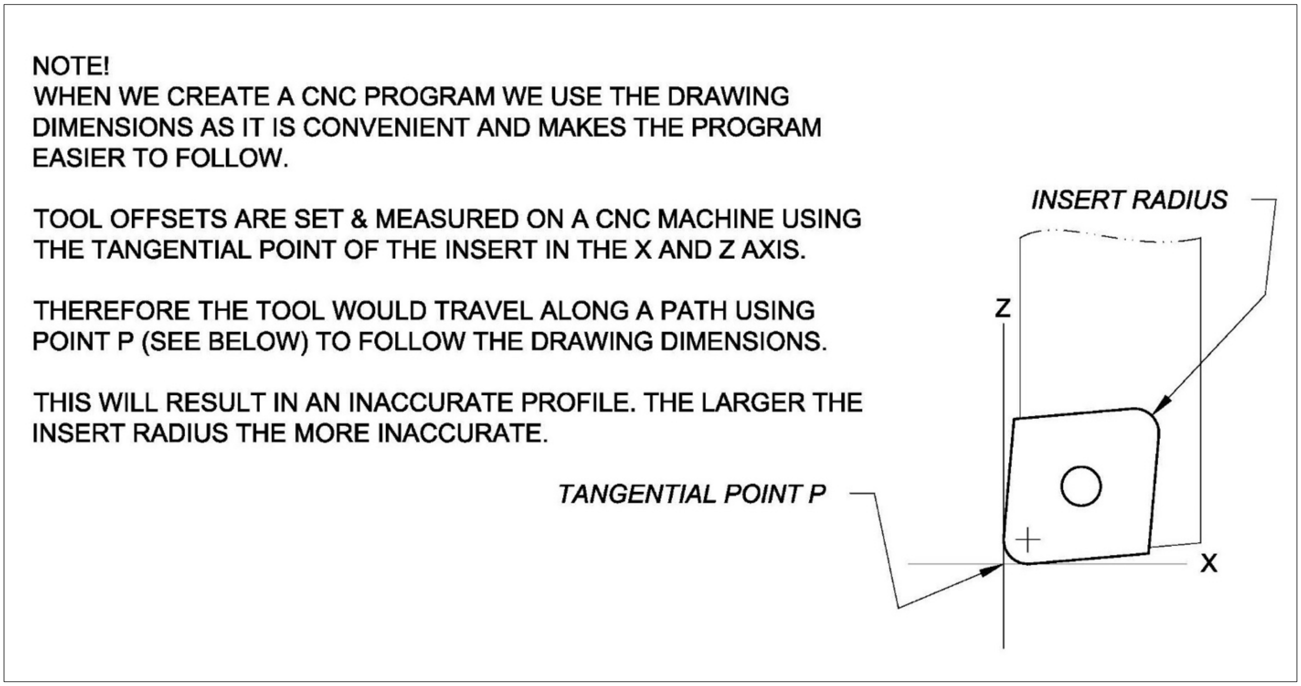
Further reminder - do not forget to change from circular to linear from element to element as the profile changes.

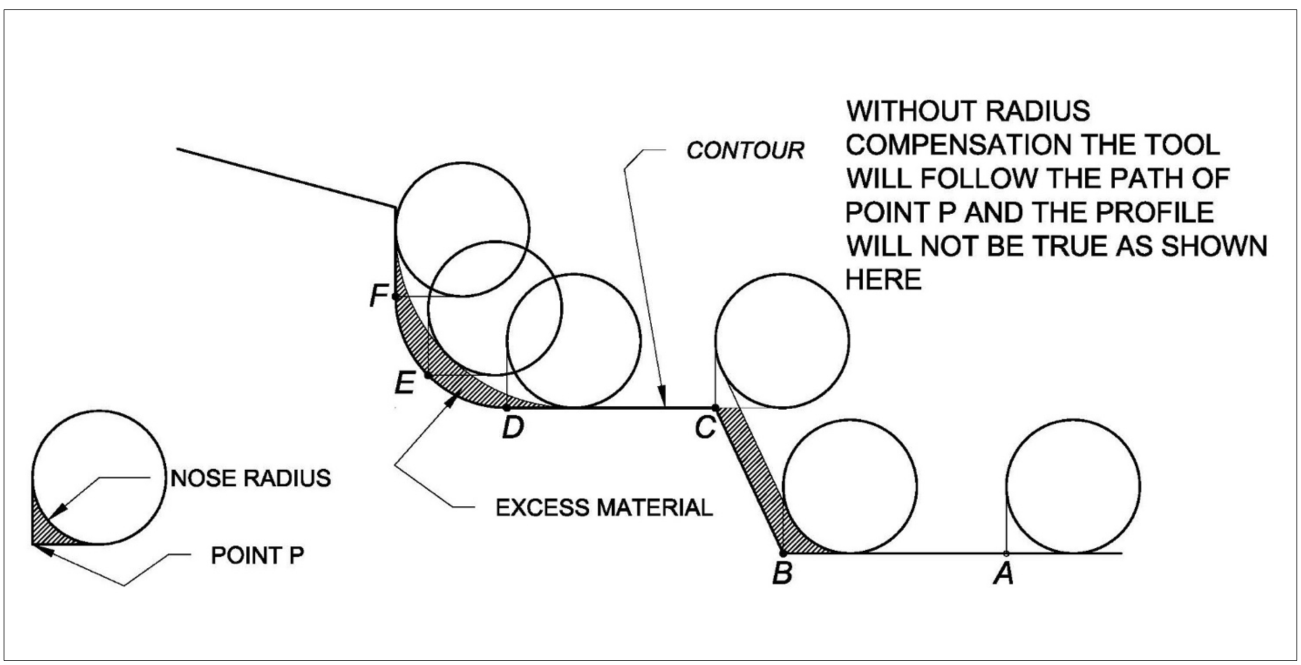
All group 1, move commands are modal e.g., after a G2 block if you were to enter X Z coordinate the CNC control will still try to move circular unless you place a G01 at the start of the block to define that this move is now linear.

CIMCO A/S
Copenhagen, Denmark
Tel: +45 45 85 60 50
Fax: +45 45 85 60 53
www.cimco.com
info@cimco.com

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.





CIMCO A/S
Copenhagen, Denmark
Tel: +45 45 85 60 50
Fax: +45 45 85 60 53
www.cimco.com
info@cimco.com

Having established to 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.

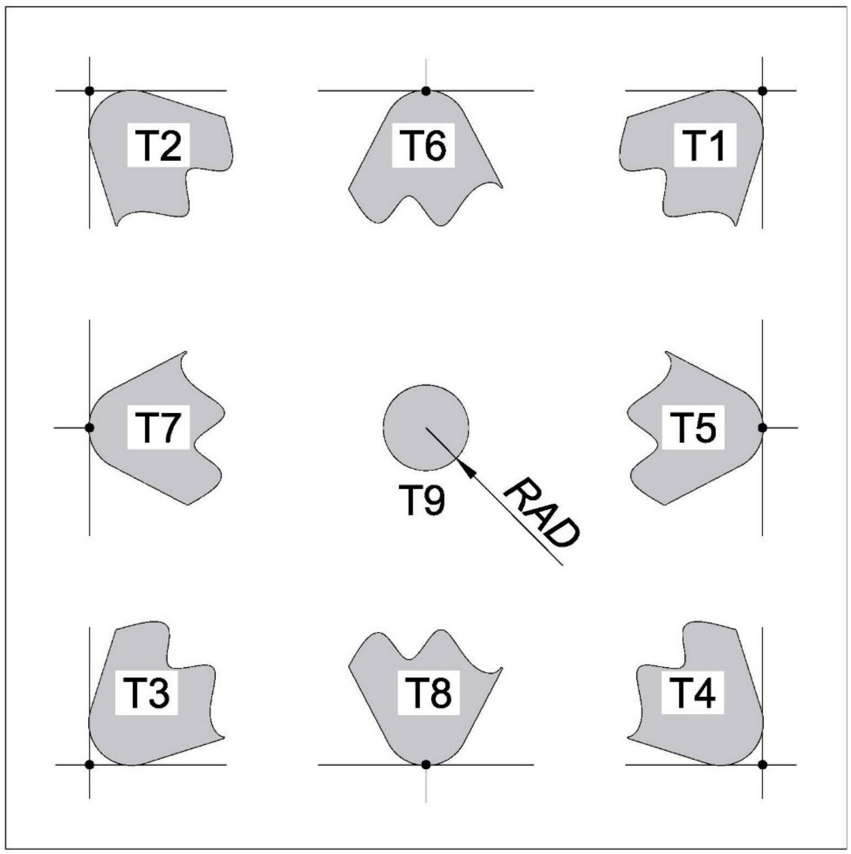
There are G commands that will activate an offset path as follows:

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.

CIMCO A/S
Copenhagen, Denmark
Tel: +45 45 85 60 50
Fax: +45 45 85 60 53
www.cimco.com
info@cimco.com

Continue

(EXTERNAL PROFILE)

N1 G0 G42 X55. Z1. (POINT 1)

See G42 inserted in the profile to activate Radius compensation.

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)

N2 G1 G40 X133. (POINT 9)

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

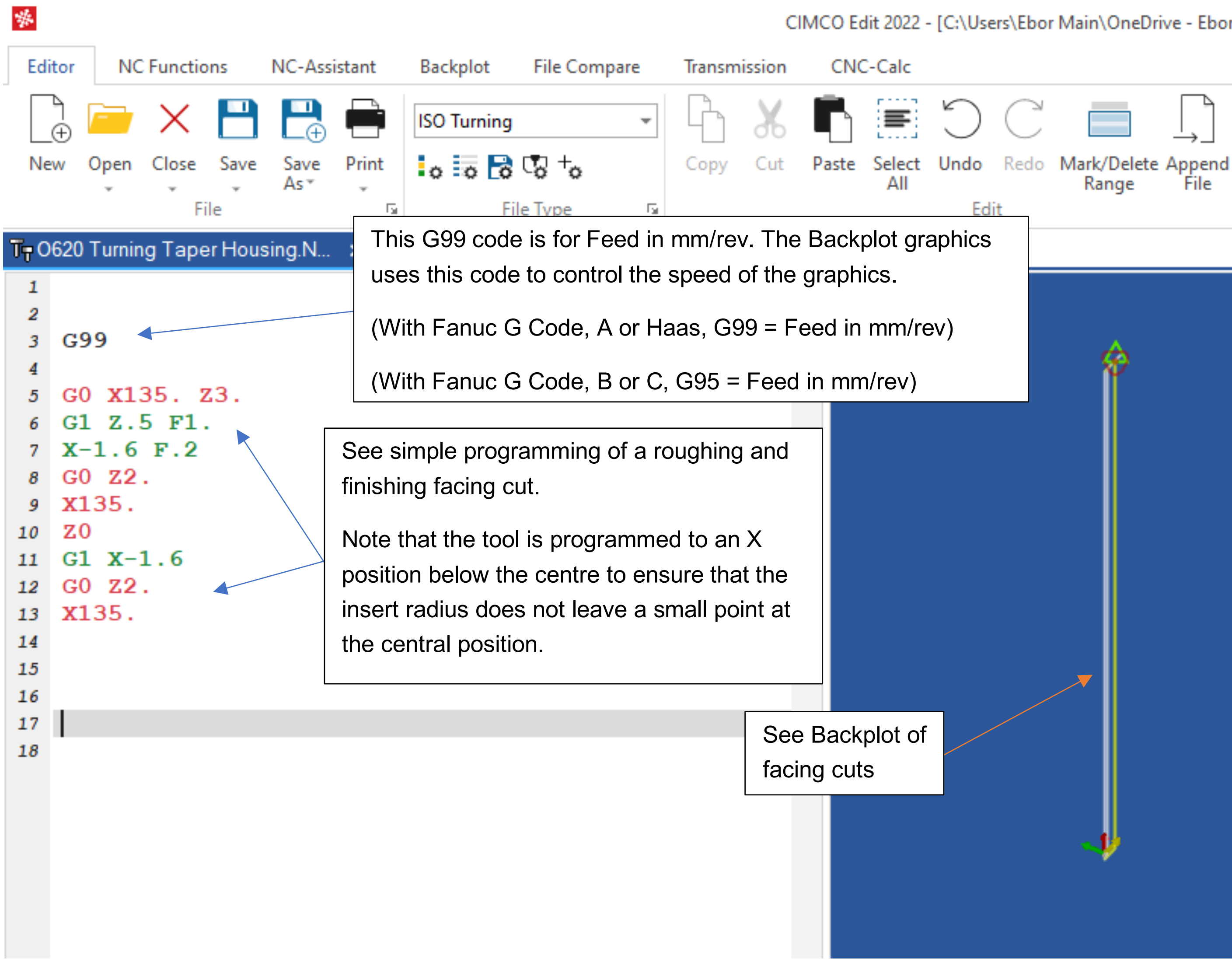
If you use block numbers put them first, G code command next, then X,Y,Z, coordinates in order, Feed rate commands next then M,S,T commands.

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.

CIMCO A/S
Copenhagen, Denmark
Tel: +45 45 85 60 50
Fax: +45 45 85 60 53
www.cimco.com
info@cimco.com

We will need to face the stock before we attempt to machine the profile to establish the front face at Z0. Any excess material on the face could cause collision problems. This simple operation can be programmed directly into the Editor, but most CNC controls have a facing cycle that can be set to take multiple facing cuts.



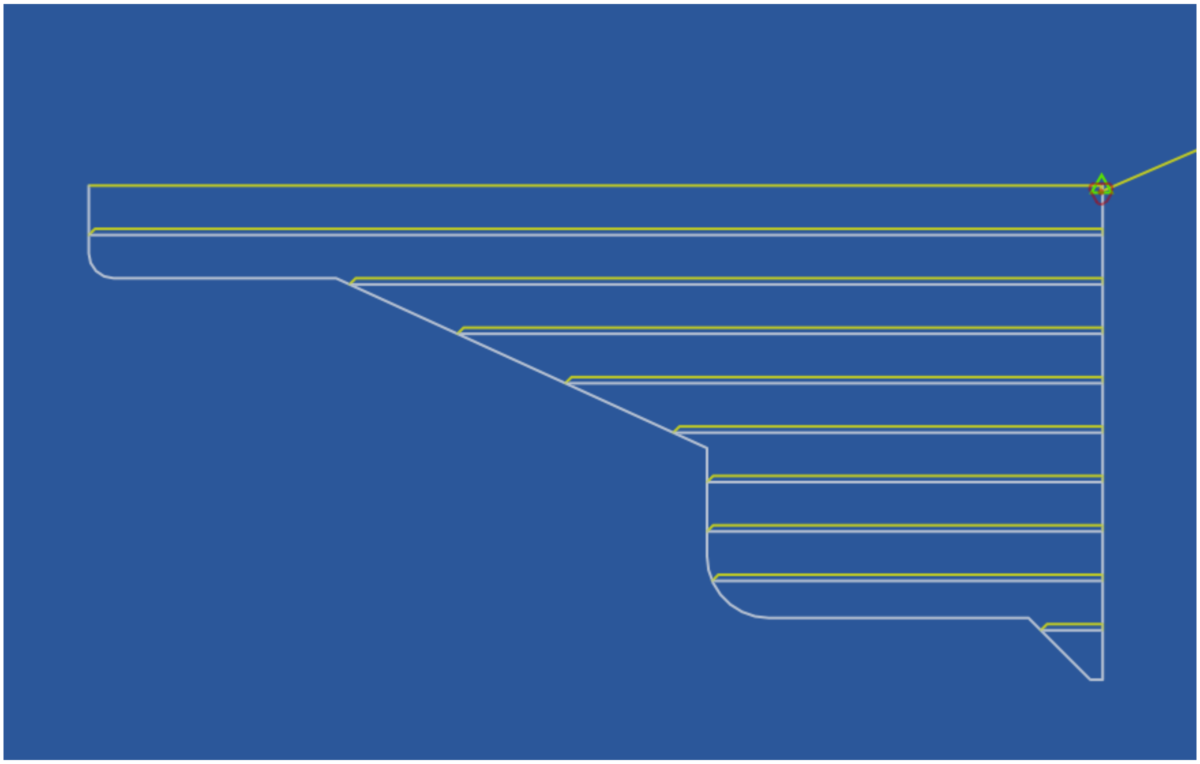
There are some differences with ISO G code as this system has evolved for the early days of NC. It is important to know on a specific CNC machine what G code system is in operation and there may be some slight edits to make if you wish to run a CNC program on more than one machine. All CNC machines will have a programming manual and you may need to refer to it. We will use here Fanuc code A which will also be generally acceptable on Haas CNC machine. The Cimco Editor can be set Fanuc code A, B, C, so that Backplot knows how to interpret the G code. At the end of this tutorial, you will find a chart showing the difference between A, B, C.

3.1 Roughing cycles

It is not generally possible to machine a profile with one cut and several cuts is usually required to rough out the bar stock to the finished profile. Most CNC controls have cycles that will carry out the operation using the cut depth that has been set in the cycle and the finish profile program to generate the final shape. See

CIMCO A/S
Copenhagen, Denmark
Tel: +45 45 85 60 50
Fax: +45 45 85 60 53
www.cimco.com
info@cimco.com

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.



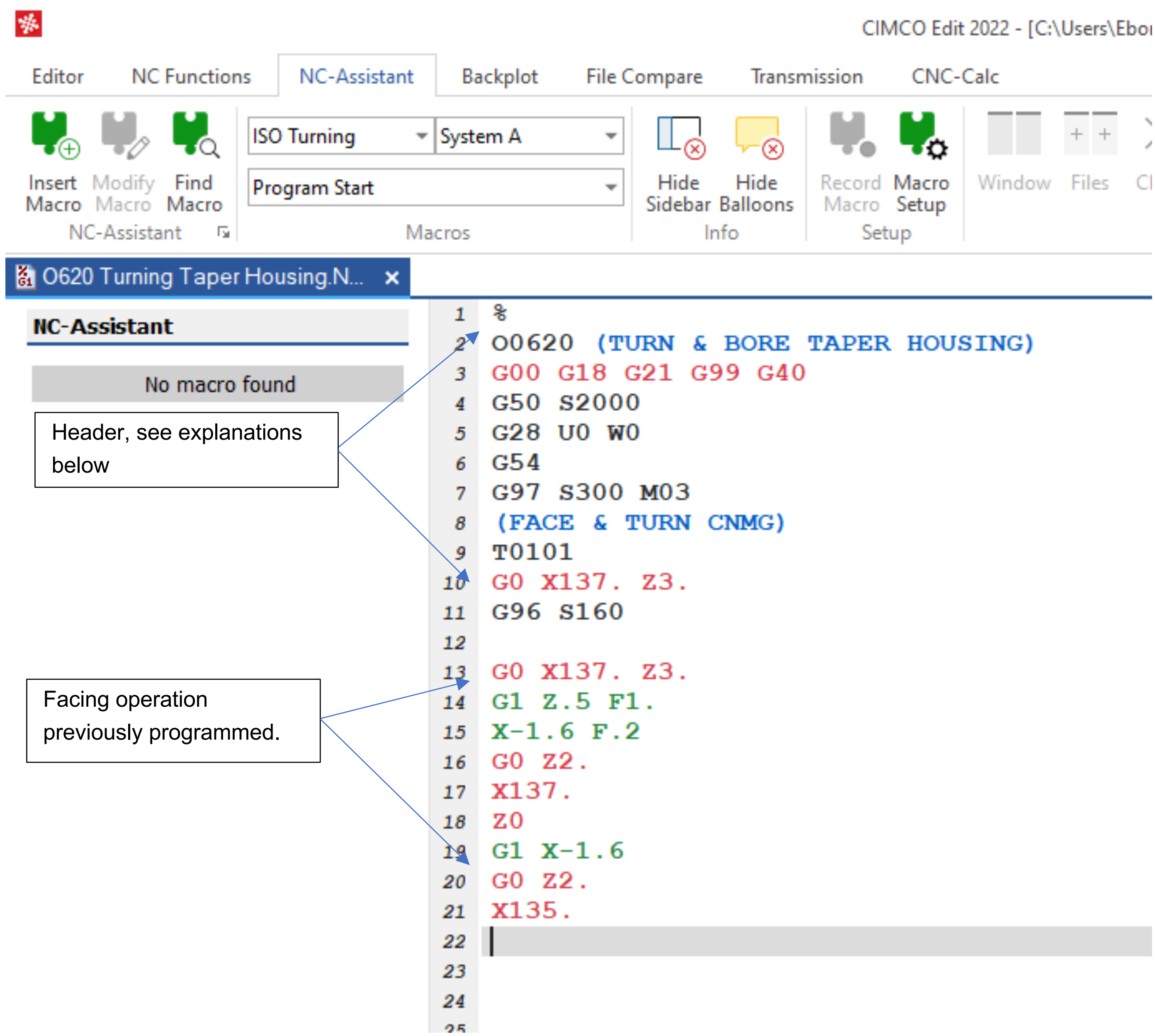
Bur before we can program the cycle, we need to look at other instructions the CNC control will need to carry out this operation.

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:

CIMCO A/S
Copenhagen, Denmark
Tel: +45 45 85 60 50
Fax: +45 45 85 60 53
www.cimco.com
info@cimco.com



% ;Program Start

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

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

CIMCO A/S
Copenhagen, Denmark
Tel: +45 45 85 60 50
Fax: +45 45 85 60 53
www.cimco.com
info@cimco.com

	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

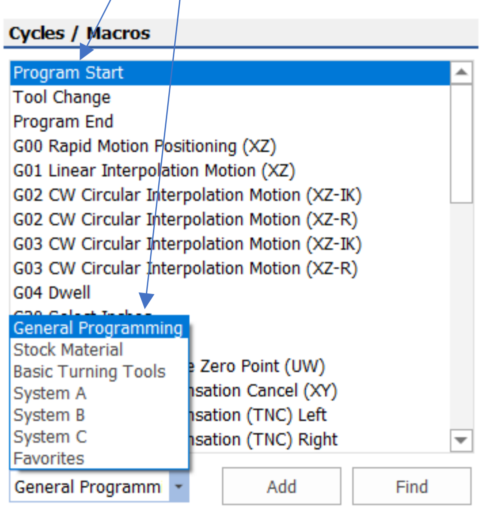
To insert a Header, see below:

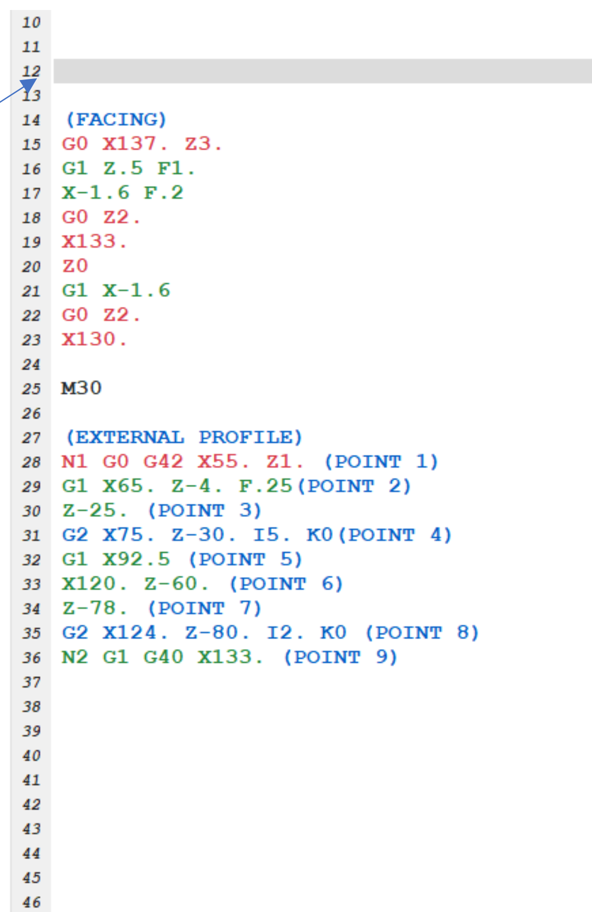
Set the cursor in the correct position.

From the NC-Assistant menu select General Programming

Click Program Start

``

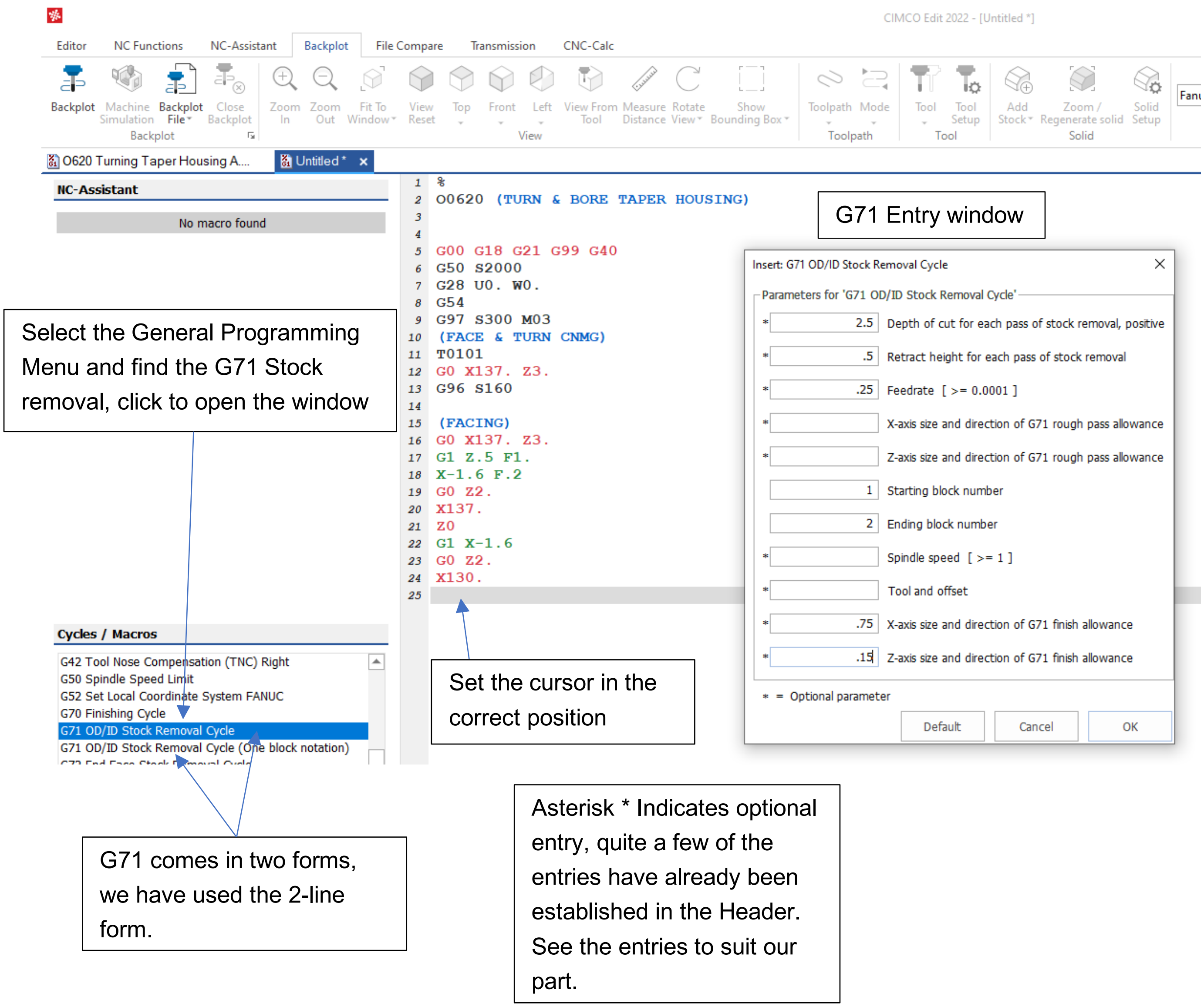




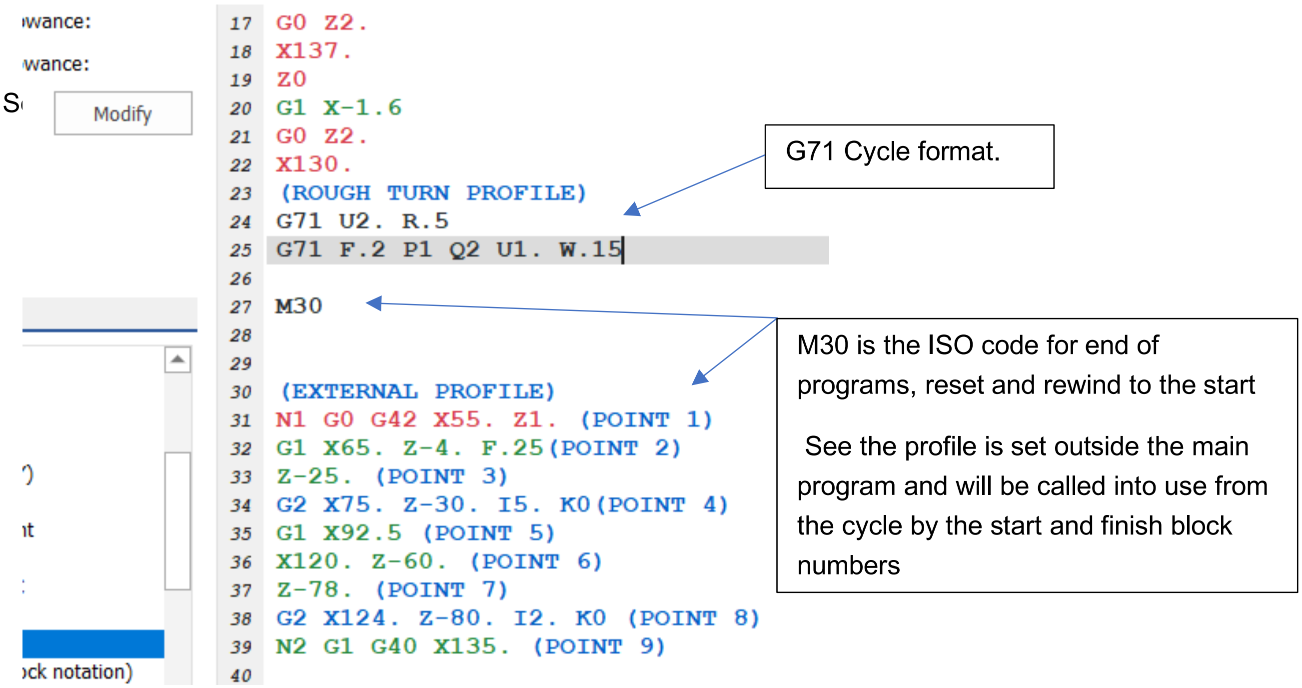
CIMCO A/S
Copenhagen, Denmark
Tel: +45 45 85 60 50
Fax: +45 45 85 60 53
www.cimco.com
info@cimco.com

4 Roughing cycle

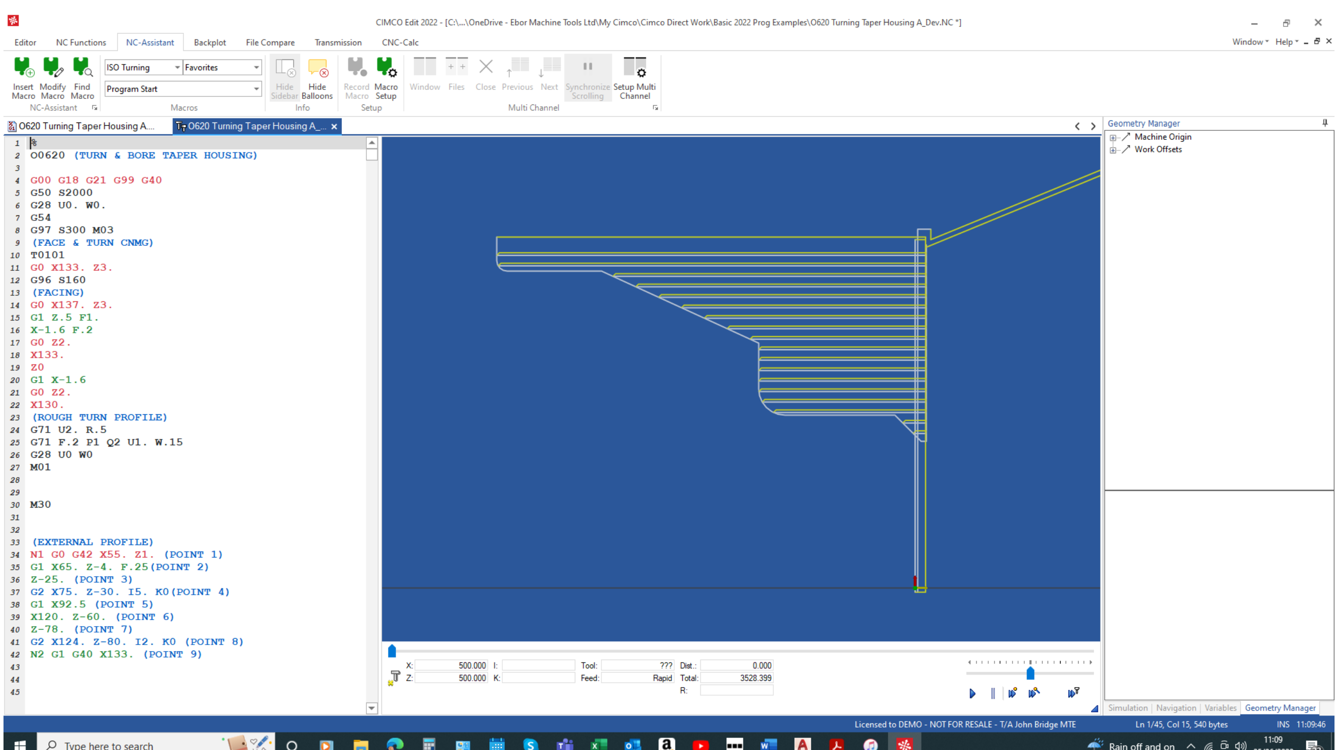
Now we have a profile. A header and facing blocks and we can proceed to include the roughing cycle. You will notice that we left 0.75 mm on the X diameter and 0.15 on the Z face so a finishing operation.



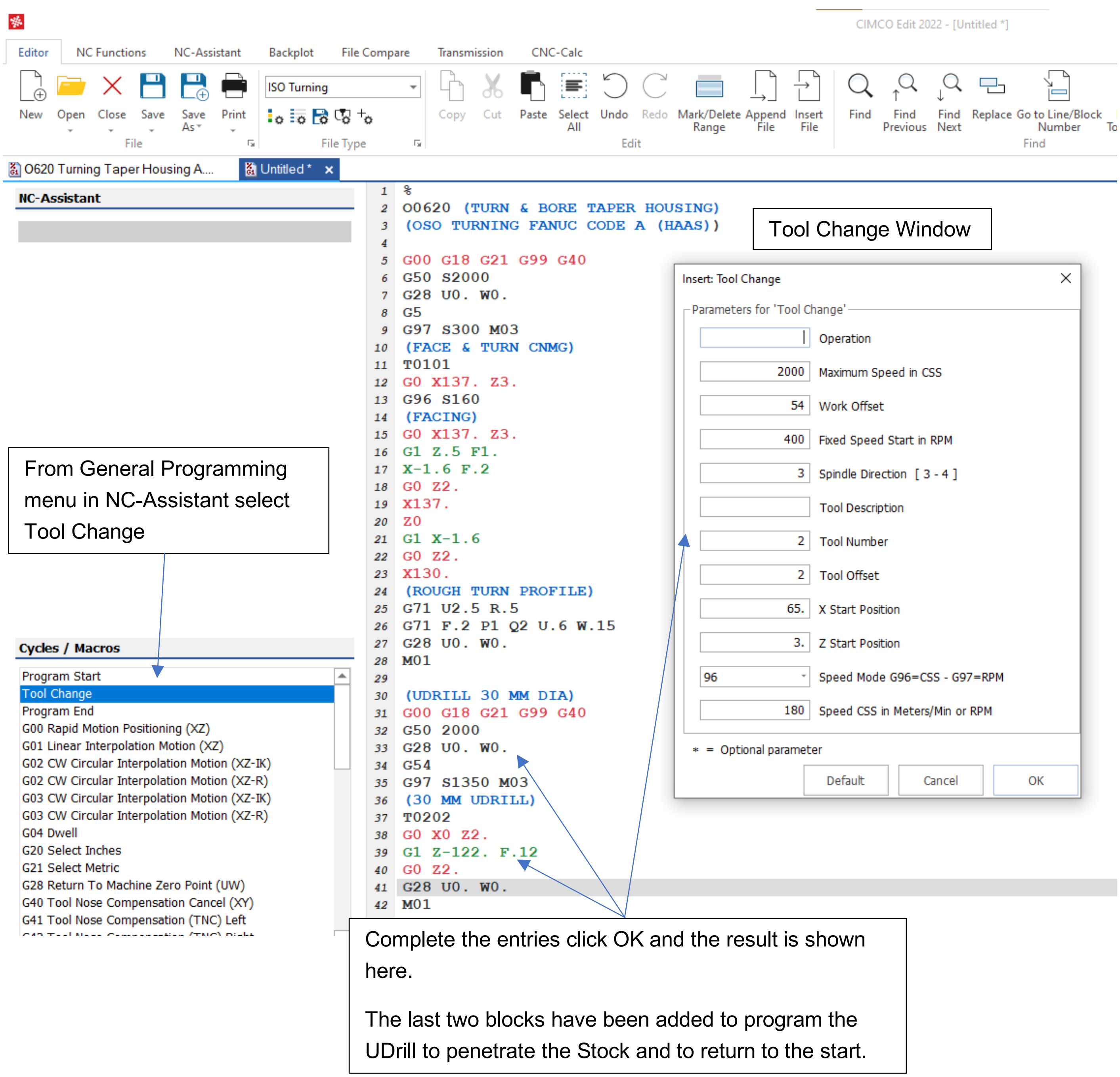
CIMCO A/S
Copenhagen, Denmark
Tel: +45 45 85 60 50
Fax: +45 45 85 60 53
www.cimco.com
info@cimco.com



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!

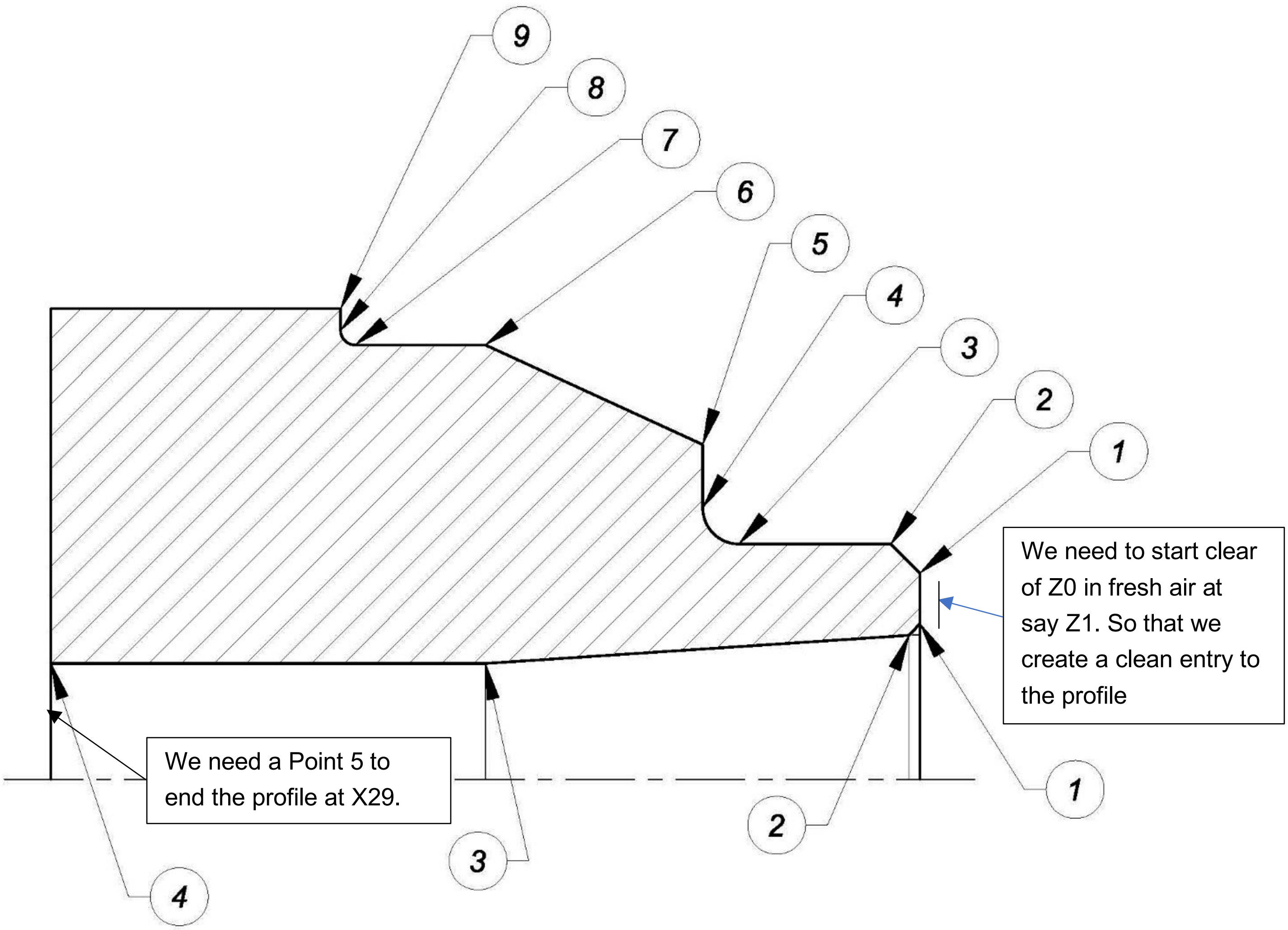


So, we now need the next operation to UDrill a hole through the Stock blank. We need a new operation and will use the Tool Change macro in the General Programming. This will terminate to the first operation and create a header for the next operation.



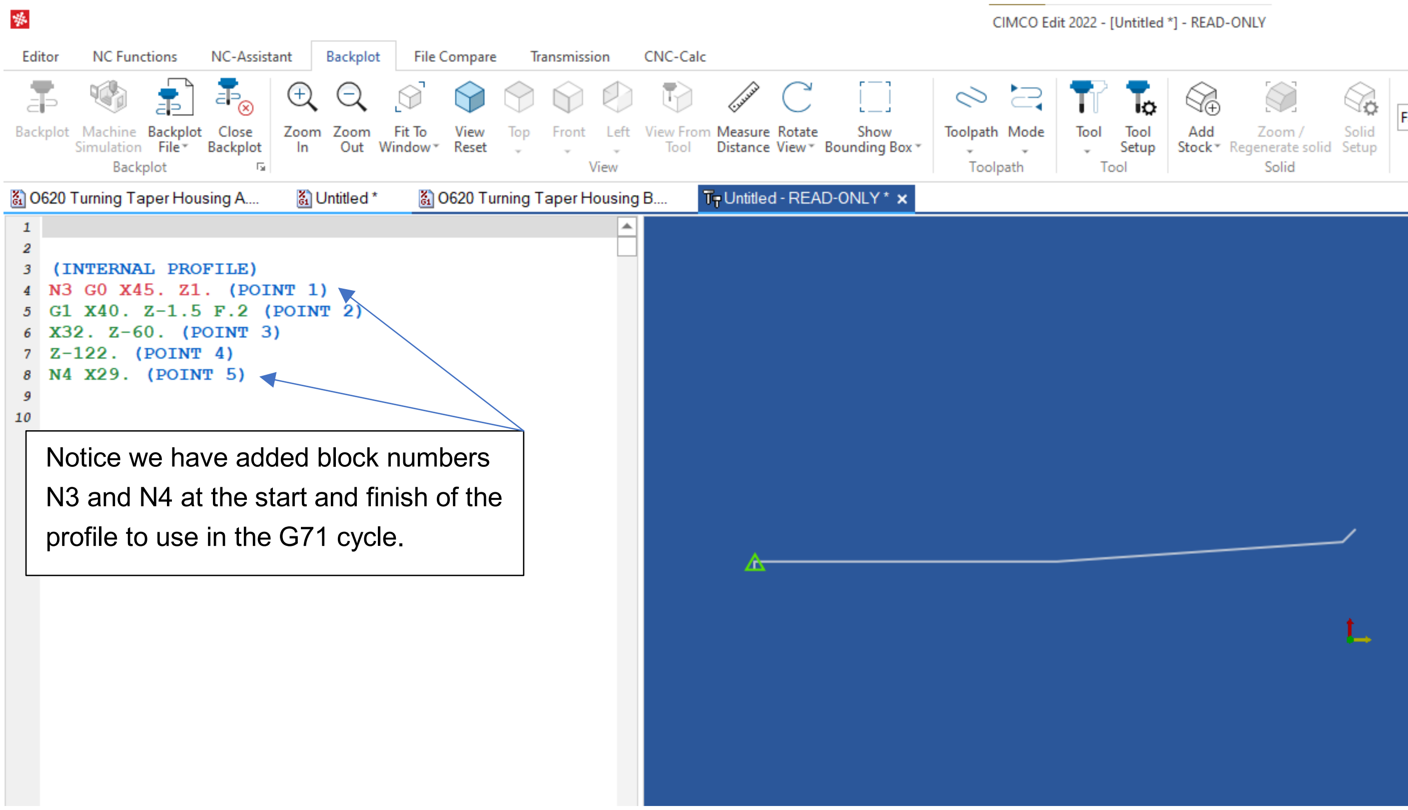
CIMCO A/S
Copenhagen, Denmark
Tel: +45 45 85 60 50
Fax: +45 45 85 60 53
www.cimco.com
info@cimco.com

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 ad 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.

CIMCO A/S
Copenhagen, Denmark
Tel: +45 45 85 60 50
Fax: +45 45 85 60 53
www.cimco.com
info@cimco.com



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.

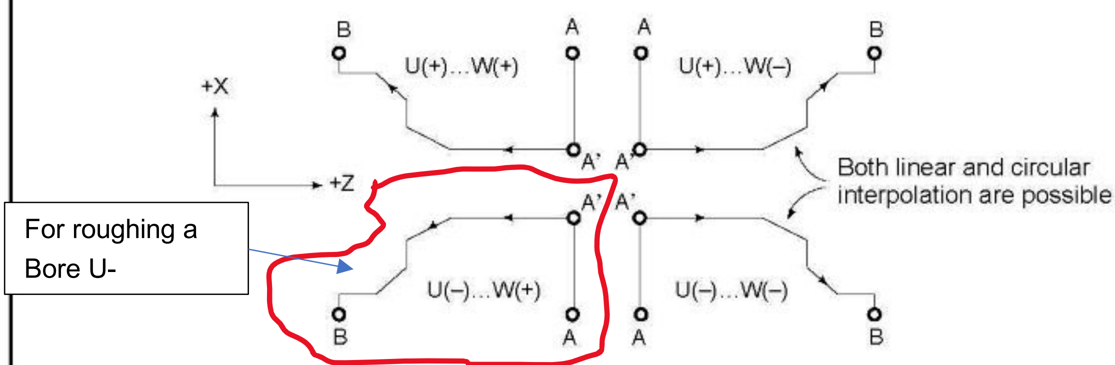
The G71 Cycle needs further information to work on an internal profile see the chart below showing the designation that will control the cycle in this respect.

CIMCO A/S
Copenhagen, Denmark
Tel: +45 45 85 60 50
Fax: +45 45 85 60 53
www.cimco.com
info@cimco.com

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.

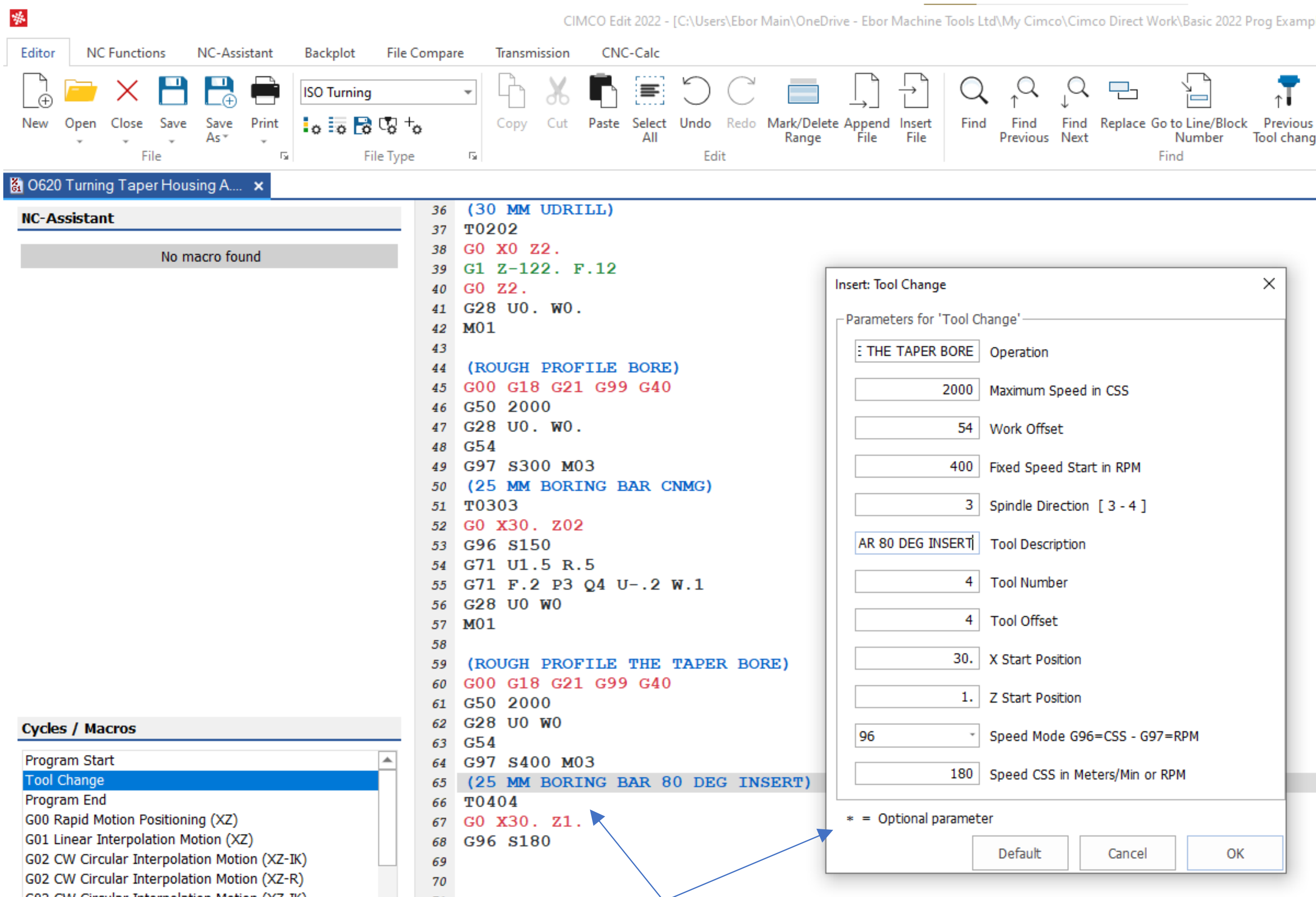
3 The subprogram cannot be called from the block between sequence number "ns" and "nf".

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)

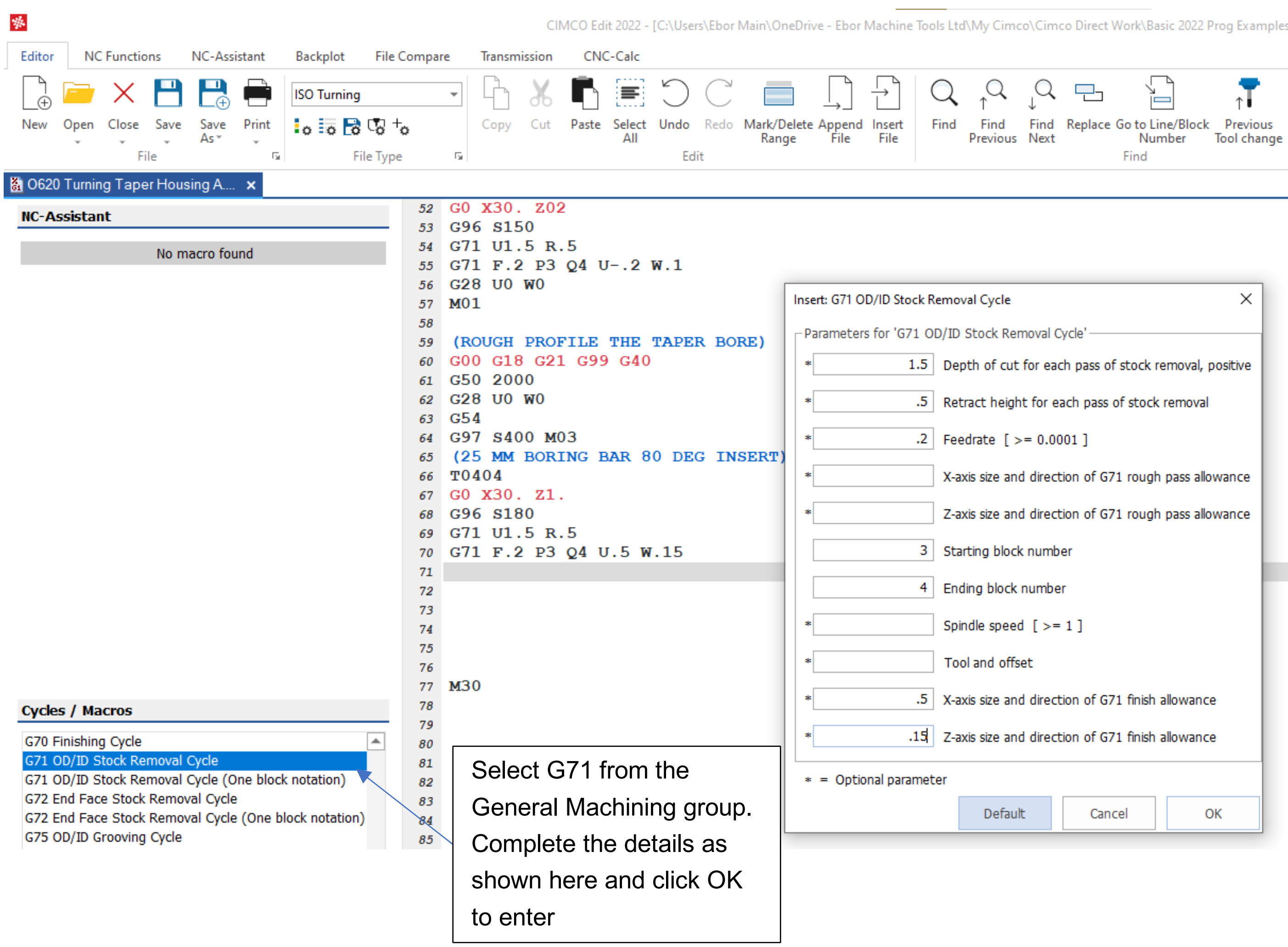
We can now program roughing of the internal profile with G71 Cycle as we know that for it to work, we need to have a negative U value. But first a tool change, see below:



Complete the Tool Change entries in the window and click OK

CIMCO A/S
Copenhagen, Denmark
Tel: +45 45 85 60 50
Fax: +45 45 85 60 53
www.cimco.com
info@cimco.com

Add the G71 Roughing Cycle

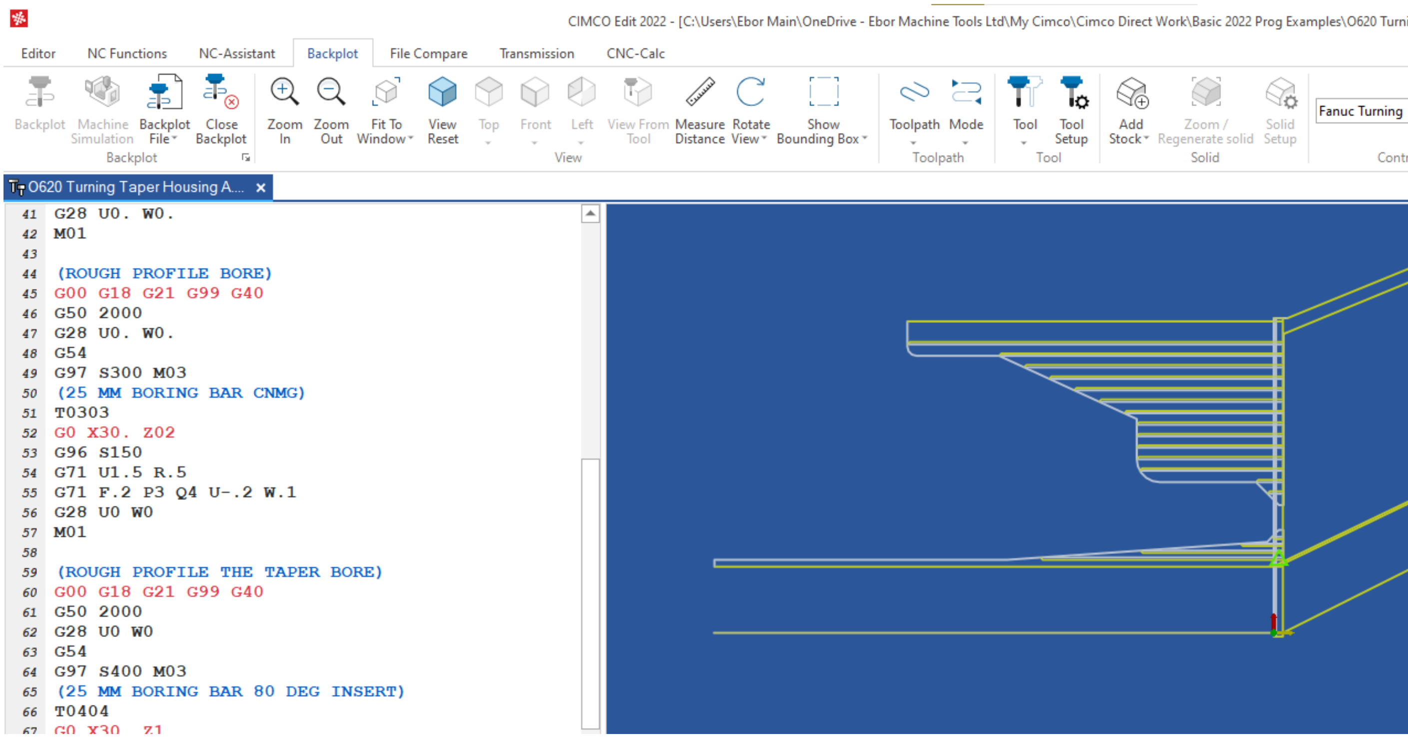


CIMCO A/S
Copenhagen, Denmark

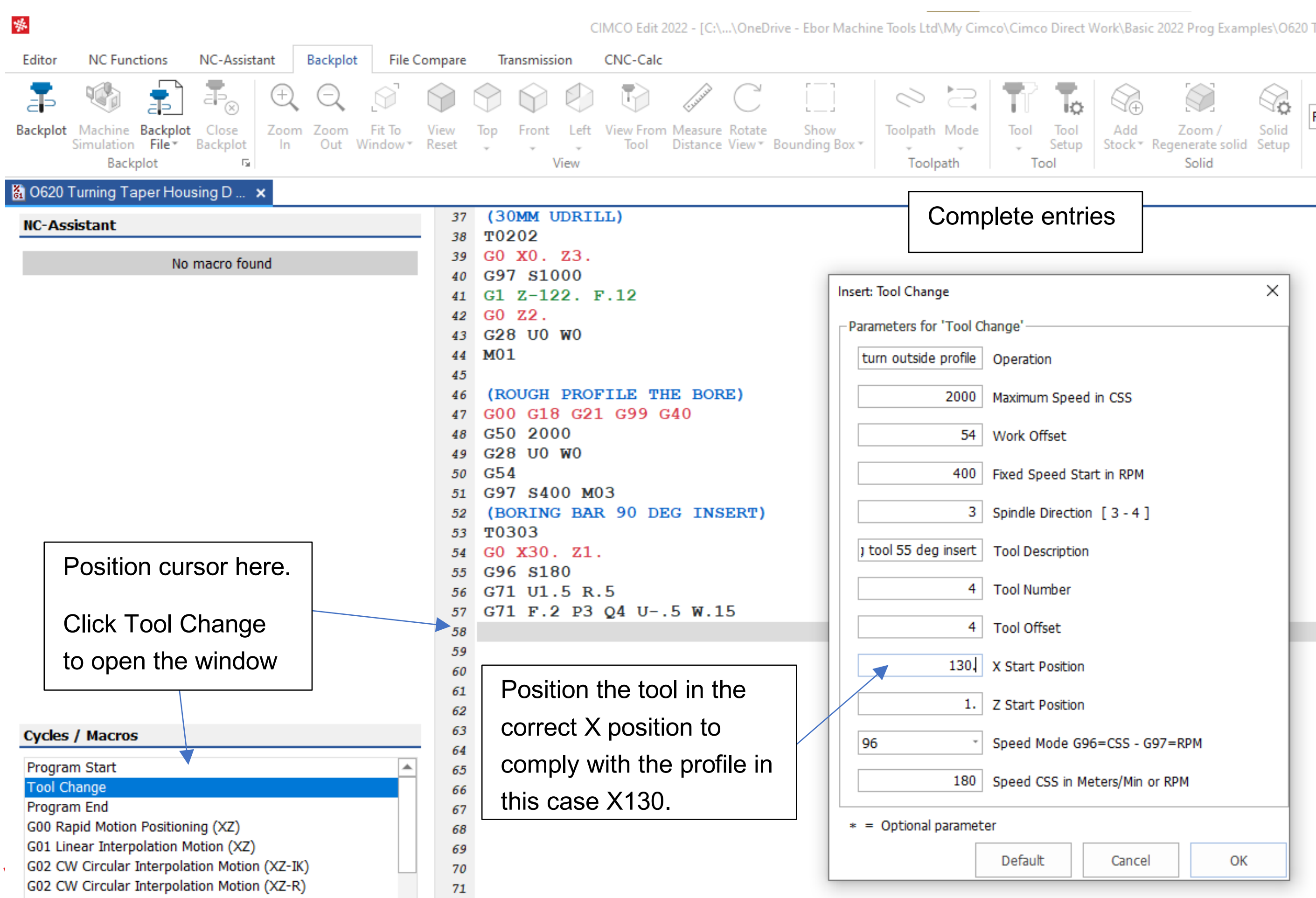
Tel: +45 45 85 60 50
Fax: +45 45 85 60 53

www.cimco.com
info@cimco.com

See Backplot results



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



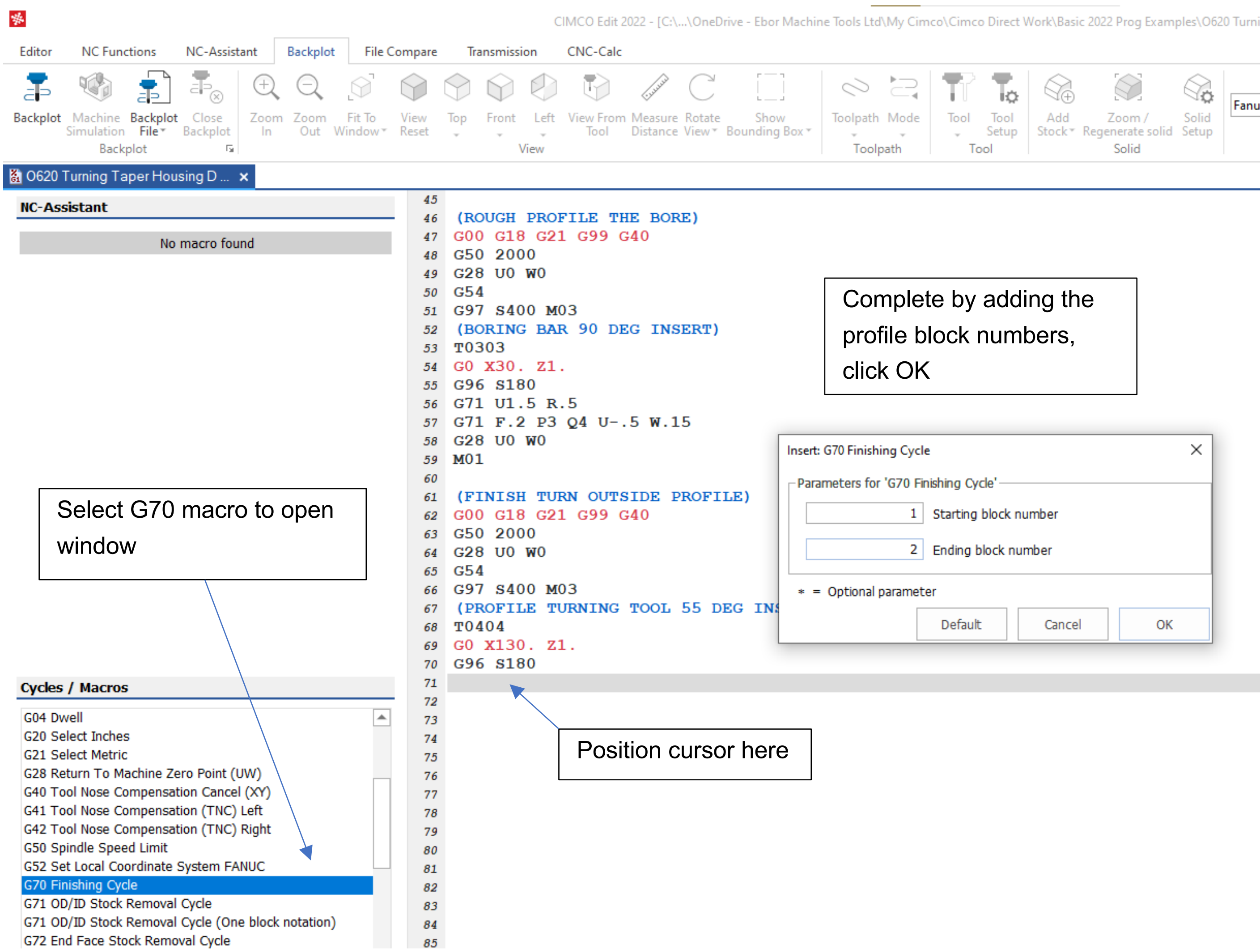
Inserted text after clicking OK now to add the finishing cycle

```
G96 S180
G71 U1.5 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
```

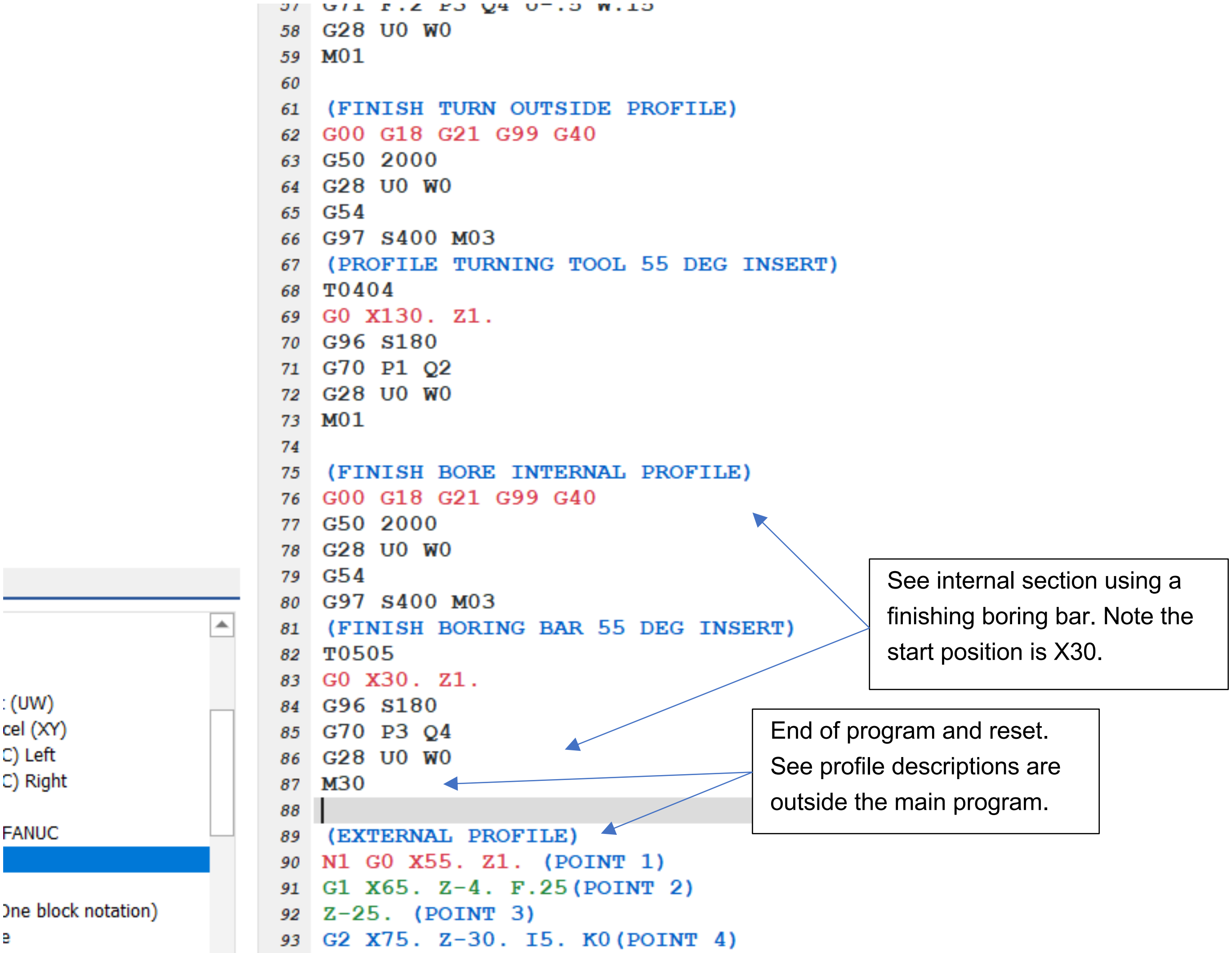

CIMCO A/S
Copenhagen, Denmark
Tel: +45 45 85 60 50
Fax: +45 45 85 60 53
www.cimco.com
info@cimco.com

Open the Finishing macro window see below



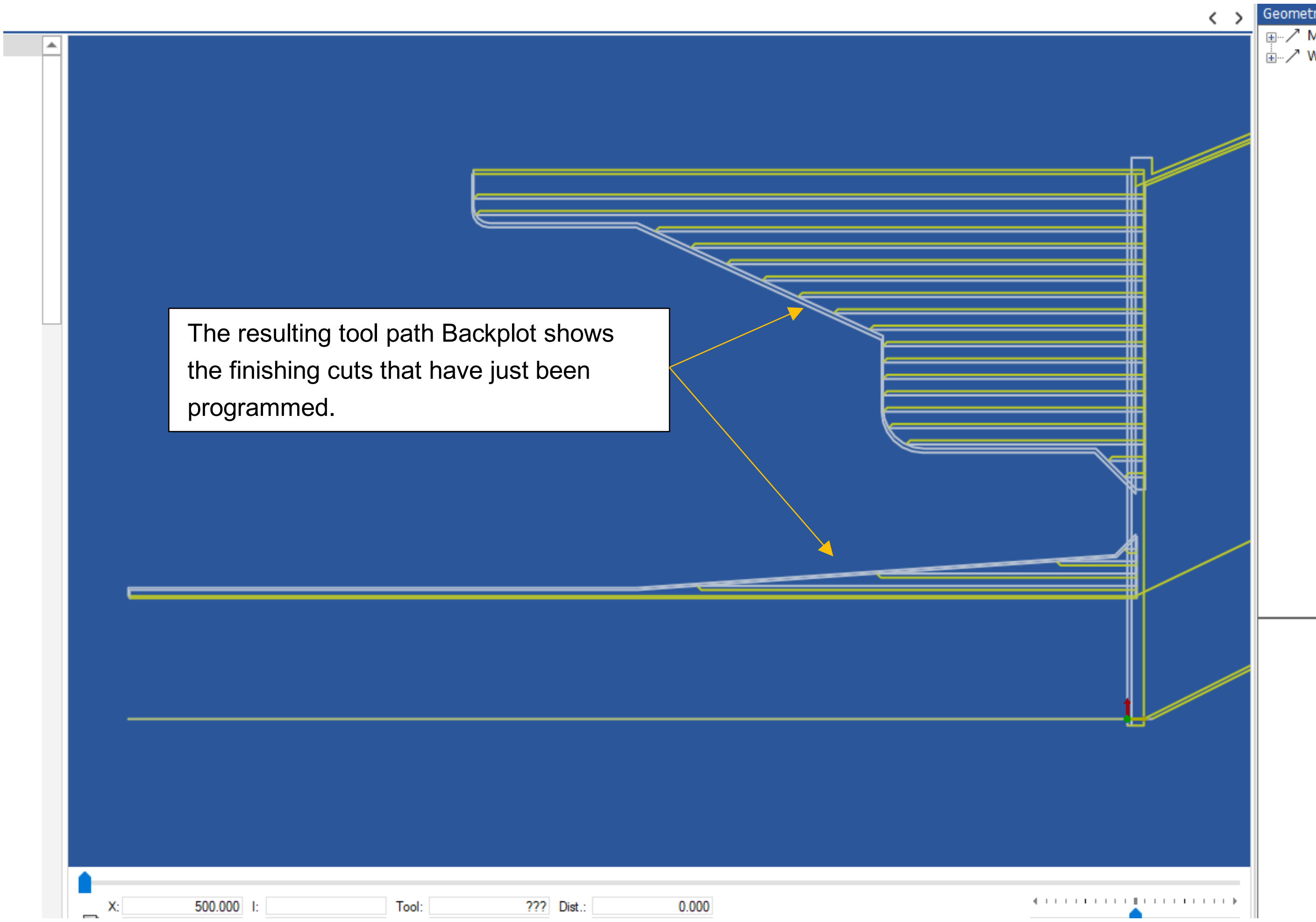
See the resulting Backplot below

CIMCO A/S
Copenhagen, Denmark
Tel: +45 45 85 60 50
Fax: +45 45 85 60 53
www.cimco.com
info@cimco.com



Repeat the process for the finishing cut to the internal profile

CIMCO A/S
Copenhagen, Denmark
Tel: +45 45 85 60 50
Fax: +45 45 85 60 53
www.cimco.com
info@cimco.com



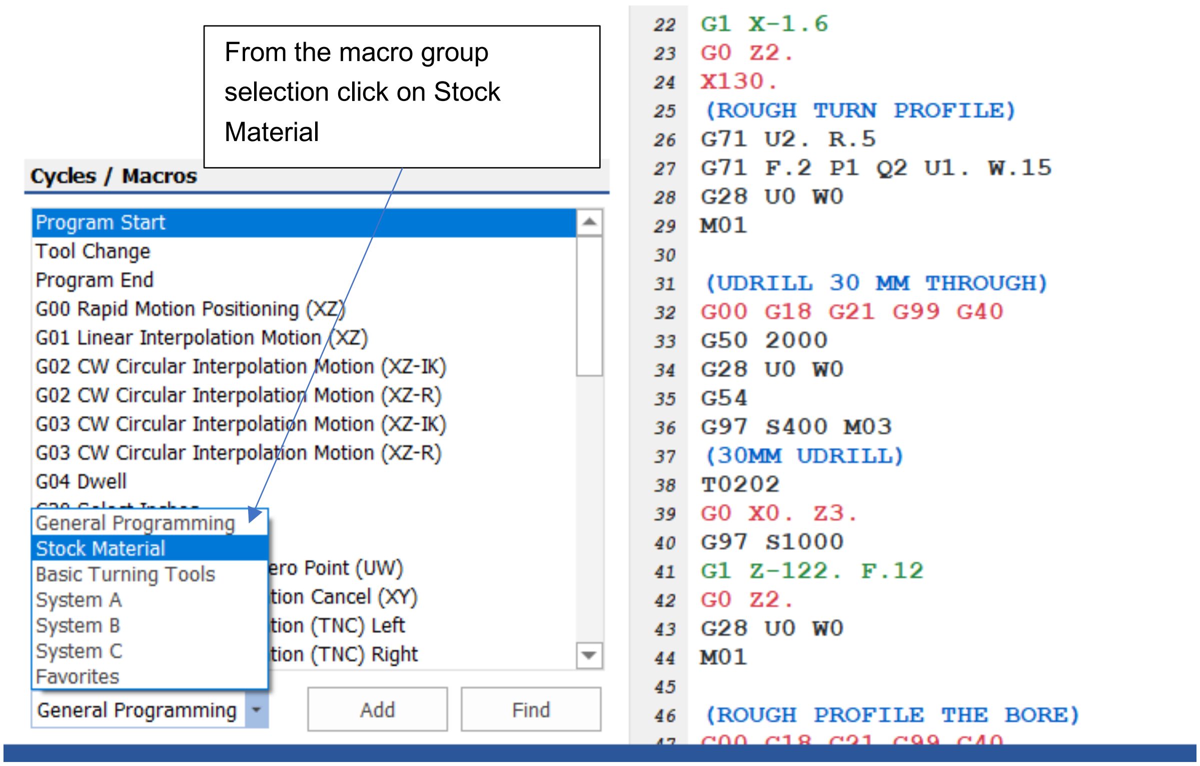
So far, we have been using the tool path Backplot to test out program. We can now add the details of Stock and Tools to achieve a 3D solid Backplot to give a true representation of the material being removed as it will happen on the CNC machine.

CIMCO A/S
Copenhagen, Denmark
Tel: +45 45 85 60 50
Fax: +45 45 85 60 53
www.cimco.com
info@cimco.com

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



Continued below

See Here above the Menu of NC-Assistant Groups.

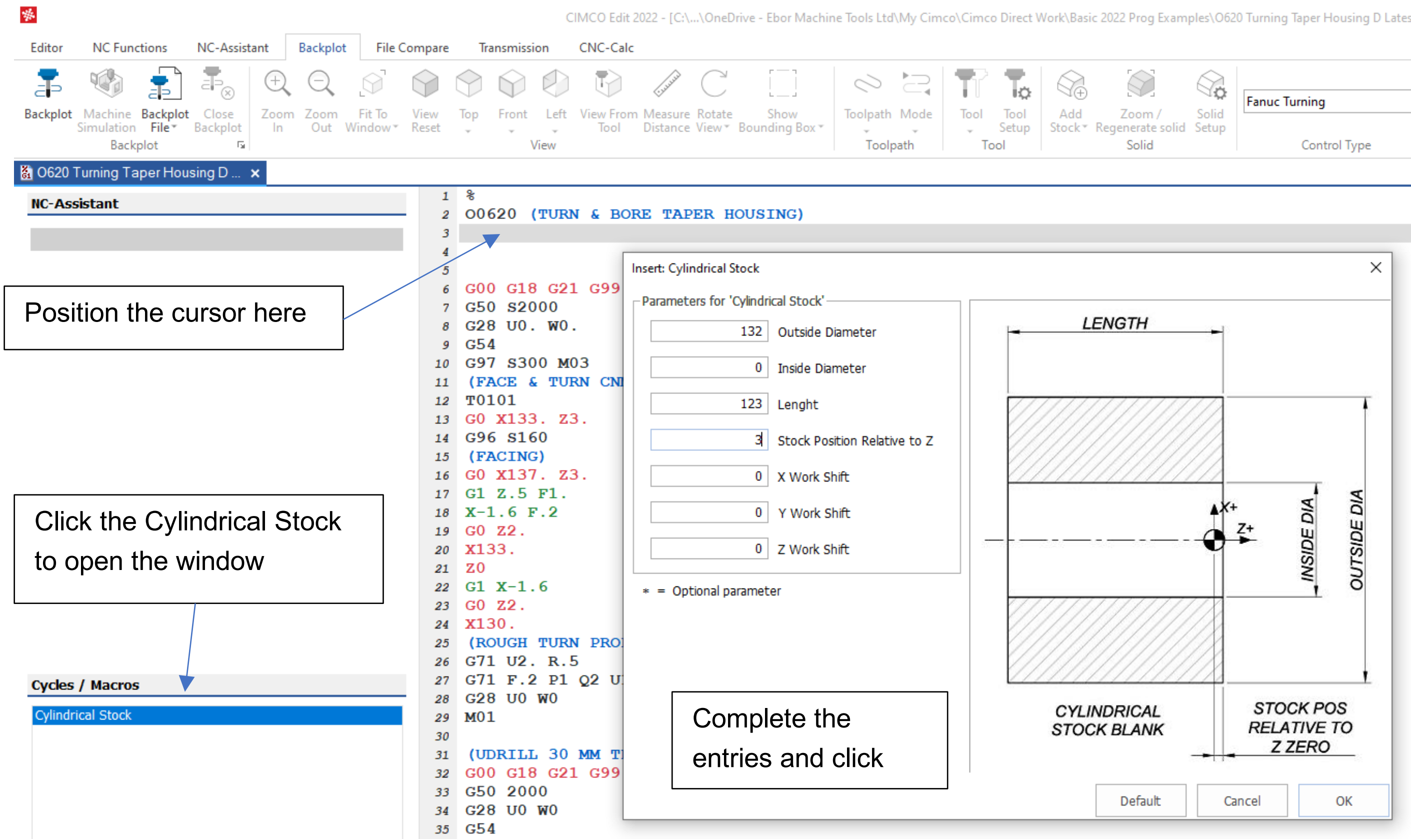
General Programming - for all the regular basic programming features.

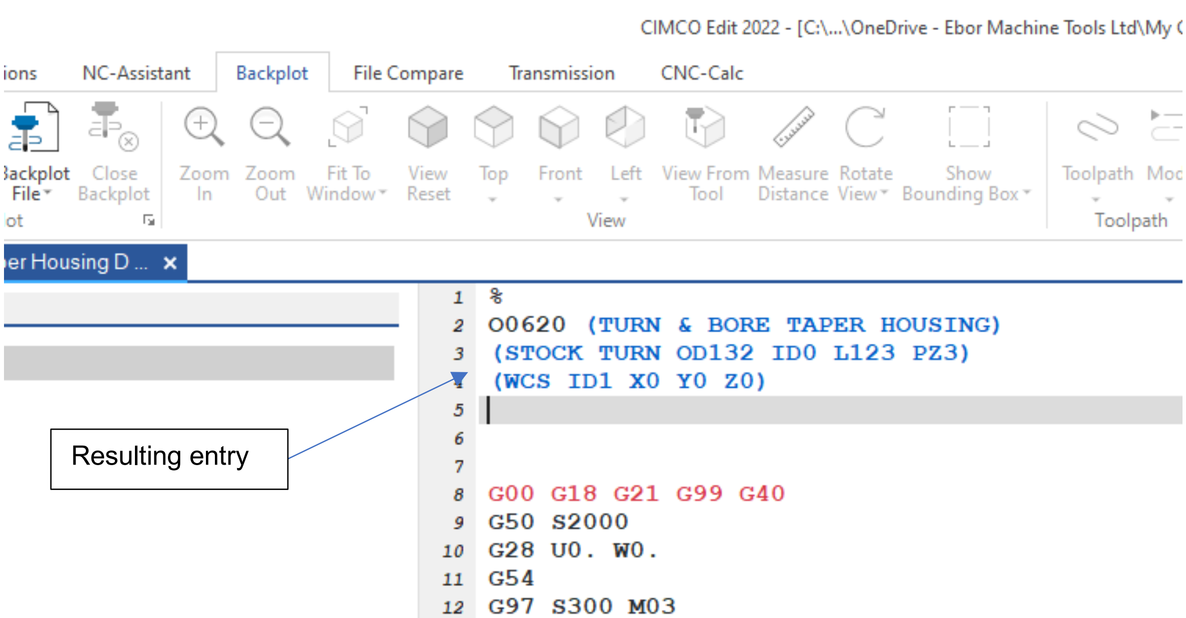
Stock Material – to establish the stock size of the part being machined

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.

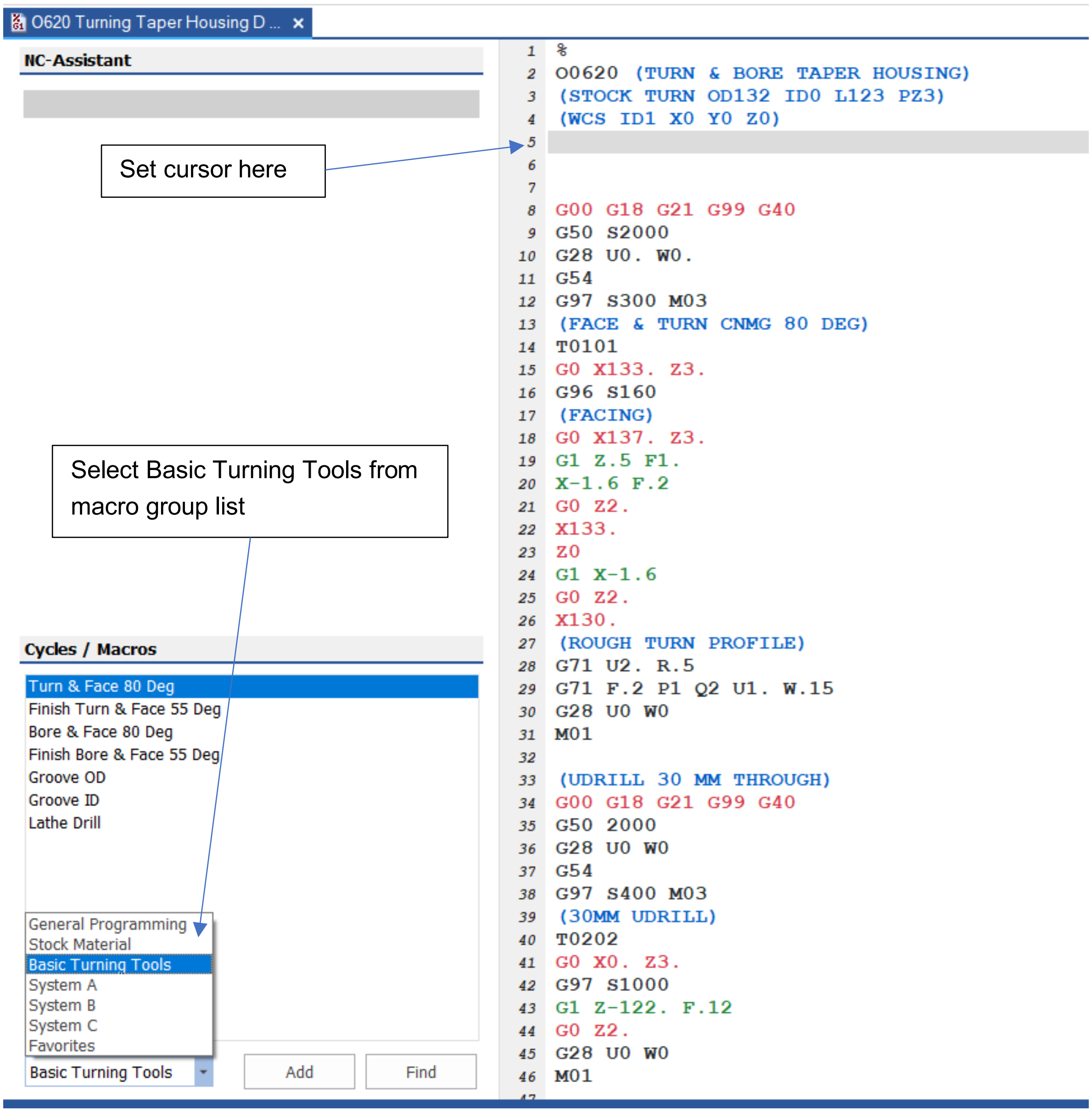
CIMCO A/S
Copenhagen, Denmark
Tel: +45 45 85 60 50
Fax: +45 45 85 60 53
www.cimco.com
info@cimco.com


Position the cursor here
Click the Cylindrical Stock
to open the window
Complete the
entries and click
Insert: Cylindrical Stock
Parameters for 'Cylindrical Stock'
132 Outside Diameter
0 Inside Diameter
123 Lenght
3 Stock Position Relative to Z
0 X Work Shift
0 Y Work Shift
0 Z Work Shift
* = Optional parameter
LENGTH
INSIDE DIA
OUTSIDE DIA
CYLINDRICAL STOCK BLANK
STOCK POS RELATIVE TO Z ZERO
Default
Cancel
OK
NC-Assistant
Cycles / Macros
Cylindrical Stock


Resulting entry
O0620 (TURN & BORE TAPER HOUSING)
(STOCK TURN OD132 ID0 L123 PZ3)
(WCS ID1 X0 Y0 Z0)
G00 G18 G21 G99 G40
G50 S2000
G28 U0. W0.
G54
G97 S300 M03

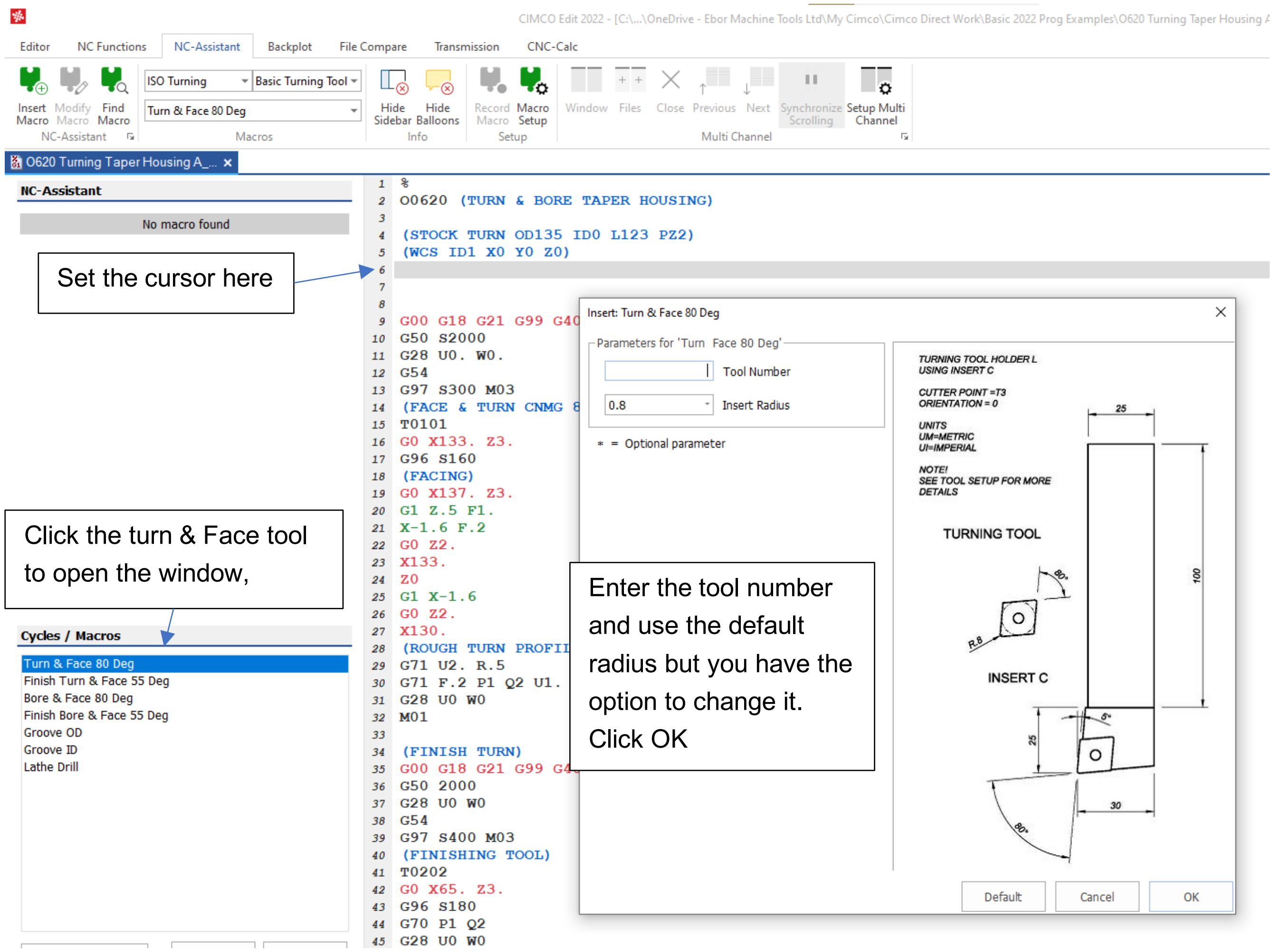
5.2 Tool setup

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.

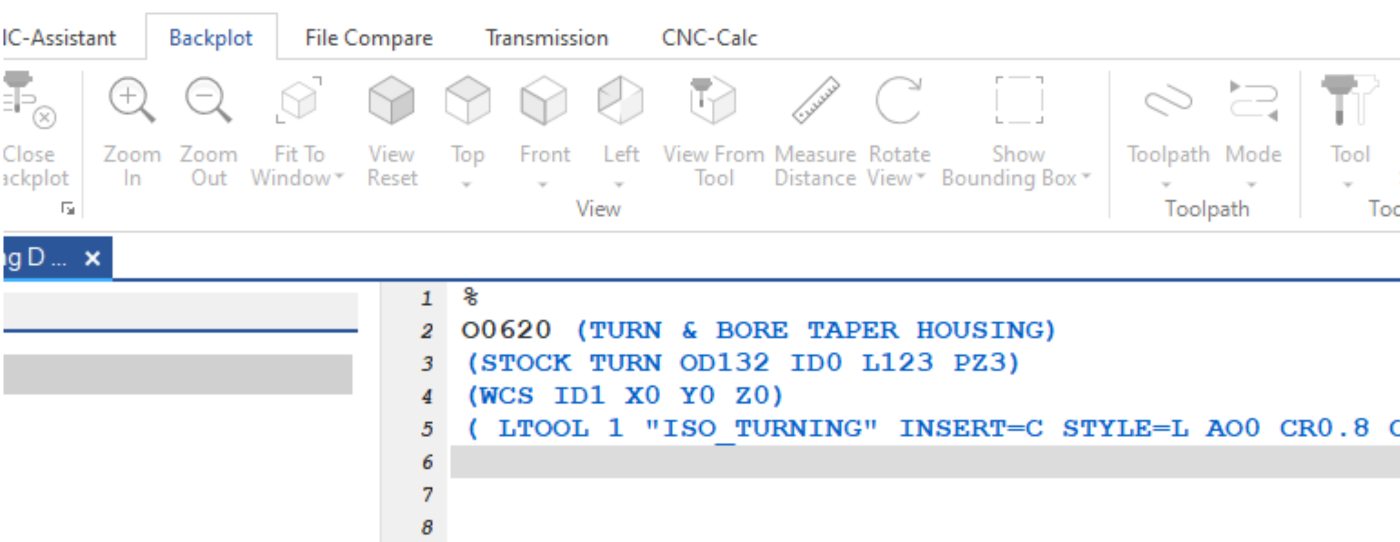


Tool number 1 is to turn and face the bar and an 80 degree rhomboid shaped insert will do both jobs

CIMCO A/S
Copenhagen, Denmark
Tel: +45 45 85 60 50
Fax: +45 45 85 60 53
www.cimco.com
info@cimco.com

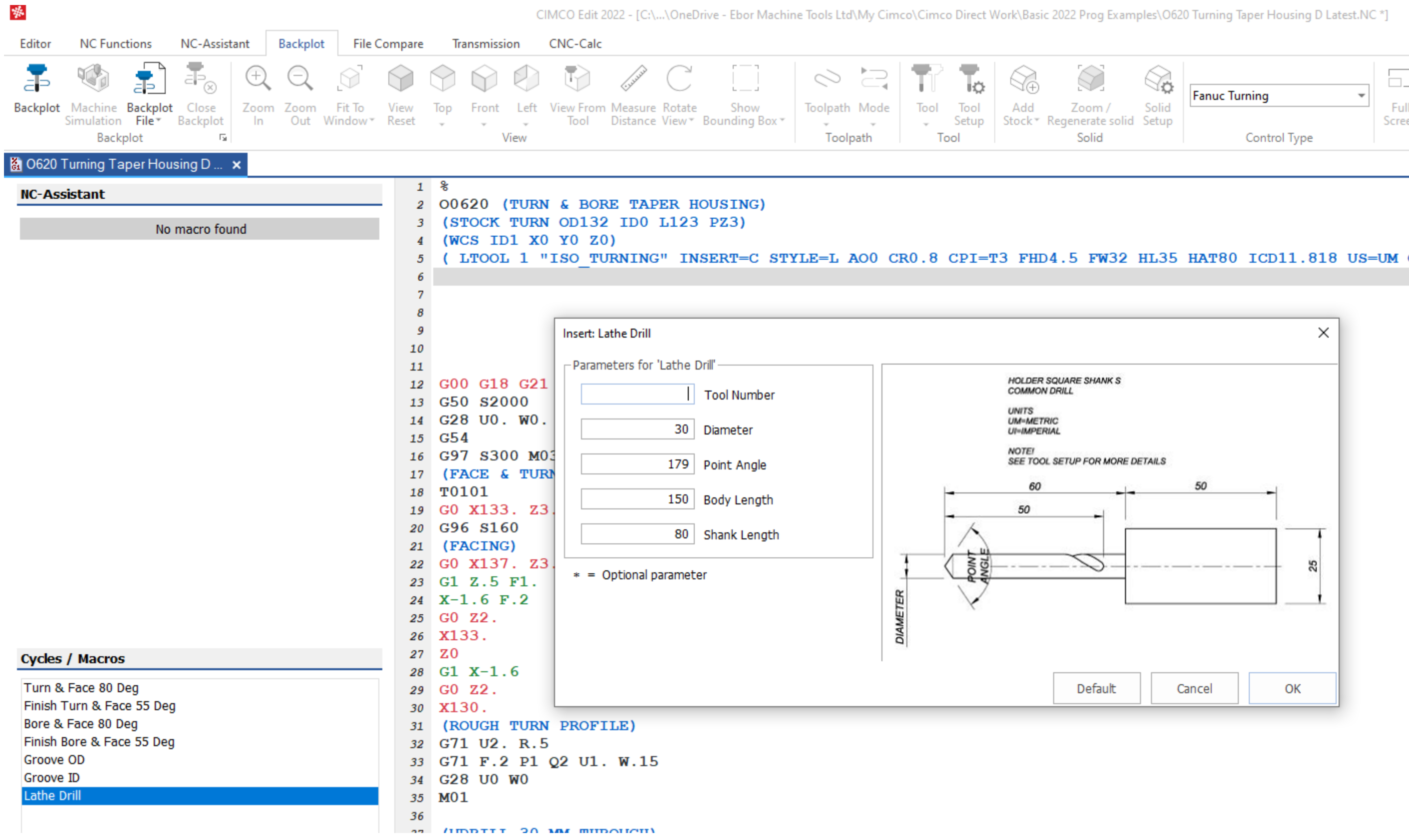


Resulting entry

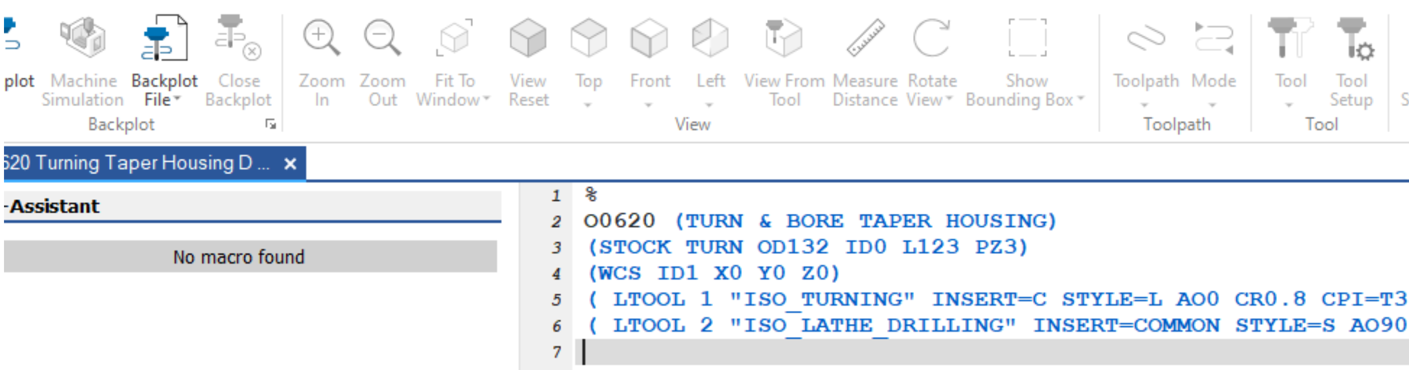


CIMCO A/S
Copenhagen, Denmark
Tel: +45 45 85 60 50
Fax: +45 45 85 60 53
www.cimco.com
info@cimco.com

Tool 2 to UDrill an 30 mm hole through the stock

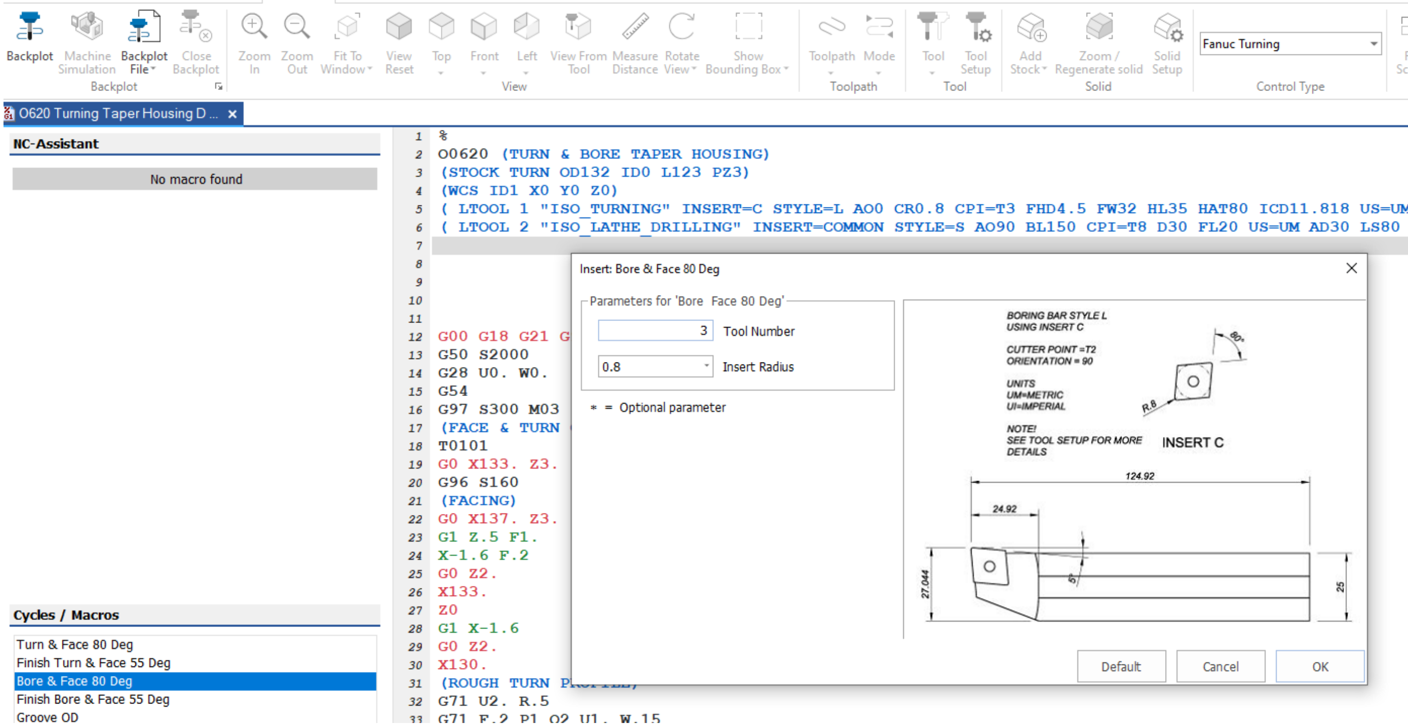


Resulting insertion

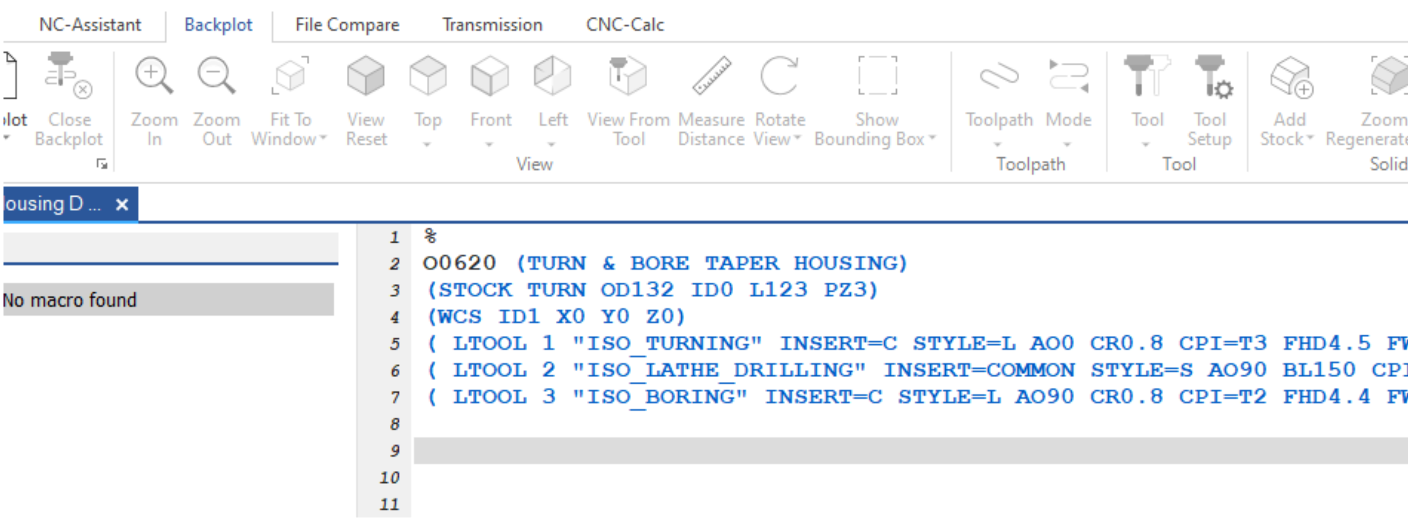


CIMCO A/S
Copenhagen, Denmark
Tel: +45 45 85 60 50
Fax: +45 45 85 60 53
www.cimco.com
info@cimco.com

Tool 3 Rough out the internal taper profile



Resulting insertion

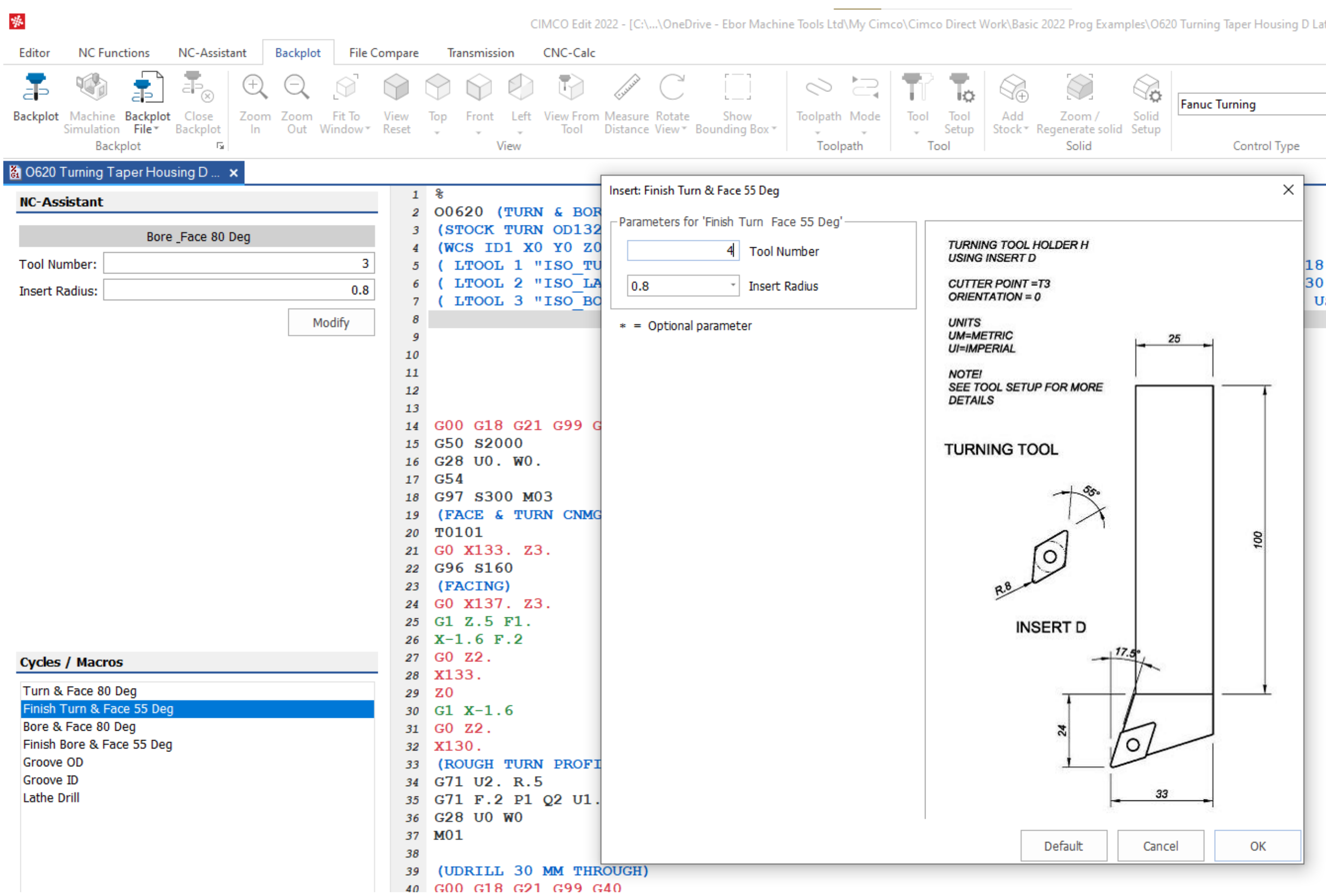


CIMCO A/S
Copenhagen, Denmark

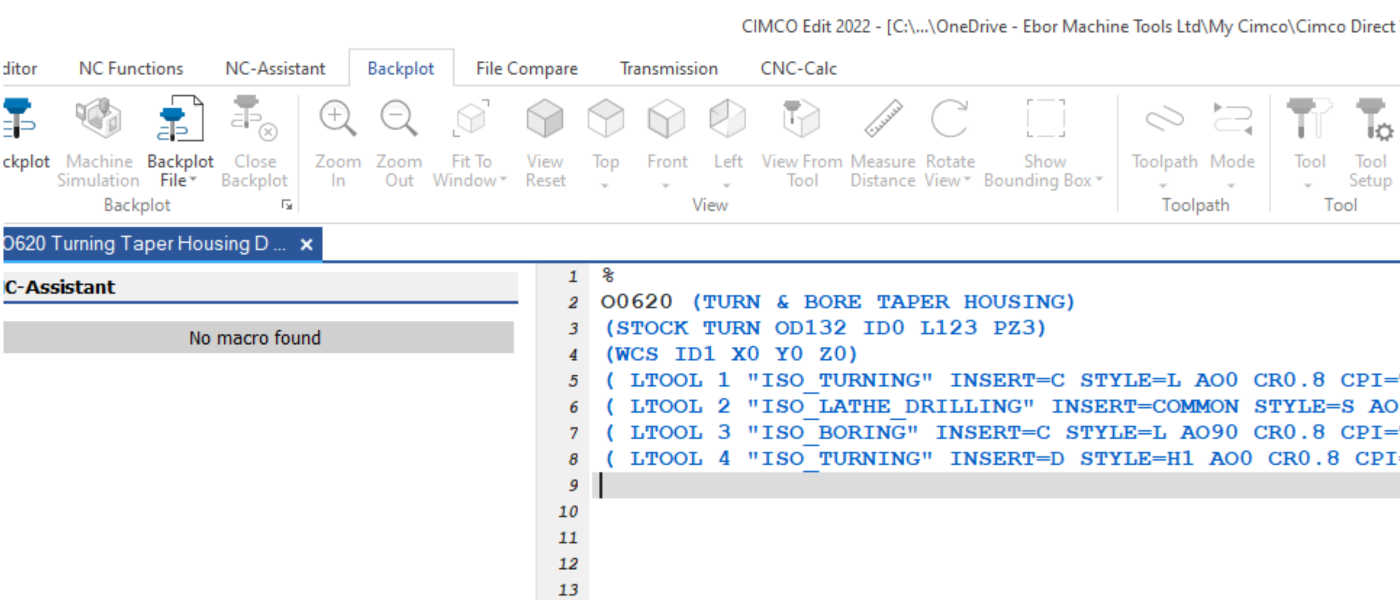
Tel: +45 45 85 60 50
Fax: +45 45 85 60 53

www.cimco.com
info@cimco.com

Tool 4 Finish turn the outside profile



Resulting insertion

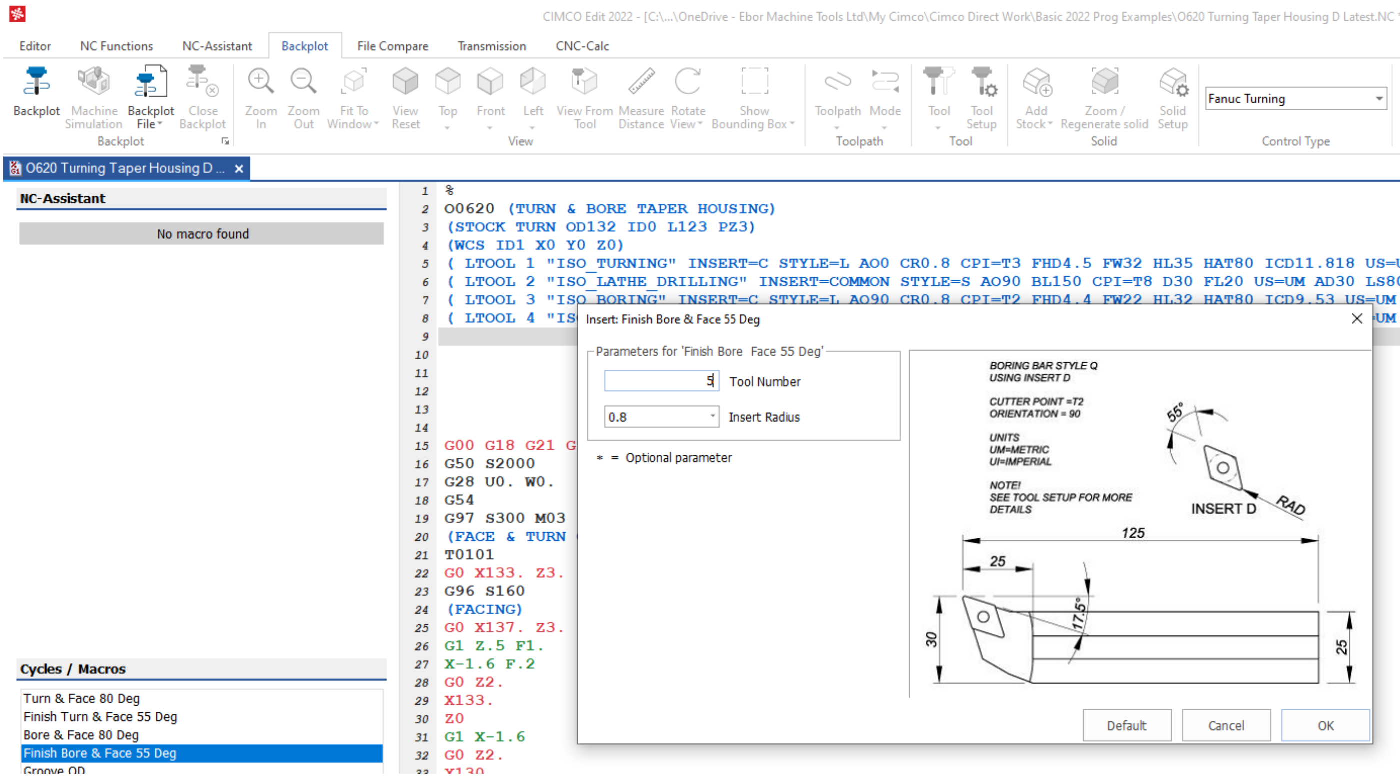


CIMCO A/S
Copenhagen, Denmark

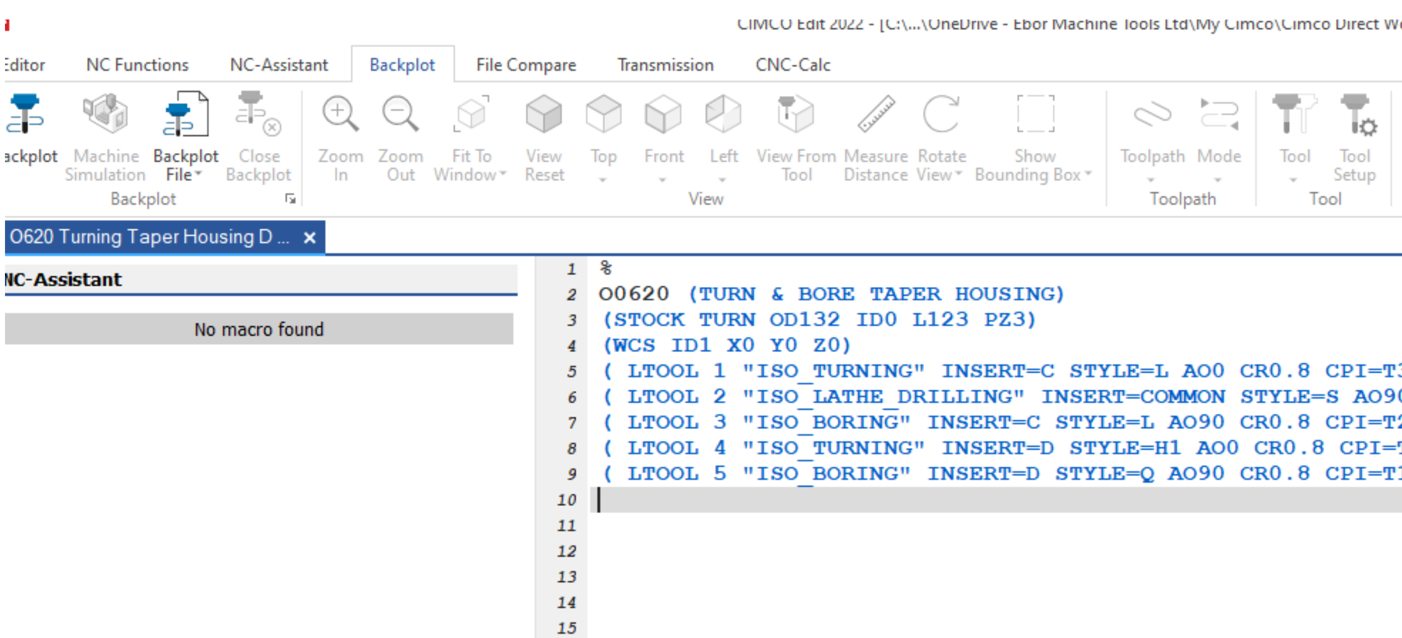
Tel: +45 45 85 60 50
Fax: +45 45 85 60 53

www.cimco.com
info@cimco.com

Tool 5 Finish bore the internal taper profile



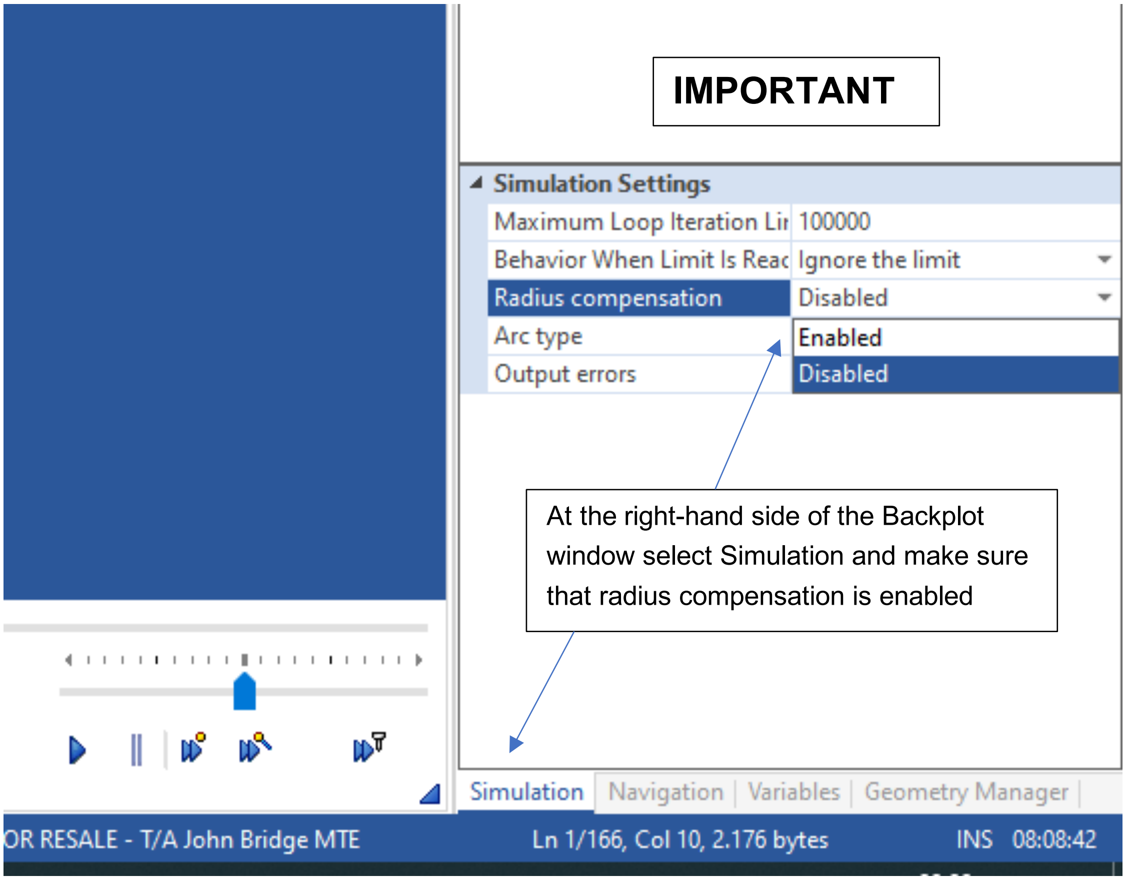
Resulting insertion



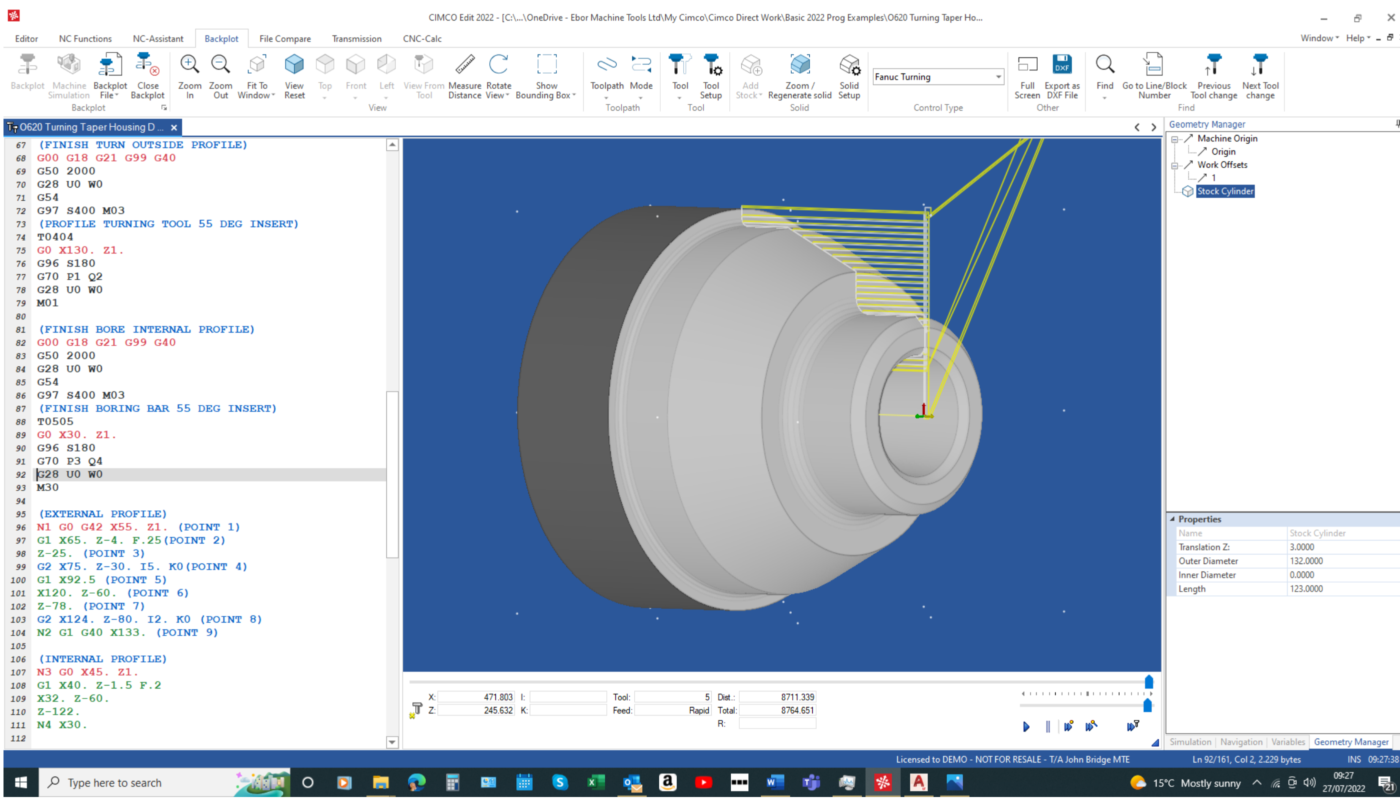
CIMCO A/S
Copenhagen, Denmark
Tel: +45 45 85 60 50
Fax: +45 45 85 60 53
www.cimco.com
info@cimco.com

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!!



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:

%

CIMCO A/S
Copenhagen, Denmark
Tel: +45 45 85 60 50
Fax: +45 45 85 60 53
www.cimco.com
info@cimco.com

```
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
```


CIMCO A/S
Copenhagen, Denmark
Tel: +45 45 85 60 50
Fax: +45 45 85 60 53
www.cimco.com
info@cimco.com

```
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)
```


CIMCO A/S
Copenhagen, Denmark
Tel: +45 45 85 60 50
Fax: +45 45 85 60 53
www.cimco.com
info@cimco.com

```
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
```


CIMCO A/S
Copenhagen, Denmark
Tel: +45 45 85 60 50
Fax: +45 45 85 60 53
www.cimco.com
info@cimco.com

```
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(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)
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.
```


CIMCO A/S
Copenhagen, Denmark
Tel: +45 45 85 60 50
Fax: +45 45 85 60 53
www.cimco.com
info@cimco.com

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)

CIMCO A/S
Copenhagen, Denmark
Tel: +45 45 85 60 50
Fax: +45 45 85 60 53
www.cimco.com
info@cimco.com

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