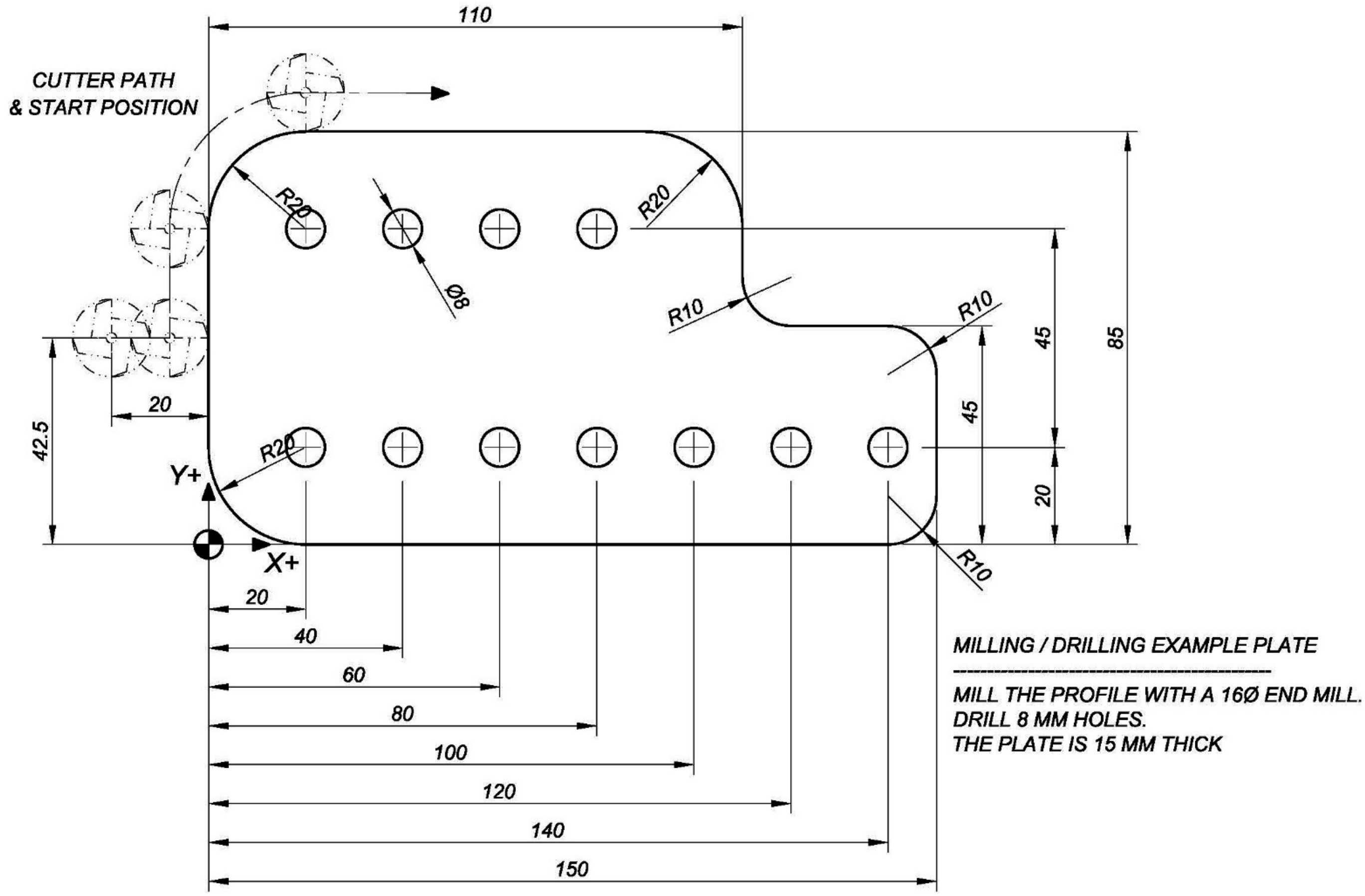
# Basic ISO Programming Exercise 1
# Milling

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

**BASIC CNC ISO PROGRAMMING TO MILL PROFILE & DRILL HOLES USING CIMCO EDITOR AND GRAPHIC BACKPLOT TO TEST**

See below a drawing of a plate with 8 mm holes. We will work through the programming using ISO G code to prepare a program to machine the outside profile and drill the holes.

ISO G code is used by many CNC control manufactures and the main groups of G codes for move commands, units 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 will consider profile milling 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:

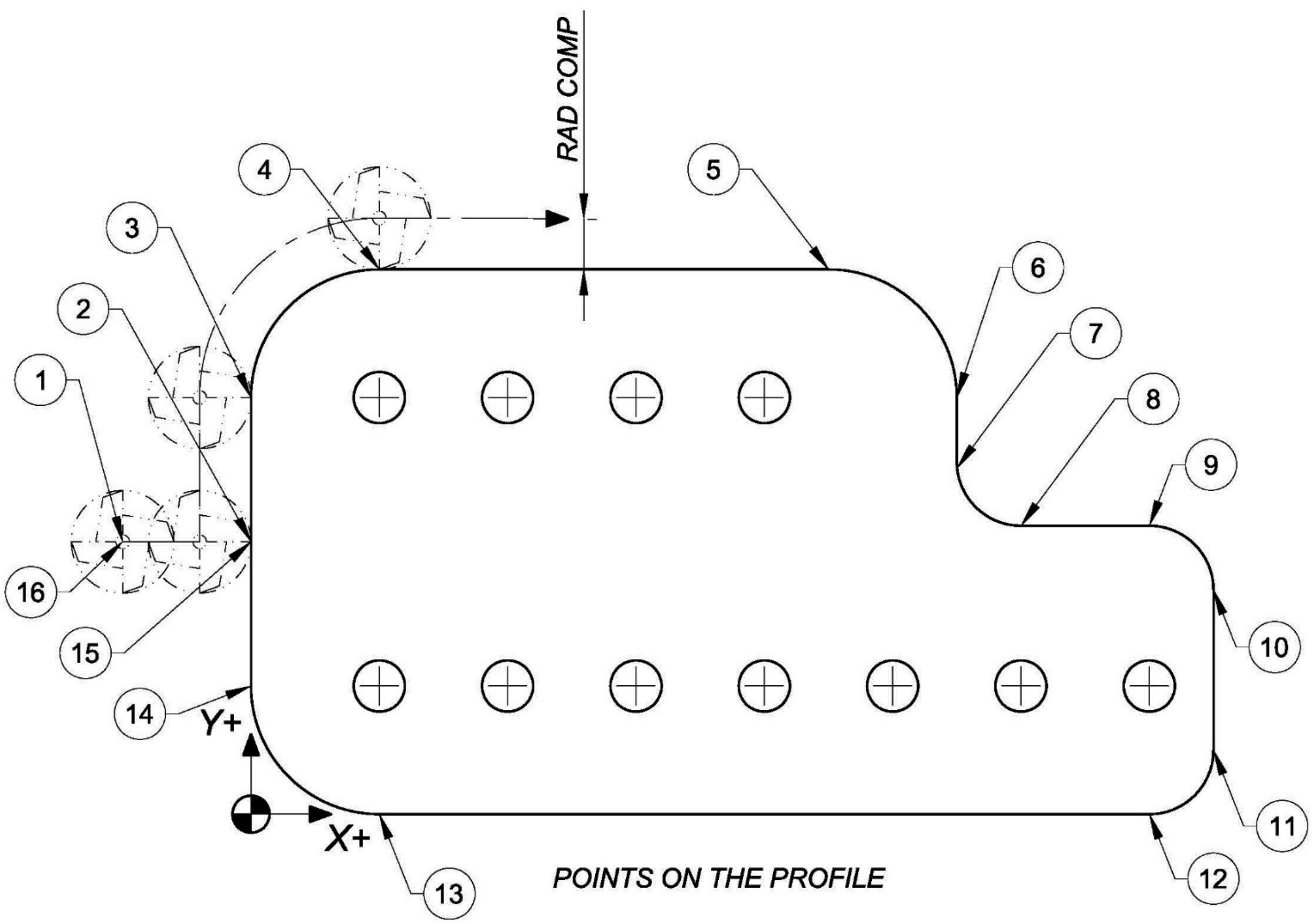
1. G00 – Straight line moves at rapid speed (On some machines this move is made in a vector line and on others a 45-degree move is followed by a single axis move to achieve the final programmed position). This is a modal command.

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 drawing below that has the profile broken down to represent the points on the profile where elements start and finish. The programming X and Y zero point is shown by the checkered circle so all dimensions will fall in the X plus, Y plus, quadrant.

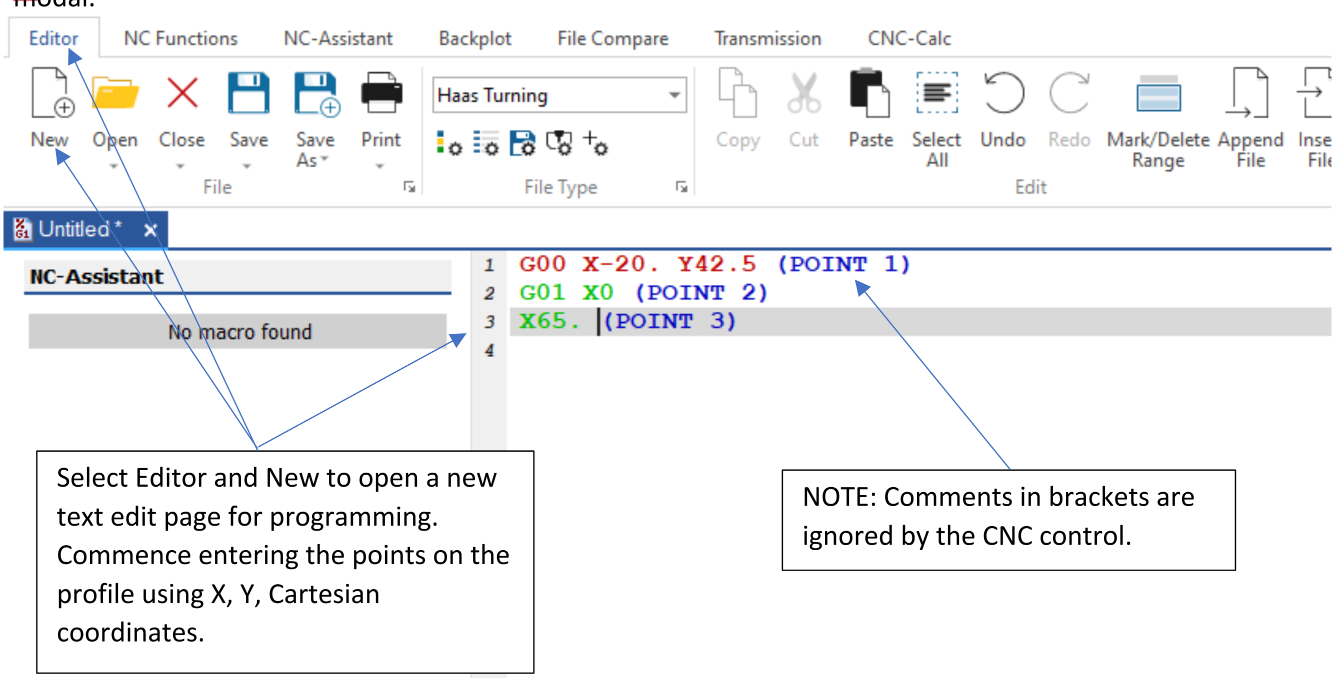


**DEFINING THE CONTOUR**

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

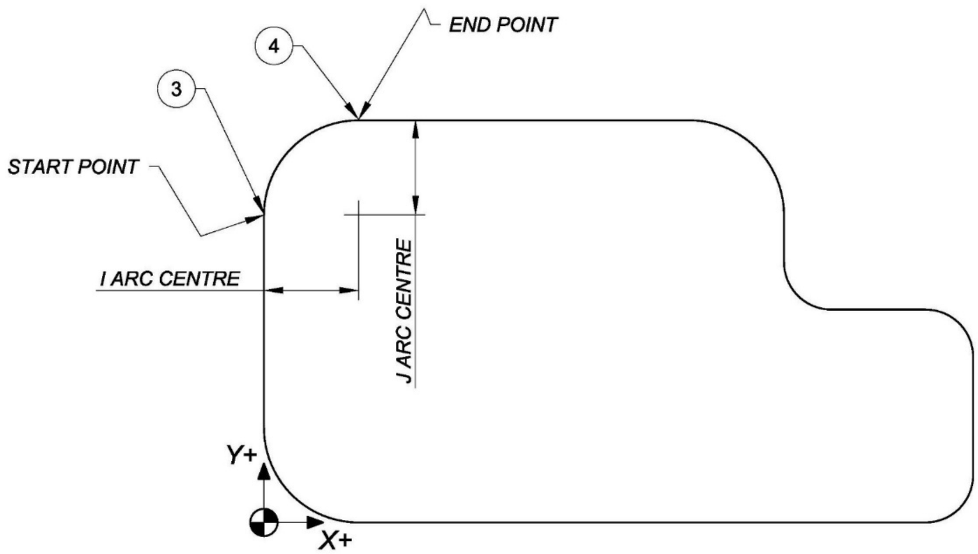
**LINEAR INTERPOLATION**

The linear interpolation points are very easy to enter. If we have only one axis command X or Y on a line, then a move in a straight line in that axis will take place. If we have and X and a Y on the same line with a G1 or G01 prefix, 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.



**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

G02 X20. Y85.

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 J for Y. So, from the start point the I and J are entered as incremental coordinates as below.

G02 X20. Y85. I20. J0

The CNC Control now has all the information to make this circular move. See that the I arc centre is a positive dimension. Had we been going in the G03 direction then the I are centre would have been I-20. Notice that the J arc centre is 0 (zero) as J is the coordinate for the arc center from the start point on the Y axis.

**Note!!** With signed plus or minus I & J arc centre designation it is possible to program a full 360-degree circle. It is also possible to use radius designation instead of I & J but then the maximin arc possible is 180 degrees. Some CNC controls prefer to use only I & J 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 & J from start point. When programming a specific CNC machine, the CNC control programming manual may need to be referred to.

When the profile entries are complete then test using Backplot. This test will show up any inaccuracies in the profile entries. Add a feed rate on the first G01 line. Feed rate is modal, as stated before and all other interpolated blocks will be move at the feed rate last designated.

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 G02 block if you were to enter X Y 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.

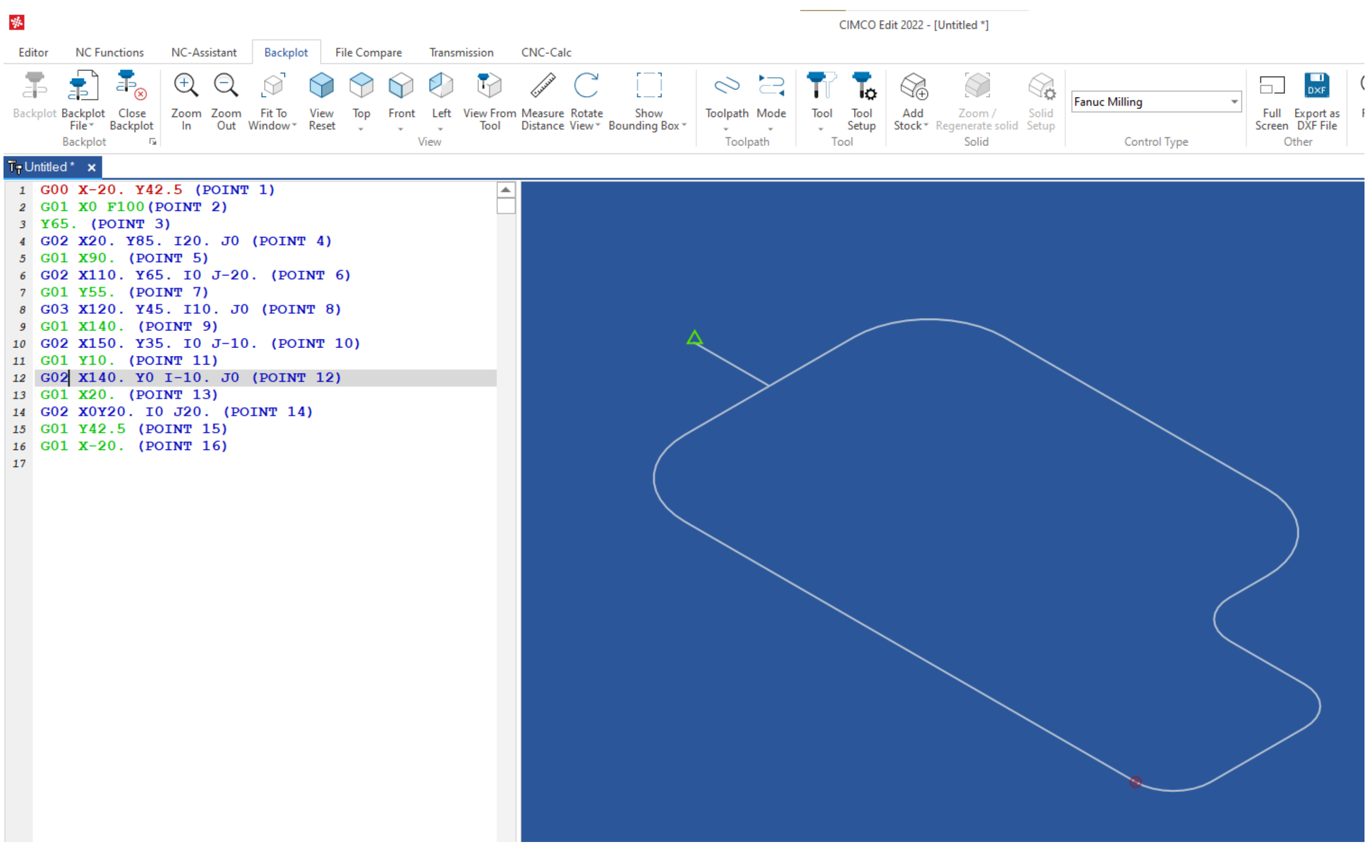
Programming good practice is to layout a block in order. This will help you see the problems when snagging a program for bugs. e.g.

N155 G01 X150. Y210.25 F200 M03 S1000

If you use block numbers put them first, G code command next in 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.

When you have finished the block entry test with the Backplot facility in the Editor and you should have a good toolpath plot as below when the all the blocks are correct.




See the correct profile syntax below to check your profile.

G00 X-20. Y42.5 (POINT 1)

G01 X0 F100 (POINT 2)

Y65. (POINT 3)

G02 X20. Y85. I20. J0 (POINT 4)

G01 X90. (POINT 5)

G02 X110. Y65. I0 J-20. (POINT 6)

G01 Y55. (POINT 7)

G03 X120. Y45. I10. J0 (POINT 8)

G01 X140. (POINT 9)

G02 X150. Y35. I0 J-10. (POINT 10)

G01 Y10. (POINT 11)

G02 X140. Y0 I-10. J0 (POINT 12)

G01 X20. (POINT 13)

G02 X0Y20. I0 J20. (POINT 14)

G01 Y42.5 (POINT 15)

G01 X-20. (POINT 16)

**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. See suggested general header below

G00 G90 G49 G98 ; Safety blocks to set start conditions of modal commands

G00 set rapid traverse

G90 set absolute coordinates

G49 cancel tool length offsets

G98 set feed in mm/min (Haas)

G28 W0 ;Send Z axis up to home. W is the incremental command for Z

G28 return to home position

W0 incremental move in Z direction (XYZ Absolute, UVW Incremental) Fanuc, Haas

G54 X0 Y0 ; move to programming X0, Y0, as set in G54 work offset table

G54 call the work offset that has been set up to establish the programming zero of the part on the machine table.

(16 MM CARBIDE END MILL) ; tool description comment

T1 M6 ; select tool number 1 with T1, put the tool in the spindle with M6

T1 or T01 selects the tool

M6 or M06 put the tool in the spindle

G97 S1000 M03 ;start the spindle to 1000 rpm in the forward direction with M3

S1000 is commanding 1000 rpm, M3 or M03 is starting the spindle forward direction

G43 H1 Z50. ; take up the tool length H1 from tool offset table with G43, position the tool 50 mm above Z0

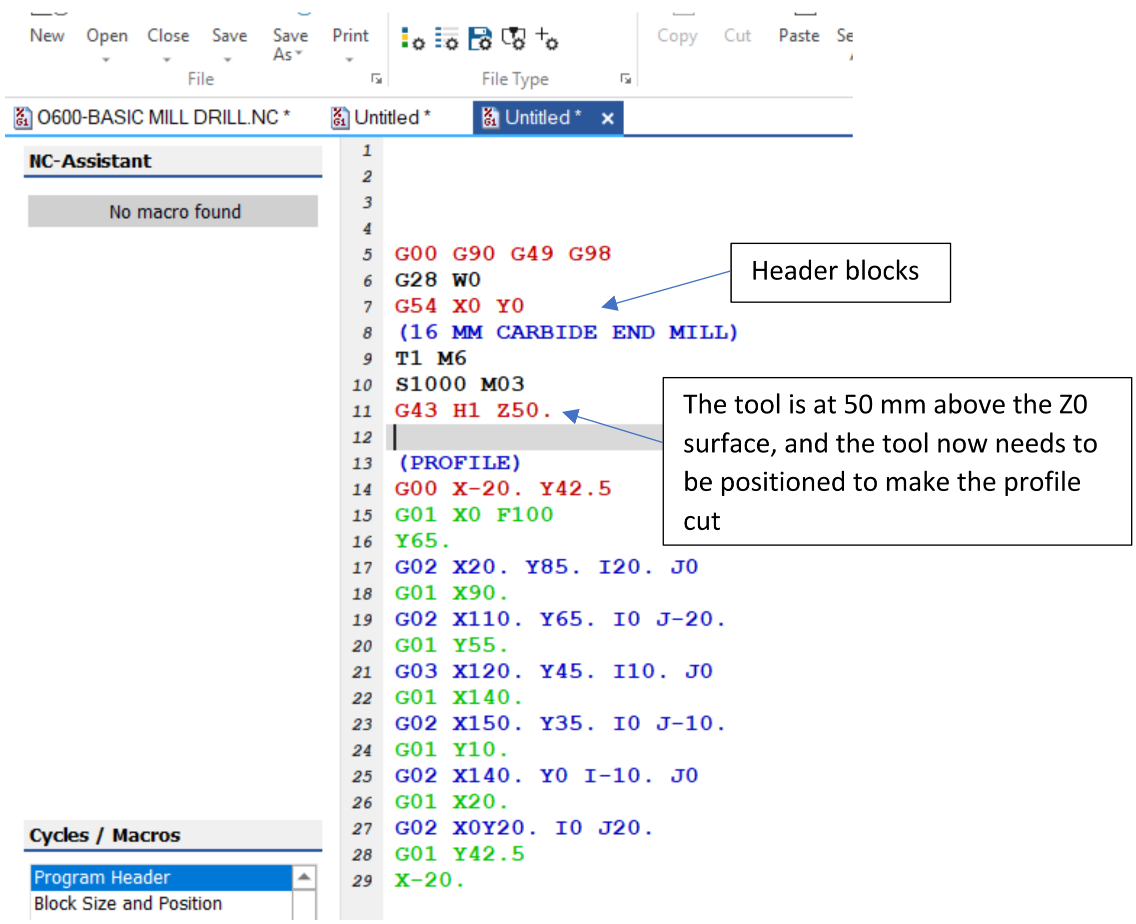
G43 is activating the tool length offset, H1 or H01 is the tool length seat in the table, Z50. Move the tool to 50 mm above the Z0 position.

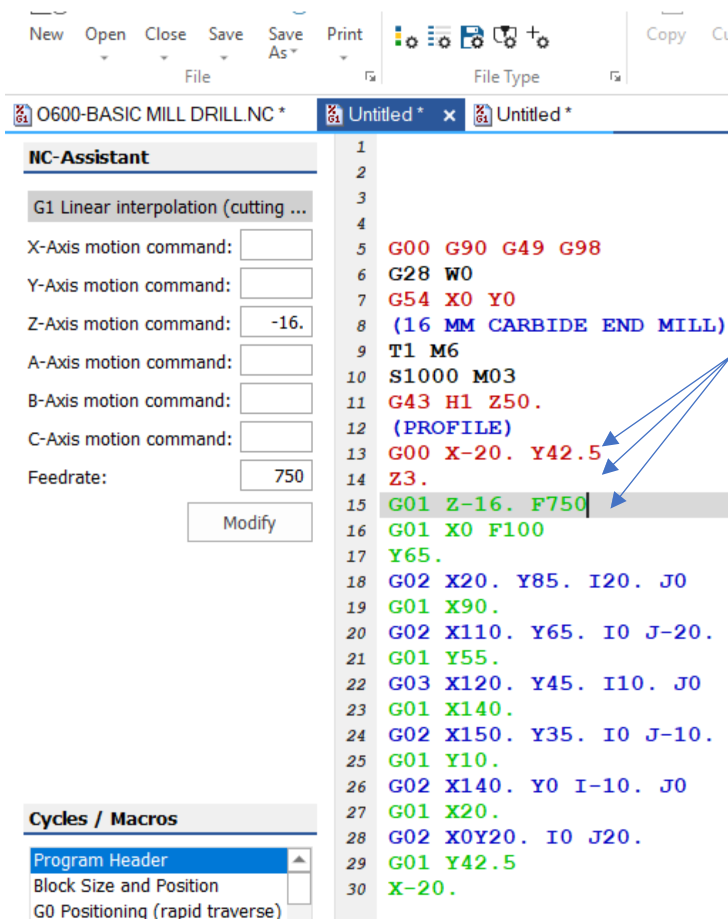
G00 X-30. Y42.5 ; position the tool over the 1st hole coordinated.

G00 is repeating the rapid traverse command as it is already modal from the 1st block.

X-30. Y42.5 move to the safe start position before traveling down in the Z axis to avoid collisions.

(PROFILE) ; comment to mark the commencement of the profile



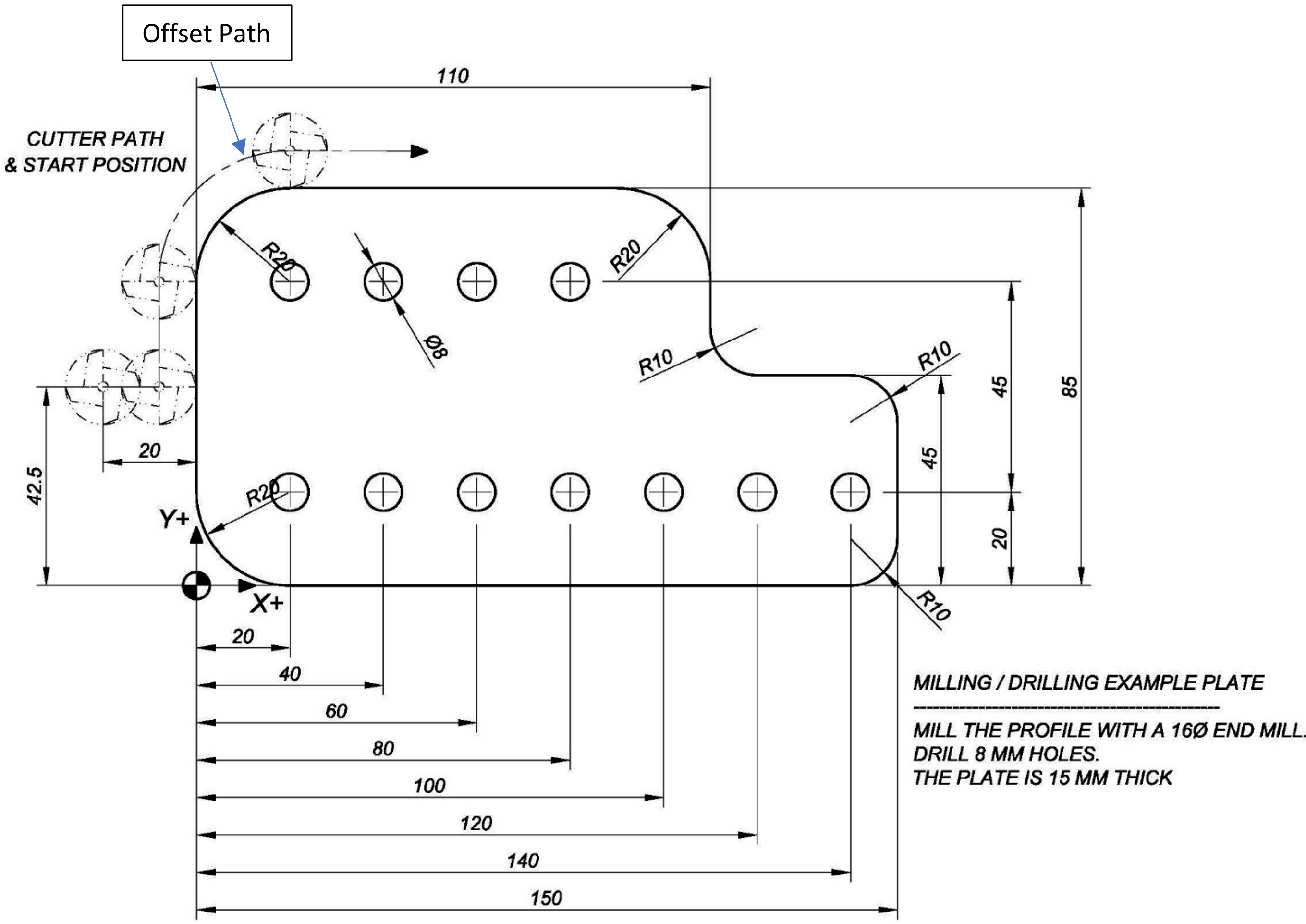



We choose to position the X, Y axes before going down in Z so that we do not collide with the work stock.

Move down in Z to 3 mm above the surface in rapid traverse and finally to Z-16. in G01 at a high feed rate. This positioning in a cautious manner is recommended.

**CUTTER RADIUS COMPENSATION**

From the drawing it can be seen that the tool path must be offset to make the finished part to the correct size.




G40 - Cancel Radius Offset Compensation

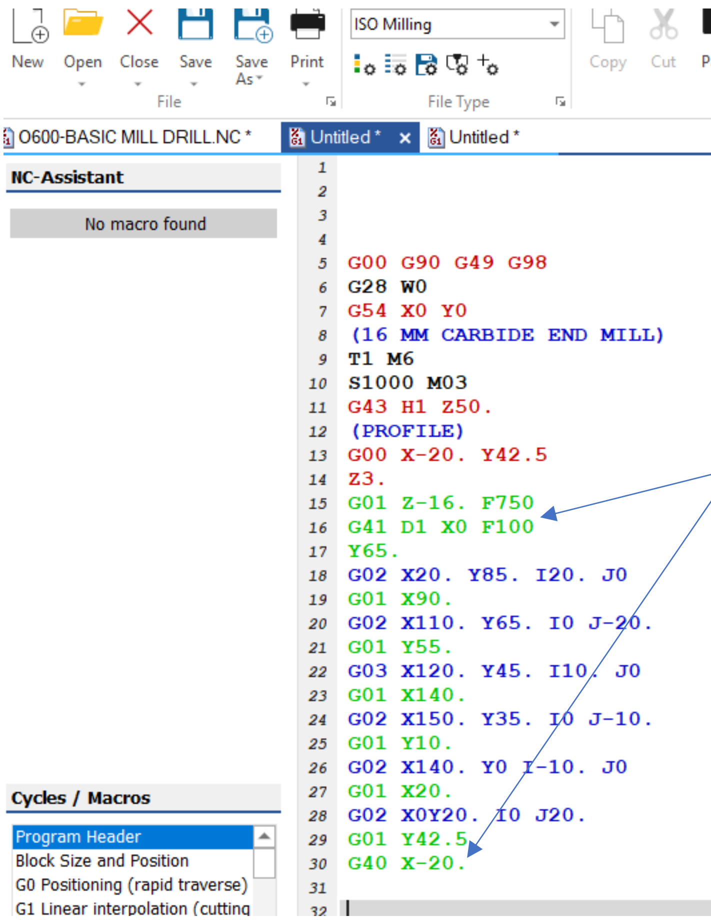
G41 - Activate a Radius Offset path (Radius Compensation is the term used) to the left in the direction of travel.

G42 - Activate a Radius Offset path 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 cutter Radius is entered. On the block where the G code is entered, we must add the D radius offset. When the G code and D radius are read by the CNC control the offset path will be activated within the move.

From the drawing we can see the direction of travel around the periphery of the part is counterclockwise. Therefore, we will offset the path to the left in the direction of travel by using G41

See below:




Place the G41 D1 in the block where the tool approaches the profile.

Cancel the offset path with G40 in the block where the tool leaves the profile.

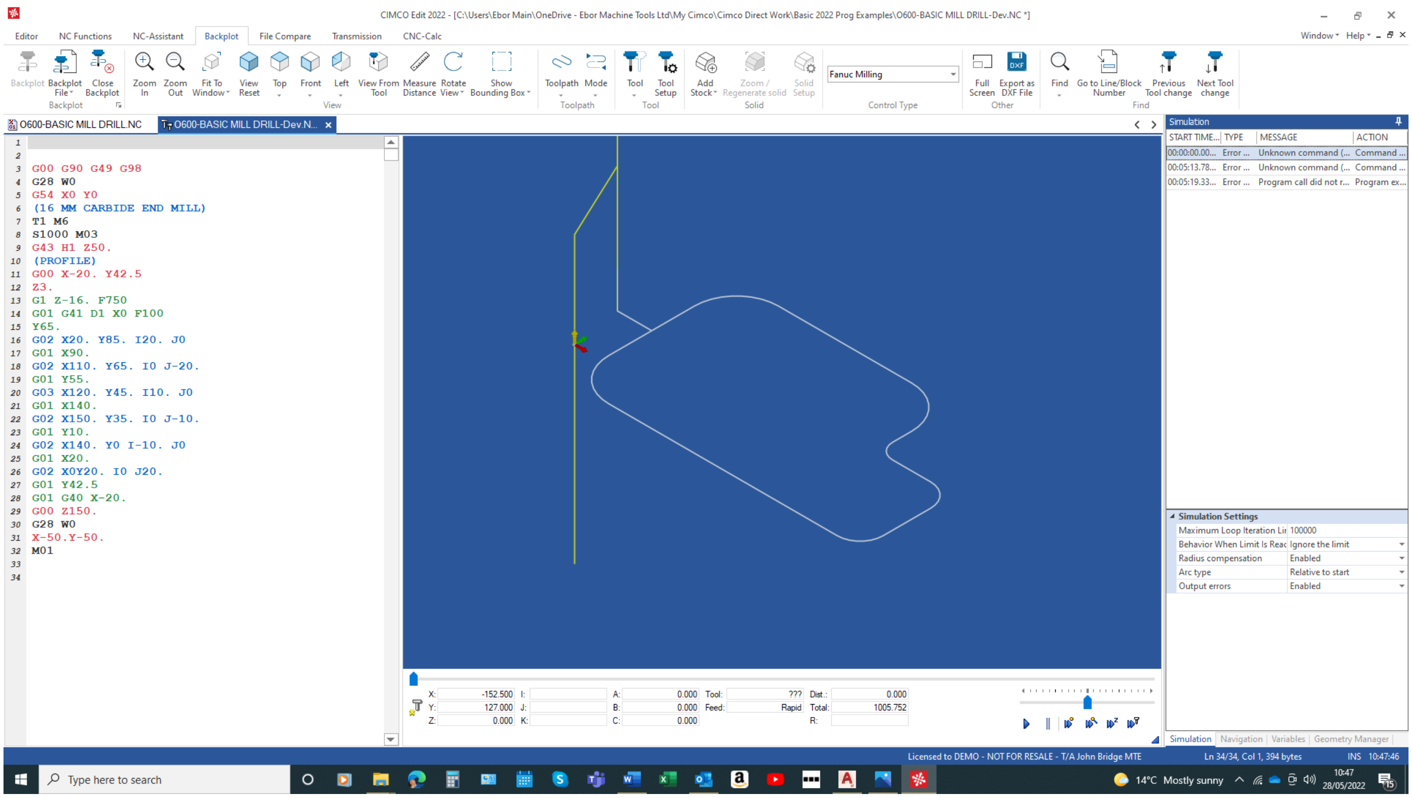
See inserted G41 to apply Tool Radius Compensation and D1 to apply the radius from the CNC table.

We have selected G41 as we are to the left of the profiles in the direction of travel.

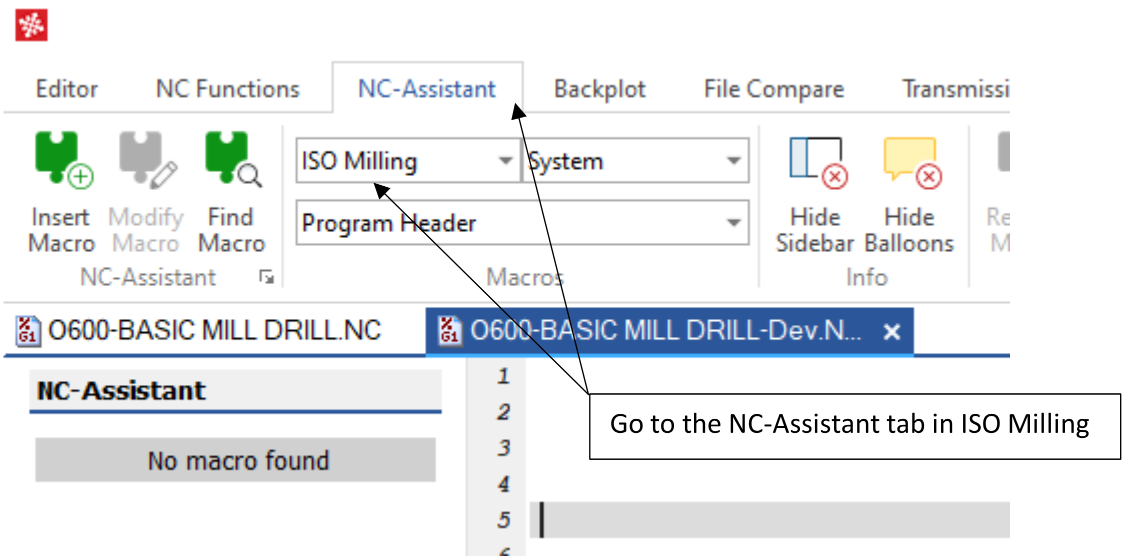
These commands should be on the block where the tool is approaching the profile and the move will be a greater distance than the cutter radius. Generally, on CNC controls this will be a G01 linear move

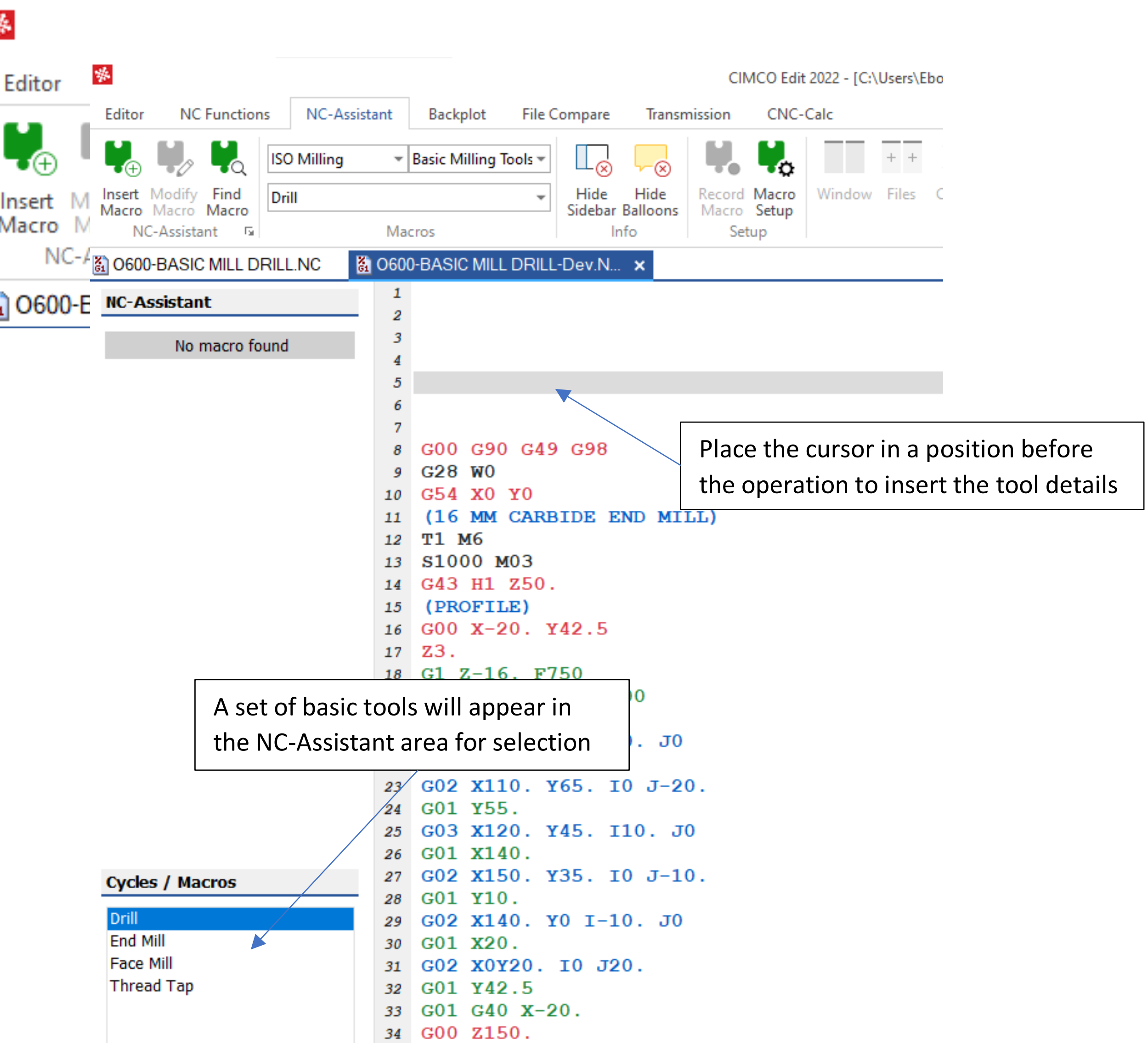
Test with Backplot and see below:

Backplot has tracked the profile as programmed and used a random tool as no tool has been setup then we have a basic tool centre path Backplot.

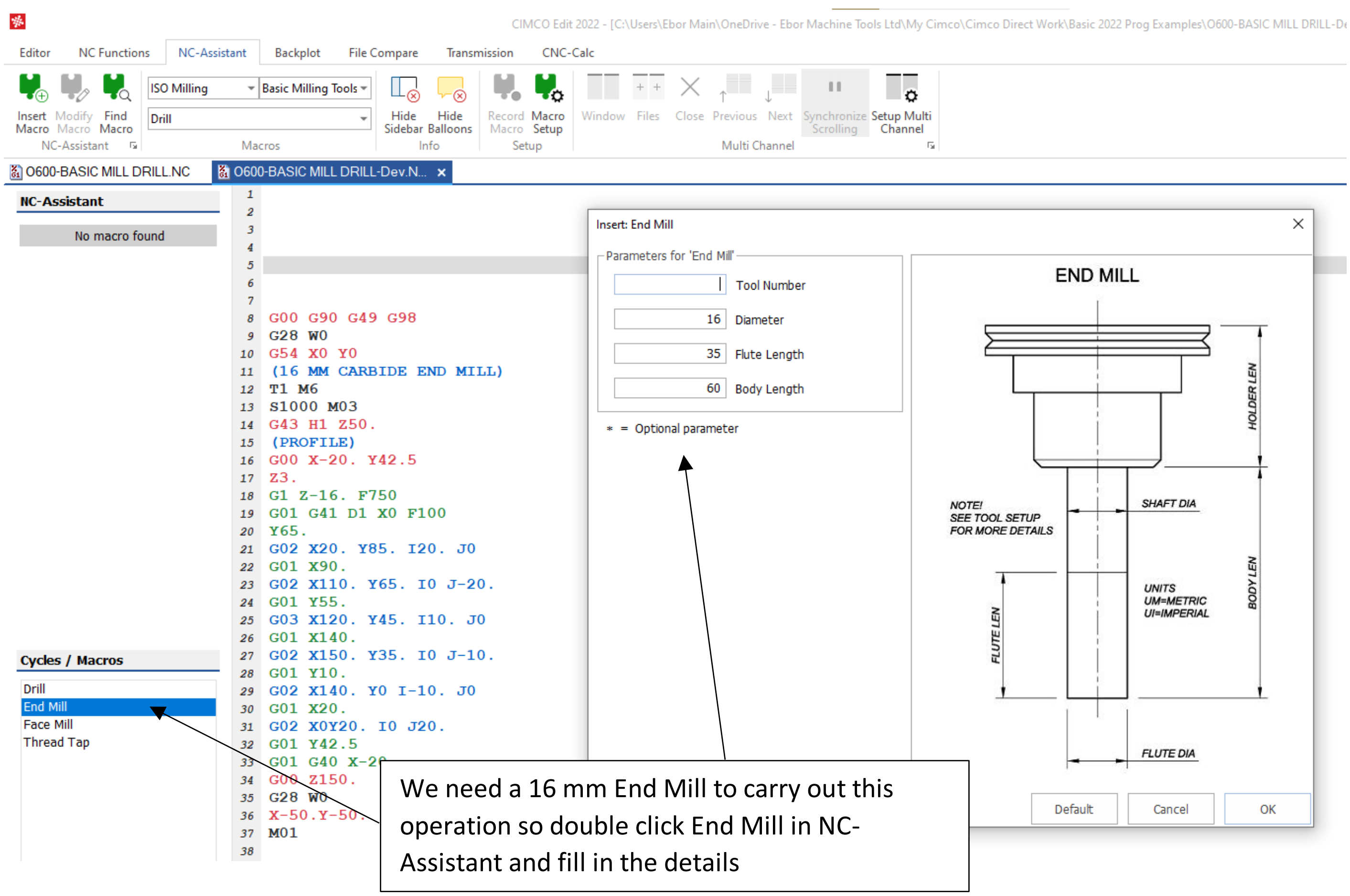


We must now setup a tool to enable the editor to give us a true finished profile. There is separate tutorial to explain in detail how to use the Tool Setup within the Backplot facility. However, there is a basic set of tools that can be configured within NC-Assistant. See below how to find and set up a tool.

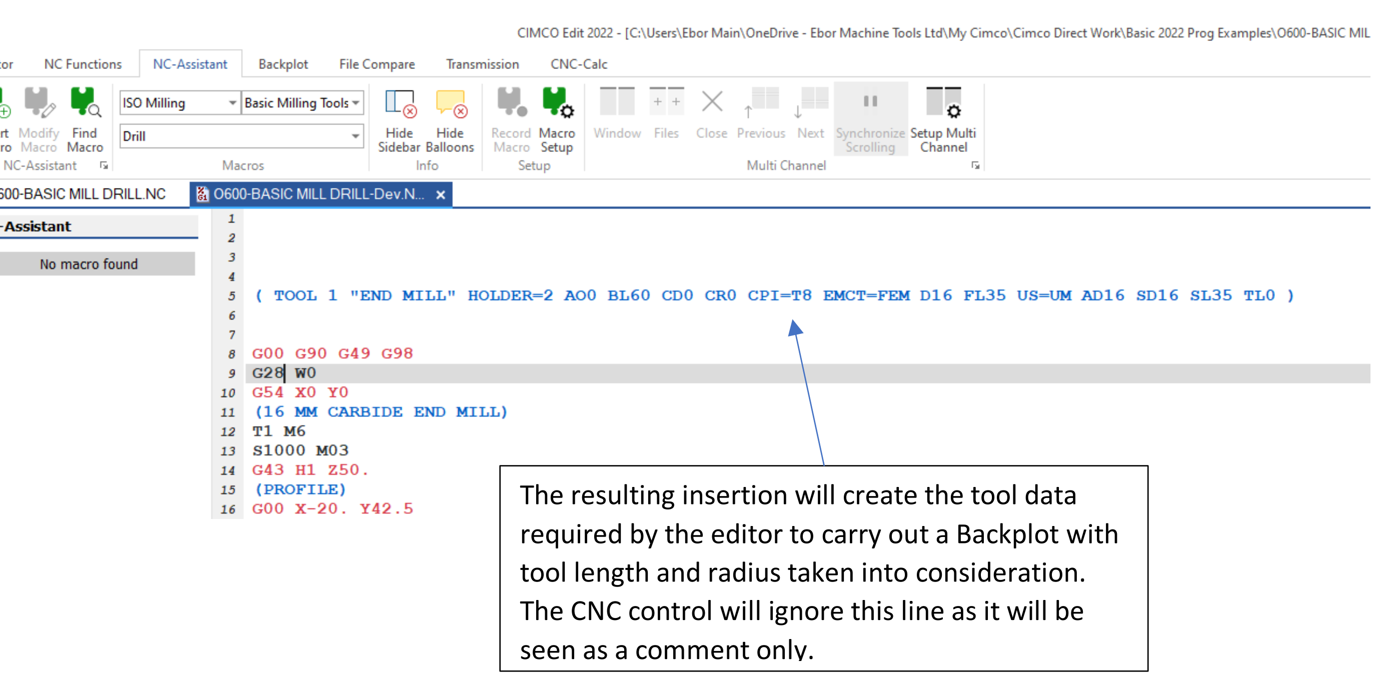


We need a 16 mm End Mill



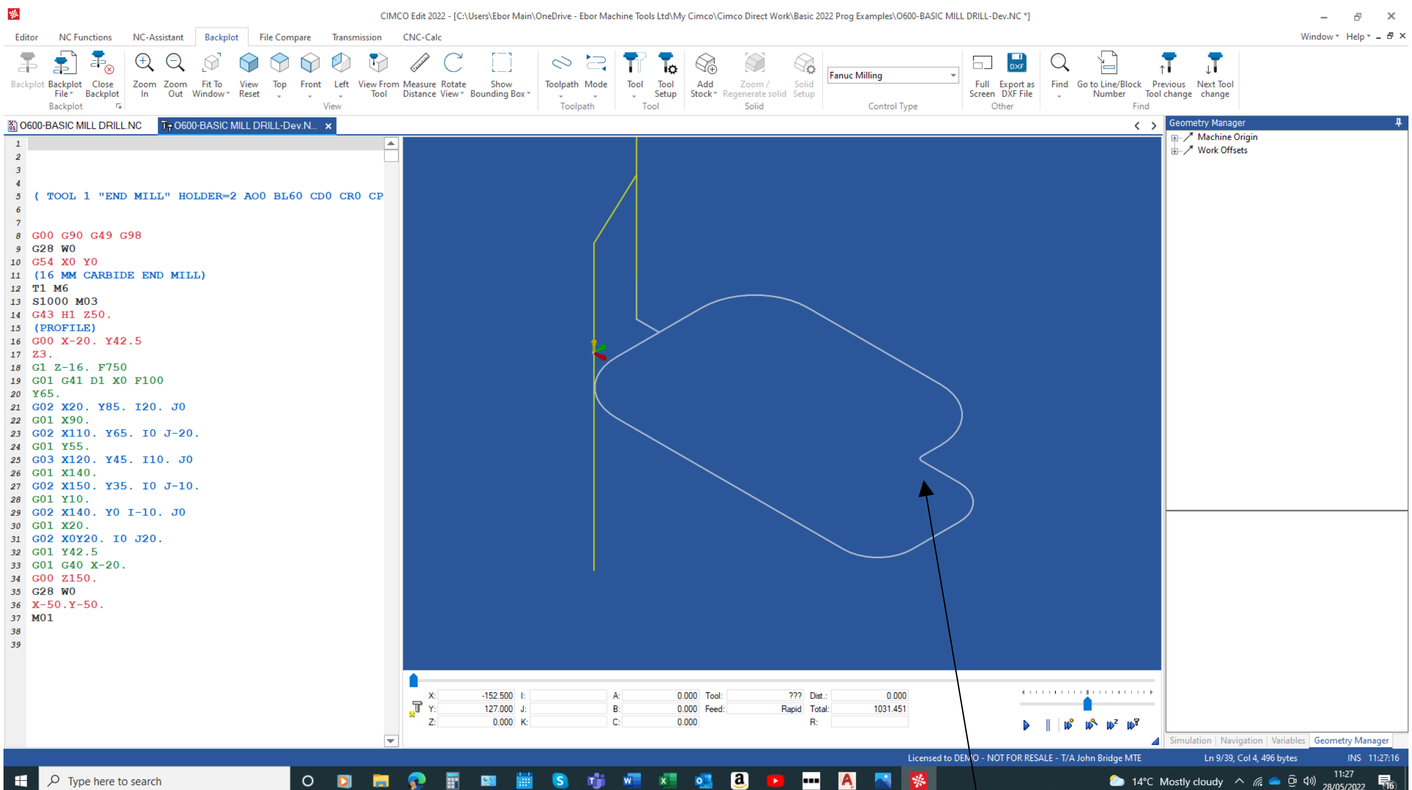
We need a 16 mm End Mill to carry out this operation so double click End Mill in NC-Assistant and fill in the details



The resulting insertion will create the tool data required by the editor to carry out a Backplot with tool length and radius taken into consideration. The CNC control will ignore this line as it will be seen as a comment only.

Test with Backplot and see below:

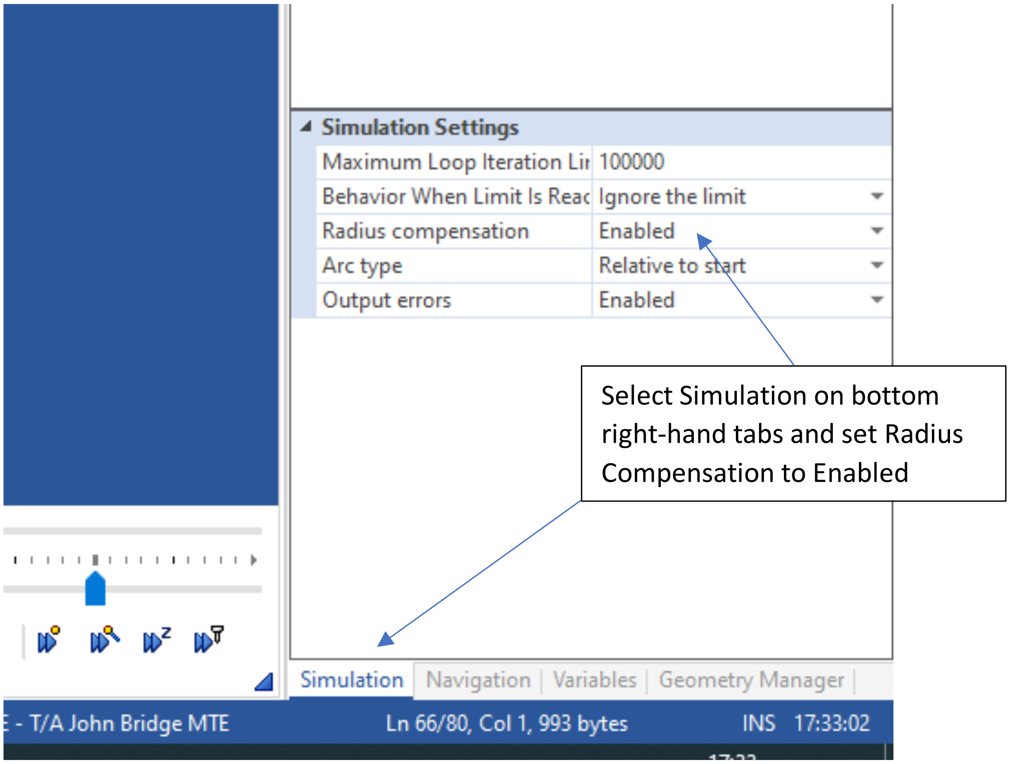
Backplot has tracked the profile by using the Tool Data provided and the Tool Path is now compensated showing the corrected path. This will become more evident when we apply the stock sizes from NC-Assistant.




Compare this to the original Backplot and it shows that the radius's are different as the Backplot is now showing compensated tool path.

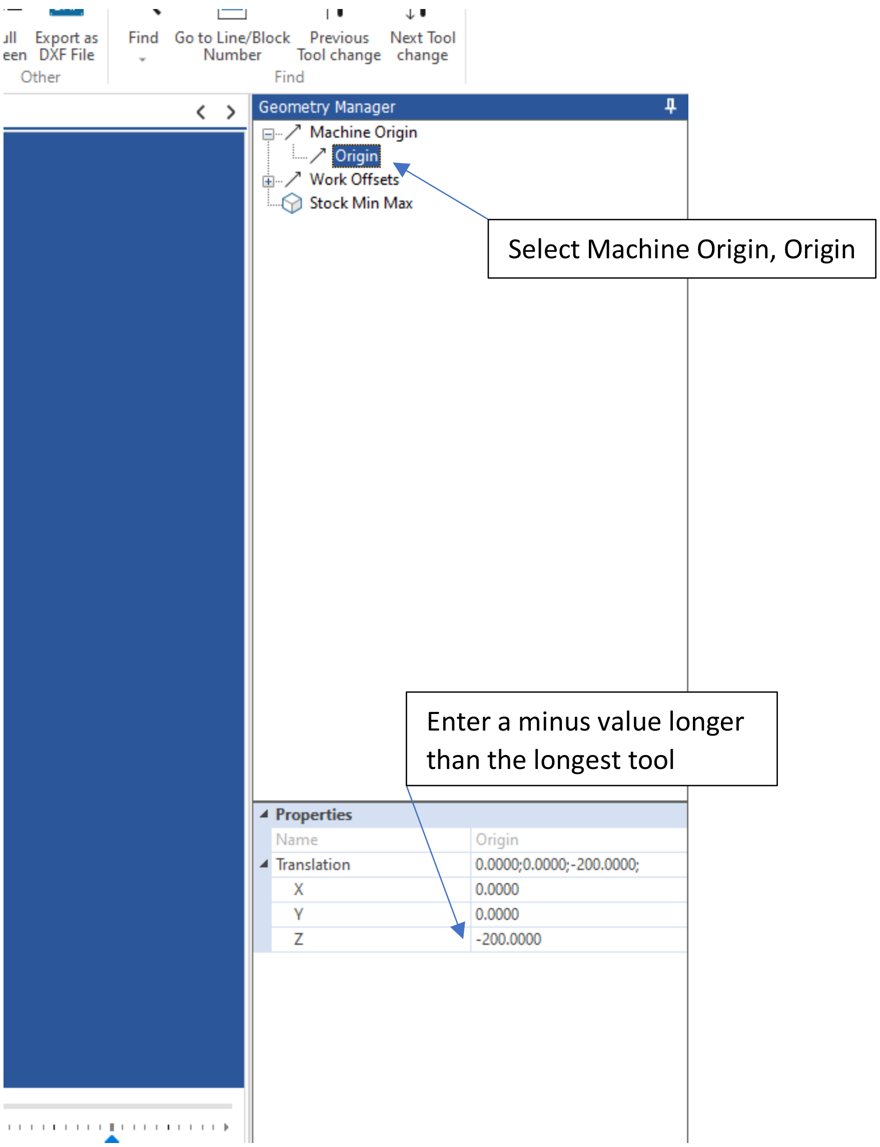
**IMPORTANT NOTE!!!**

Make sure that the Radius compensartion in the Similation settings is set to Enabled

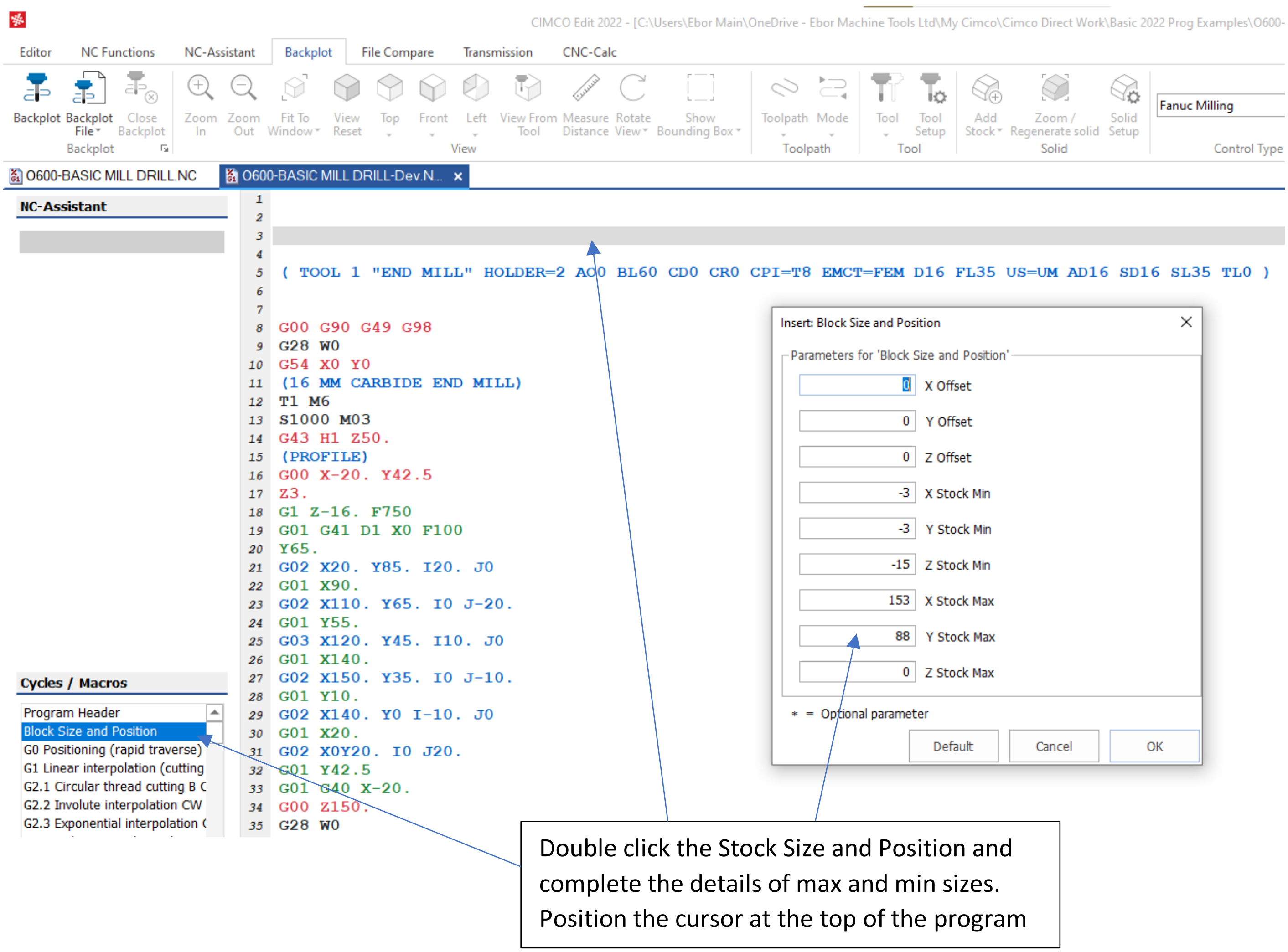


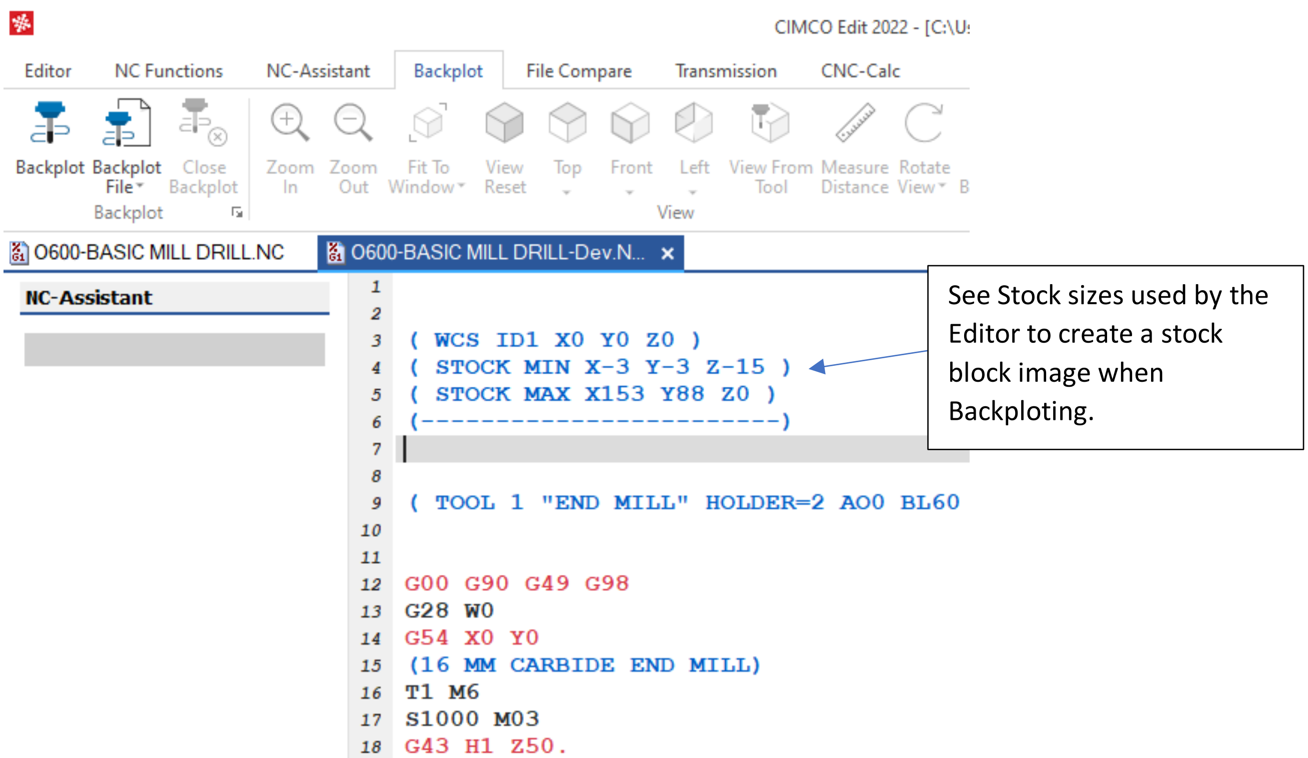

Continued

We also need to set a machine origin for the Z axis otherwise the Tool Change will place the Tool holder flange at Z0 in Backplot and it will look as though the Tool is buried in the Stock. The size of the origin should be a little longer that the longest tool and this will lift the Tools above the job in Backplot. On a CNC machine this is all taken care of by the Machine Coordinates, Work Shifts, and Tool Length offsets.

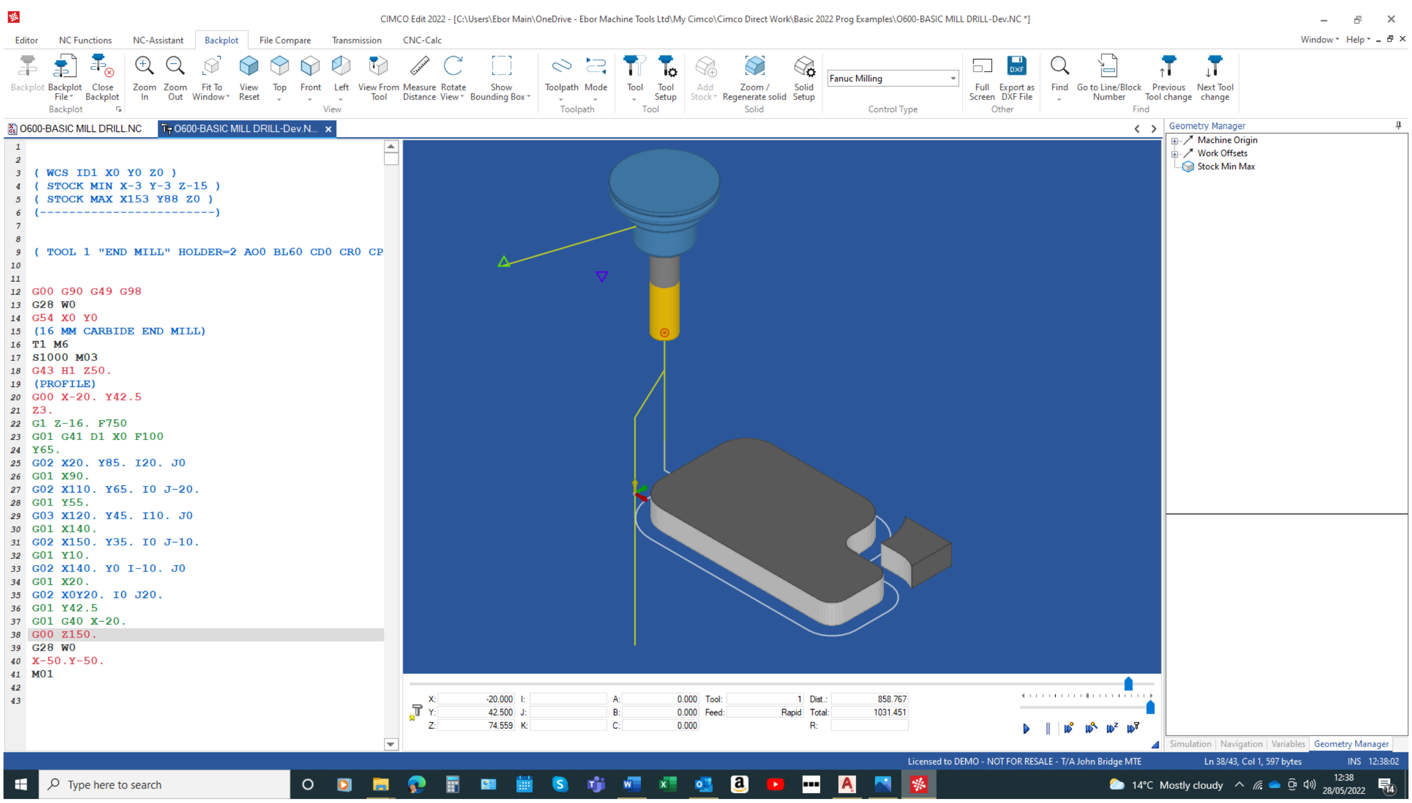


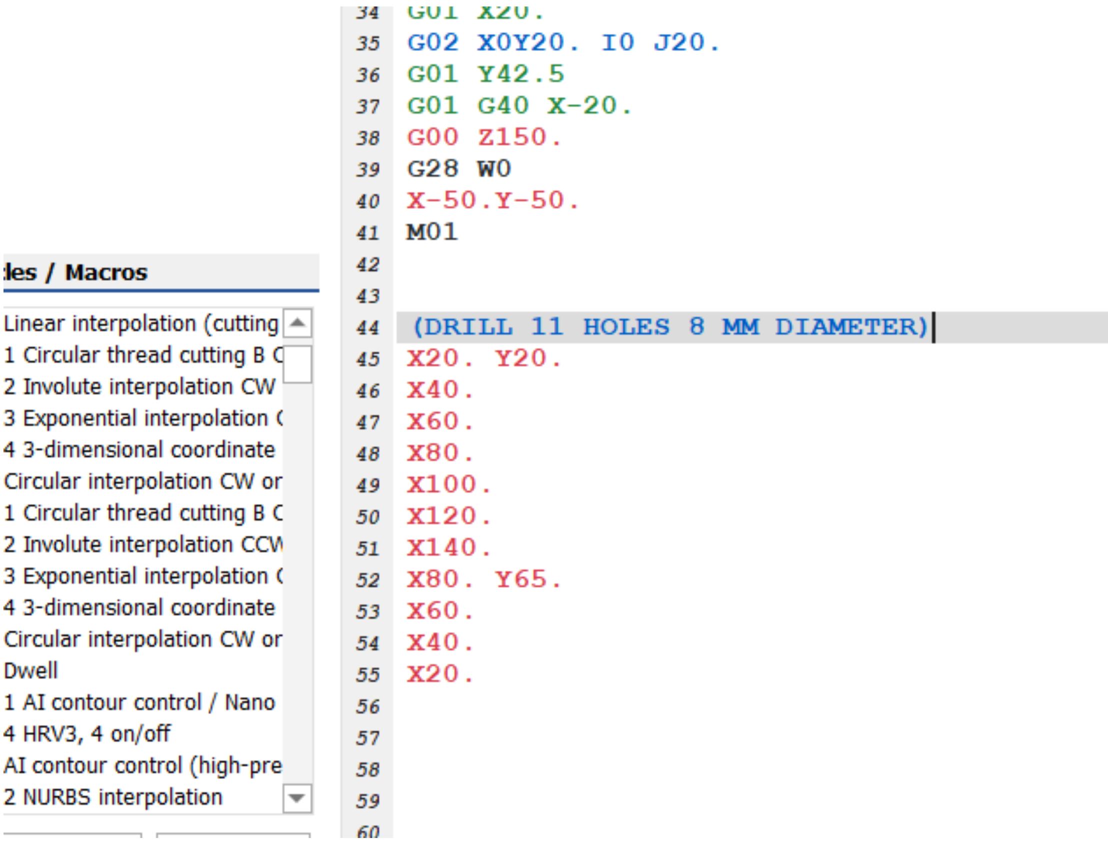
Let's now apply the stock sizes to the program. This again will be in the form of comments so the CNC control will ignore these blocks, but the Editor will use the sizes to create a stock block when Backploting.



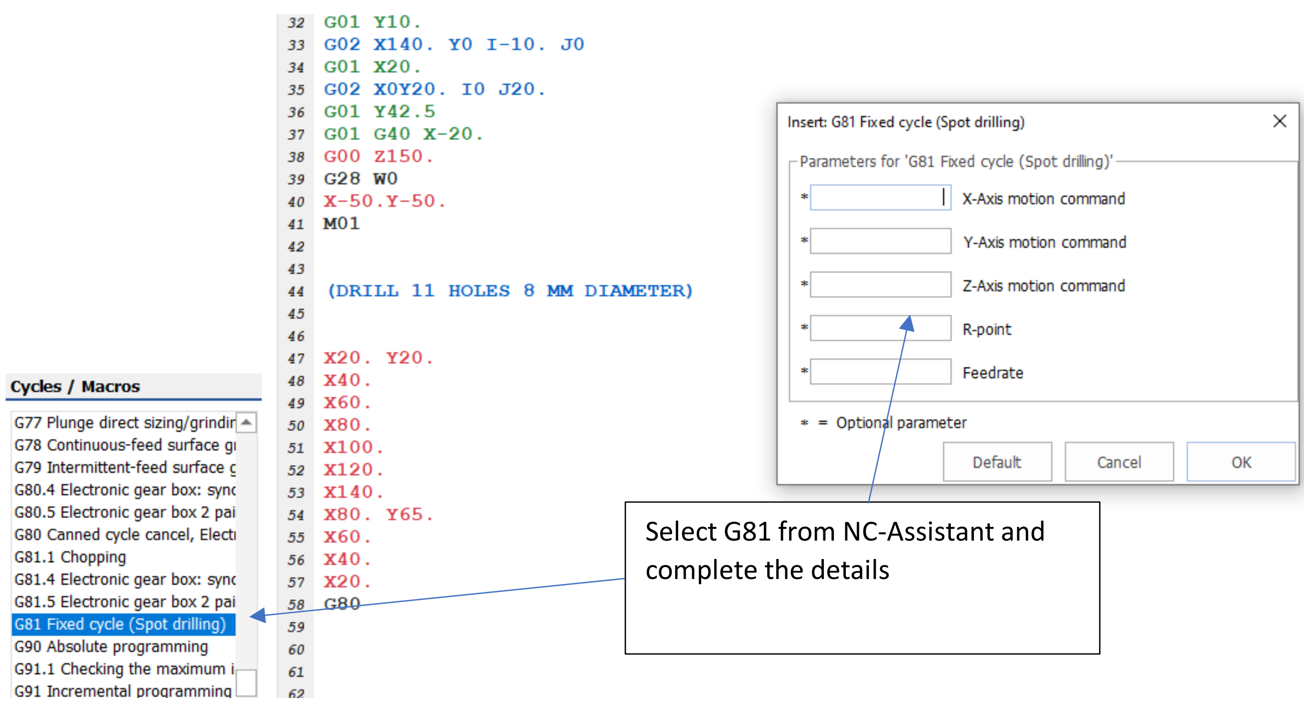



With the Stock Size we now have an accurate Backplot of the part as it would be profiled on the CNC Machine.






Now we will apply the drilling canned cycle G81 that will be used by the CNC control to set up a modal drilling cycle at the end of every coordinate move until the code G80 is read to stop the drilling operation.



**Cycles / Macros**

- G77 Plunge direct sizing/grindir
- G78 Continuous-feed surface gı
- G79 Intermittent-feed surface ç
- G80.4 Electronic gear box: sync
- G80.5 Electronic gear box 2 pai
- G80 Canned cycle cancel, Electı
- G81.1 Chopping
- G81.4 Electronic gear box: sync
- G81.5 Electronic gear box 2 pai
- G81 Fixed cycle (Spot drilling)
- G90 Absolute programming

```
M01


(DRILL 11 HOLES 8 MM DIAMETER)

G81 X20. Y20. Z-20. R2. F65
X20. Y20.
X40.
X60.
X80.
X100.
X120.
X140.
X80. Y65.
X60.
X40.
X20.
G80
```

We have incorporated the first hole into the cycle, so we do not need the next X20. Y20. line and can delete it.

Notice Z is the final depth and R is the position the tool rapids down to before drilling the hole at Feedrate F.

On completion of the drilling operations enter the line G80. This will cancel further drilling of a hole after the next positioning move.

NOTE!

This canned cycle is suitable for a Fanuc and other CNC controls but there are several CNC controls that merely set up the depths, feeds, etc. and do not execute the canned cycle until after the next block coordinate move has taken place.

**NC-Assistant**

No macro found

**Cycles / Macros**

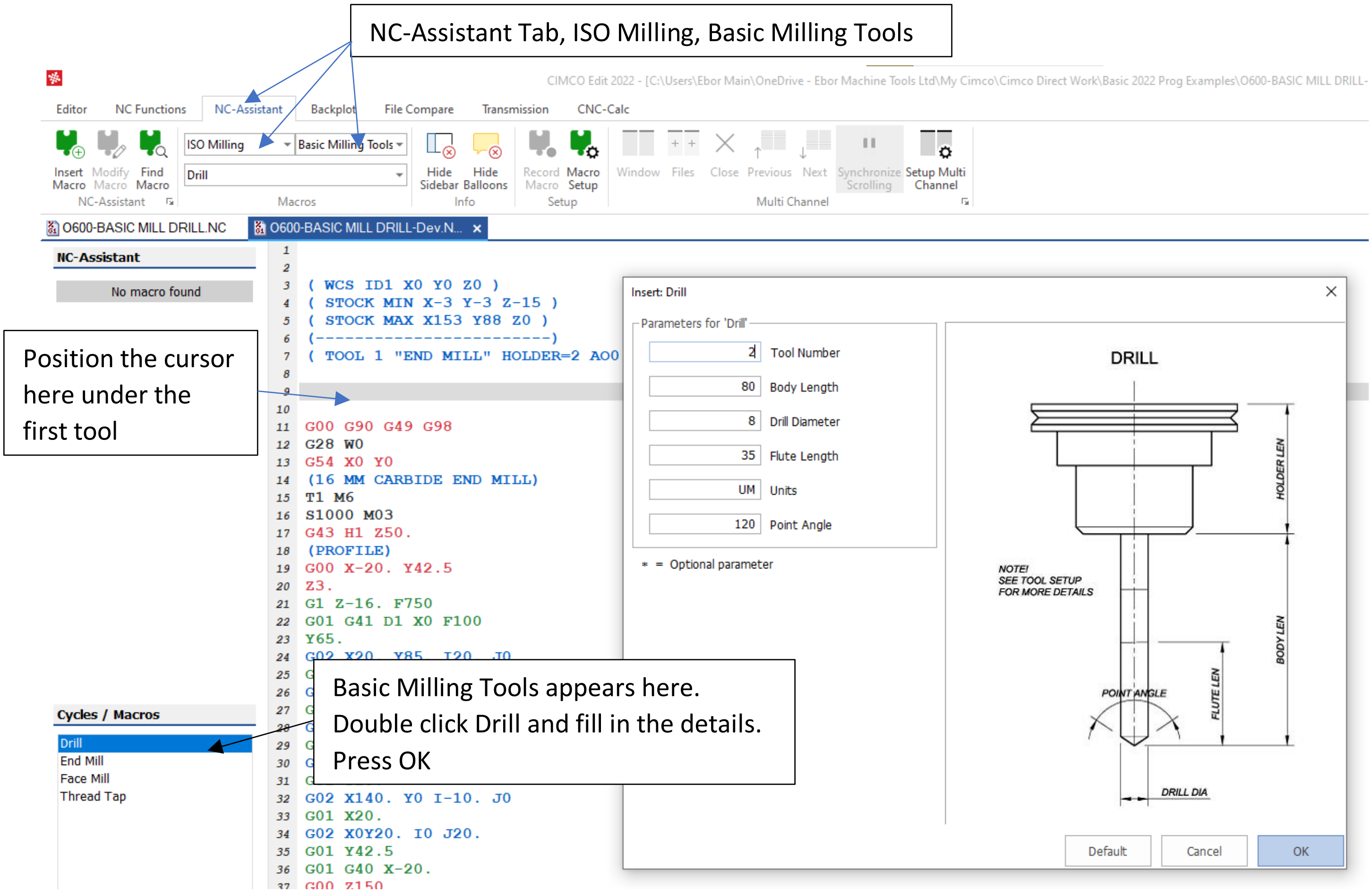
- G81.4 Electronic gear box: sync
- G81.5 Electronic gear box 2 pai
- G81 Fixed cycle (Spot drilling)
- G90 Absolute programming
- G91.1 Checking the maximum i
- G91 Incremental programming
- G92.1 Workpiece coordinate sy
- G92 Setting for workpiece coor

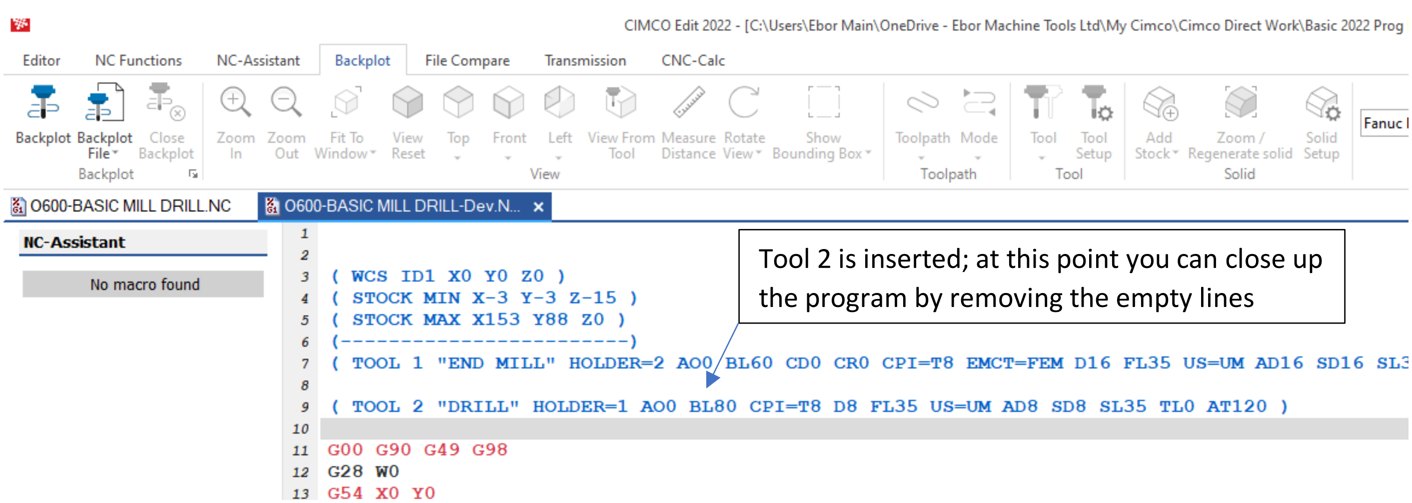
```
G01 G40 X-20.
G00 Z150.
G28 W0
X-50.Y-50.
M01

(DRILL 11 HOLES 8 MM DIAMETER)
G00 G90 G49 G98
G28 W0
G54 X0 Y0
(8 MM CARBIDE DRILL)
T2 M6
S1000 M03
G43 H2 Z50.
(PROFILE)
G00 X-20. Y20.
Z50.
G81 X20. Y20. Z-20. R2. F65
X20. Y20.
X40.
X60.
X80.
X100.
X120.
X140.
X80. Y65.
X60.
X40.
X20.
G80
G00 Z150.
G28 W0
X-50.Y-50.
M01
```

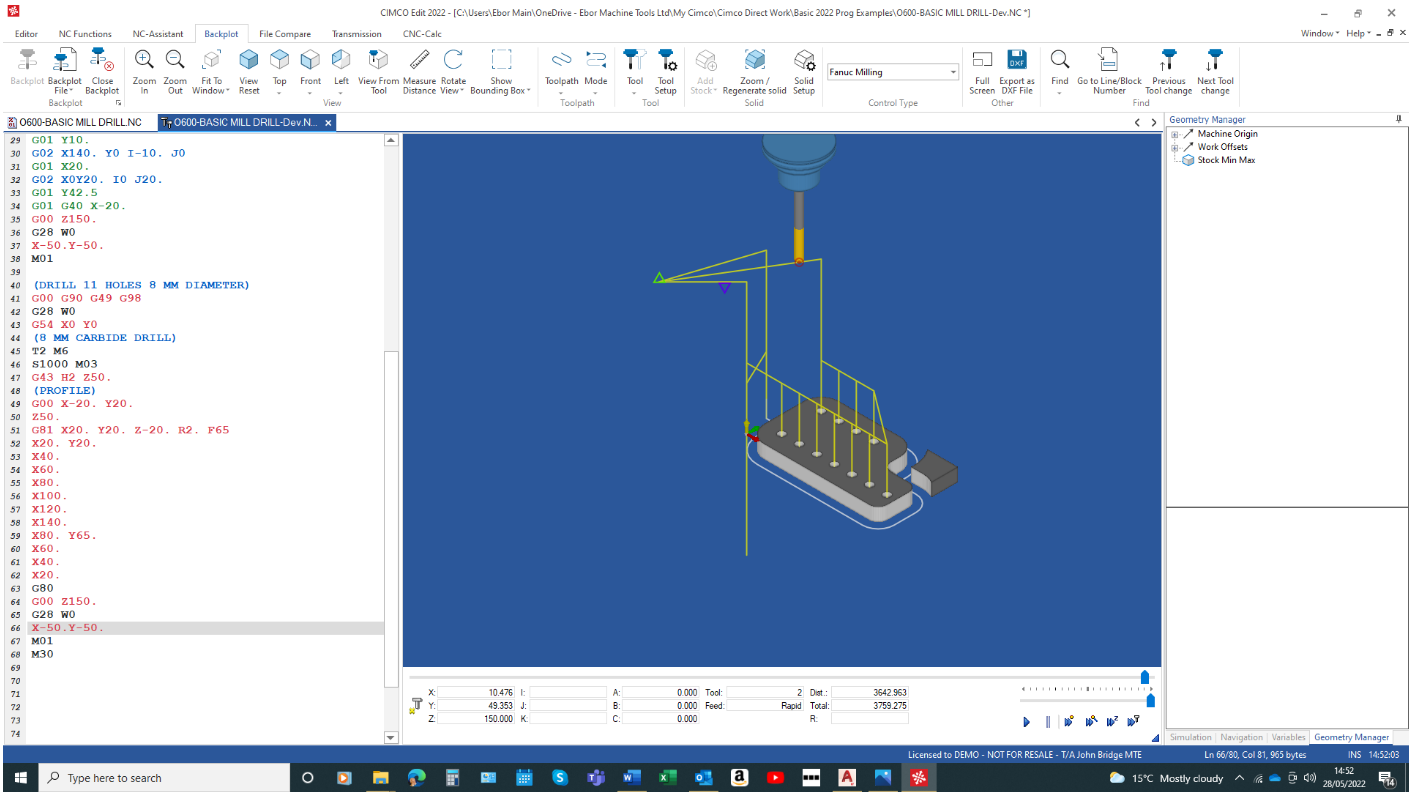
We have inserted a header and a trailer by copying for the first tool header and making the simple edits to suit the drilling operation.

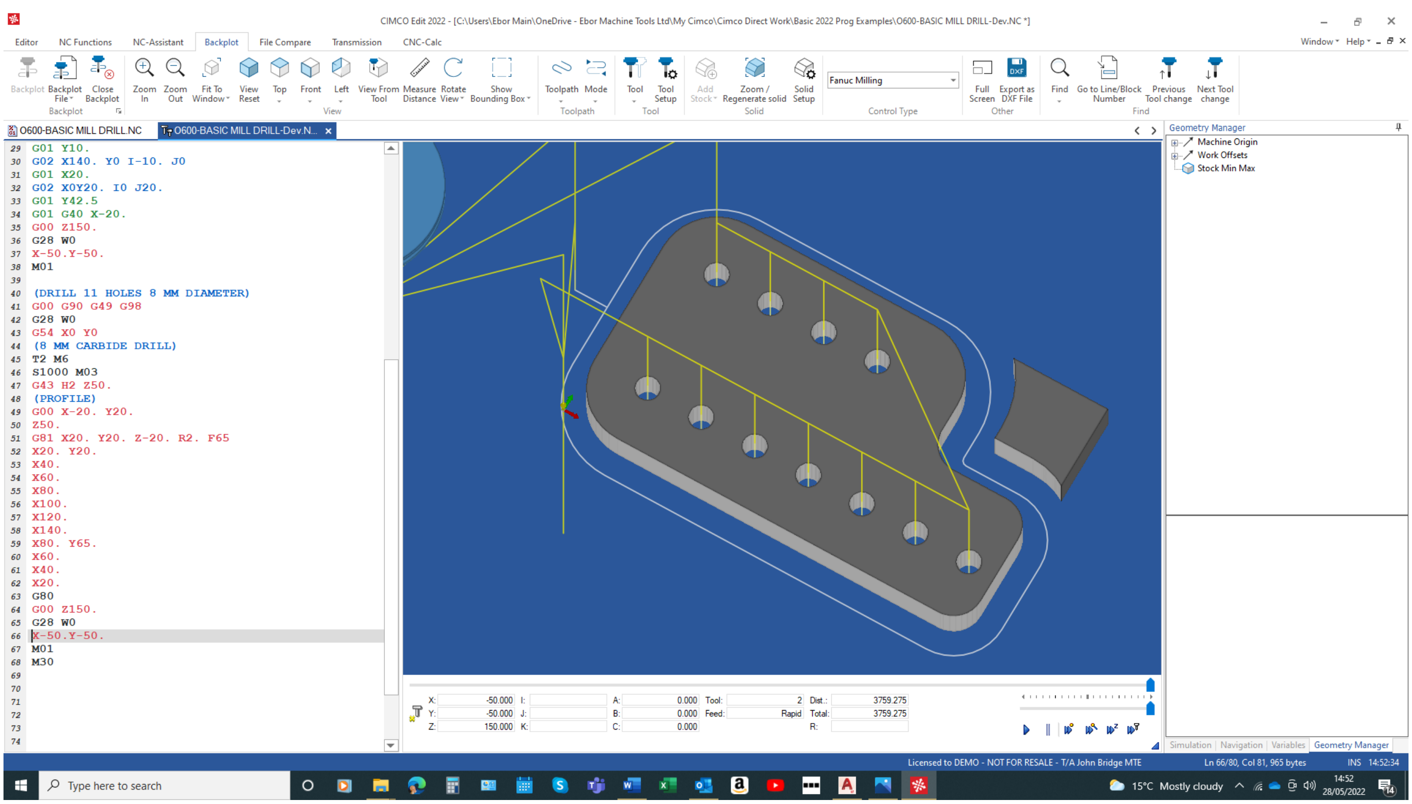
We now have to setup the Tool for this drilling operation so we will visit the Basic Milling Tool list see below.



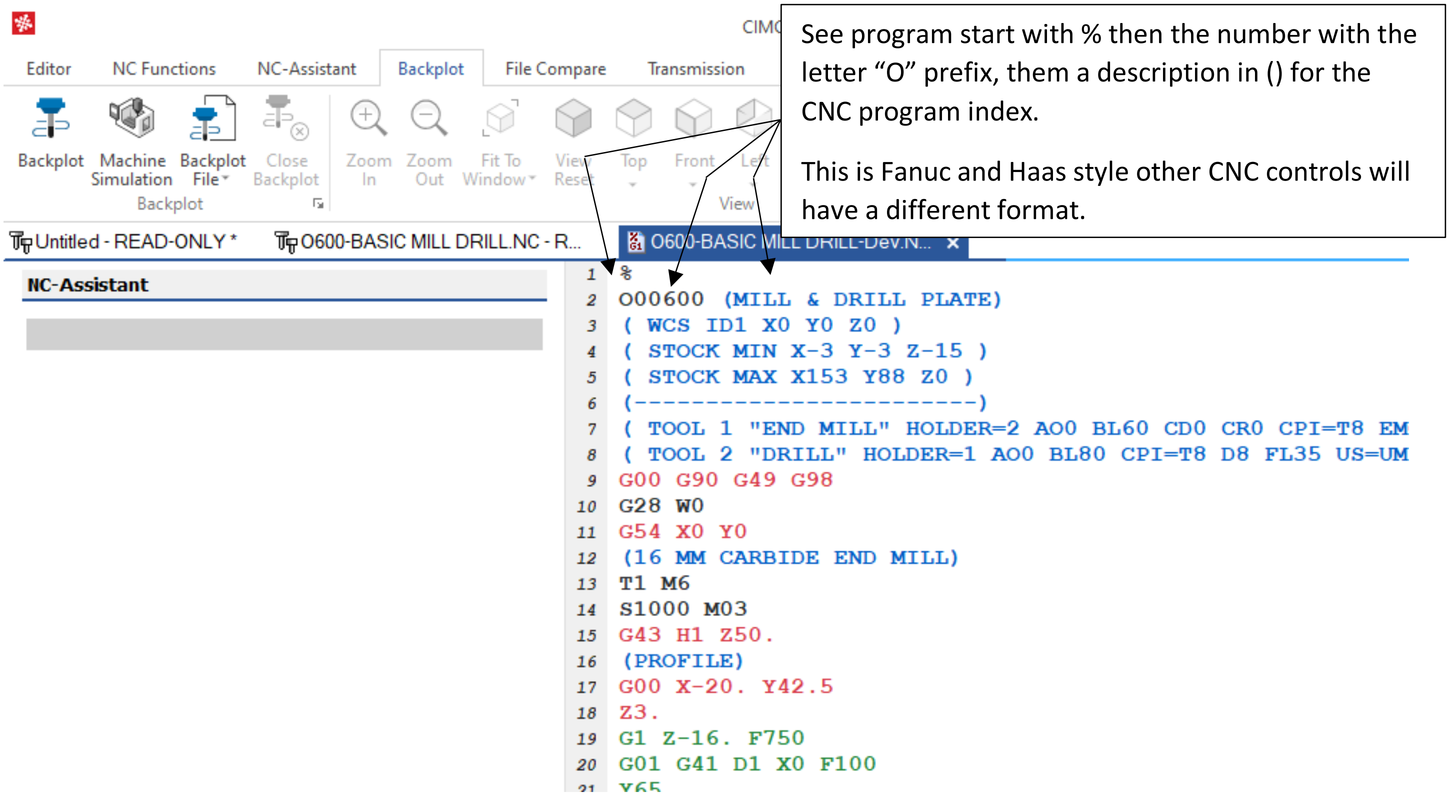



Now complete with a full Backplot to see the program complete with Radius and Length Compensation



We can add a program number and the start of program % sign that is used when transferring a CNC part program to and from the CNC control memory to a peripheral device like a PC, see below




See below the full CNC program text

%

O00600 (MILL & DRILL PLATE)

( WCS ID1 X0 Y0 Z0 )

( STOCK MIN X-3 Y-3 Z-15 )

( STOCK MAX X153 Y88 Z0 )

(-----------------------)

( TOOL 1 "END MILL" HOLDER=2 AO0 BL60 CD0 CR0 CPI=T8 EMCT=FEM D16 FL35 US=UM AD16 SD16 SL35 TL0 )

( TOOL 2 "DRILL" HOLDER=1 AO0 BL80 CPI=T8 D8 FL35 US=UM AD8 SD8 SL35 TL0 AT120 )

G00 G90 G49 G98

G28 W0

G54 X0 Y0

(16 MM CARBIDE END MILL)

T1 M6

```
S1000 M03
G43 H1 Z50.
(PROFILE)
G00 X-20. Y42.5
Z3.
G1 Z-16. F750
G01 G41 D1 X0 F100
Y65.
G02 X20. Y85. I20. J0
G01 X90.
G02 X110. Y65. I0 J-20.
G01 Y55.
G03 X120. Y45. I10. J0
G01 X140.
G02 X150. Y35. I0 J-10.
G01 Y10.
G02 X140. Y0 I-10. J0
G01 X20.
G02 X0Y20. I0 J20.
G01 Y42.5
G01 G40 X-20.
G00 Z150.
G28 W0
X-50.Y-50.
M01

(DRILL 11 HOLES 8 MM DIAMETER)
G00 G90 G49 G98
G28 W0
G54 X0 Y0
```

```
(8 MM CARBIDE DRILL)
T2 M6
S1000 M03
G43 H2 Z50.
(PROFILE)
G00 X-20. Y20.
Z50.
G81 X20. Y20. Z-20. R2. F65
X40.
X60.
X80.
X100.
X120.
X140.
X80. Y65.
X60.
X40.
X20.
G80
G00 Z150.
G28 W0
X-50.Y-50.
M01
M30
```