



Basic ISO Programming Exercise 1

Milling

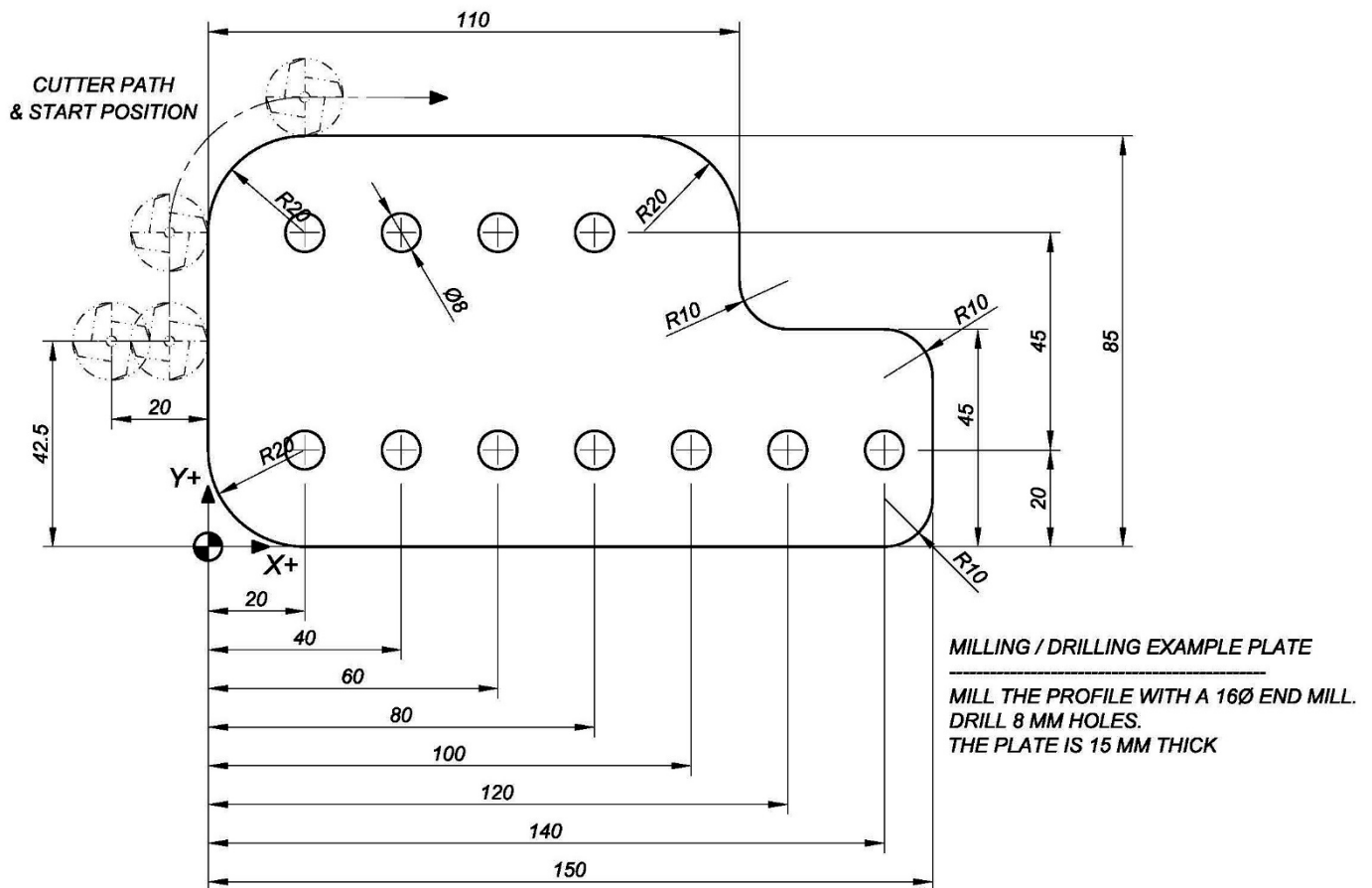


This document is made available as a preliminary version (draft).
Questions and feedback should be sent to support@cimco.com

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.

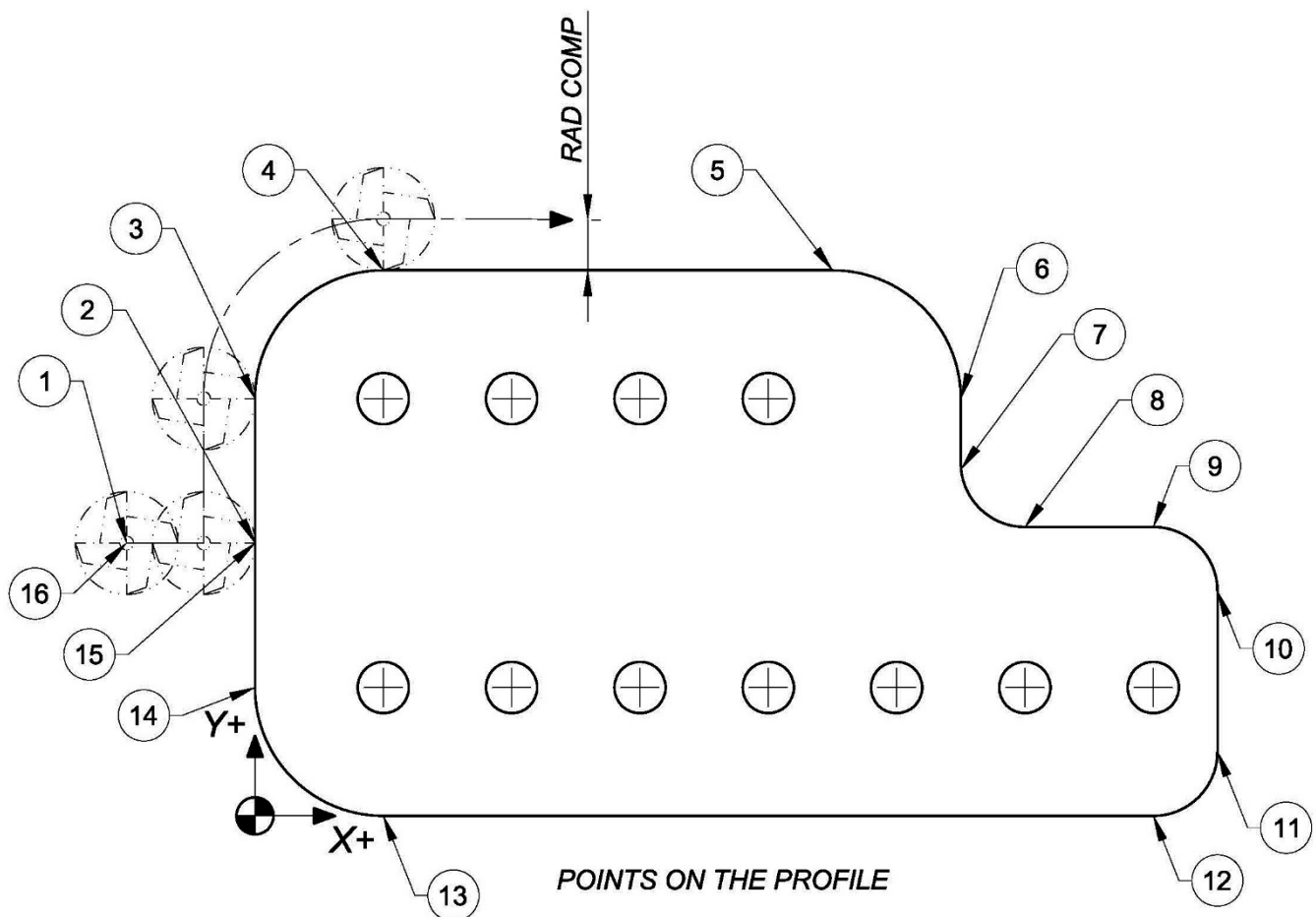


We will consider profile milling first. When programming the part, we will use the drawing dimensions to describe the tool path. The program will be made up of line by line (block by block is the term used) information that will comprise straight lines and arcs. The basic move commands that make up Group 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 an 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

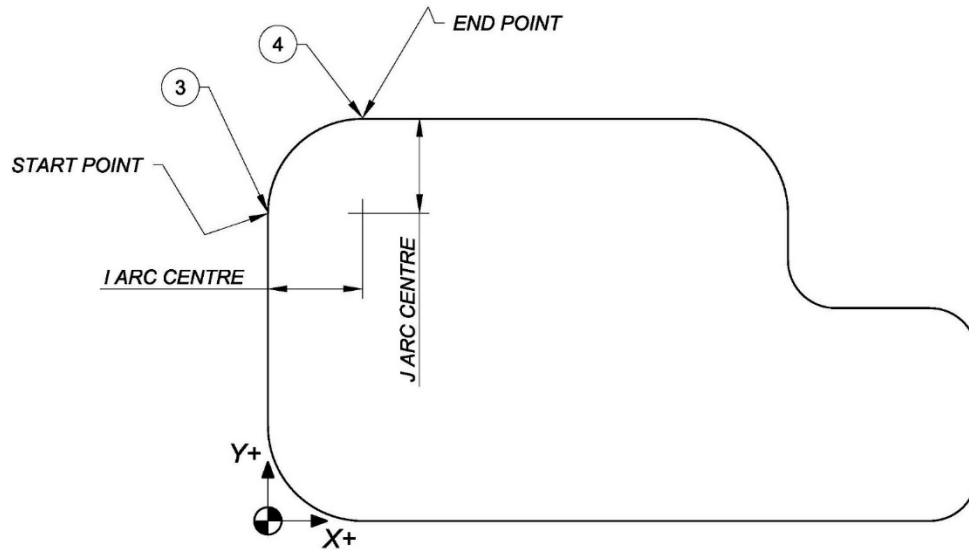
modal.

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.

NOTE: Comments in brackets are ignored by the CNC control.

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 arc 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 maximum 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.

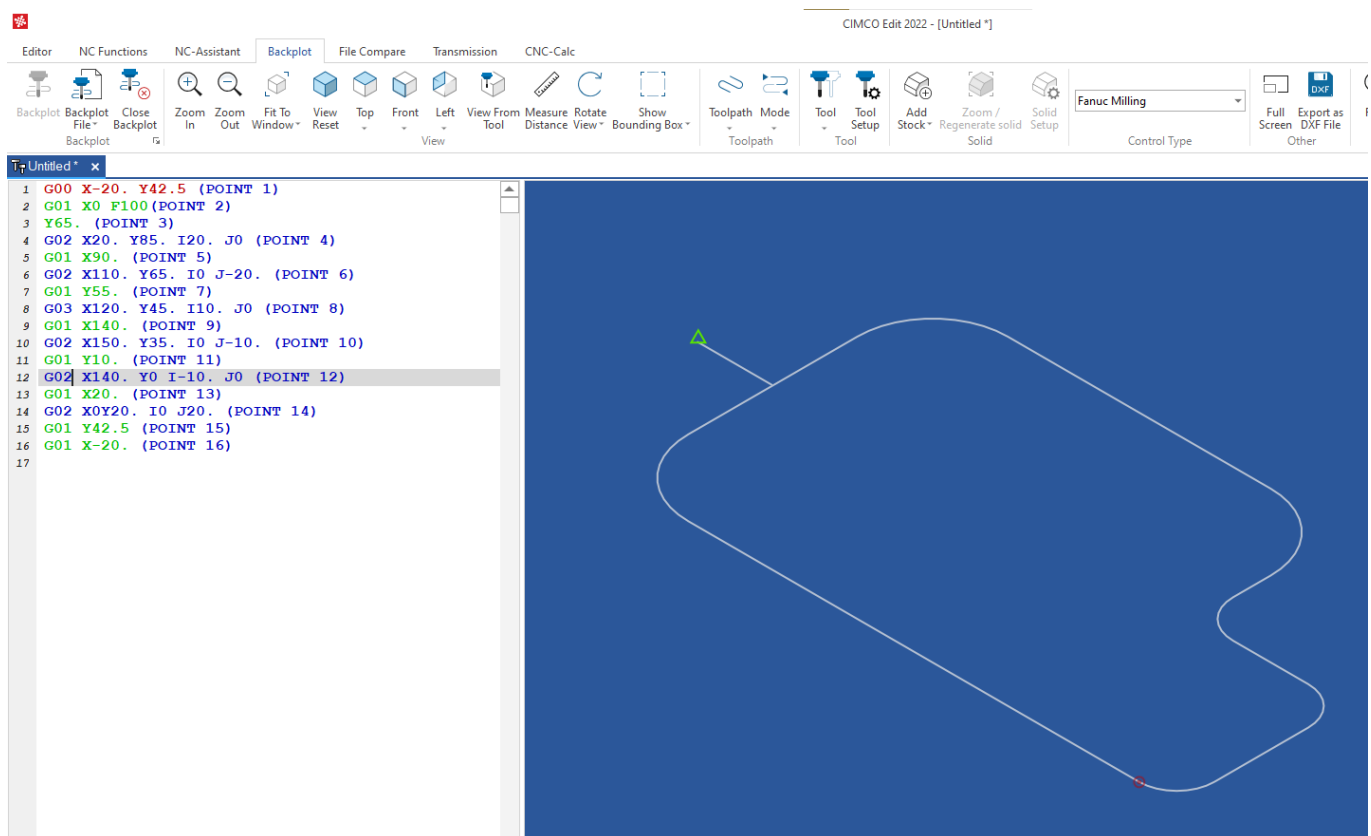
Programming good practice is to layout a block in order. This will help you see the problems when snagging a program for bugs. e.g.

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

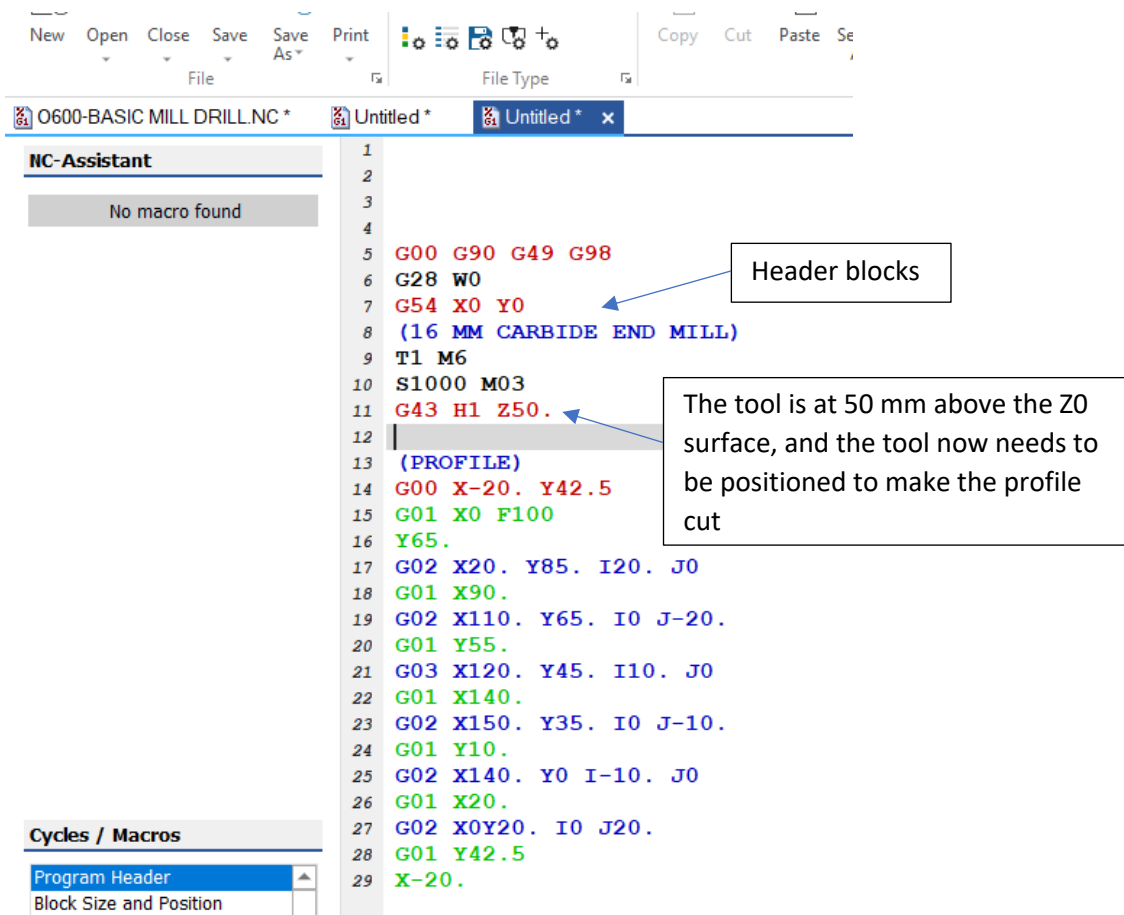
G43 is activating the tool length offset, H1 or H01 is the tool length seat in the table, Z50. Move the tool to 50 mm above the Z0 position.

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



New Open Close Save Save As Print
 File File Type

O600-BASIC MILL DRILL.NC * Untitled * x Untitled *

NC-Assistant
 G1 Linear interpolation (cutting ...
 X-Axis motion command:
 Y-Axis motion command:
 Z-Axis motion command:
 A-Axis motion command:
 B-Axis motion command:
 C-Axis motion command:
 Feedrate:
 Modify

Cycles / Macros
 Program Header
 Block Size and Position
 G0 Positioning (rapid traverse)

```

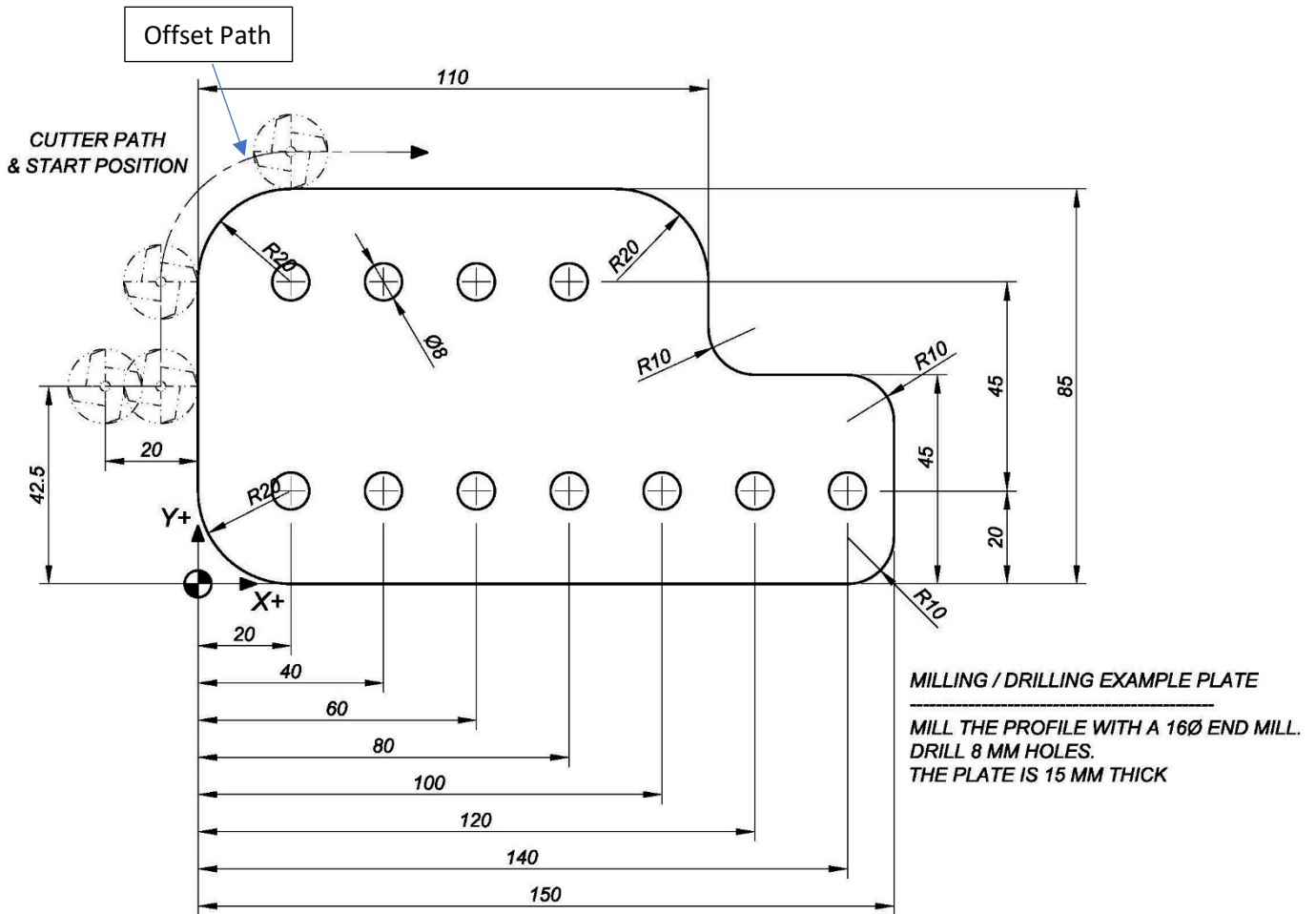
1
2
3
4
5 G00 G90 G49 G98
6 G28 W0
7 G54 X0 Y0
8 (16 MM CARBIDE END MILL)
9 T1 M6
10 S1000 M03
11 G43 H1 Z50.
12 (PROFILE)
13 G00 X-20. Y42.5
14 Z3.
15 G01 Z-16. F750
16 G01 X0 F100
17 Y65.
18 G02 X20. Y85. I20. J0
19 G01 X90.
20 G02 X110. Y65. I0 J-20.
21 G01 Y55.
22 G03 X120. Y45. I10. J0
23 G01 X140.
24 G02 X150. Y35. I0 J-10.
25 G01 Y10.
26 G02 X140. Y0 I-10. J0
27 G01 X20.
28 G02 X0Y20. I0 J20.
29 G01 Y42.5
30 X-20.
  
```

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

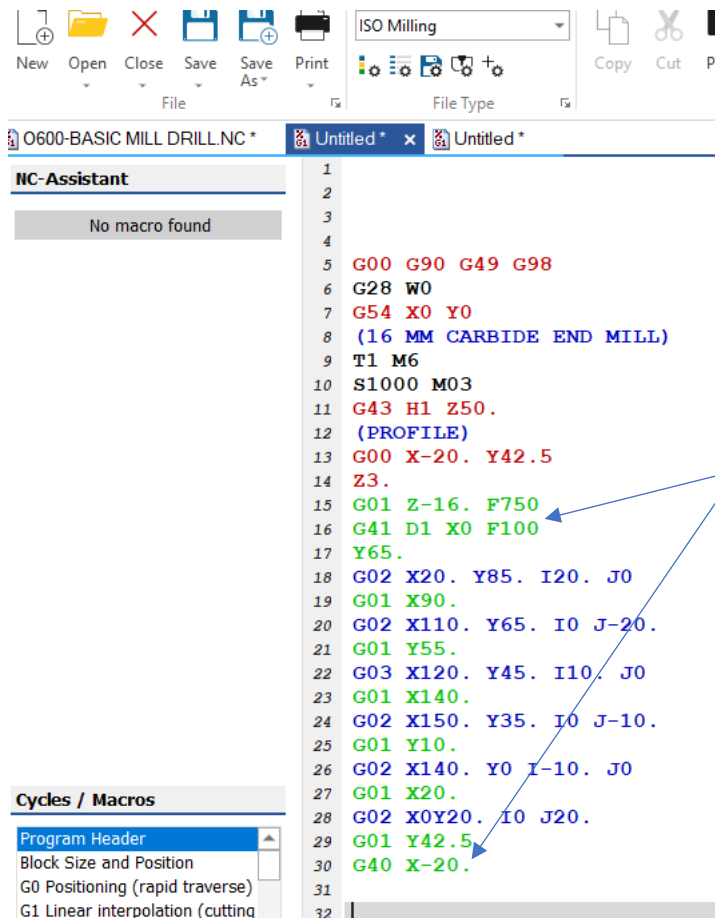
G41 - Activate a Radius Offset path (Radius Compensation is the term used) to the left in the direction of travel.

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.

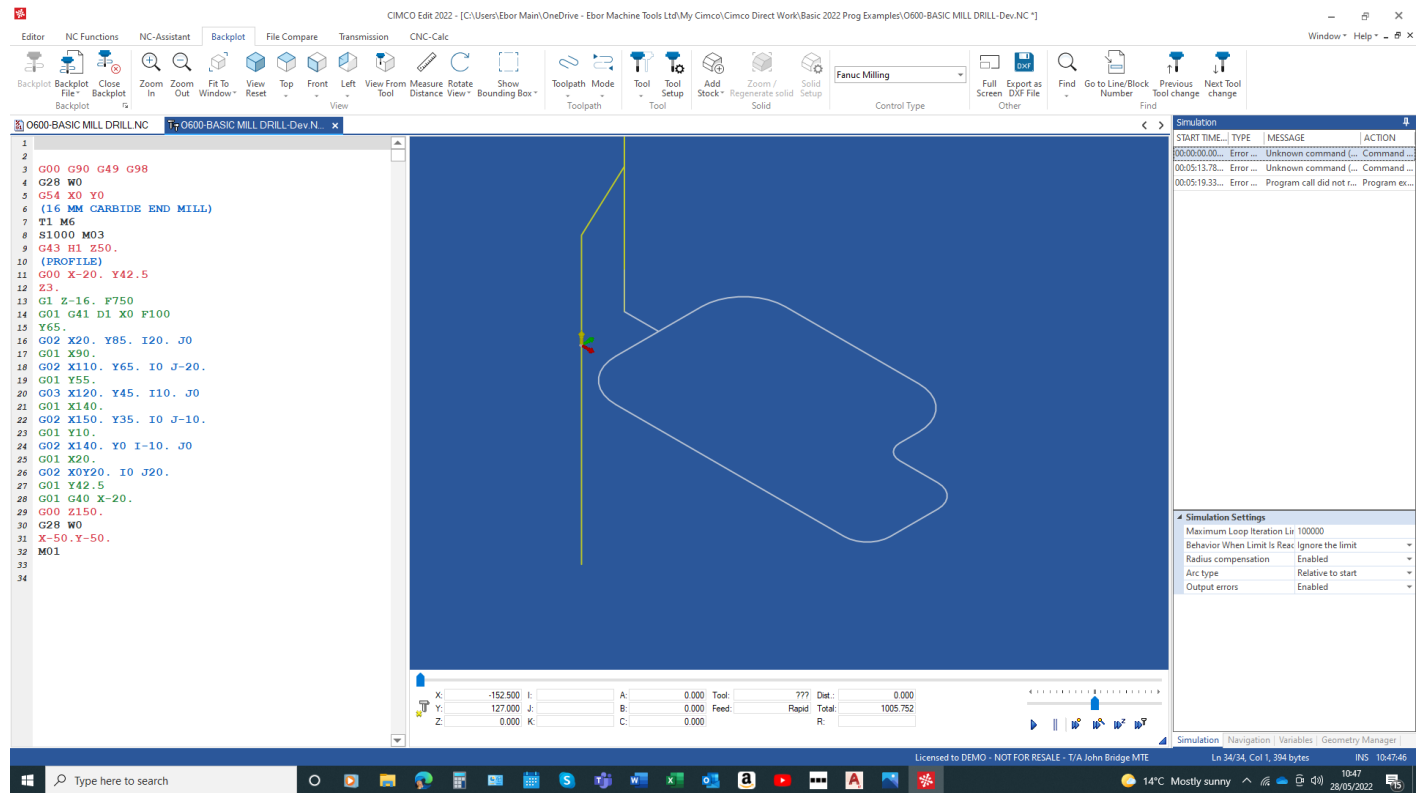
From the drawing we can see the direction of travel around the periphery of the part is counterclockwise. Therefore, we will offset the path to the left in the direction of travel by using G41

See below:

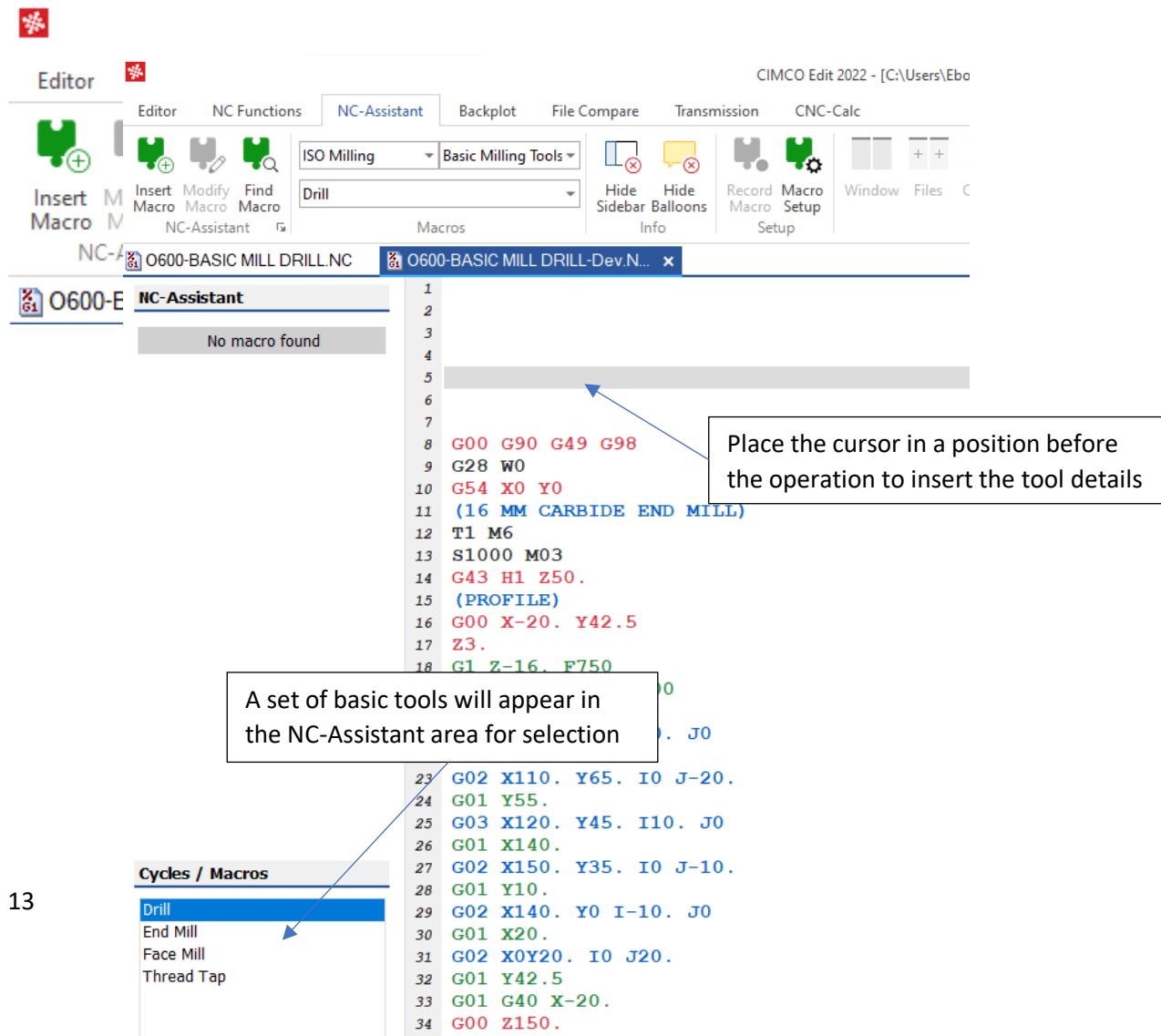
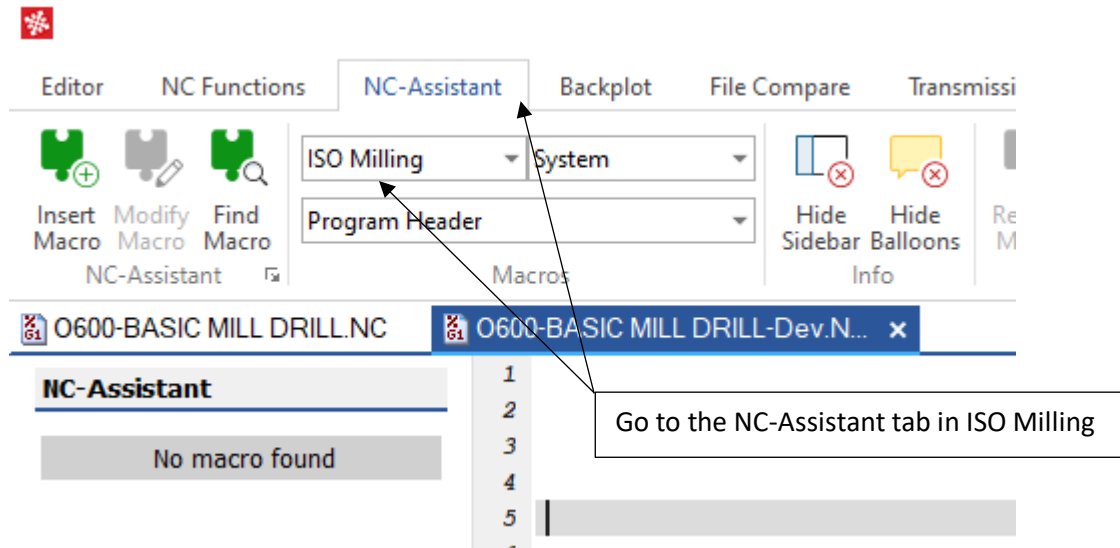


Test with Backplot and see below:

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

CIMCO Edit 2022 - [C:\Users\Ebor Main\OneDrive - Ebor Machine Tools Ltd\My Cimco\Cimco Direct Work\Basic 2022 Prog Examples\O600-BASIC MILL DRILL-Dr

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

ISO Milling Basic Milling Tools Drill

Insert Macro Find Macro NC-Assistant Macros

Hide SideBar Hide Balloons Record Macro Macro Setup Setup

Window Files Close Previous Next Synchronize Scrolling Setup Multi Channel

O600-BASIC MILL DRILL.NC O600-BASIC MILL DRILL-Dev.N

NC-Assistant

No macro found

1
2
3
4
5
6
7
8 G00 G90 G49 G98
9 G28 W0
10 G54 X0 Y0
11 (16 MM CARBIDE END MILL)
12 T1 M6
13 S1000 M03
14 G43 H1 Z50.
15 (PROFILE)
16 G00 X-20. Y42.5
17 Z3.
18 G1 Z-16. F750
19 G01 G41 D1 X0 F100
20 Y65.
21 G02 X20. Y85. I20. J0
22 G01 X90.
23 G02 X110. Y65. I0 J-20.
24 G01 Y55.
25 G03 X120. Y45. I10. J0
26 G01 X140.
27 G02 X150. Y35. I0 J-10.
28 G01 Y10.
29 G02 X140. Y0 I-10. J0
30 G01 X20.
31 G02 X0Y20. I0 J20.
32 G01 Y42.5
33 G01 G40 X-20.
34 G00 Z150.
35 G28 W0
36 X-50. Y-50.
37 M01
38

Cycles / Macros

Drill
End Mill
Face Mill
Thread Tap

Insert: End Mill

Parameters for 'End Mill'

Tool Number
16 Diameter
35 Flute Length
60 Body Length

* = Optional parameter

END MILL

NOTE! SEE TOOL SETUP FOR MORE DETAILS

FLUTE LEN
HOLDER LEN
BODY LEN
SHAFT DIA
FLUTE DIA
UNITS UM=METRIC UI=IMPERIAL

Default Cancel OK

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

CIMCO Edit 2022 - [C:\Users\Ebor Main\OneDrive - Ebor Machine Tools Ltd\My Cimco\Cimco Direct Work\Basic 2022 Prog Examples\O600-BASIC MIL

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

ISO Milling Basic Milling Tools Drill

Insert Macro Find Macro NC-Assistant Macros

Hide SideBar Hide Balloons Record Macro Macro Setup Setup

Window Files Close Previous Next Synchronize Scrolling Setup Multi Channel

O600-BASIC MILL DRILL.NC O600-BASIC MILL DRILL-Dev.N

NC-Assistant

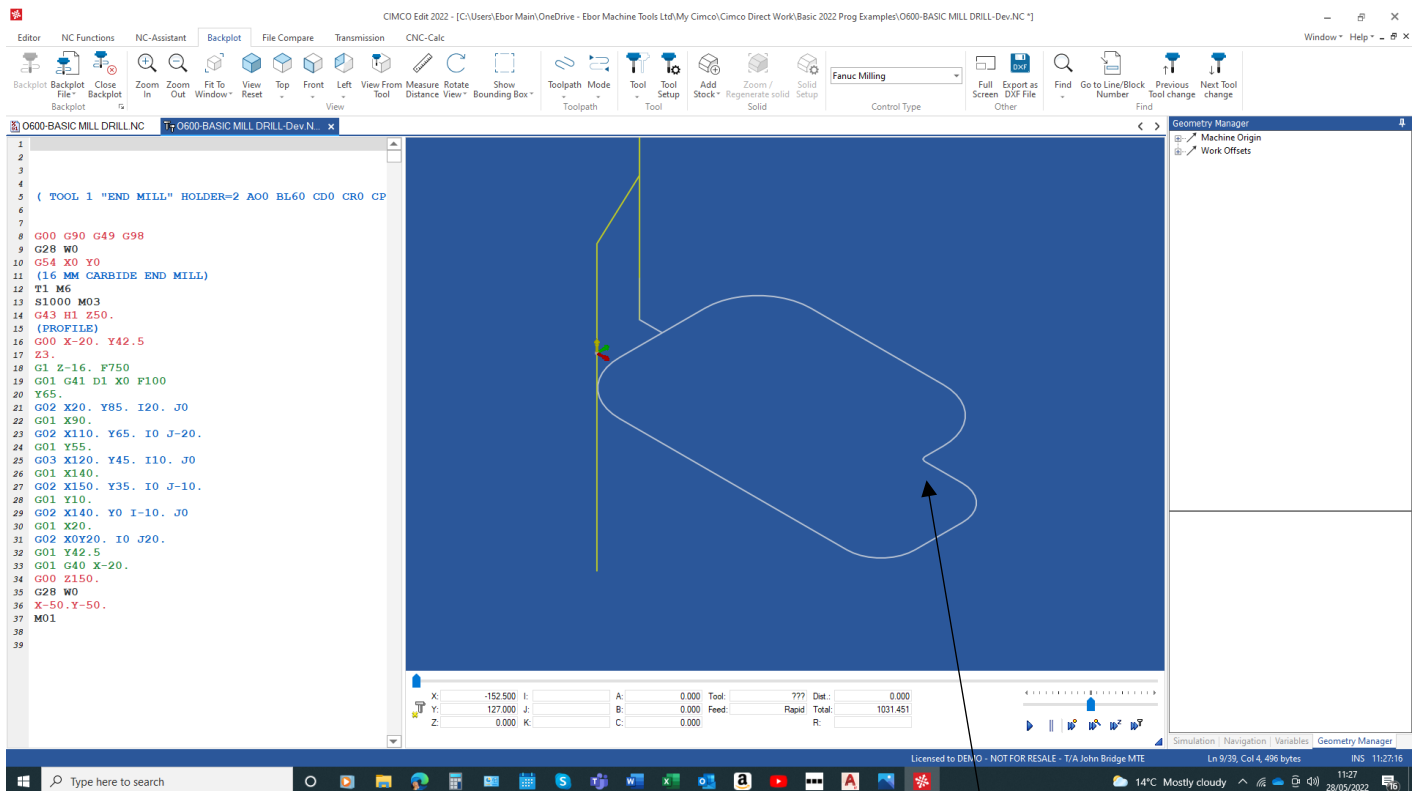
No macro found

1
2
3
4
5 (TOOL 1 "END MILL" HOLDER=2 AO0 BL60 CD0 CRO CPI=T8 EMCT=FEM D16 FL35 US=UM AD16 SD16 SL35 TL0)
6
7
8 G00 G90 G49 G98
9 G28 W0
10 G54 X0 Y0
11 (16 MM CARBIDE END MILL)
12 T1 M6
13 S1000 M03
14 G43 H1 Z50.
15 (PROFILE)
16 G00 X-20. Y42.5

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:

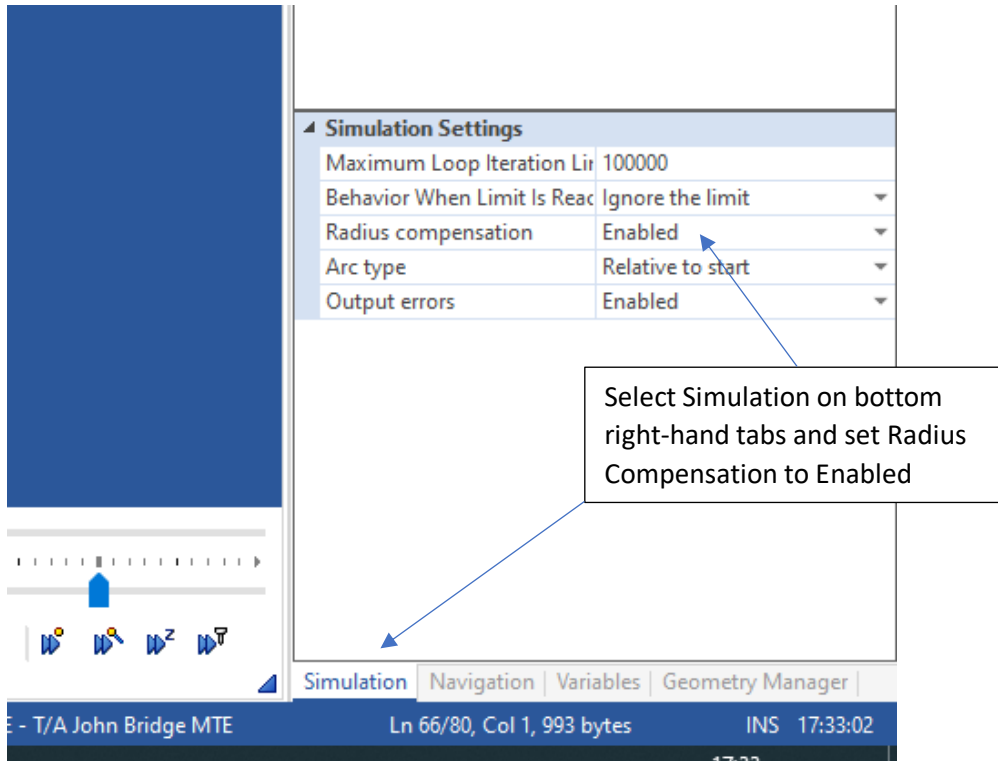
Backplot has tracked the profile by using the Tool Data provided and the Tool Path is now compensated showing the corrected path. This will become more evident when we apply the stock sizes from NC-Assistant.



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 compensation in the Simulation 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 than 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.

Geometry Manager

- Machine Origin
 - Origin
 - Work Offsets
 - Stock Min Max

Select Machine Origin, Origin

Properties

Properties	
Name	Origin
Translation	0.0000;0.0000;-200.0000;
X	0.0000
Y	0.0000
Z	-200.0000

Enter a minus value longer than the longest tool

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 Backplotting.

The screenshot shows the CIMCO Edit 2022 interface. The top menu bar includes Editor, NC Functions, NC-Assistant, Backplot, File Compare, Transmission, and CNC-Calc. The NC-Assistant tab is active, showing a list of macros on the left and a code editor on the right. The code editor displays a CNC program with various G-codes and comments. A dialog box titled "Insert: Block Size and Position" is open, showing parameters for X, Y, and Z offsets and stock sizes. A text box with arrows points to the "Block Size and Position" macro in the list and the top of the code editor, with instructions to double-click it and position the cursor at the top of the program.

NC-Assistant

Cycles / Macros

- Program Header
- Block Size and Position**
- G0 Positioning (rapid traverse)
- G1 Linear interpolation (cutting)
- G2.1 Circular thread cutting B C
- G2.2 Involute interpolation CW
- G2.3 Exponential interpolation C

Code Editor:

```

1
2
3
4
5 ( TOOL 1 "END MILL" HOLDER=2 A00 BL60 CD0 CR0 CPI=T8 EMCT=FEM D16 FL35 US=UM AD16 SD16 SL35 TL0 )
6
7
8 G00 G90 G49 G98
9 G28 W0
10 G54 X0 Y0
11 (16 MM CARBIDE END MILL)
12 T1 M6
13 S1000 M03
14 G43 H1 Z50.
15 (PROFILE)
16 G00 X-20. Y42.5
17 Z3.
18 G1 Z-16. F750
19 G01 G41 D1 X0 F100
20 Y65.
21 G02 X20. Y85. I20. J0
22 G01 X90.
23 G02 X110. Y65. I0 J-20.
24 G01 Y55.
25 G03 X120. Y45. I10. J0
26 G01 X140.
27 G02 X150. Y35. I0 J-10.
28 G01 Y10.
29 G02 X140. Y0 I-10. J0
30 G01 X20.
31 G02 X0Y20. I0 J20.
32 G01 Y42.5
33 G01 G40 X-20.
34 G00 Z150.
35 G28 W0
  
```

Insert: Block Size and Position

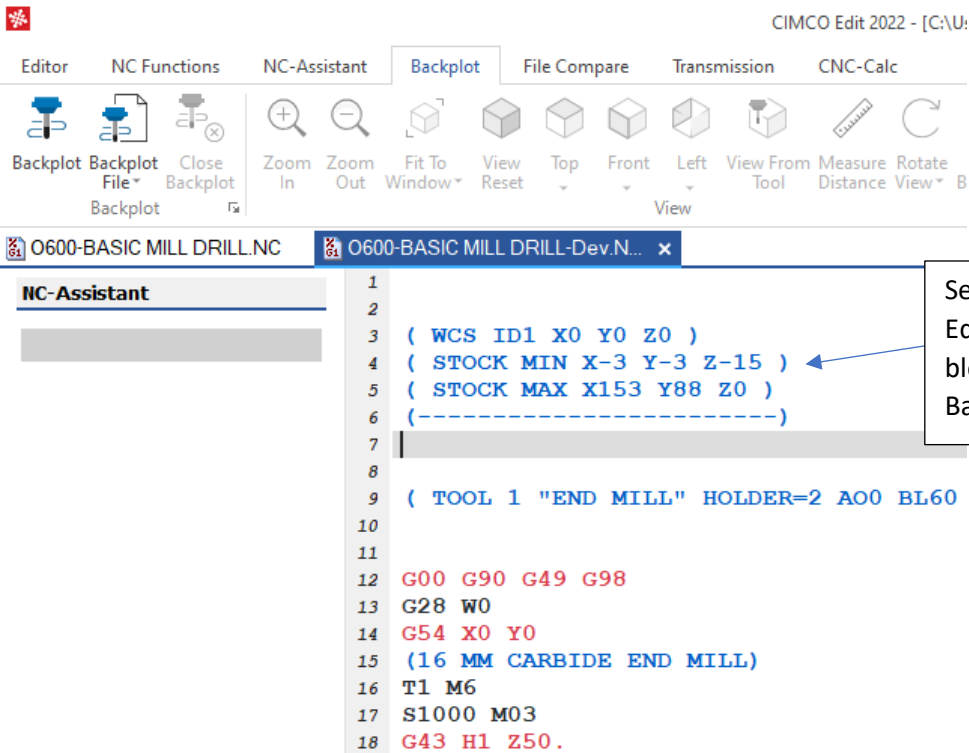
Parameters for 'Block Size and Position'

- X Offset: []
- Y Offset: [0]
- Z Offset: [0]
- X Stock Min: [-3]
- Y Stock Min: [-3]
- Z Stock Min: [-15]
- X Stock Max: [153]
- Y Stock Max: [88]
- Z Stock Max: [0]

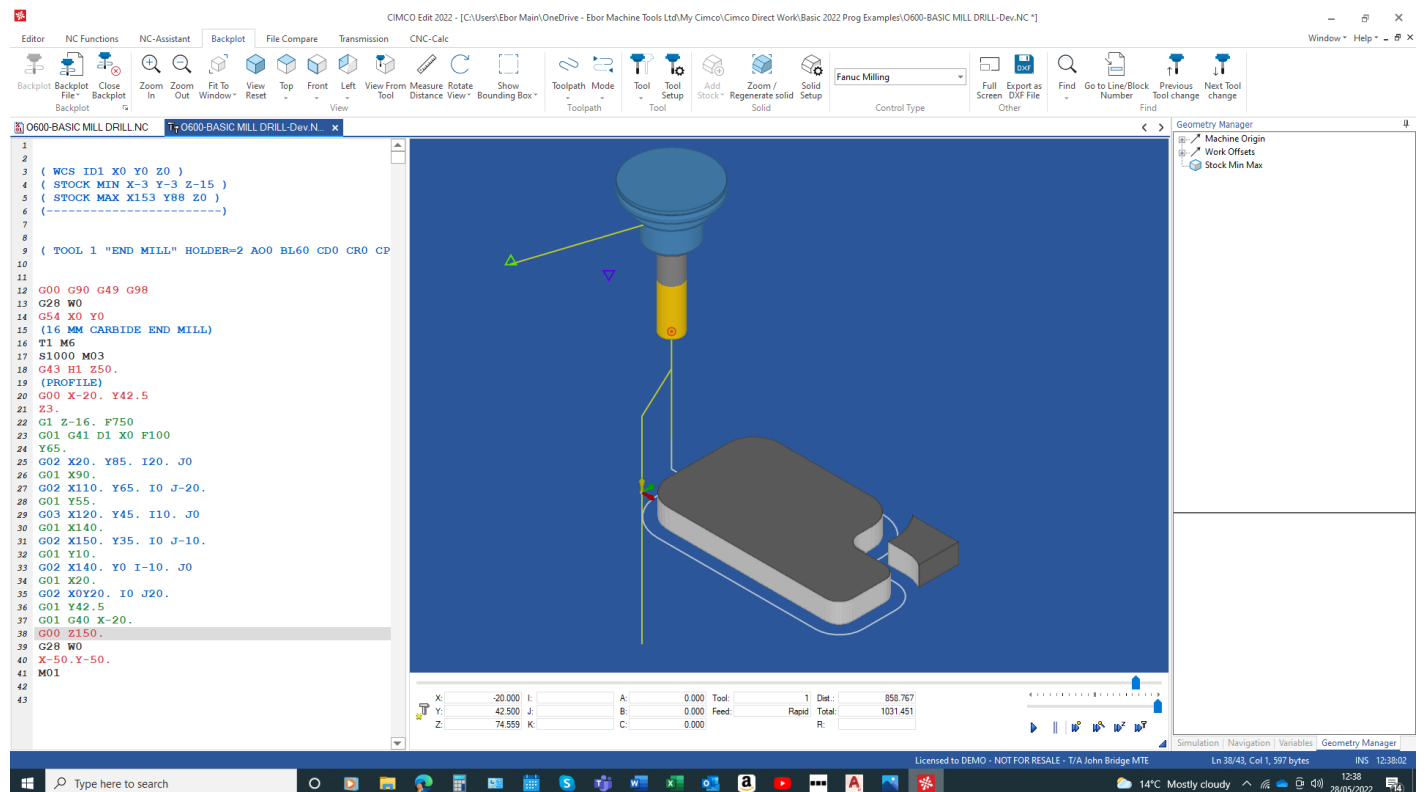
* = Optional parameter

Default Cancel OK

Double click the Stock Size and Position and complete the details of max and min sizes. Position the cursor at the top of the program



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



les / Macros

- Linear interpolation (cutting)
- 1 Circular thread cutting B C
- 2 Involute interpolation CW
- 3 Exponential interpolation (
- 4 3-dimensional coordinate
- Circular interpolation CW or
- 1 Circular thread cutting B C
- 2 Involute interpolation CCV
- 3 Exponential interpolation (
- 4 3-dimensional coordinate
- Circular interpolation CW or
- Dwell
- 1 AI contour control / Nano
- 4 HRV3, 4 on/off
- AI contour control (high-pre
- 2 NURBS interpolation

```

34 G01 X20.
35 G02 X0Y20. I0 J20.
36 G01 Y42.5
37 G01 G40 X-20.
38 G00 Z150.
39 G28 W0
40 X-50.Y-50.
41 M01
42
43
44 (DRILL 11 HOLES 8 MM DIAMETER)
45 X20. Y20.
46 X40.
47 X60.
48 X80.
49 X100.
50 X120.
51 X140.
52 X80. Y65.
53 X60.
54 X40.
55 X20.
56
57
58
59
60

```

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.

```

32 G01 Y10.
33 G02 X140. Y0 I-10. J0
34 G01 X20.
35 G02 X0Y20. I0 J20.
36 G01 Y42.5
37 G01 G40 X-20.
38 G00 Z150.
39 G28 W0
40 X-50.Y-50.
41 M01
42
43
44 (DRILL 11 HOLES 8 MM DIAMETER)
45
46
47 X20. Y20.
48 X40.
49 X60.
50 X80.
51 X100.
52 X120.
53 X140.
54 X80. Y65.
55 X60.
56 X40.
57 X20.
58 G80
59
60
61
62

```

Cycles / Macros

- G77 Plunge direct sizing/grindin
- G78 Continuous-feed surface gr
- G79 Intermittent-feed surface g
- G80.4 Electronic gear box: sync
- G80.5 Electronic gear box 2 pai
- G80 Canned cycle cancel, Elect
- G81.1 Chopping
- 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

Insert: G81 Fixed cycle (Spot drilling)

Parameters for 'G81 Fixed cycle (Spot drilling)'

X-Axis motion command

Y-Axis motion command

Z-Axis motion command

R-point

Feedrate

Optional parameter

Default

Cancel

OK

Select G81 from NC-Assistant and complete the details

Cycles / Macros

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

```

41 M01
42
43
44 (DRILL 11 HOLES 8 MM DIAMETER)
45
46 G81 X20. Y20. Z-20. R2. F65
47 X20. Y20.
48 X40.
49 X60.
50 X80.
51 X100.
52 X120.
53 X140.
54 X80. Y65.
55 X60.
56 X40.
57 X20.
58 G80
59
60
  
```

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

```

37 G01 G40 X-20.
38 G00 Z150.
39 G28 W0
40 X-50.Y-50.
41 M01
42
43 (DRILL 11 HOLES 8 MM DIAMETER)
44 G00 G90 G49 G98
45 G28 W0
46 G54 X0 Y0
47 (8 MM CARBIDE DRILL)
48 T2 M6
49 S1000 M03
50 G43 H2 Z50.
51 (PROFILE)
52 G00 X-20. Y20.
53 Z50.
54 G81 X20. Y20. Z-20. R2. F65
55 X20. Y20.
56 X40.
57 X60.
58 X80.
59 X100.
60 X120.
61 X140.
62 X80. Y65.
63 X60.
64 X40.
65 X20.
66 G80
67 G00 Z150.
68 G28 W0
69 X-50.Y-50.
70 M01
71
  
```

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.

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

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

NC-Assistant Tab, ISO Milling, Basic Milling Tools

Position the cursor here under the first tool

Basic Milling Tools appears here. Double click Drill and fill in the details. Press OK

Insert: Drill

Parameters for 'Drill'

Tool Number: 2

Body Length: 80

Drill Diameter: 8

Flute Length: 35

Units: UM

Point Angle: 120

* = Optional parameter

DRILL

HOLDER LEN

BODY LEN

FLUTE LEN

POINT ANGLE

DRILL DIA

NOTE! SEE TOOL SETUP FOR MORE DETAILS

Default Cancel OK

Tool 2 is inserted; at this point you can close up the program by removing the empty lines

NC-Assistant

No macro found

Cycles / Macros

Drill

End Mill

Face Mill

Thread Tap

NC-Assistant

1

2

3 (WCS ID1 X0 Y0 Z0)

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

5 (STOCK MAX X153 Y88 Z0)

6 (-----)

7 (TOOL 1 "END MILL" HOLDER=2 AO0 BL60 CD0 CR0 CPI=T8 EMCT=FEM D16 FL35 US=UM AD16 SD16 SL3

8

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

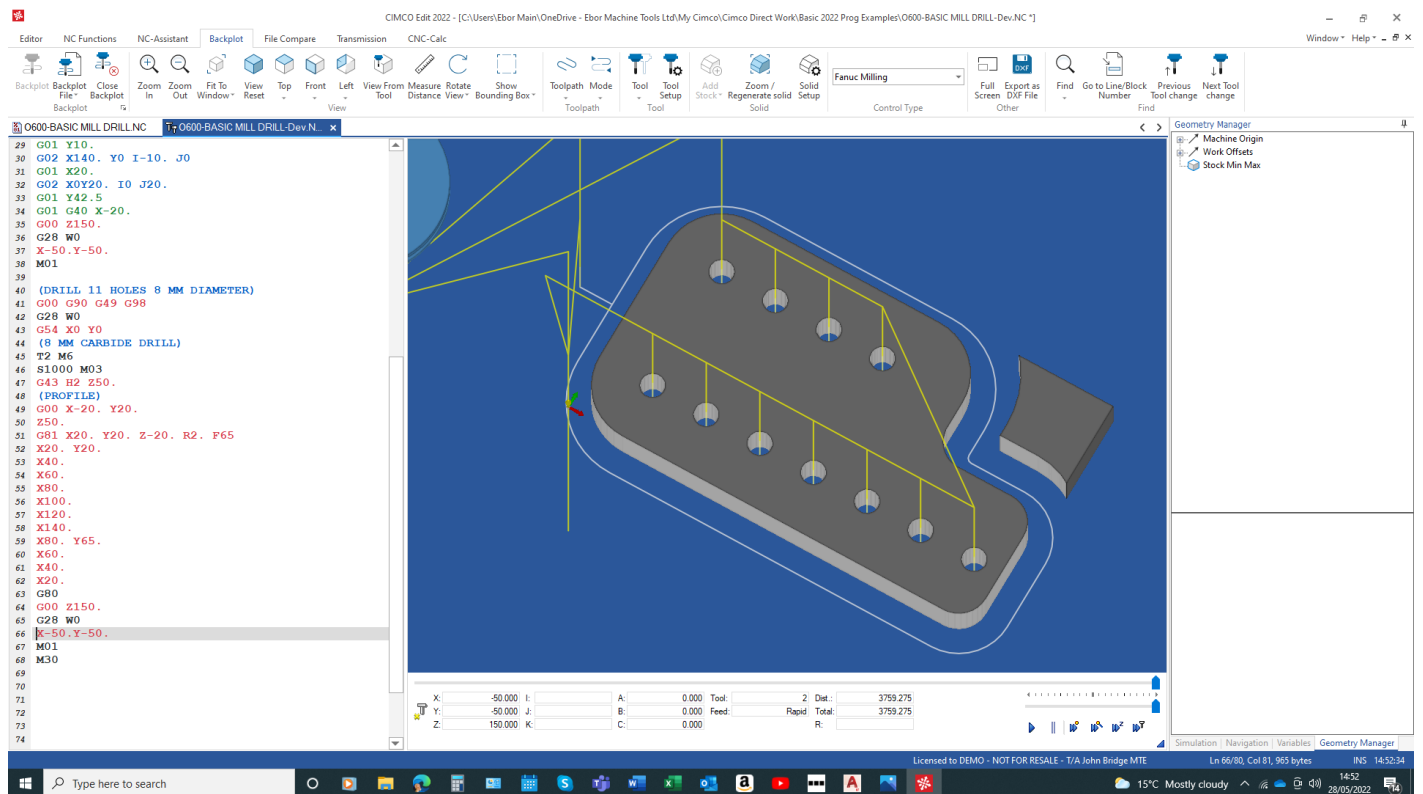
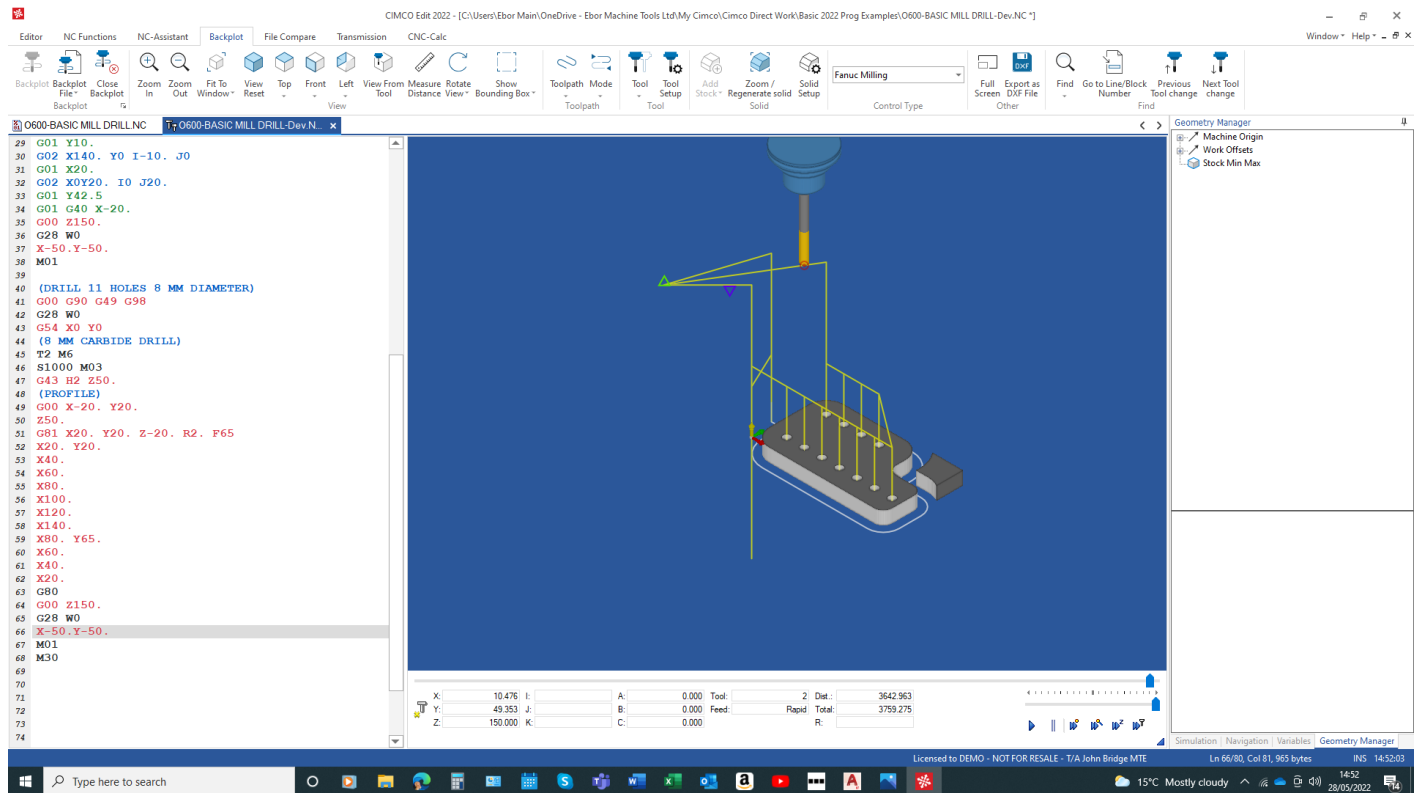
10

11 G00 G90 G49 G98

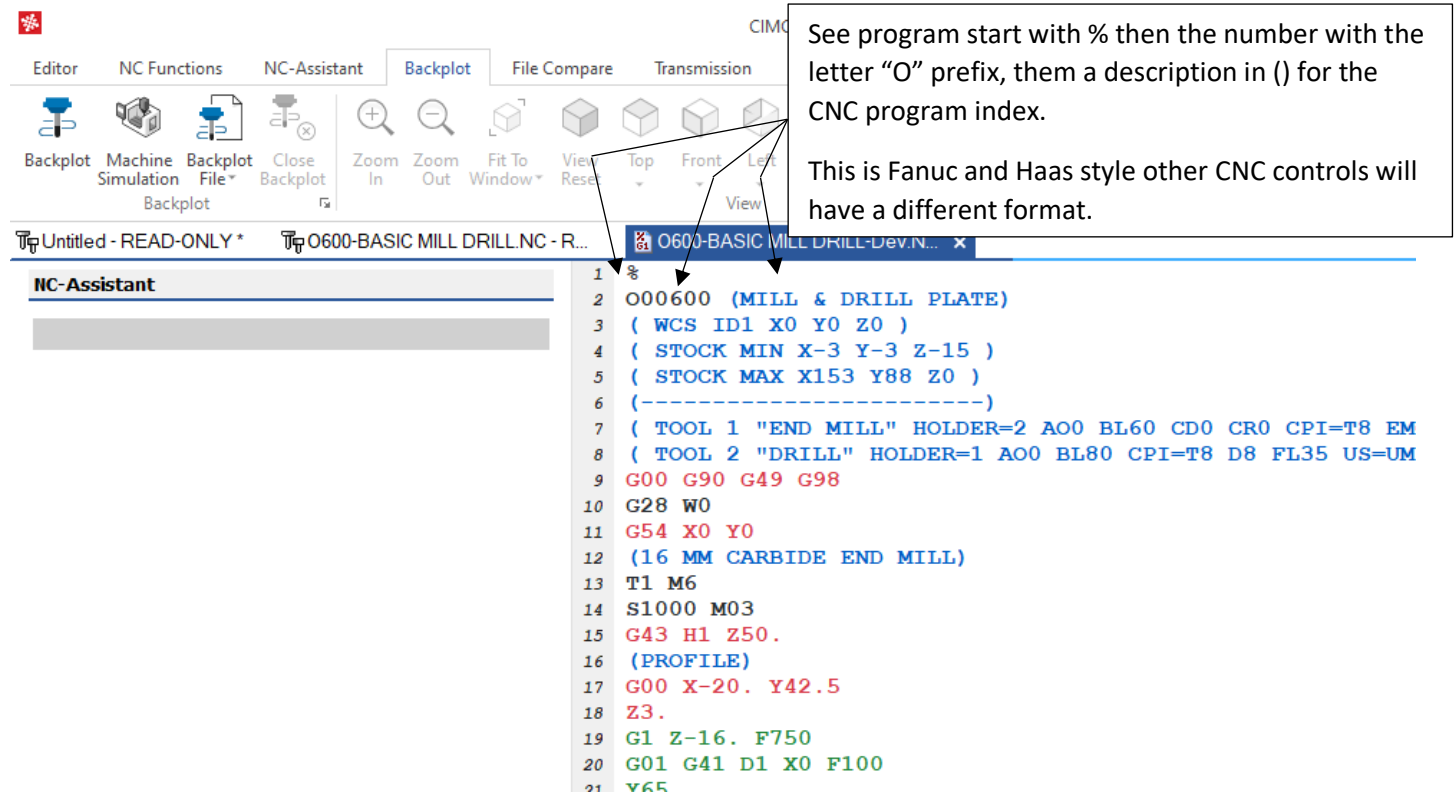
12 G28 W0

13 G54 X0 Y0

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 program start with % then the number with the letter "O" prefix, them a description in () for the CNC program index.
This is Fanuc and Haas style other CNC controls will have a different format.

```

1 %
2 O00600 (MILL & DRILL PLATE)
3 ( WCS ID1 X0 Y0 Z0 )
4 ( STOCK MIN X-3 Y-3 Z-15 )
5 ( STOCK MAX X153 Y88 Z0 )
6 (-----)
7 ( TOOL 1 "END MILL" HOLDER=2 AO0 BL60 CD0 CR0 CPI=T8 EM
8 ( TOOL 2 "DRILL" HOLDER=1 AO0 BL80 CPI=T8 D8 FL35 US=UM
9 G00 G90 G49 G98
10 G28 W0
11 G54 X0 Y0
12 (16 MM CARBIDE END MILL)
13 T1 M6
14 S1000 M03
15 G43 H1 Z50.
16 (PROFILE)
17 G00 X-20. Y42.5
18 Z3.
19 G1 Z-16. F750
20 G01 G41 D1 X0 F100
21 G65
  
```

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
24
  
```


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