EDIT

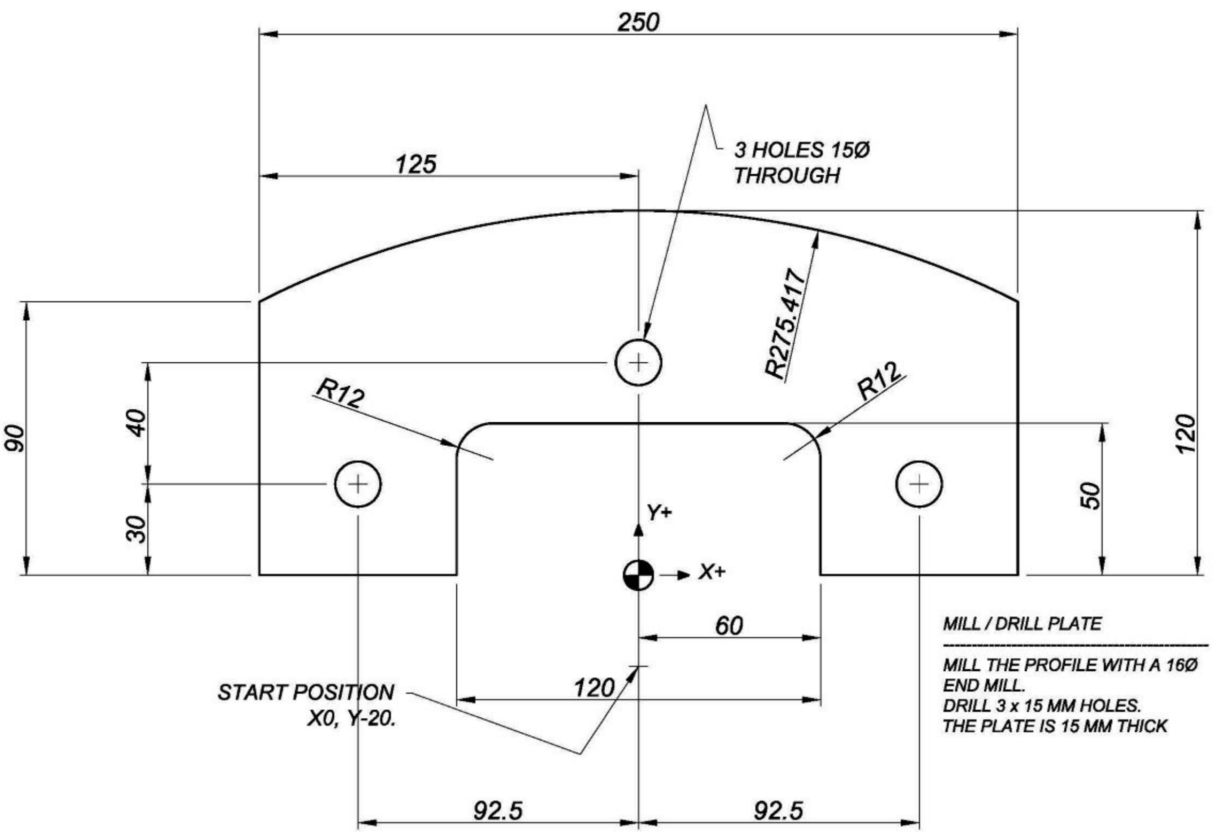
Basic ISO Programming Exercise 2
Milling

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

EXERCISE 2: 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 15 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, 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 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:

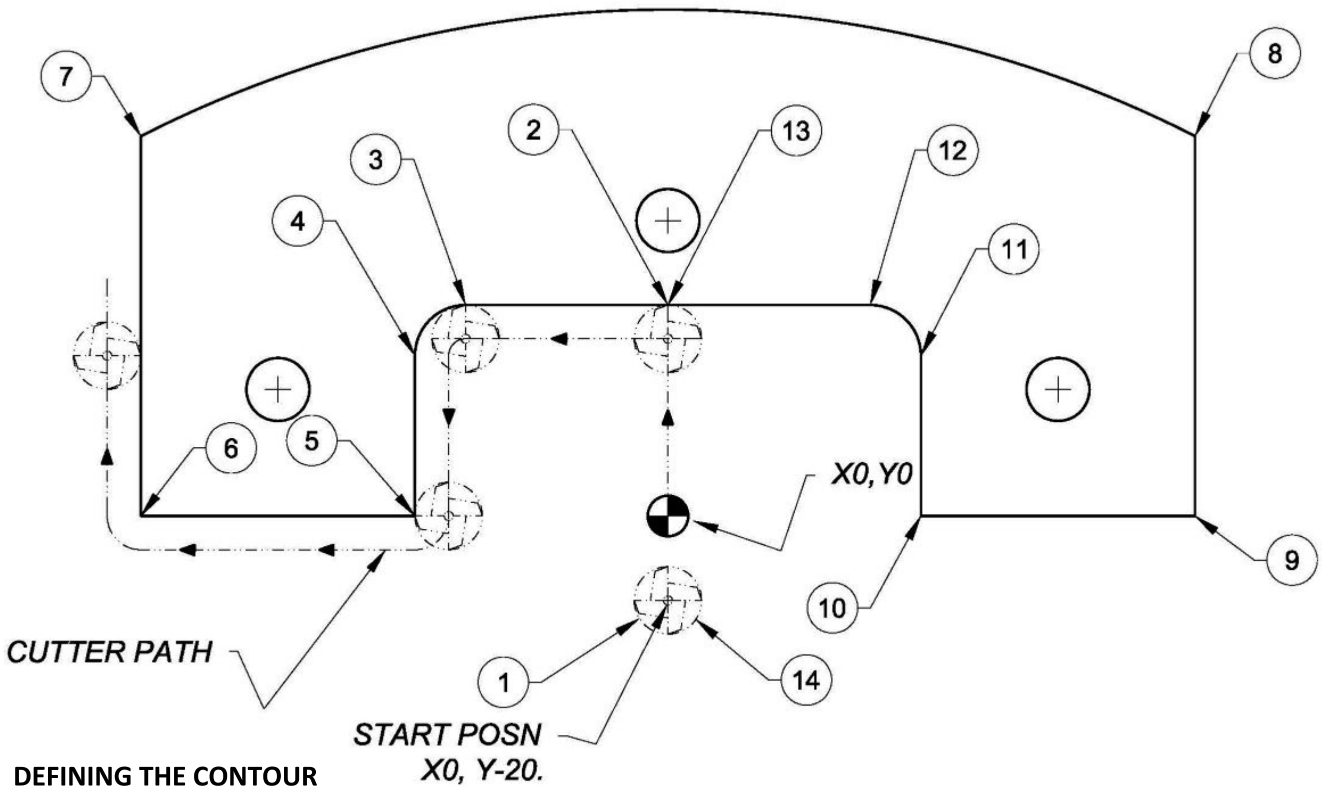
1. G00 – Straight line moves at rapid speed (On some machines this move is made in a vector line and on others a 45-degree move is followed by a single axis move to achieve the final programmed position). This is a modal command.

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 the dimensions will fall in the X+, Y+ quadrant and the X-, Y+ quadrant.



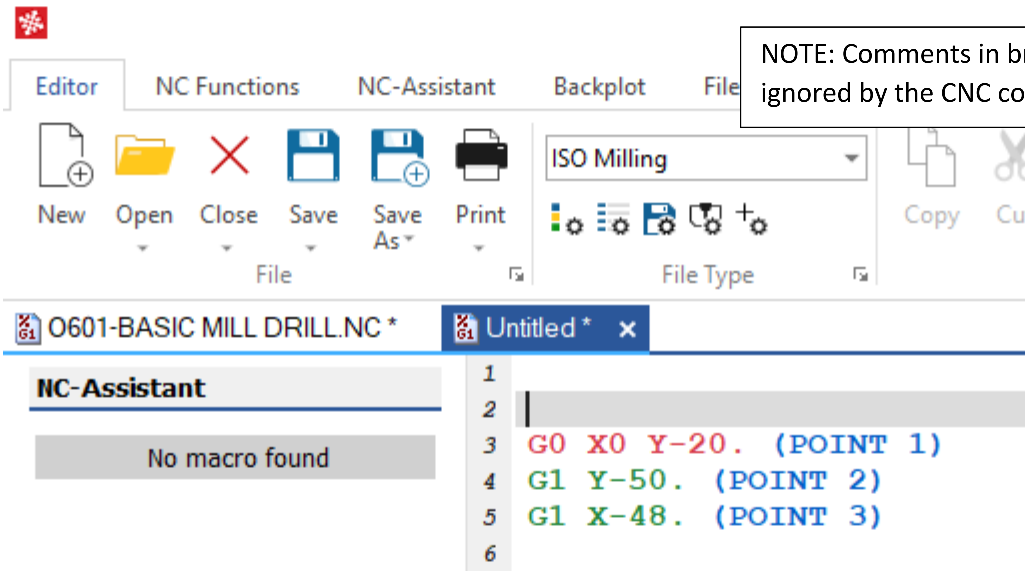
DEFINING THE CONTOUR

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

LINEAR INTERPOLATION

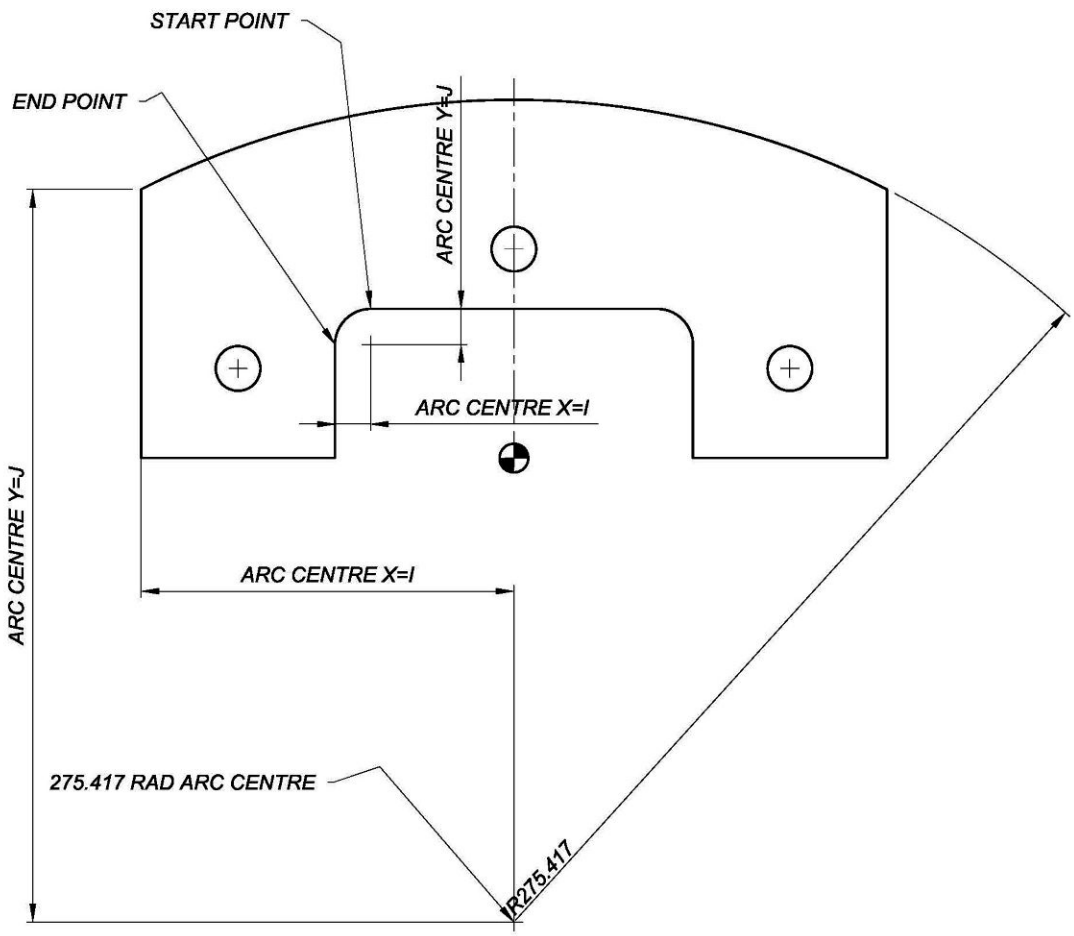
The linear interpolation points are very easy to enter. If we have only one axis command X or Y on a line, then a move in a straight line in that axis will take place. If we have and X and a Y on the same line with a G1 or G01 prefix, then a linear interpolation will take place with both axis moving 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. Enter the first few lines of linear moves, see below:

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 counterclockwise direction the block will start

G03 X-60. Y38.

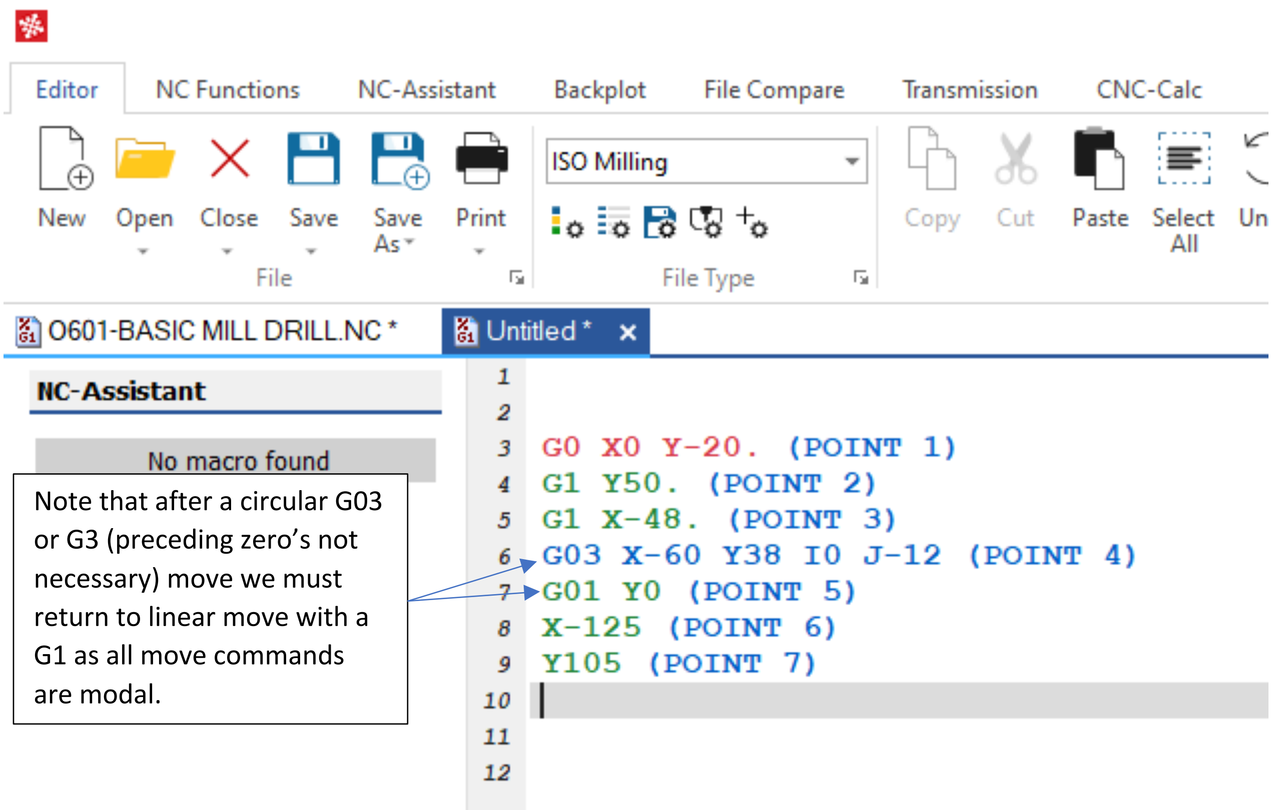
The CNC control must have the arc centre fixed to be able to interpolate this move. The arc centre is fixed generally by its coordinates from the start point. The arc centre coordinates have the designation I for X and J for Y. So, from the start point the I and J are entered as incremental coordinates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

G03 X20. Y85. I0. J-12.

Note that the I dimension is 0 (zero) and remember that arc centers are incremental from the start position and as I = X arc centre then I = 0. Note that J = -12. which is the position of the arc centre on the Y axis from the start position. The CNC Control now has all the information to make this circular move.

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.

Carry on with the coordinate entry in the editor



The next block is for the large radius circular move, it is a clockwise move G2 so we are at the start position at point 7 and going to the radius end position at point 8:

G2 X125. Y90.

From the part drawing and looking at the diagram above the arc centre positions can be seen as:

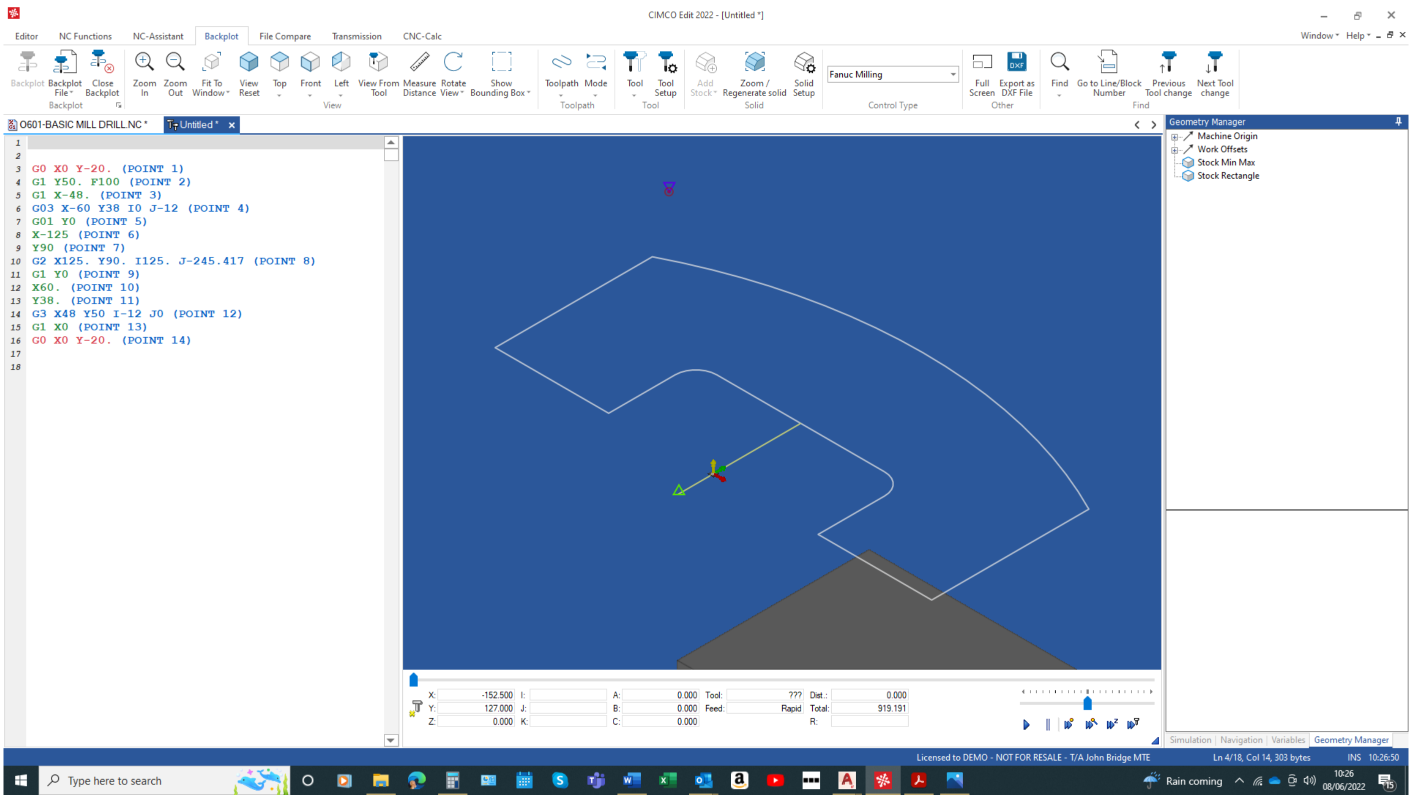
I = 125.

J = 275.417-30. (Being the difference between 120-90 dimensions, so the block completed will look like this:

G2 X125. Y90. I125. J245.417

Continue entering the coordinates and there is one more circular interpolation between points 11 and 12. When you have completed the profile up to point 14, we can test using the Backplot facility. To make the Backplot work add a feed rate to the first G1 block. Remember feed rate is modal so all the move blocks will assume the feed rate of the last feed entered.

When you have all blocks correct the profile will look like this below



A 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 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.

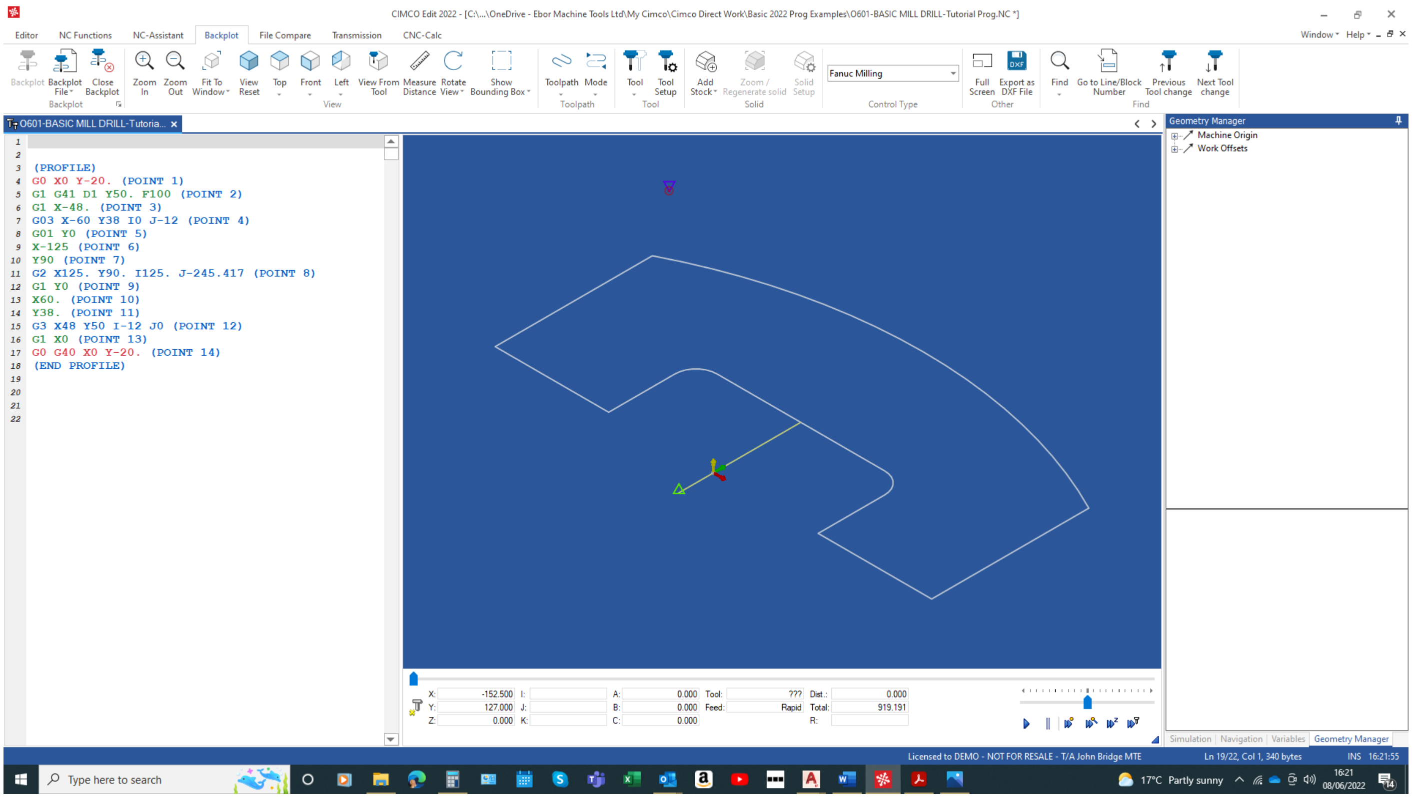
Programming good practice is to layout a block in order. This will help you see the problems when snagging a program for bugs. e.g.

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, check your profile if you are having problems?

G0 X0 Y-20. (POINT 1)

G1 Y50. F100 (POINT 2)

G1 X-48. (POINT 3)

G03 X-60 Y38 I0 J-12 (POINT 4)

G01 Y0 (POINT 5)

X-125 (POINT 6)

Y90 (POINT 7)

G2 X125. Y90. I125. J-245.417 (POINT 8)

G1 Y0 (POINT 9)

X60. (POINT 10)

Y38. (POINT 11)

G3 X48 Y50 I-12 J0 (POINT 12)

G1 X0 (POINT 13)

G0 X0 Y-20. (POINT 14)

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/mim (Haas)

G28 W0 ;Send Z axis up to home. W is the incremental command for Z (on some machine it may be Z0)

G28 return to home position

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

(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

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

G00 G54 X0 Y-20 ; move in X and Y axis at rapid to the programming start position

G54 is the distance set in the work shift table from the machine zero to the programming zero point.

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 set in the table, Z50. Move the tool end point to 50 mm above the Z0 position.

G00 X-30. Y42.5 ; position the tool in a safe X, Y position to start the machining.

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.

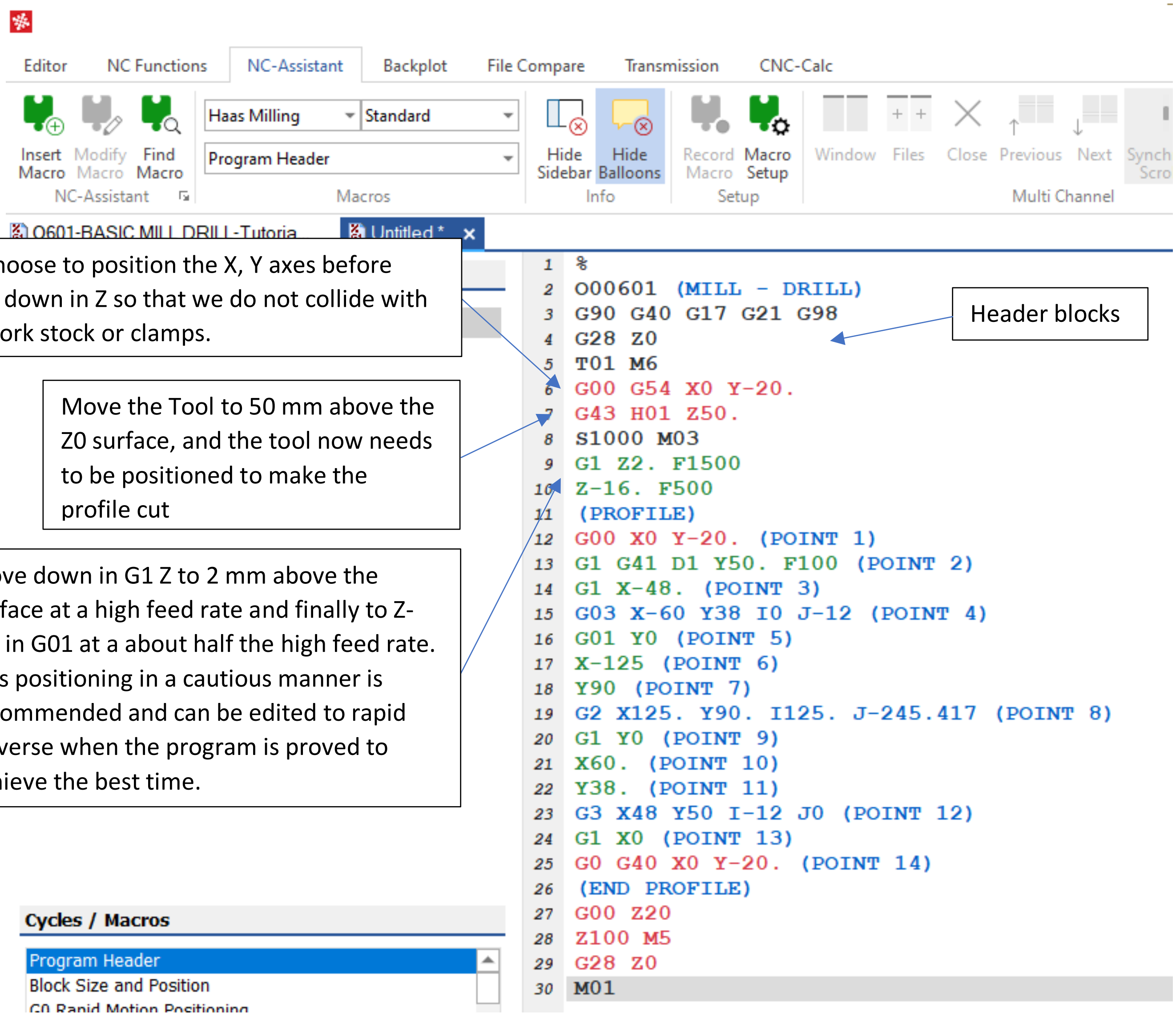
(PROFILE) ; comment to mark the commencement of the profile

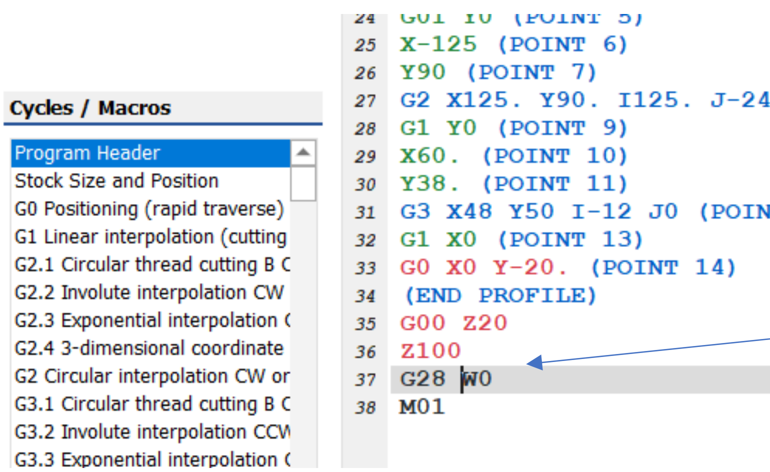
We choose to position the X, Y axes before going down in Z so that we do not collide with the work stock or clamps.

Move the Tool to 50 mm above the Z0 surface, and the tool now needs to be positioned to make the profile cut

Move down in G1 Z to 2 mm above the surface at a high feed rate and finally to Z-16. in G01 at a about half the high feed rate. This positioning in a cautious manner is recommended and can be edited to rapid traverse when the program is proved to achieve the best time.

Header blocks



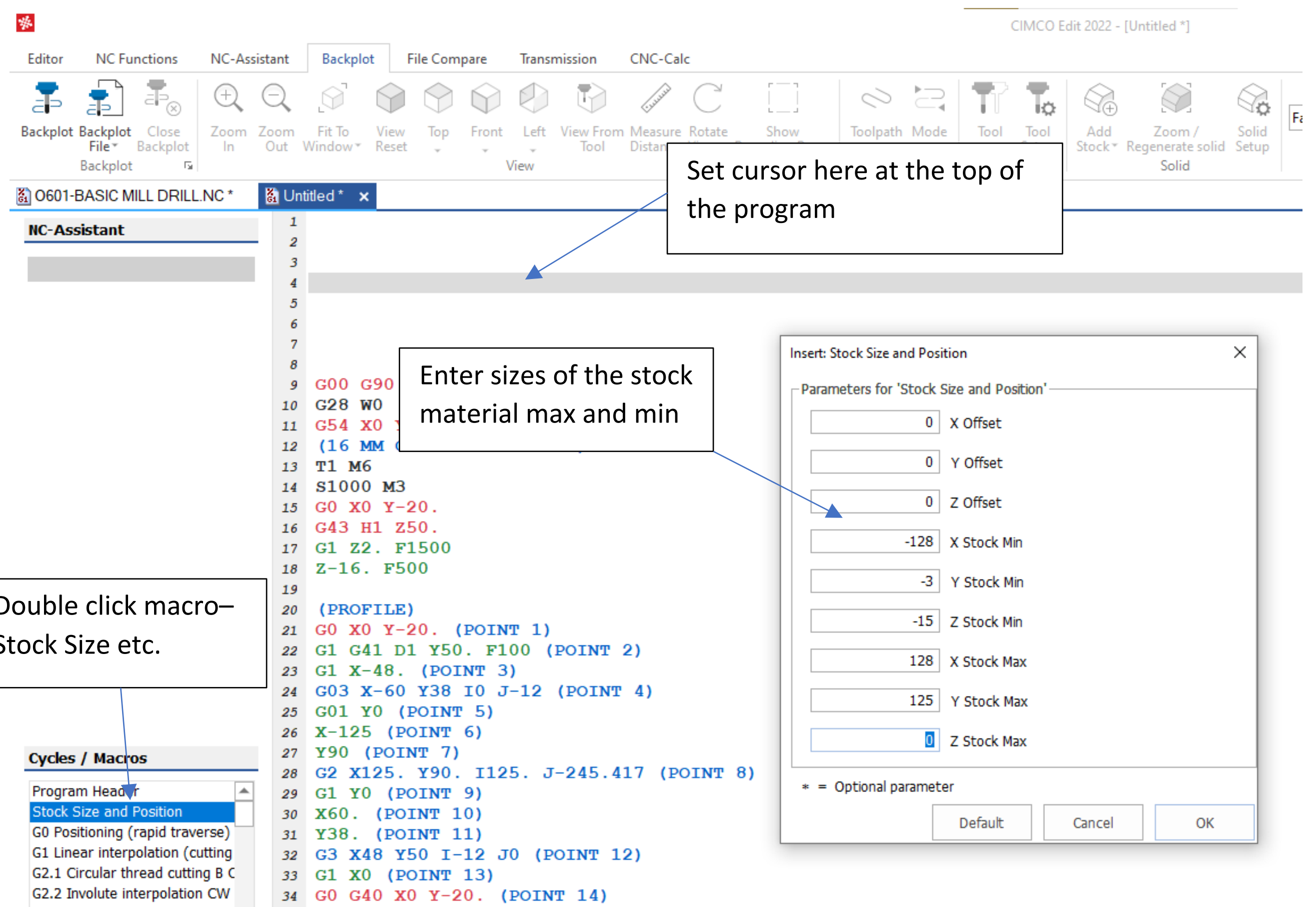


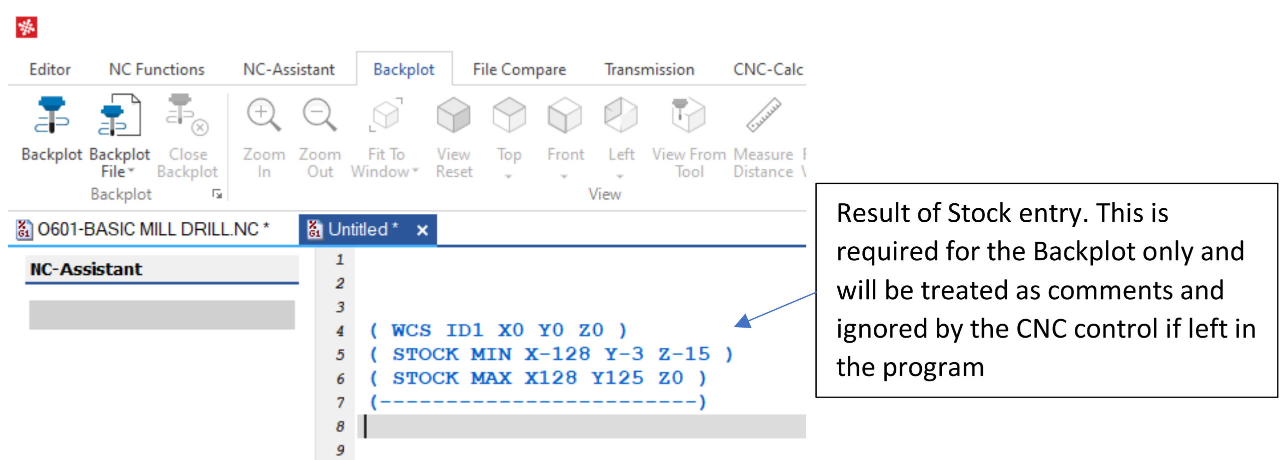
Add a few blocks to end the use of this tool. Move the tool up to Z20. Then up to home position with G28 W0, then M01 option stop, to stop the program to check the part if the option switch is set.

SET UP A TOOL AND STOCK DIMENSIONS

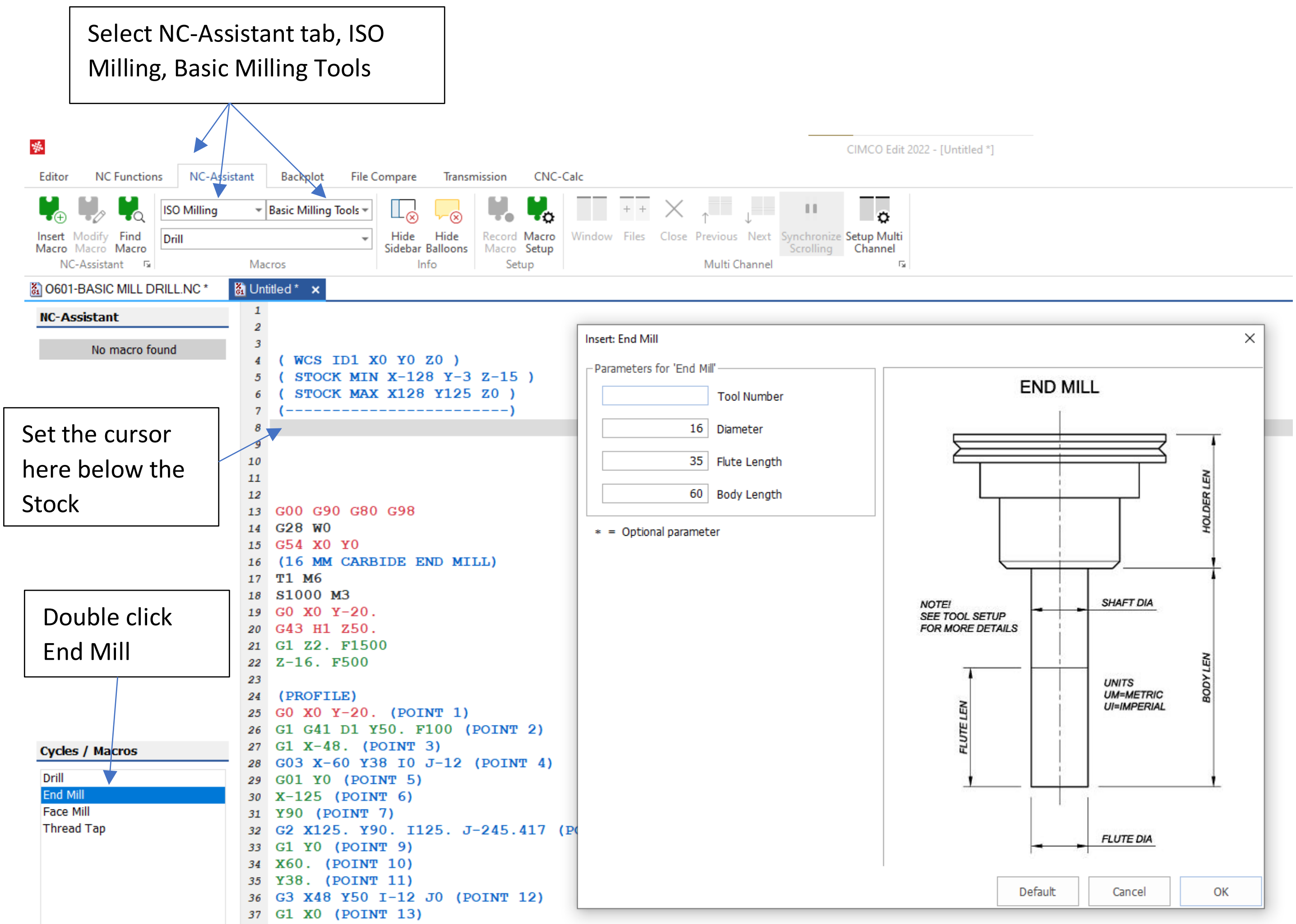
At this point we will set up a tool to enable the Backplot to show the cutter path is accurately and how it would operate on the CNC machine when we apply Tool Radius Compensation. We need a tool set up for this. We also need to see the block of material before machining. Then Backplot shows thew metal removal to our program commands.

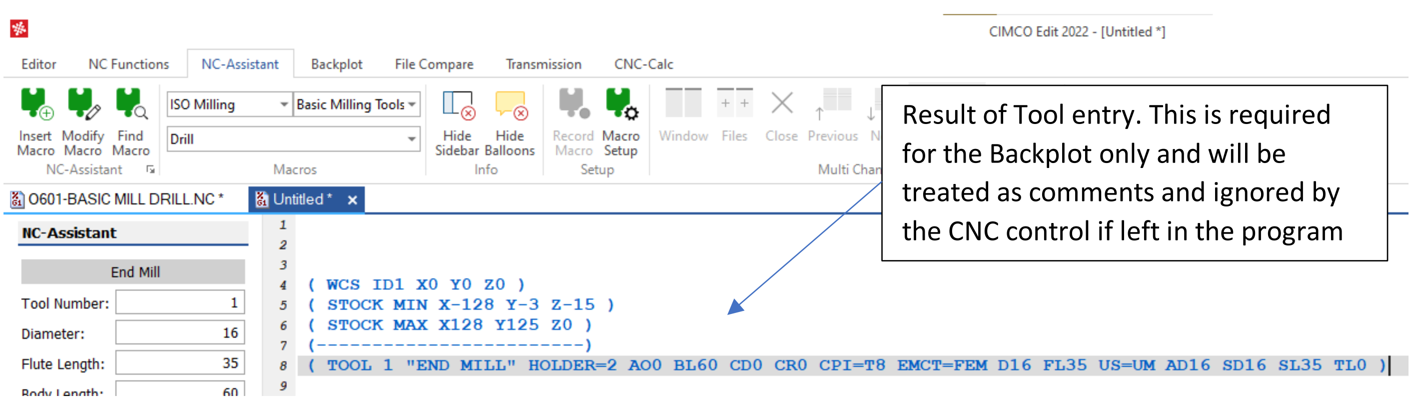
STOCK SETUP



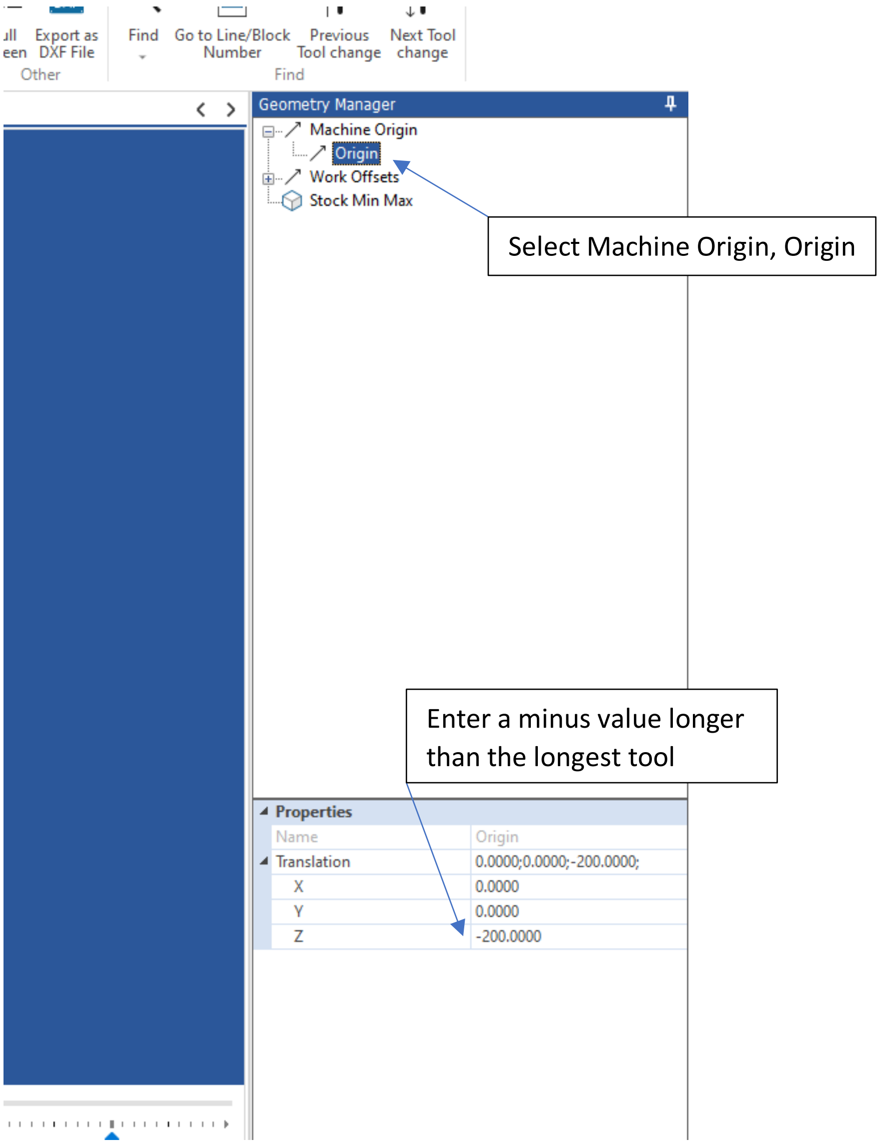


TOOL SETUP

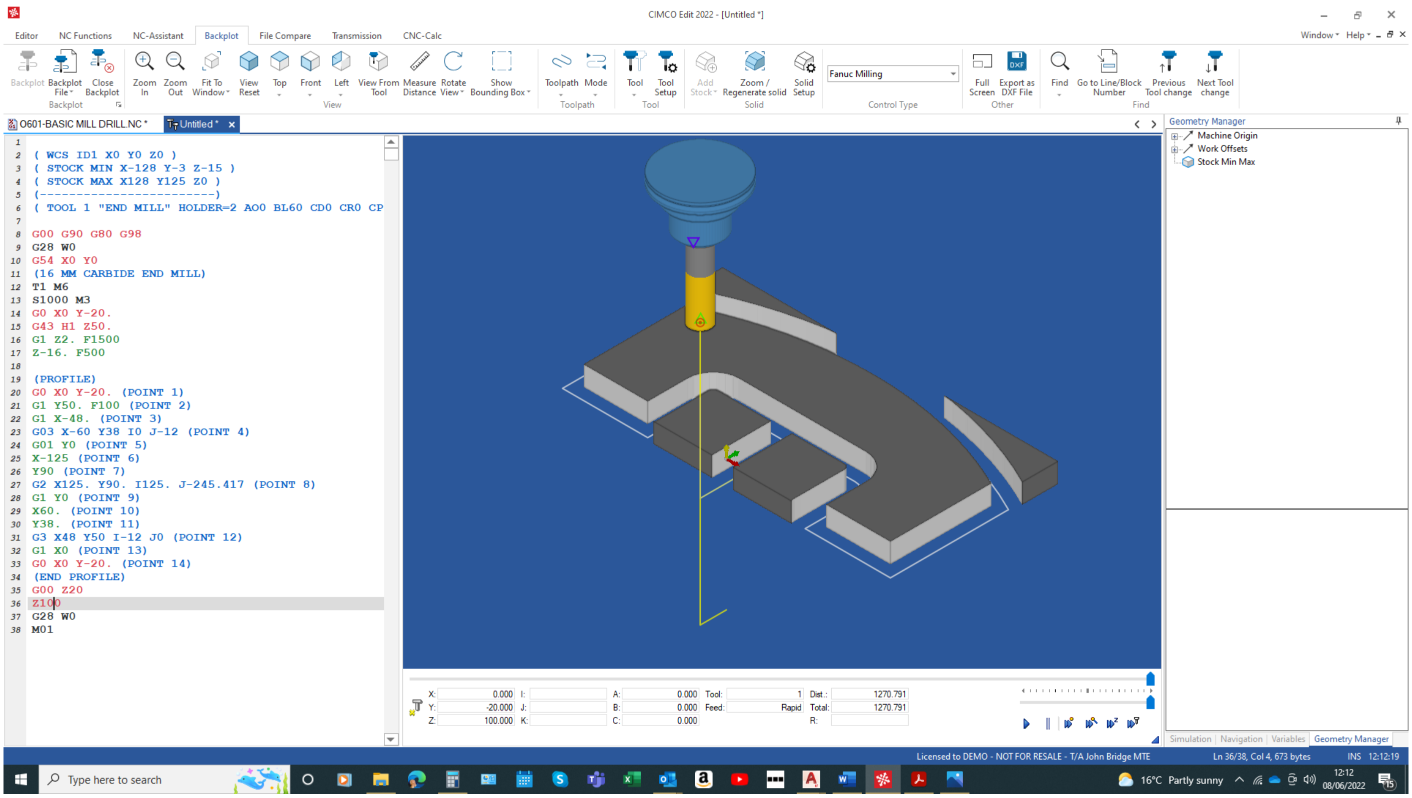

Select NC-Assistant tab, ISO Milling, Basic Milling Tools
Set the cursor here below the Stock
Double click End Mill
CIMCO Edit 2022 - [Untitled *]
Editor
NC Functions
NC-Assistant
Backplot
File Compare
Transmission
CNC-Calc
ISO Milling
Basic Milling Tools
Drill
Cycles / Macros
Drill
End Mill
Face Mill
Thread Tap
No macro found
(WCS ID1 X0 Y0 Z0)
(STOCK MIN X-128 Y-3 Z-15)
(STOCK MAX X128 Y125 Z0)
G00 G90 G80 G98
G28 W0
G54 X0 Y0
(16 MM CARBIDE END MILL)
T1 M6
S1000 M3
Insert: End Mill
Parameters for 'End Mill'
Tool Number
16 Diameter
35 Flute Length
60 Body Length
* = Optional parameter
END MILL
HOLDER LEN
SHAFT DIA
NOTE! SEE TOOL SETUP FOR MORE DETAILS
UNITS UM=METRIC UI=IMPERIAL
BODY LEN
FLUTE LEN
FLUTE DIA
Default
Cancel
OK


Result of Tool entry. This is required for the Backplot only and will be treated as comments and ignored by the CNC control if left in the program
CIMCO Edit 2022 - [Untitled *]
End Mill
Tool Number: 1
Diameter: 16
Flute Length: 35
Body Length: 60
(WCS ID1 X0 Y0 Z0)
(STOCK MIN X-128 Y-3 Z-15)
(STOCK MAX X128 Y125 Z0)
(TOOL 1 "END MILL" HOLDER=2 AO0 BL60 CD0 CR0 CPI=T8 EMCT=FEM D16 FL35 US=UM AD16 SD16 SL35 TL0)

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.

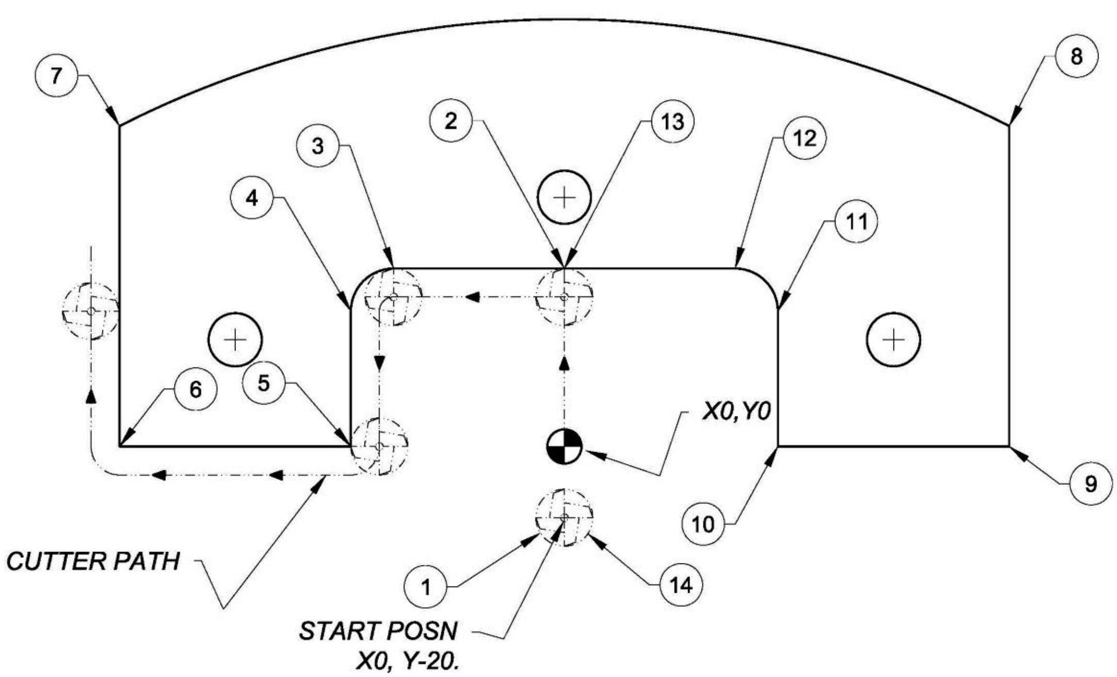


If we Backplot now we will see that the Stock block is shown, and the tool is shown but the path is not compensated, and the part will be too small as the tool is following the centre path.



CUTTER RADIUS COMPENSATION

From the diagram below it can be seen that the tool path (cutter path) must be offset to make the finished part to the correct size.



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

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 in the block by the CNC control the offset path will be activated within the move and the tool will take up the offset position.

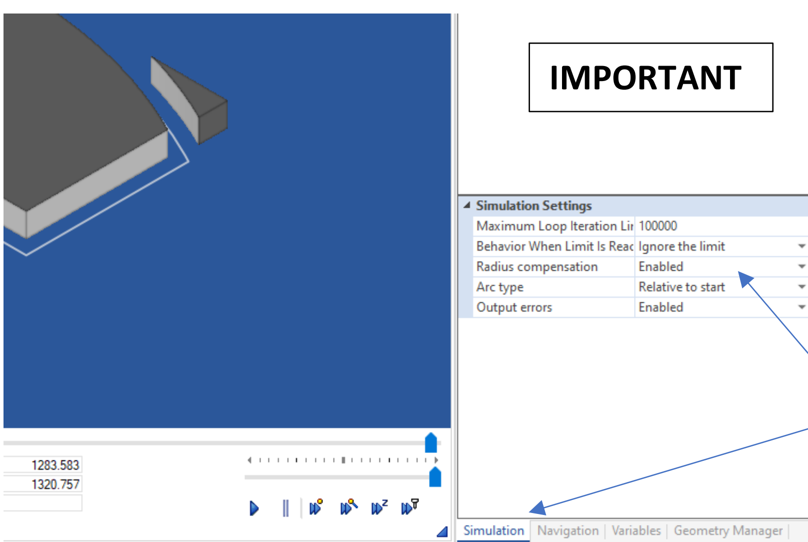
From the drawing we can see the direction of travel around the periphery of the part is counterclockwise. Therefore, we will offset the path to the left in the direction of travel by using G41

See below:

```

(PROFILE)
G0 X0 Y-20. (POINT 1)
G1 G41 D1 Y50. F100 (POINT 2)
G1 X-48. (POINT 3)
G03 X-60 Y38 I0 J-12 (POINT 4)
G01 Y0 (POINT 5)
X-125 (POINT 6)
Y90 (POINT 7)
G2 X125. Y90. I125. J-245.417 (POINT 8)
G1 Y0 (POINT 9)
X60. (POINT 10)
Y38. (POINT 11)
G3 X48 Y50 I-12 J0 (POINT 12)
G1 X0 (POINT 13)
G0 G40 X0 Y-20. (POINT 14)
(END PROFILE)
G00 Z20
Z100
G28 W0
M01
```

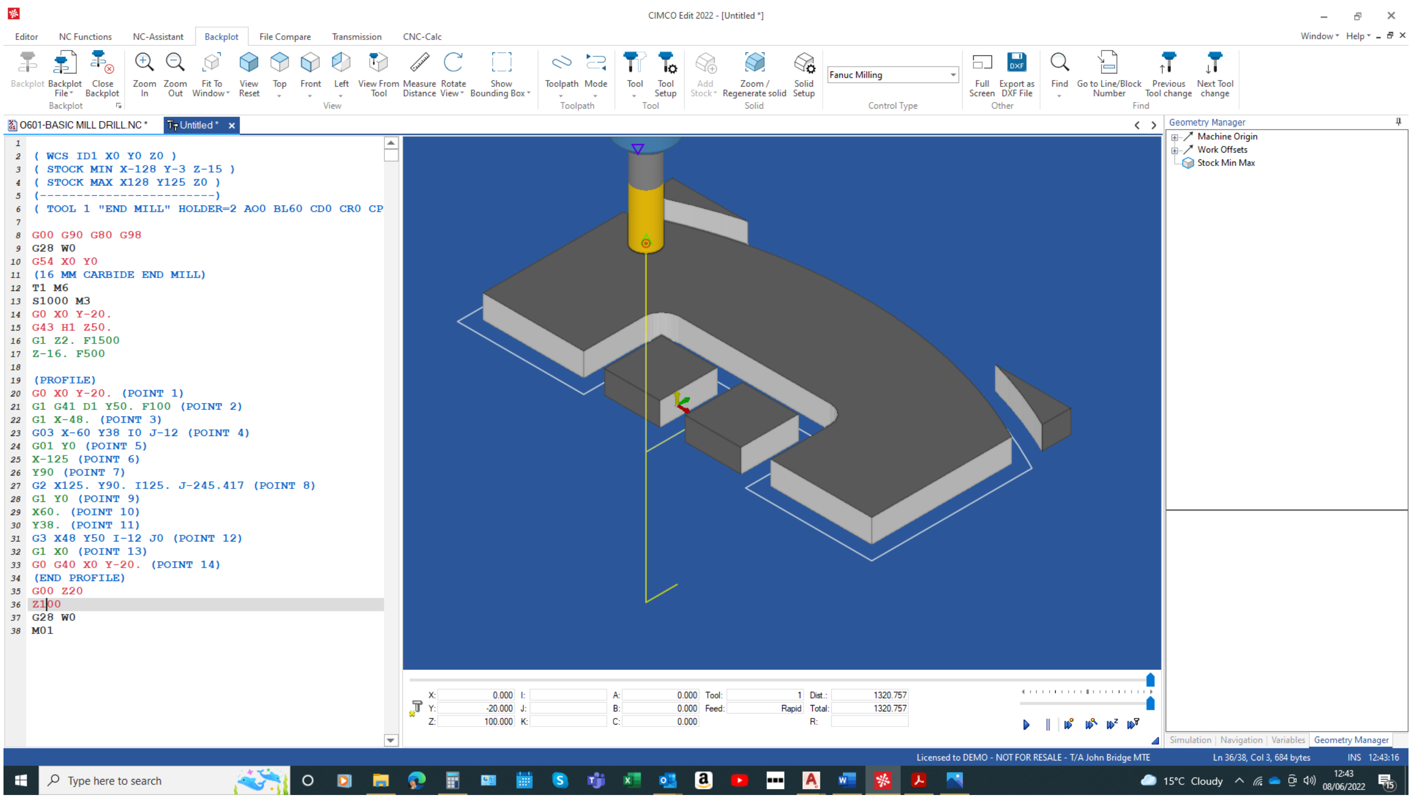
See G41 and D1 added to apply radius compensation

See G40 added to cancel radius compensation



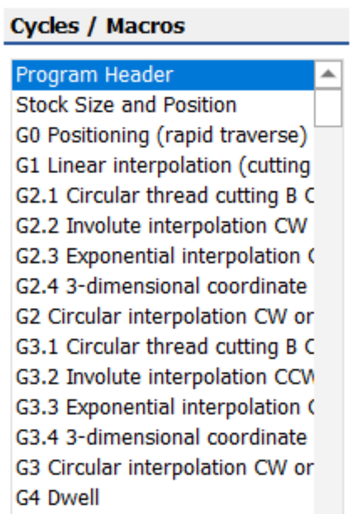
Make sure that Radius Compensation is enabled. Select Simulation tab and enabled.

Backplot is now showing a fully compensated path and we would need to add more lines to the program if we wanted to machine away the excess material shown, and you can add these blocks as an additional exercise if you wish.



DRILLING OPERATION

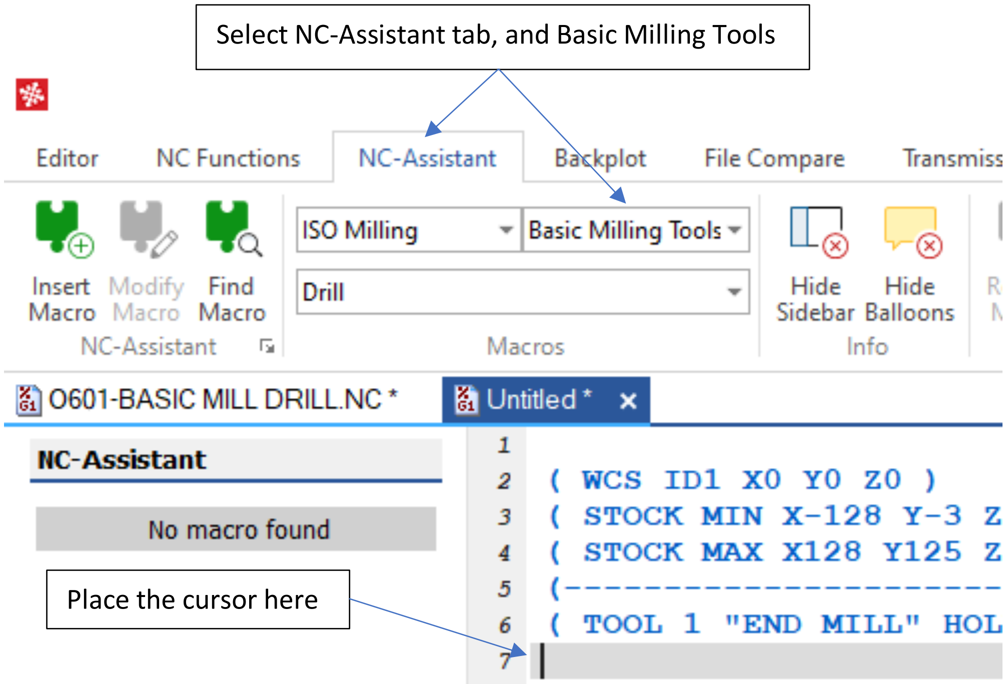
The 15 mm holes can be easily programmed using a canned cycle to that will execute the drilling move every time a new set of coordinated is reached. We will start by copying the Heade for the first operation and editing to suit.

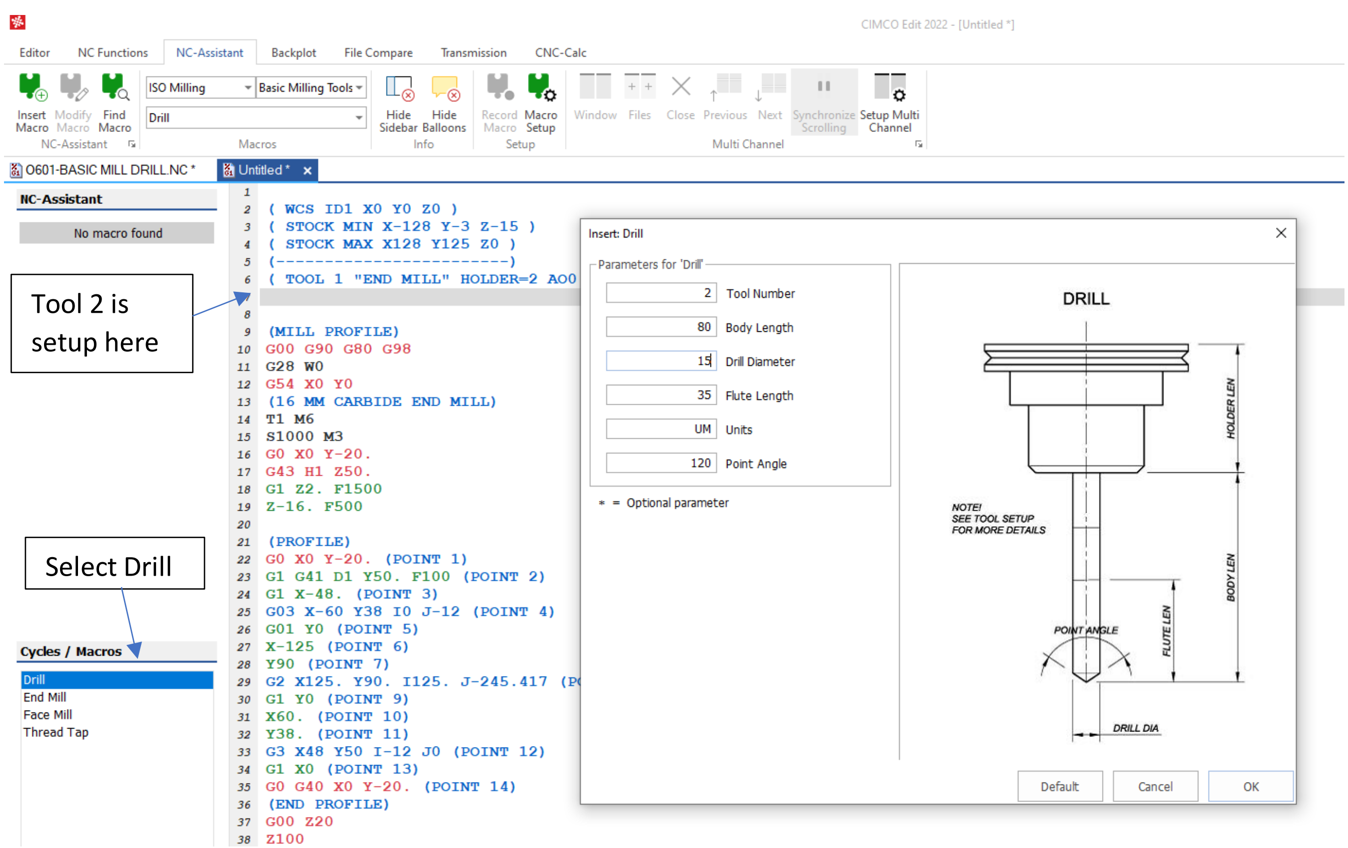



```
G2 X125. Y90. I125. J-245.417 (POINT 8)
G1 Y0 (POINT 9)
X60. (POINT 10)
Y38. (POINT 11)
G3 X48 Y50 I-12 J0 (POINT 12)
G1 X0 (POINT 13)
G0 G40 X0 Y-20. (POINT 14)
(END PROFILE)
G00 Z20
Z100
G28 W0
M01

(DRILL 15 MM HOLES)
G00 G90 G80
G28 W0
G54 X0 Y0
(15 MM CARBIDE DRILL)
T2 M6
S1800 M3
G00 X-92.5 Y30
G43 H1 Z50.
```

New operation comment, new Tool description, New Tool number, new speed, and the coordinates for the first hole, then positioning the drill 50 mm above the part ready to carry out the drilling operation.

We need to set up a new tool for this drilling operation.

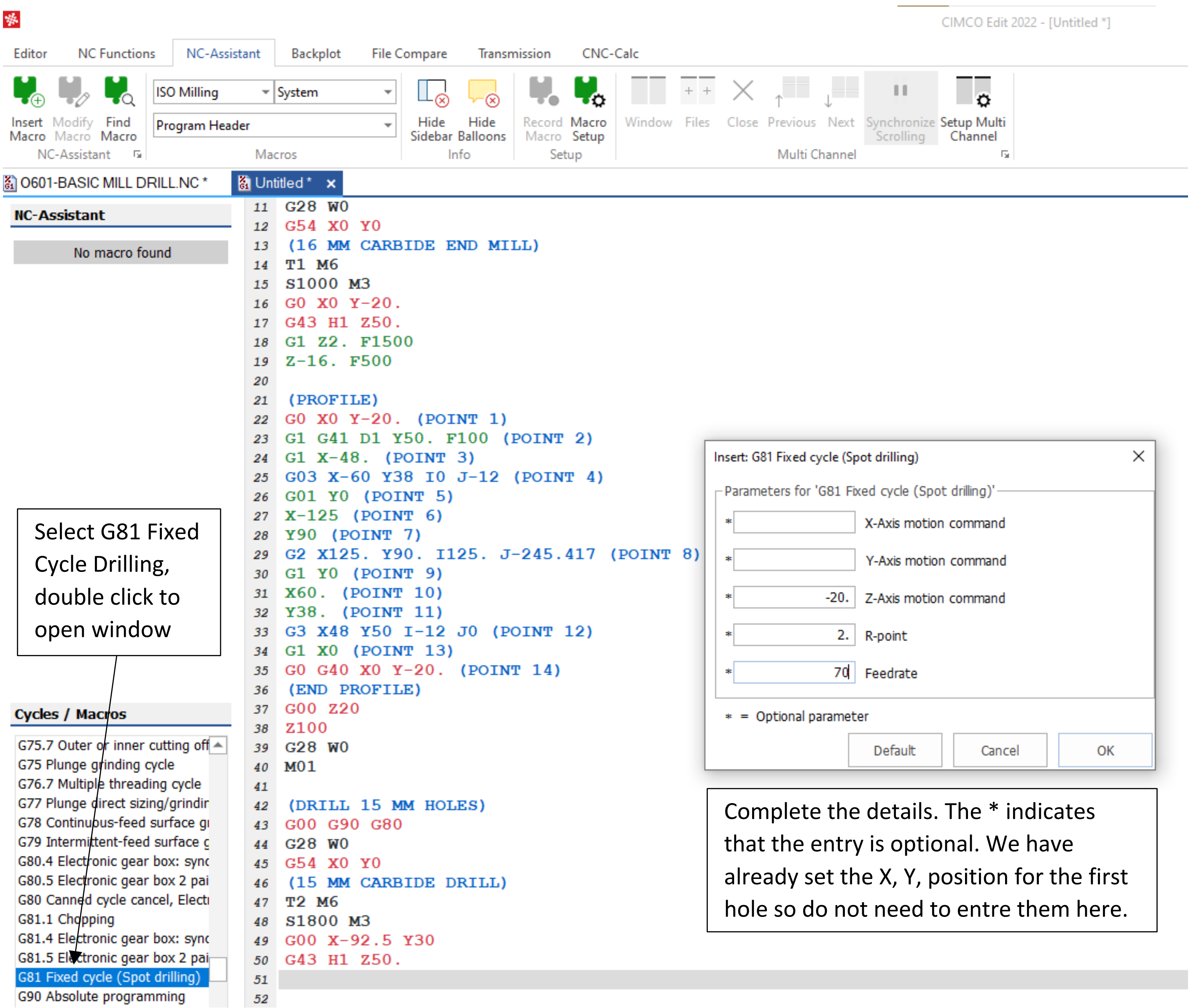




See the result of Tool 2 setup

```
%
O00601 (MILL & DRILL PLATE)
( WCS ID1 X0.000 Y0.000 Z0.000 )
( STOCK MIN X-128 Y-3 Z-15 )
( STOCK MAX X128 Y125 Z0 )
(-----------------------)
( T 1 "END MILL" HOLDER=2 AO0 BL60 CD0 CR0 CPI=T8 EMCT=FEM D16 FL40 US=UM AD16 SD16 SL40 TL0 )
( T 2 "DRILL" HOLDER=2 AO0 BL100 CPI=T8 D15 FL80 US=UM AD15 SD15 SL100 TL0 AT120 )
```

Now we can proceed with the drilling operation by setting up the canned cycle and after completing the other coordinate blocks. 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.



See result below

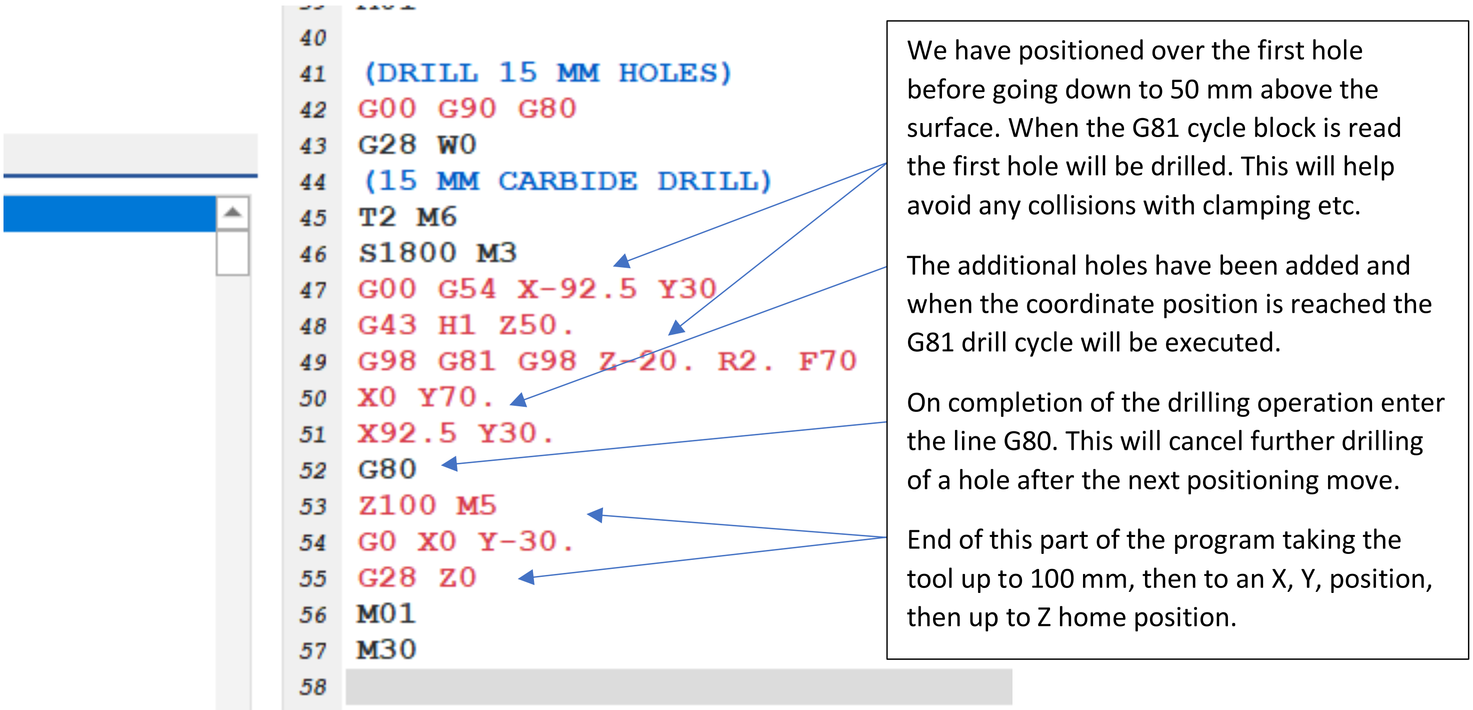
ng cycle
eading cycle
sizing/grindir
eed surface gi
feed surface g
jear box: sync
jear box 2 pai
cancel, Electi
jear box: sync
jear box 2 pai
Spot drilling)
gramming
1e maximum i

```
M01

(DRILL 15 MM HOLES)
G00 G90 G80
G28 W0
G54 X0 Y0
(15 MM CARBIDE DRILL)
T2 M6
S1800 M3
G00 X-92.5 Y30
G43 H1 Z50.
N0 G81 Z-20. R2. F70
```

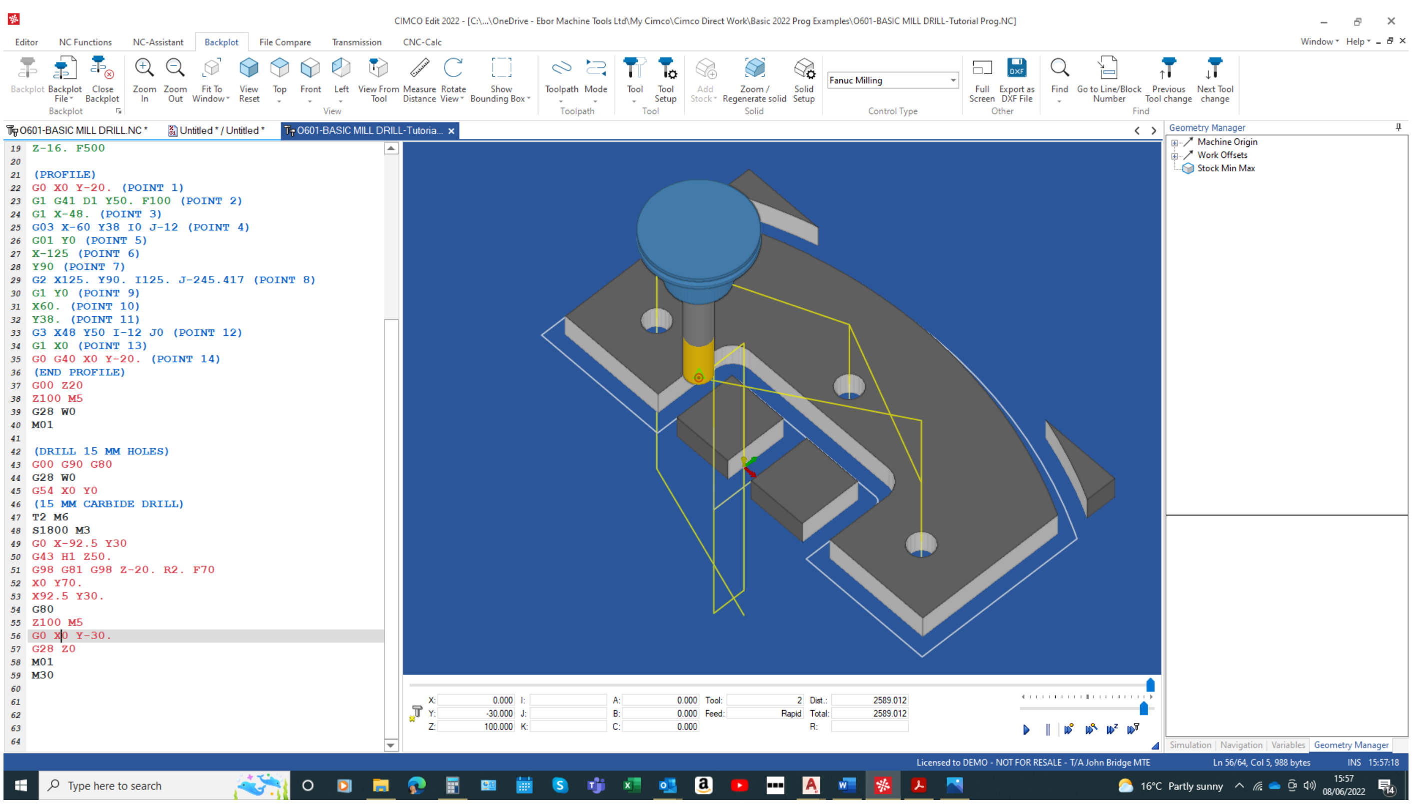
G81 Cycle inserted

The resulting canned cycle is set to rapid down to 2 mm (R) above the drilling surface and then drill 20 mm (Z-) deep and rapid back up to 50 mm above the drilling surface. We do not need the block number so it can be edited out.

Now complete the coordinates for the other two holes.



Now a carry out a full Backplot to test the program.



Here is a full copy of the program for you to check yours against. This program should run on most CNC machines with Fanuc or Haas CNC controls. There may be a small amount of editing to do from machine-to-machine dependent on the Machine Tool Builders application of the CNC control. The edits will apply generally to the header and trailer format and syntax.

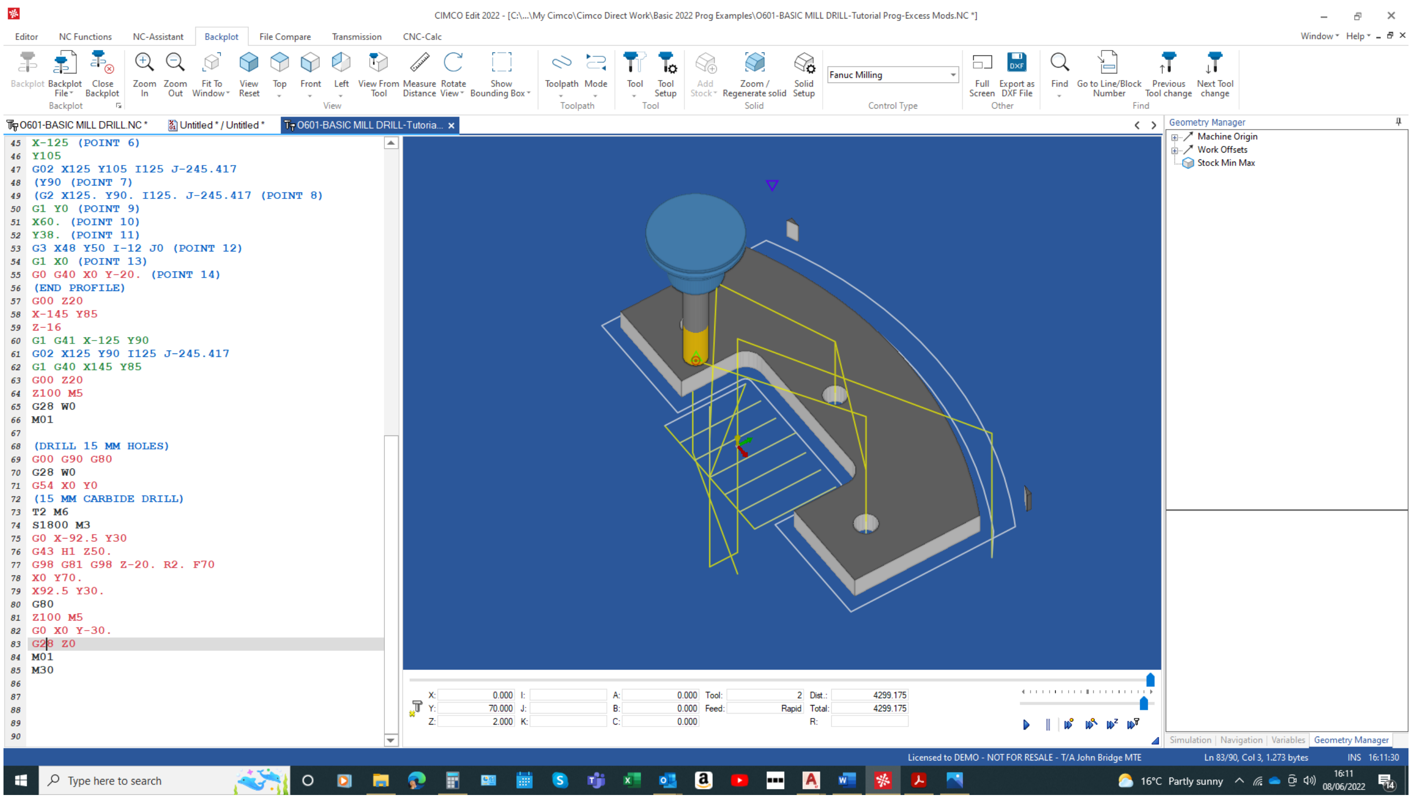
```
%
O00601 (MILL & DRILL PLATE)
( WCS ID1 X0.000 Y0.000 Z0.000 )
( STOCK MIN X-128 Y-3 Z-15 )
( STOCK MAX X128 Y125 Z0 )
(------------------------)
( T 1 "END MILL" HOLDER=2 AO0 BL60 CD0 CR0 CPI=T8 EMCT=FEM D16 FL40 US=UM AD16 SD16 SL40 TL0 )
( T 2 "DRILL" HOLDER=2 AO0 BL100 CPI=T8 D15 FL80 US=UM AD15 SD15 SL100 TL0 AT120 )

(MILL PROFILE)
G00 G90 G80 G98
G28 W0
(16 MM CARBIDE END MILL)
T1 M6
S1000 M3
G0 G54 X0 Y-20.
G43 H1 Z50.
G1 Z2. F1500
Z-16. F500
(PROFILE)
G00 X0 Y-20. (POINT 1)
G1 G41 D1 Y50. F100 (POINT 2)
G1 X-48. (POINT 3)
G03 X-60 Y38 I0 J-12 (POINT 4)
G01 Y0 (POINT 5)
X-125 (POINT 6)
```

```
Y90 (POINT 7)

G2 X125. Y90. I125. J-245.417 (POINT 8)

G1 Y0 (POINT 9)

X60. (POINT 10)

Y38. (POINT 11)

G3 X48 Y50 I-12 J0 (POINT 12)

G1 X0 (POINT 13)

G0 G40 X0 Y-20. (POINT 14)

(END PROFILE)

G00 Z20

Z100 M5

G28 Z0

M01


(DRILL 15 MM HOLES)

G00 G90 G80

G28 W0

(15 MM CARBIDE DRILL)

T2 M6

S1800 M3

G00 G54 X-92.5 Y30

G43 H1 Z50.

G98 G81 G98 Z-20. R2. F70

X0 Y70.

X92.5 Y30.

G80

Z100 M5

G0 X0 Y-30.

G28 Z0

M01
```

M30

ADDITIONAL PROGRAMMING TO CLEAR EXCESS MATERIAL

See Backplot



Copy of program with additional blocks for material clearance.

(WCS ID1 X0 Y0 Z0)

(STOCK MIN X-128 Y-3 Z-15)

(STOCK MAX X128 Y125 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 D15 FL35 US=UM AD15 SD15 SL35 TL0 AT120)

%

O00601 (MILL & DRILL PLATE)

(WCS ID1 X0.000 Y0.000 Z0.000)

```
( STOCK MIN X-128 Y-3 Z-15 )
( STOCK MAX X128 Y125 Z0 )
(------------------------)
( T 1 "END MILL" HOLDER=2 AO0 BL60 CD0 CR0 CPI=T8 EMCT=FEM D16 FL40 US=UM AD16 SD16 SL40 TL0 )
( T 2 "DRILL" HOLDER=2 AO0 BL100 CPI=T8 D15 FL80 US=UM AD15 SD15 SL100 TL0 AT120 )
(MILL PROFILE)
G00 G90 G80 G98
G28 W0
(16 MM CARBIDE END MILL)
T1 M6
S1000 M3
G0 G54 X0 Y-20.
G43 H1 Z50.
G1 Z2. F1500
Z-16. F500
G01 Y38 F100
G0 Y-20
X16
G01 Y38 F100
G0 Y-20
X32
G01 Y38 F100
G0 Y-20
X48
G01 Y38 F100
G0 Y-20
X-16
G01 Y38 F100
G0 Y-20
X-32
```

```
G01 Y38 F100

G0 Y-20

X-48

G01 Y38 F100

(PROFILE)

G0 X0 Y-20. (POINT 1)

G1 G41 D1 Y50. F100 (POINT 2)

G1 X-48. (POINT 3)

G03 X-60 Y38 I0 J-12 (POINT 4)

G01 Y0 (POINT 5)

X-125 (POINT 6)

Y105

G02 X125 Y105 I125 J-245.417

(Y90 (POINT 7)

(G2 X125. Y90. I125. J-245.417 (POINT 8)

G1 Y0 (POINT 9)

X60. (POINT 10)

Y38. (POINT 11)

G3 X48 Y50 I-12 J0 (POINT 12)

G1 X0 (POINT 13)

G0 G40 X0 Y-20. (POINT 14)

(END PROFILE)

G00 Z20

X-145 Y85

Z-16

G1 G41 X-125 Y90

G02 X125 Y90 I125 J-245.417

G1 G40 X145 Y85

G00 Z20

Z100 M5
```

G28 W0

M01

(DRILL 15 MM HOLES)

G00 G90 G80

G28 W0

(15 MM CARBIDE DRILL)

T2 M6

S1800 M3

G00 G54 X-92.5 Y30

G43 H1 Z50.

G98 G81 G98 Z-20. R2. F70

X0 Y70.

X92.5 Y30.

G80

Z100 M5

G0 X0 Y-30.

G28 Z0

M01

M30

(DRILL 15 MM HOLES)

G00 G90 G80

G28 W0

G54 X0 Y0

(15 MM CARBIDE DRILL)

```
T2 M6
S1800 M3
G0 X-92.5 Y30
G43 H1 Z50.
G98 G81 G98 Z-20. R2. F70
X0 Y70.
X92.5 Y30.
G80
Z100 M5
G0 X0 Y-30.
G28 Z0
M01
M30
```

End of Tutorial.