EDIT

Mill/turn with C-axis programming sections

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

INTRODUCTION

The aim of this tutorial is to familiarise the trainee on the CNC programming of turned and milled parts using C axis in conjunction with X and Z axis to create turned parts with milled contours and drilled holes on the end face and on the periphery of parts on a CNC Lathe with C axis and live tool facilities. The trainee can follow the steps to program and evaluate the program using the graphic Backplot facilities within the CIMCO 2022 Editor.

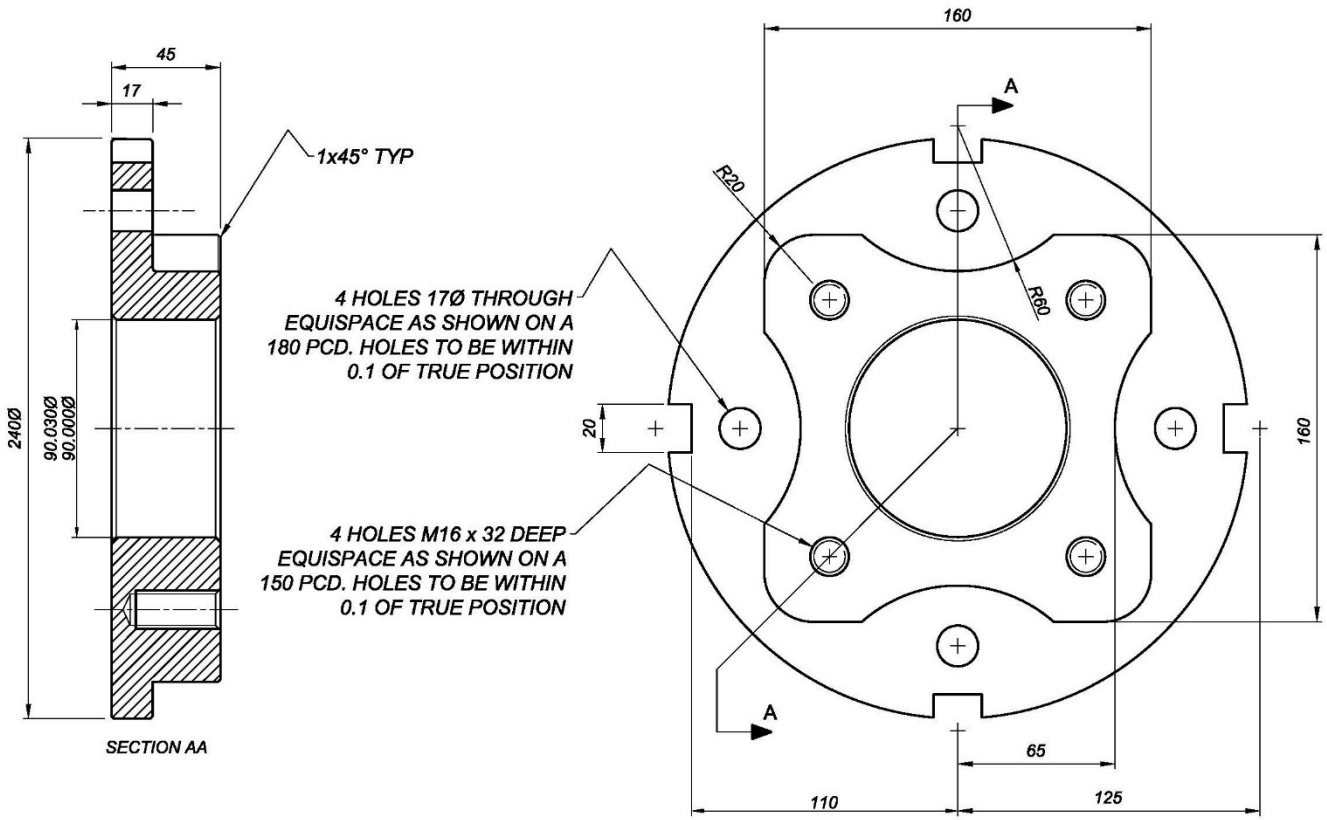
The original stock material can be set in the CNC program to record the size and position of the stock. Here we will guide the trainee through the procedure to setup the stock size and position.

The correct shape of tool to perform the specific operation can be setup in Tool Setup. To achieve an accurate animated Backplot, the trainee will be taken through the step-by-step procedure to setup individual tools by modifying default tools and or introducing new tools.

To achieve the animated graphic representation of the machining process the stock and tool data is generated and saved as comments at the start of the CNC part program. This will not affect the CNC program's operation as comments are ignored by CNC controls. It is possible to save this data in the tool setup library. If the tool data is not wanted in the part CNC part program, then it can be deleted for the start of the program. If you wish to check the program again later, then the stock and tool data can be restored for the tool library.

In the sample programs below, we have selected Control Types, Haas Turning in the Editor. When we evaluate with Backplot later you will see we chosen File Type Haas NCG Turning.

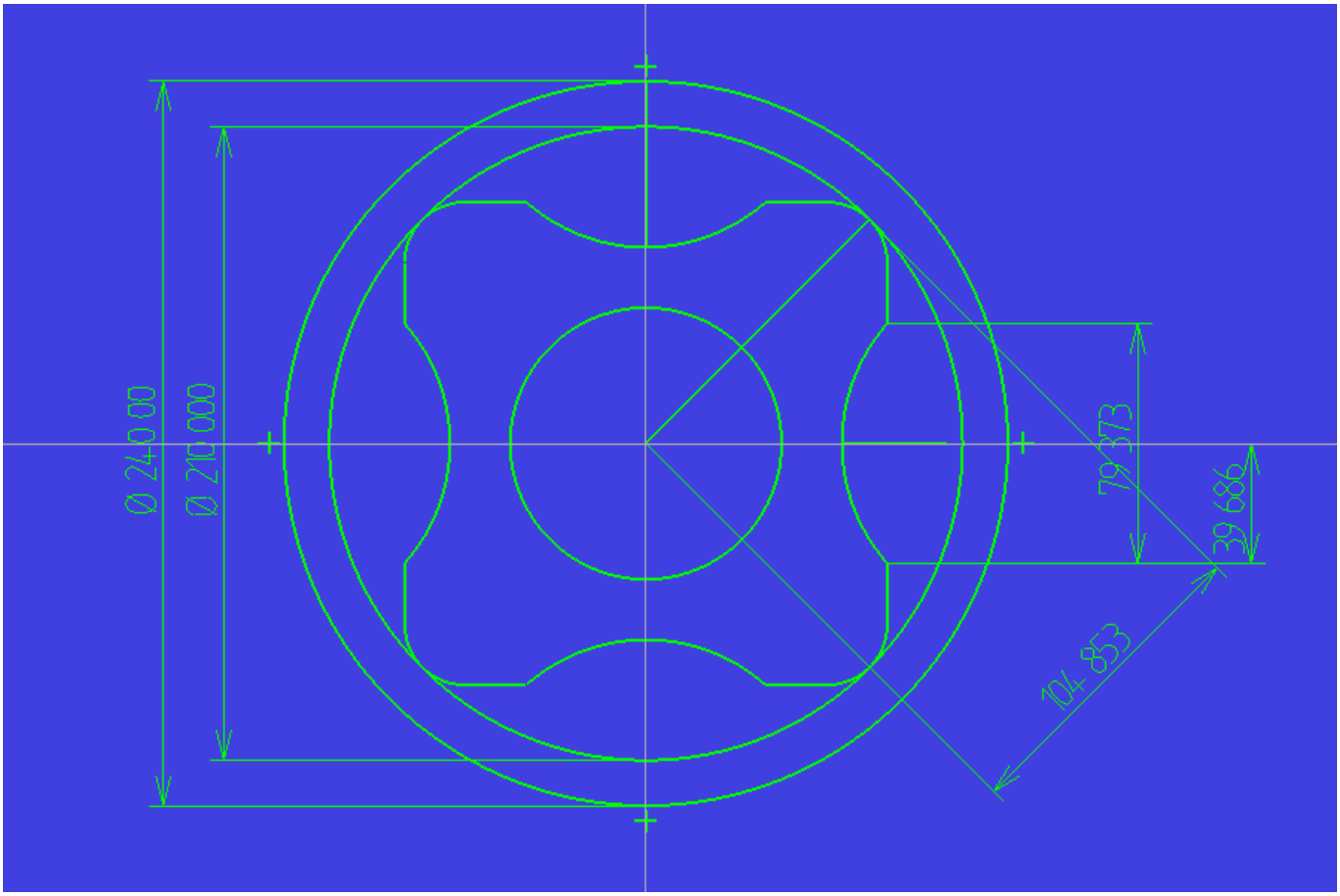
We will first look at the Half Coupling, see the drawing below. We are going to program operation 1 that will face, turn, and bore the blank to a size. Then we will use the live tooling to mill the profile and drill/Tap the holes. Haas and Fanuc provide a G code that will permit the use of X, Y coordinated to describe the profiles and transform them into X, C polar coordinates to create the profile shape. The drilling and tapping operation can be easily programed by the trainee using direct C and X polar axis coordinates to position the tool.



HALF COUPLING - EN8 - M/C ALLOVER

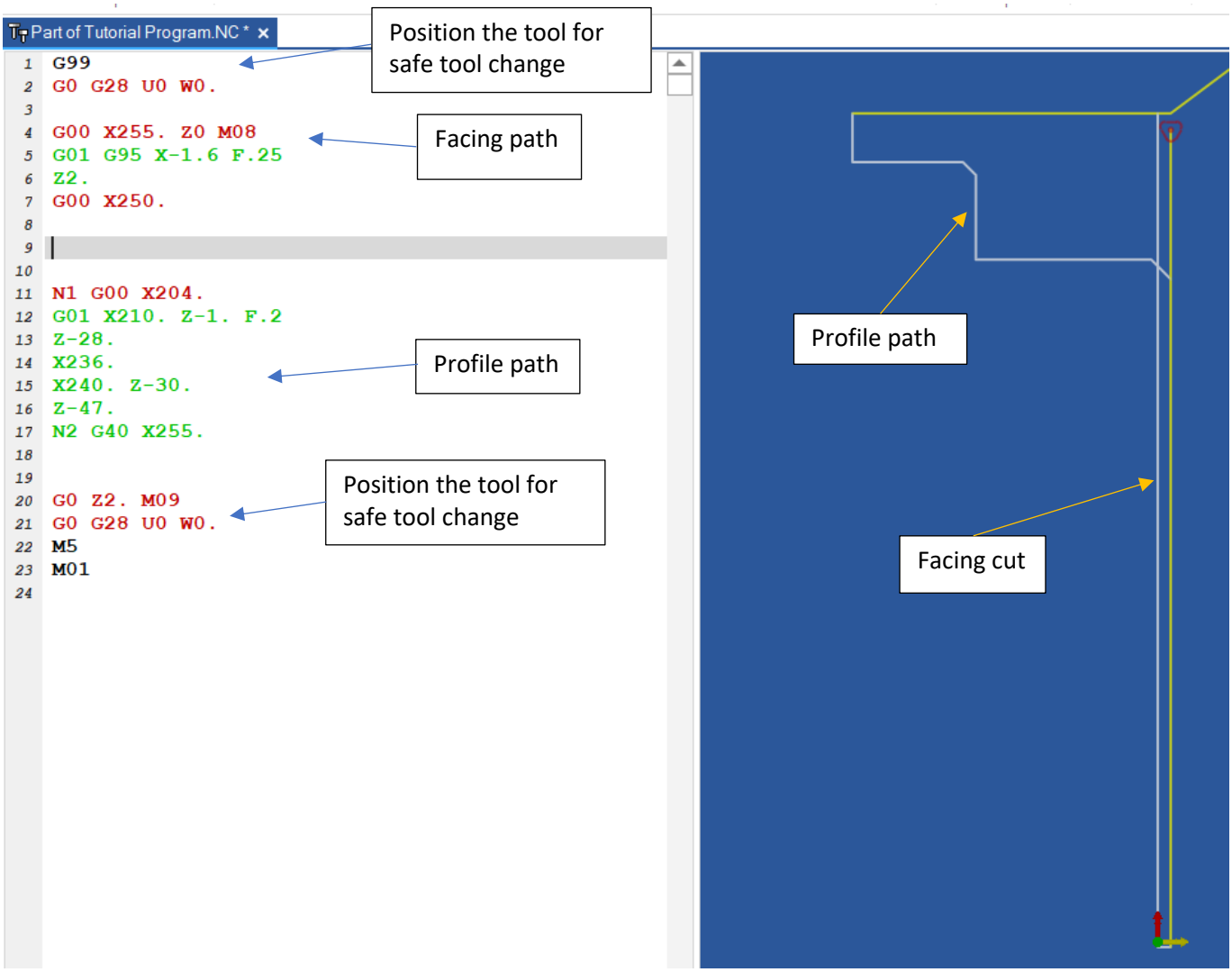
We have started by creating a drawing for programming use. We will be able to gain information we need for the first Turning operation and later for the milling. We have created a drawing in CNC-Calc showing the final profile

path and have drawn a radial line to the 20 mm radius and measured it at 104.853 so if we take a radius of 105 x 2, we will turn the diameter down to 210 mm to remove the excess material before milling the profile.

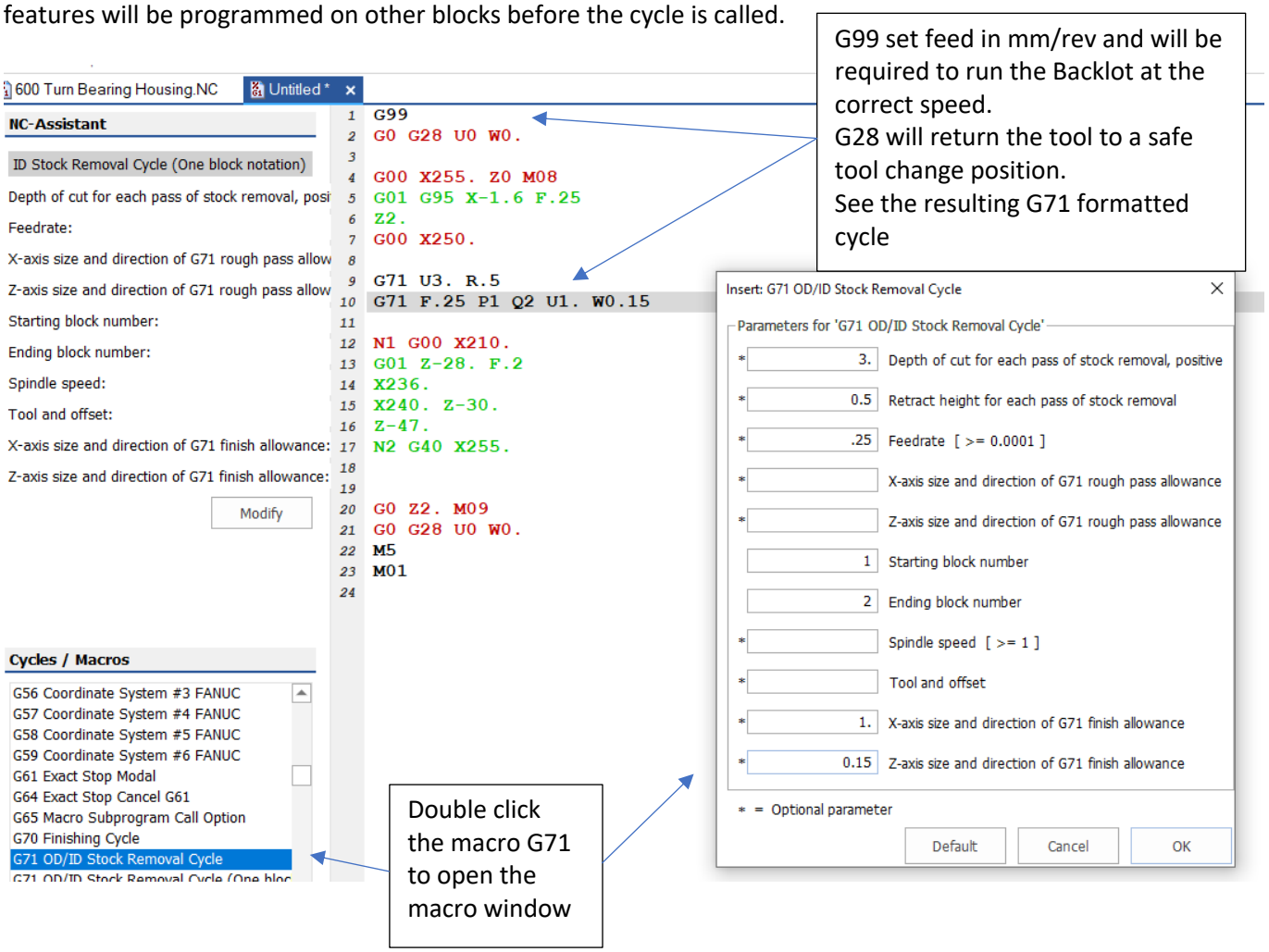


COMMENCE PROGRAMMING USING THE EDITOR

Start by programming the simple moves block by block to turn the front face and create a profile for the first turning operation and then add the roughing cycle to achieve a series of cuts to the final profile and test with Backplot. (See below) Test the Facing section first, then the profile section, then add the Roughing Cycle, until the Backplot is as below. Select the Backplot tab a click the Backplot icon.



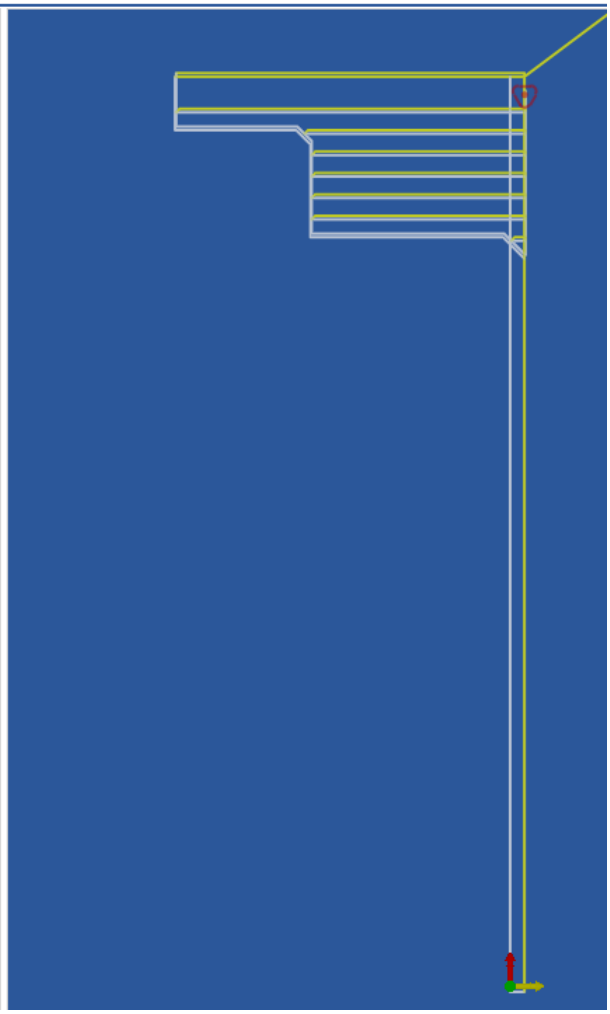
Now we can add the roughing canned cycle to create the turning passes that will achieve the final profile. The two-line G71 cycle is selected here, and you will note that some of the optional fields have been left as these features will be programmed on other blocks before the cycle is called.



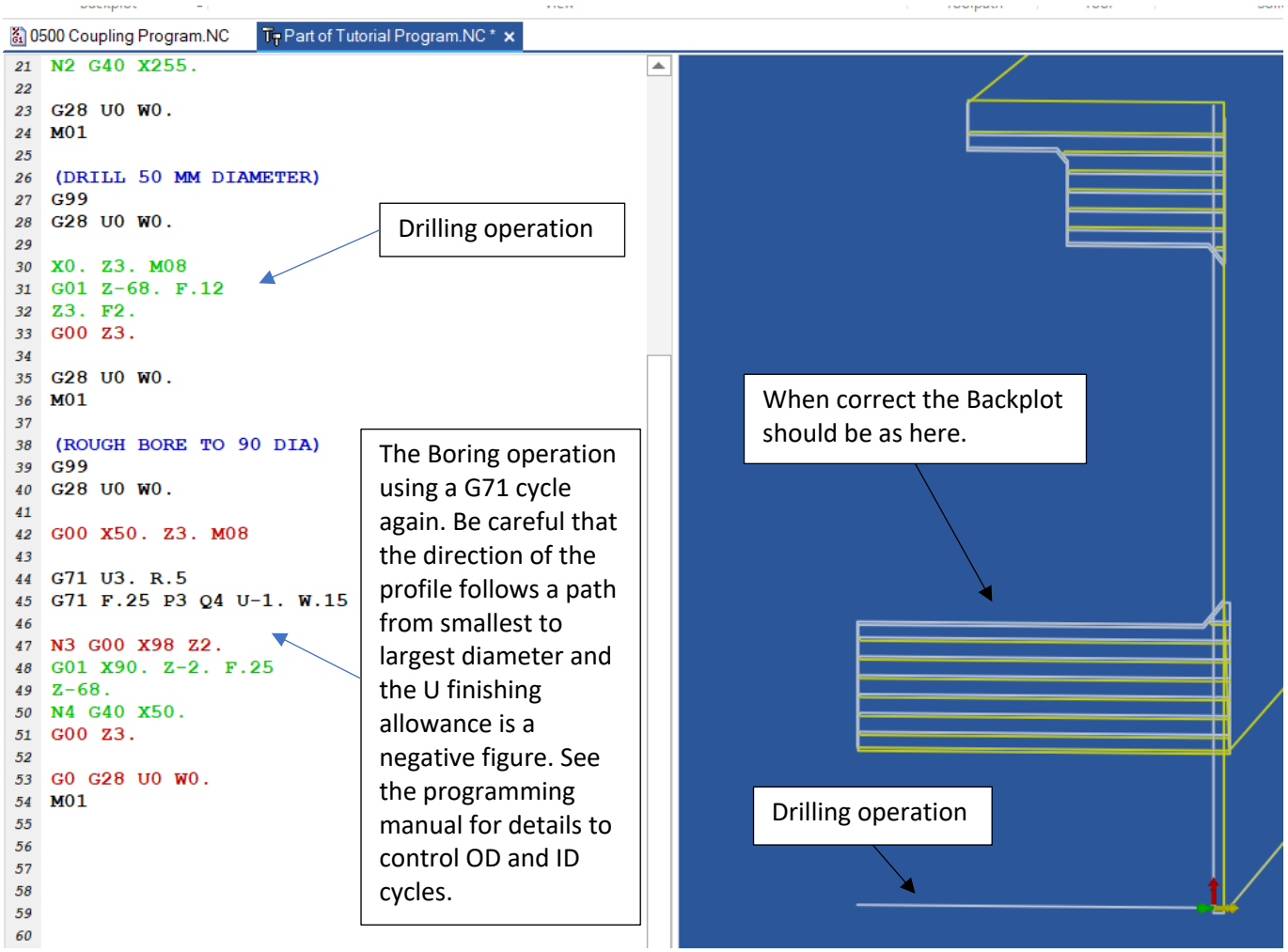
When the Roughing Cycle is correct the Backplot will be as below:

Part of Tutorial Program.NC *

```
G99
G0 G28 U0 W0.

G00 X255. Z0 M08
G01 G95 X-1.6 F.25
Z2.
G00 X250.

G71 U3. R.5
G71 F.25 P1 Q2 U1. W.15

N1 G00 X204.
G01 X210. Z-1. F.2
Z-28.
X236.
X240. Z-30.
Z-47.
N2 G40 X255.


G0 Z2. M09
G0 G28 U0 W0.
M5
M01

```

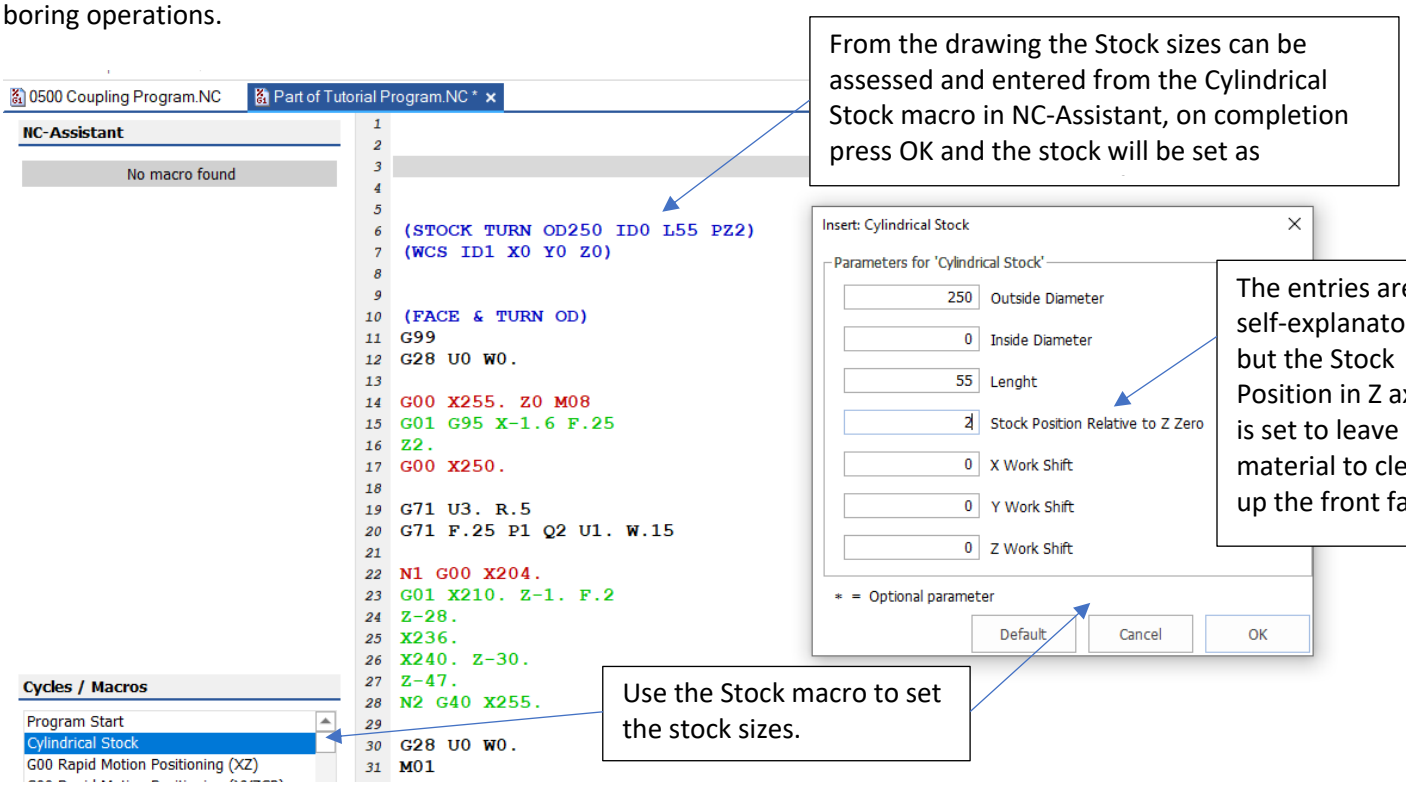


Continue with the Block-by-Block programming to complete the boring operation. We will first drill a 50 mm holes then use a boring bar to open the bore to the drawing diameter.



SETTING UP THE STOCK SIZE

Before we move onto the C axis milling programming, we will consider applying a stock size and setting the tools so that we can generate a solid animated image to give a true representation of the operation of the turning and boring operations.



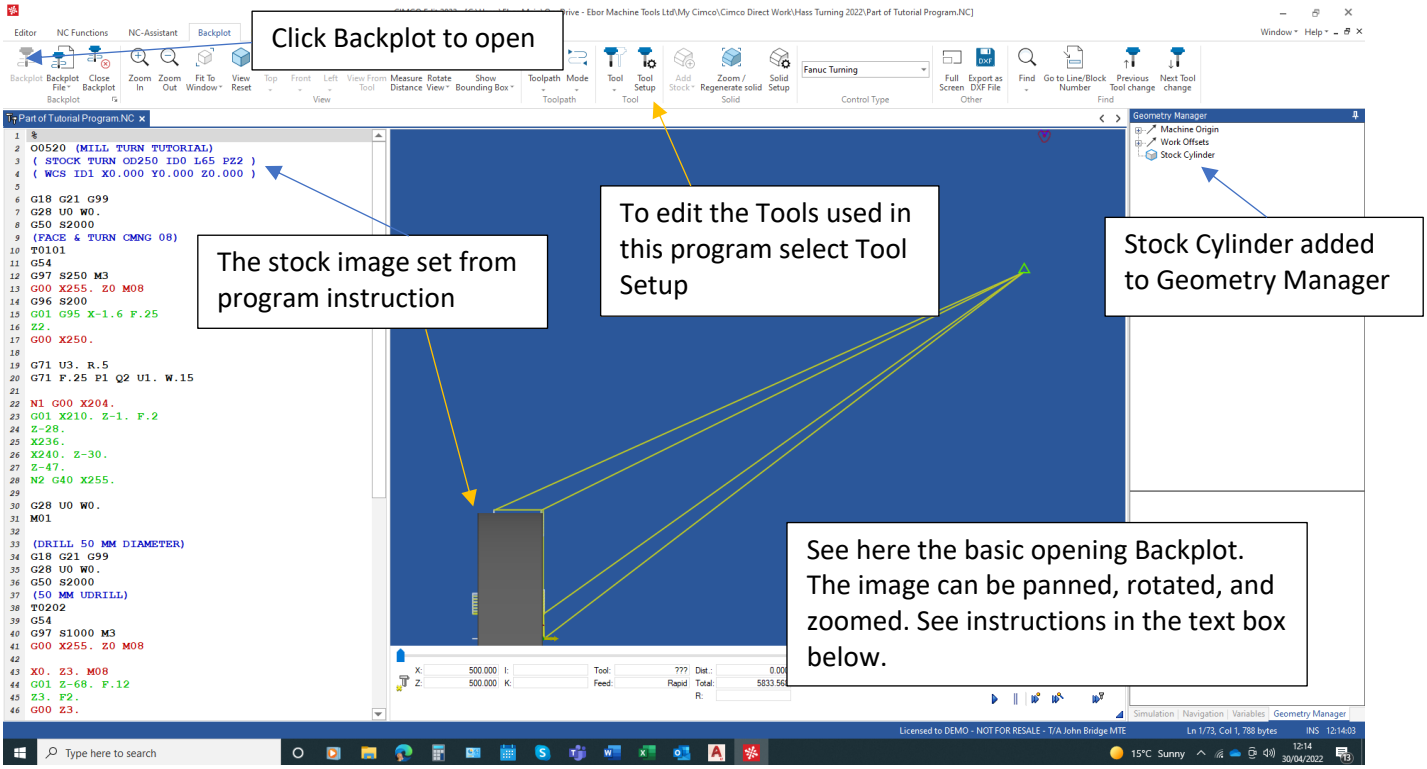
(STOCK TURN OD250 ID0 L80 PZ2)
(WCS ID1 X0.000 Y0.000 Z0.000)

Resulting Stock blank sizes and Coordinate system work shift details are set in the program as comments

Stock Details
Outside Diameter = 250
Inside Diameter = 0
Length = 80
Z Position = 2 (2 mm to face of the front to clean up at Z zero)

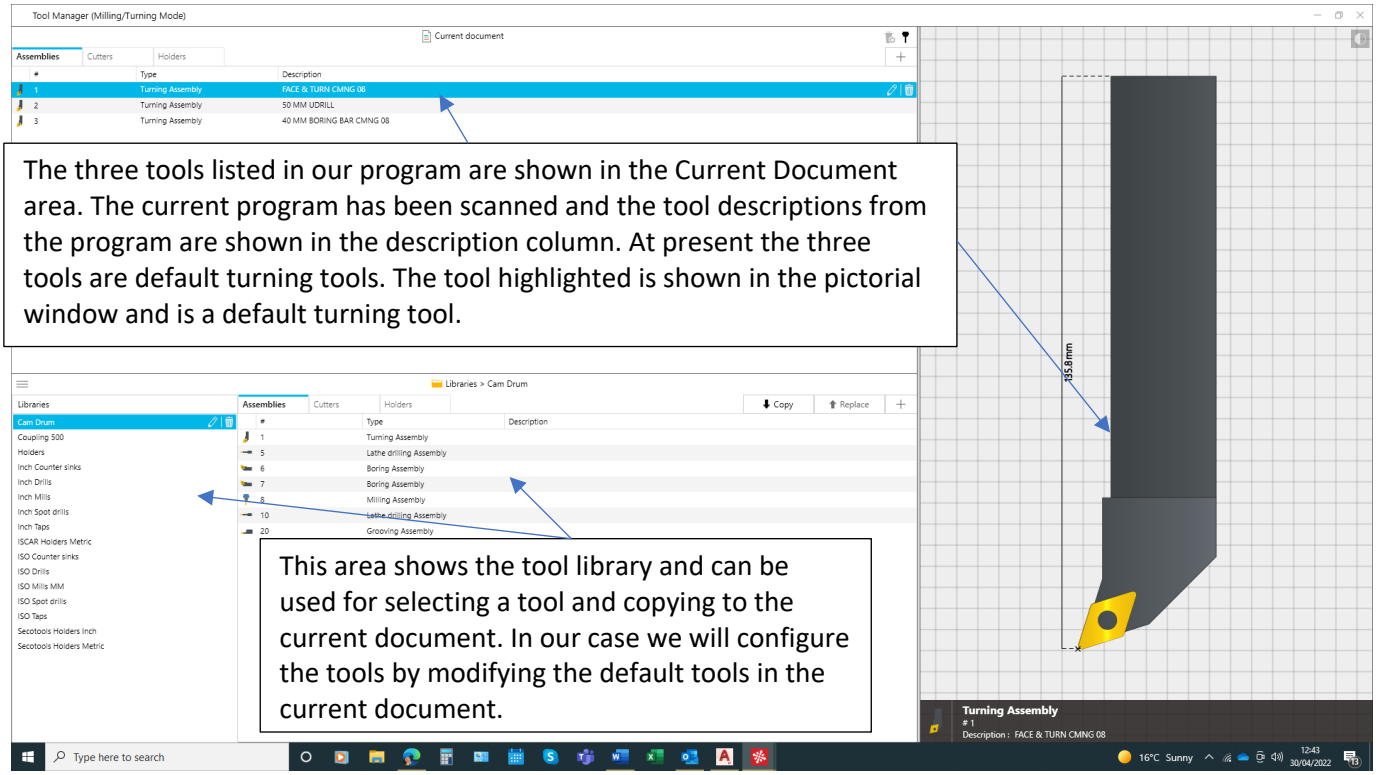
SETTING UP TOOLS

To setup the tools to face, turn, drill, and bore the part as programmed so far, we will go to Backplot. See below

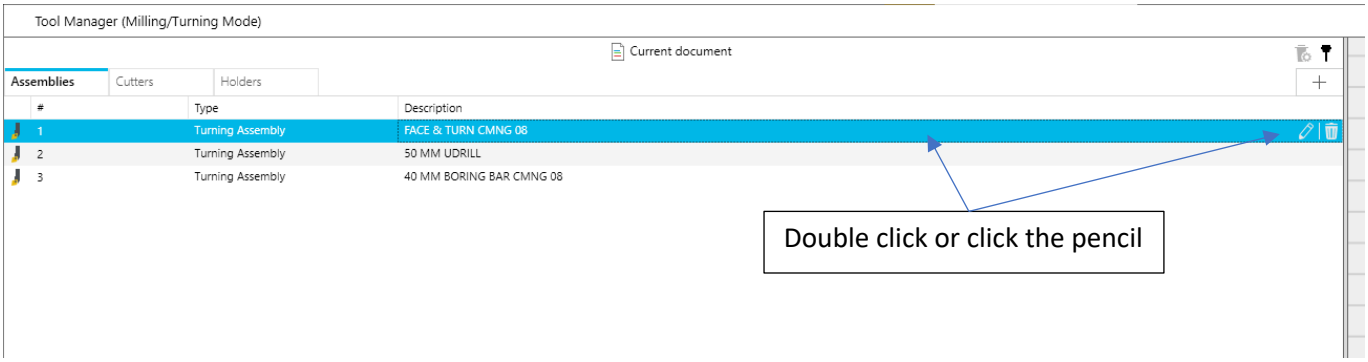


CONTROL OF THE IMAGE
Hold down mouse left key to rotate the image!
Hold down mouse right key to pan the image!
Roll the mouse wheel to zoom

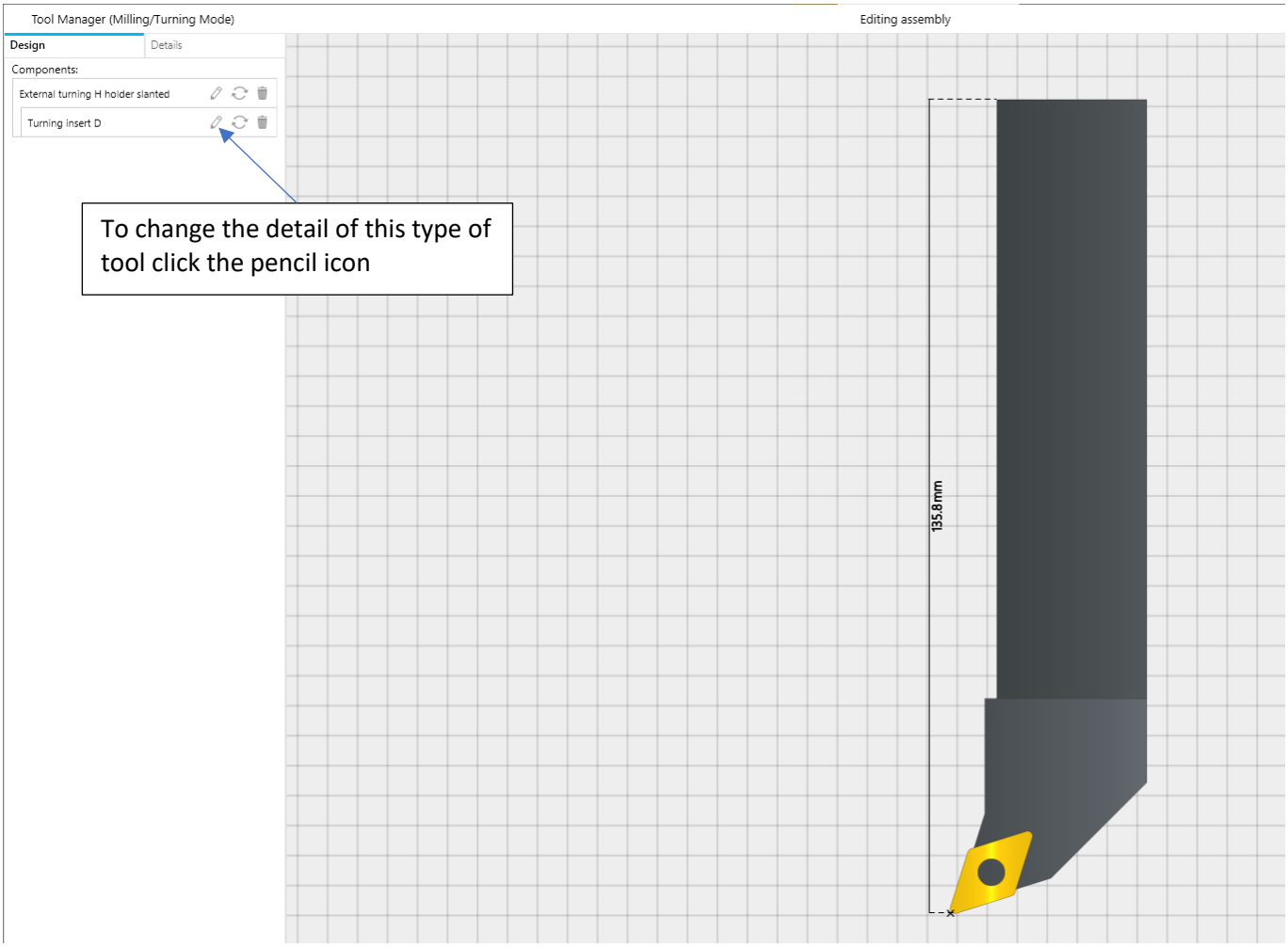
The Tool Setup page will open when clicking the Tool Setup icon. We will need to set the tools to the correct type, shape, and orientation with the correct cutting insert. When we first open Tool Setup we will see below this page.



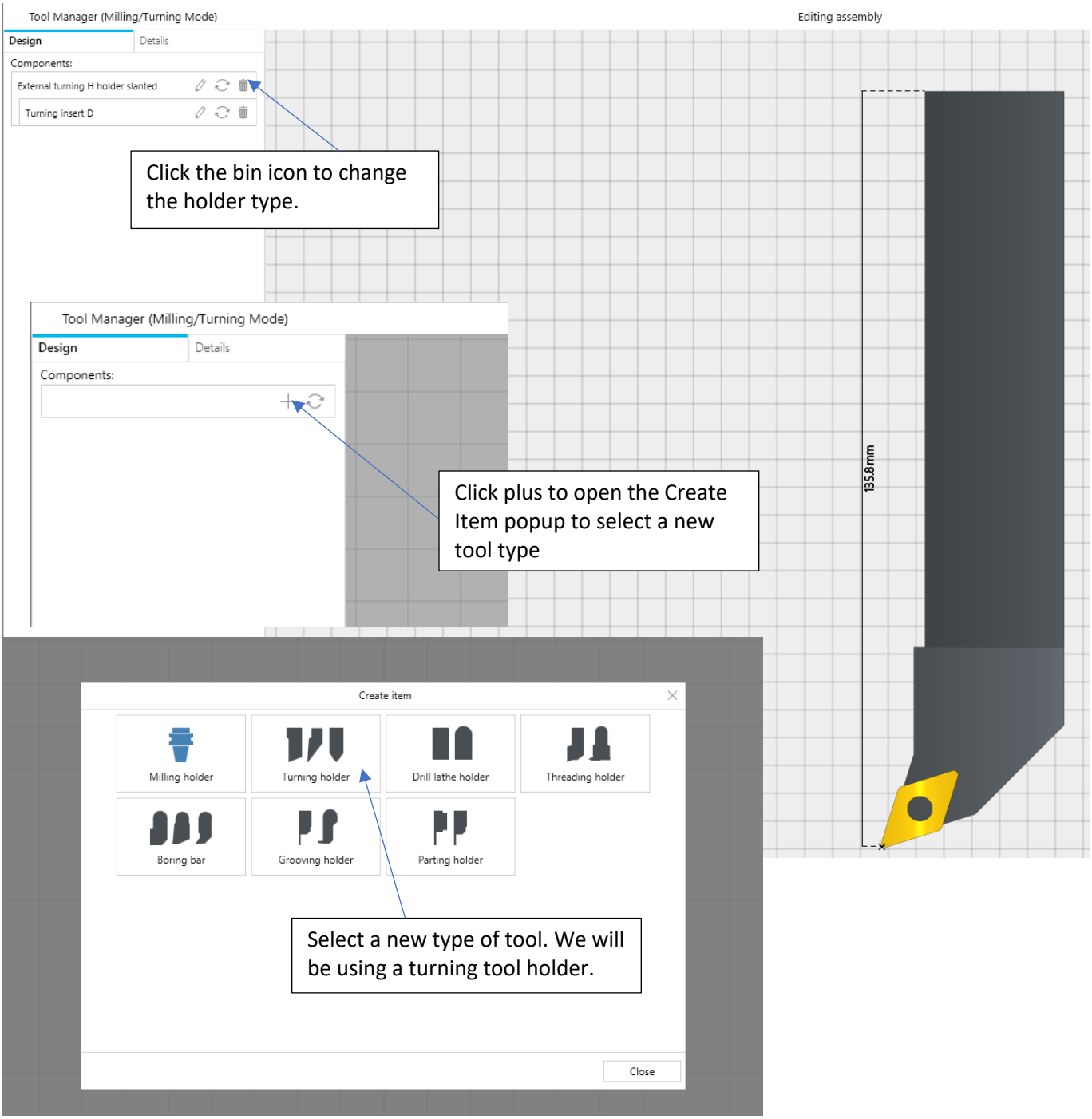
The default tool number 1 is not the one we want to use. The tool can be edited by double clicking the highlighted tool or clicking on the pencil icon (The default tool has a profiling insert which we will change to a CNMG 80 degree Turning and Facing insert and holder.



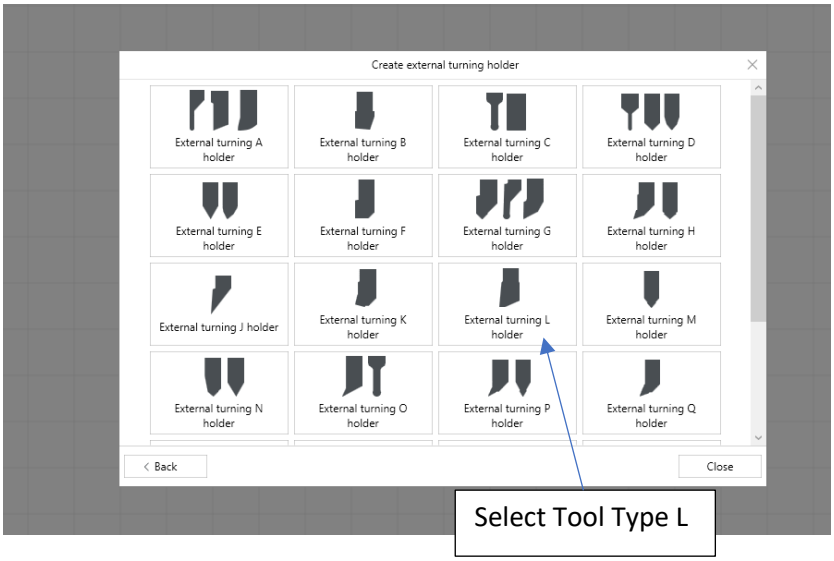
If the type of tool is correct, then we can edit the details of this type of tool by clicking the pencil icon for either the holder or insert but if the type is not correct, we need to select a new type. Continue:



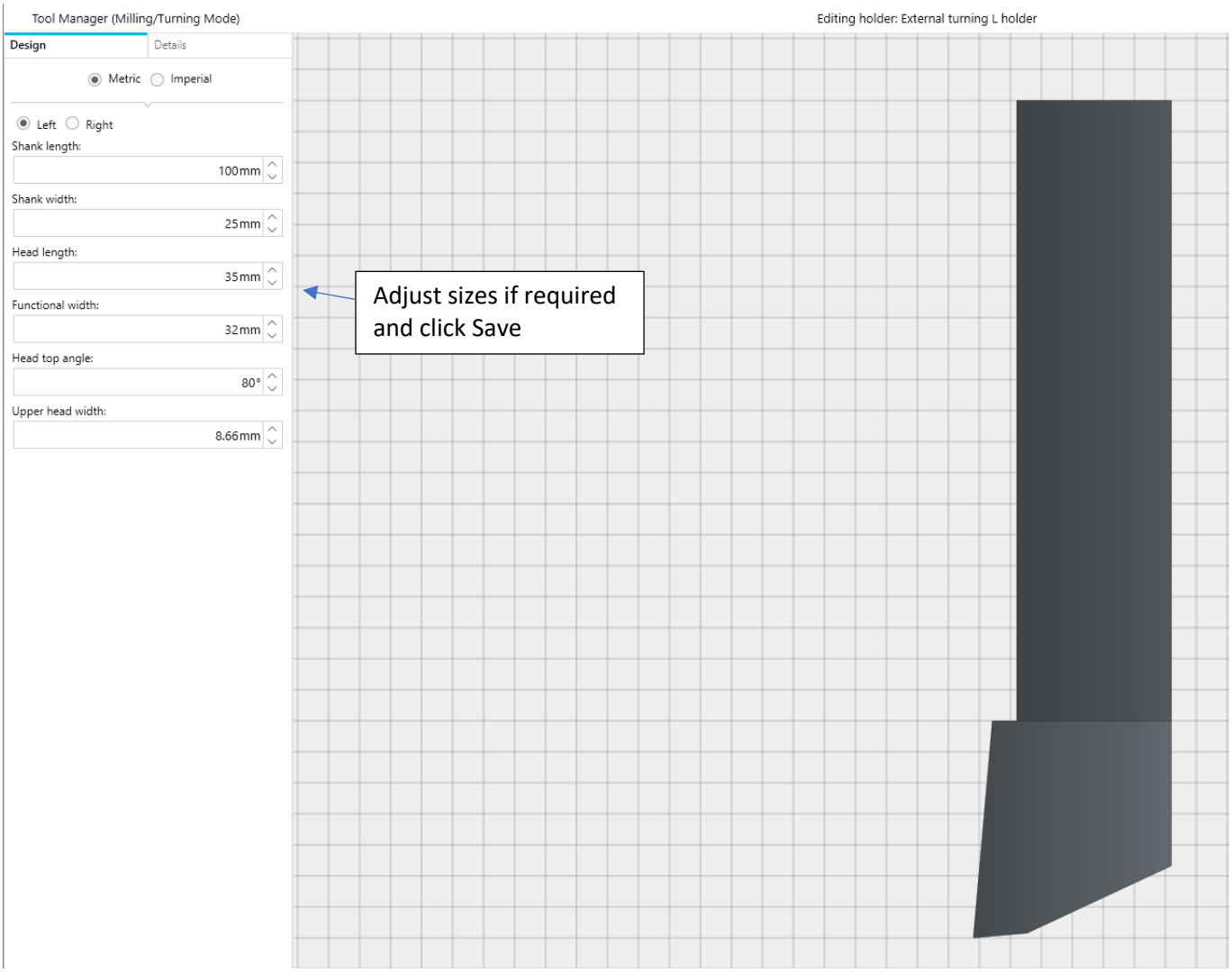
To completely change the type of tool



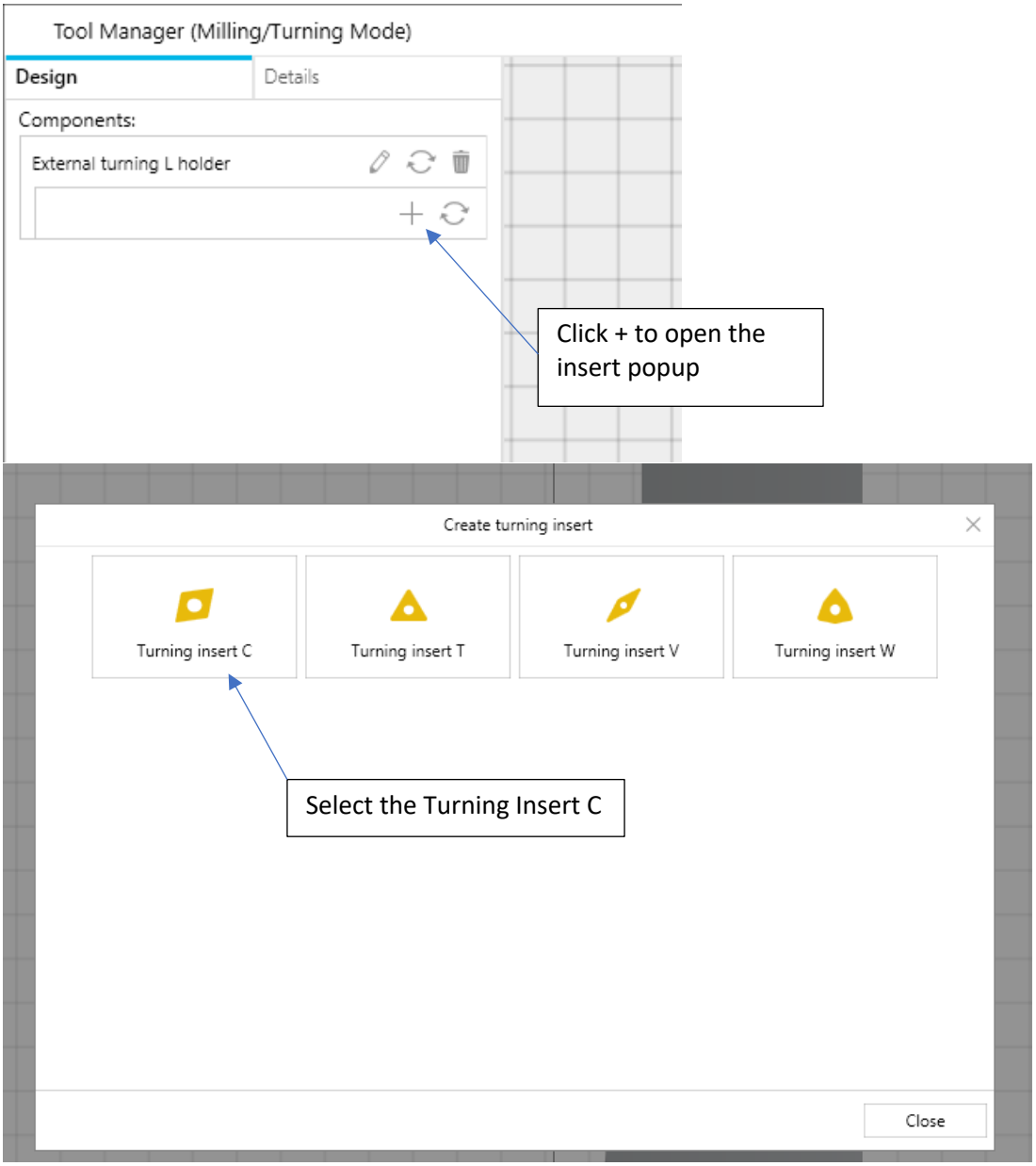
Select Turning Tool holder L

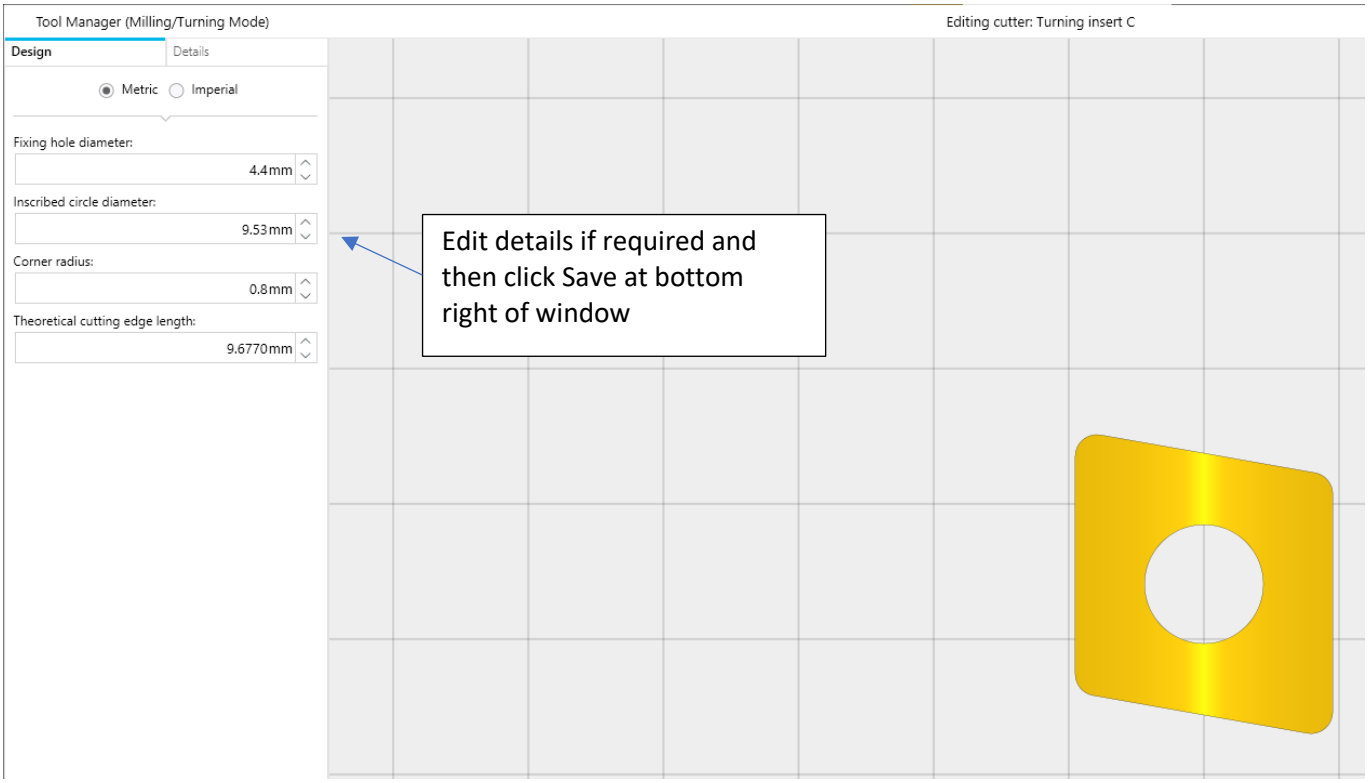


Adjust sizes if required

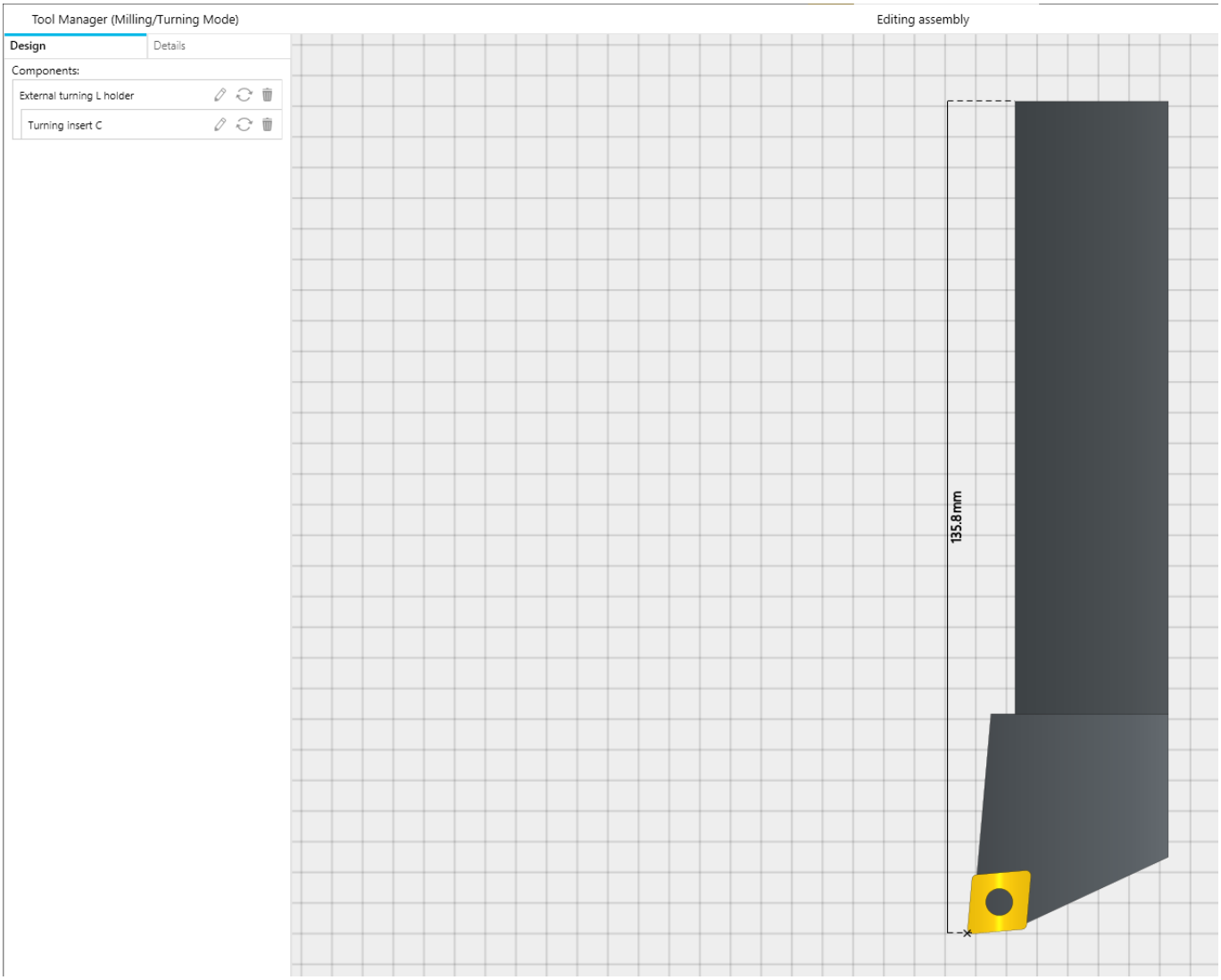


Now select the insert required

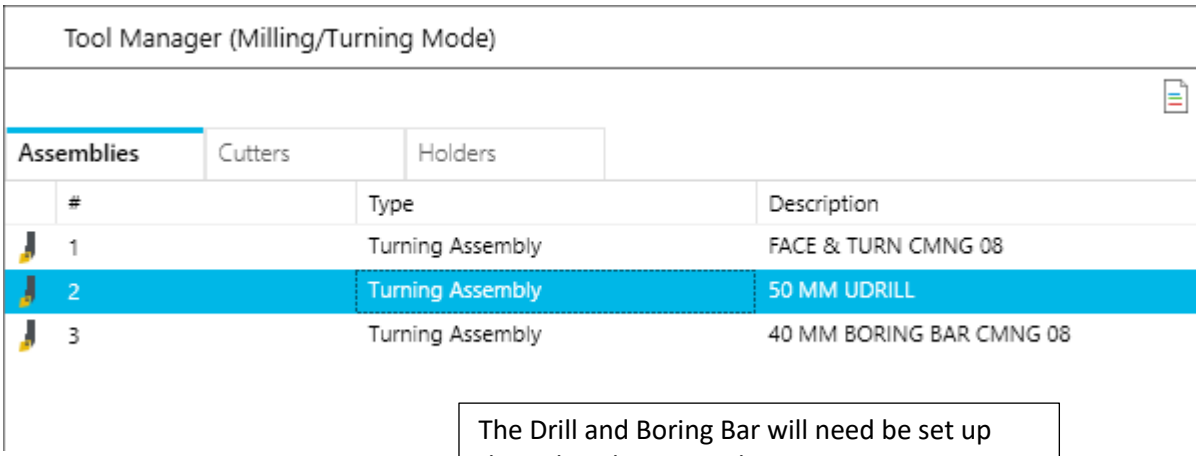



Tool Manager (Milling/Turning Mode)
Editing cutter: Turning insert C
Design
Details
Metric
Imperial
Fixing hole diameter:
4.4mm
Inscribed circle diameter:
9.53mm
Corner radius:
0.8mm
Theoretical cutting edge length:
9.6770mm
Edit details if required and then click Save at bottom right of window

The tool holder and insert are selected click Save bottom right of window.

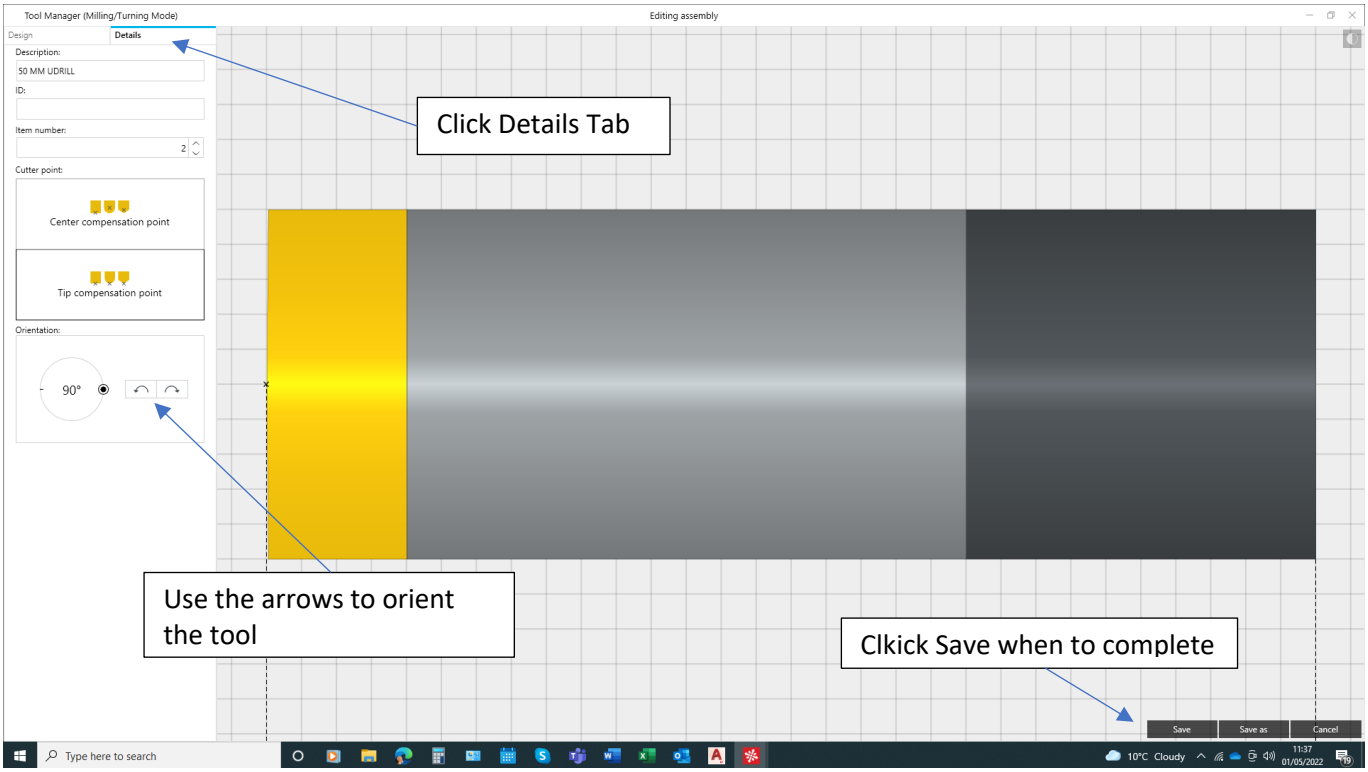


Now the other tool must be configured.

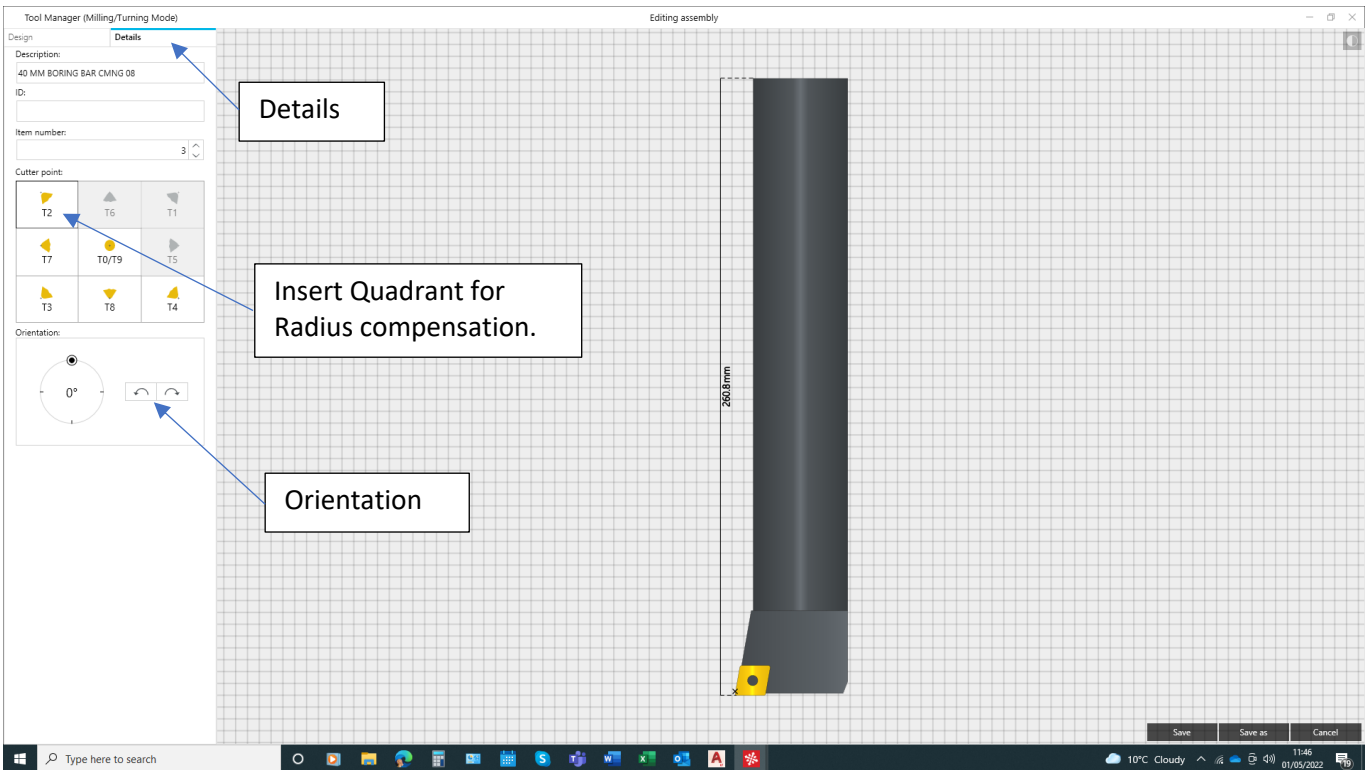


The Drill and Boring Bar will need be set up then placed in an axial orientation.

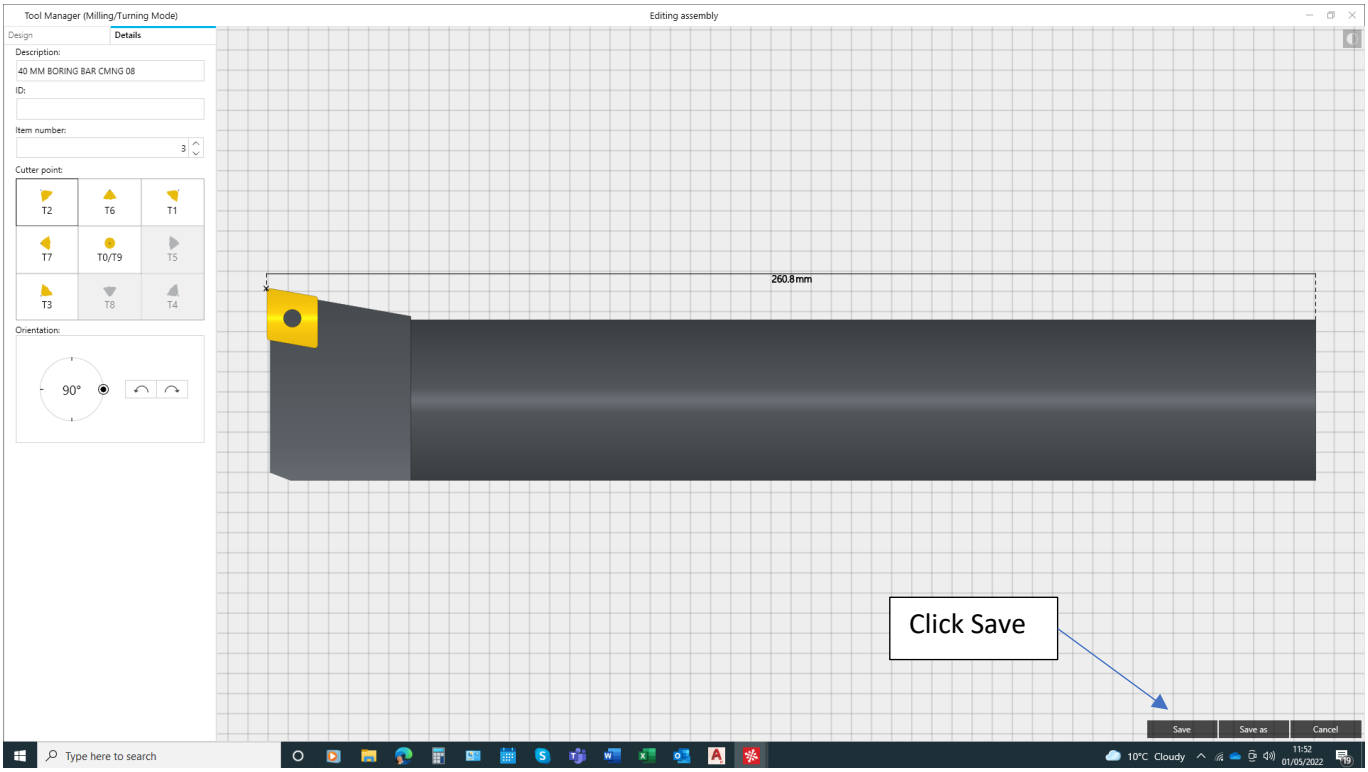
Having setup, the drill at 50 mm diameter in the same way as previously described we use the details tab to orient the tool to work axially.



The Boring Bar can now be set up. Select Details to set the tool insert quadrant and the orientation.



Final setting for the boring bar



When all our tools are set, see below, confirmation of correctness can be made.

Tool Manager (Milling/Turning Mode)

Current doc

Assemblies	Cutters	Holders

#	Type	Description
1	Turning Assembly	FACE & TURN CMNG 08
2	Lathe drilling Assembly	50 MM UDRILL
3	Boring Assembly	40 MM BORING BAR CMNG 08

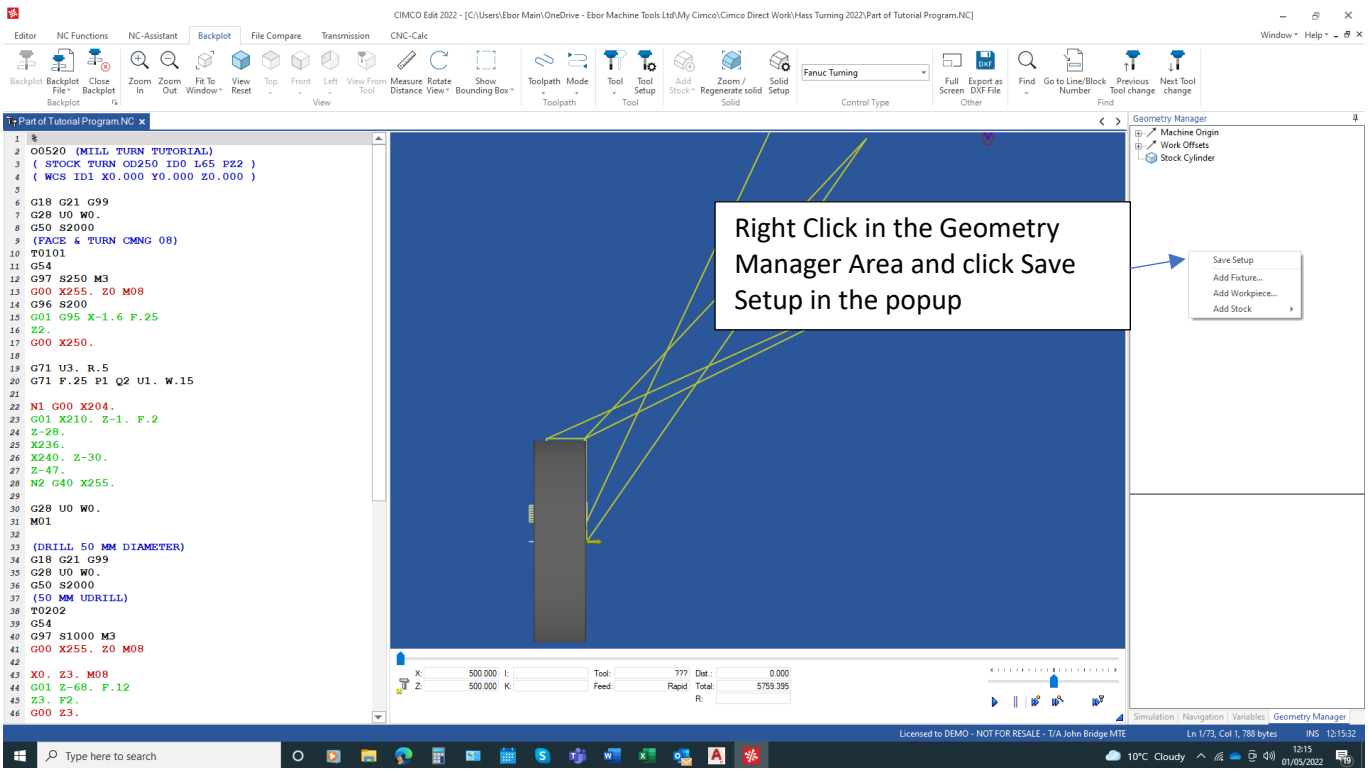
Thumbnail pictorial view to confirm the shape.

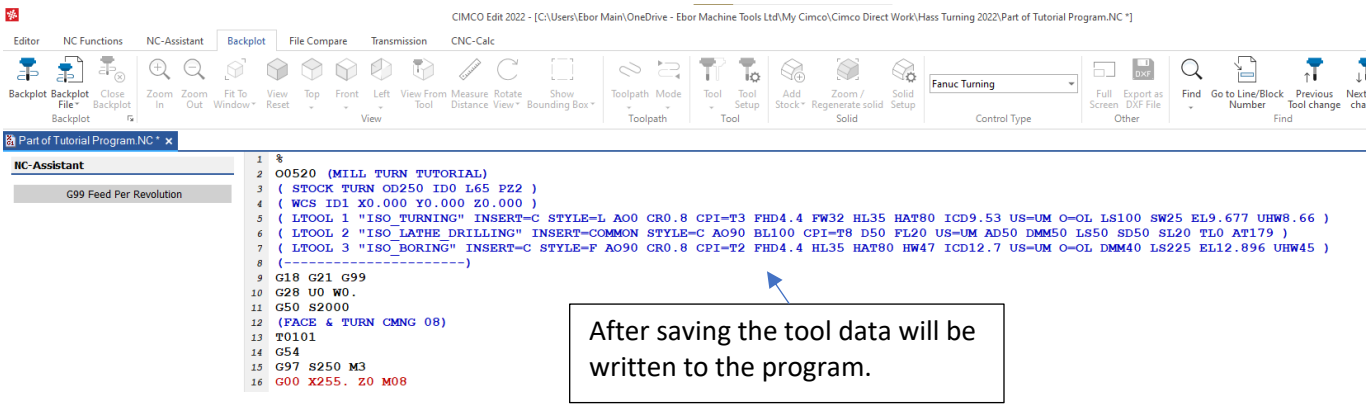
Tool description from the Tool assemblies set up.

Tool Description text scanned from the new CNC program

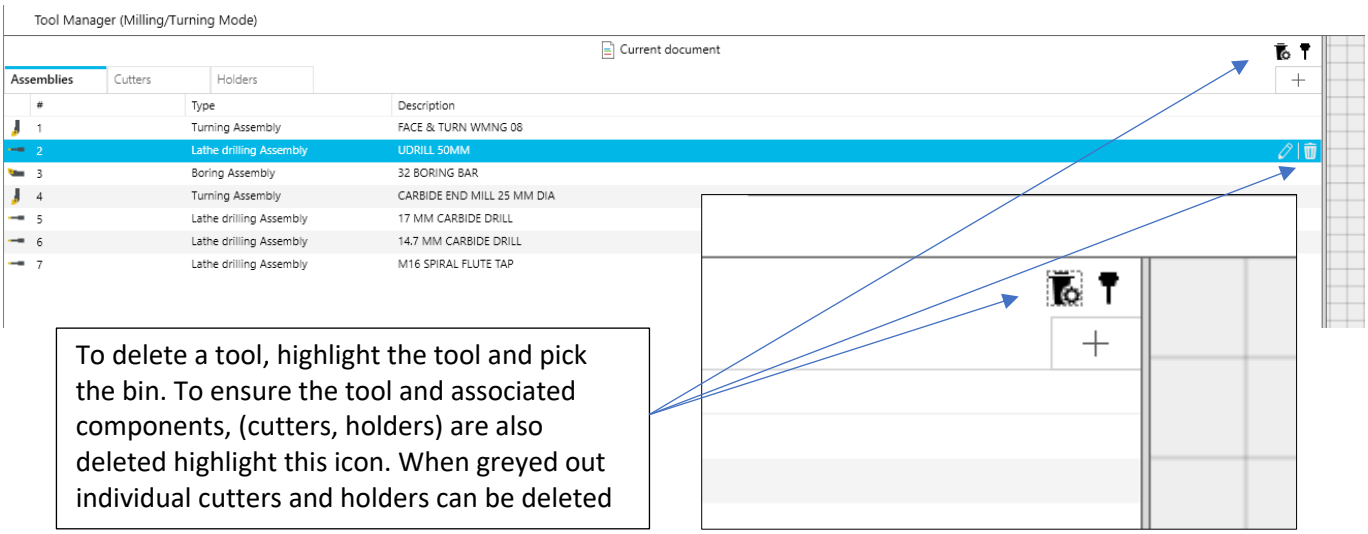
IMPORTANT

To write the Tool setup to the CNC program so that when we Backplot the correct tool is used we must save the setup. The Tool setup window can be closed, (click close top right X) then right click in the Geometry Manager area see below:



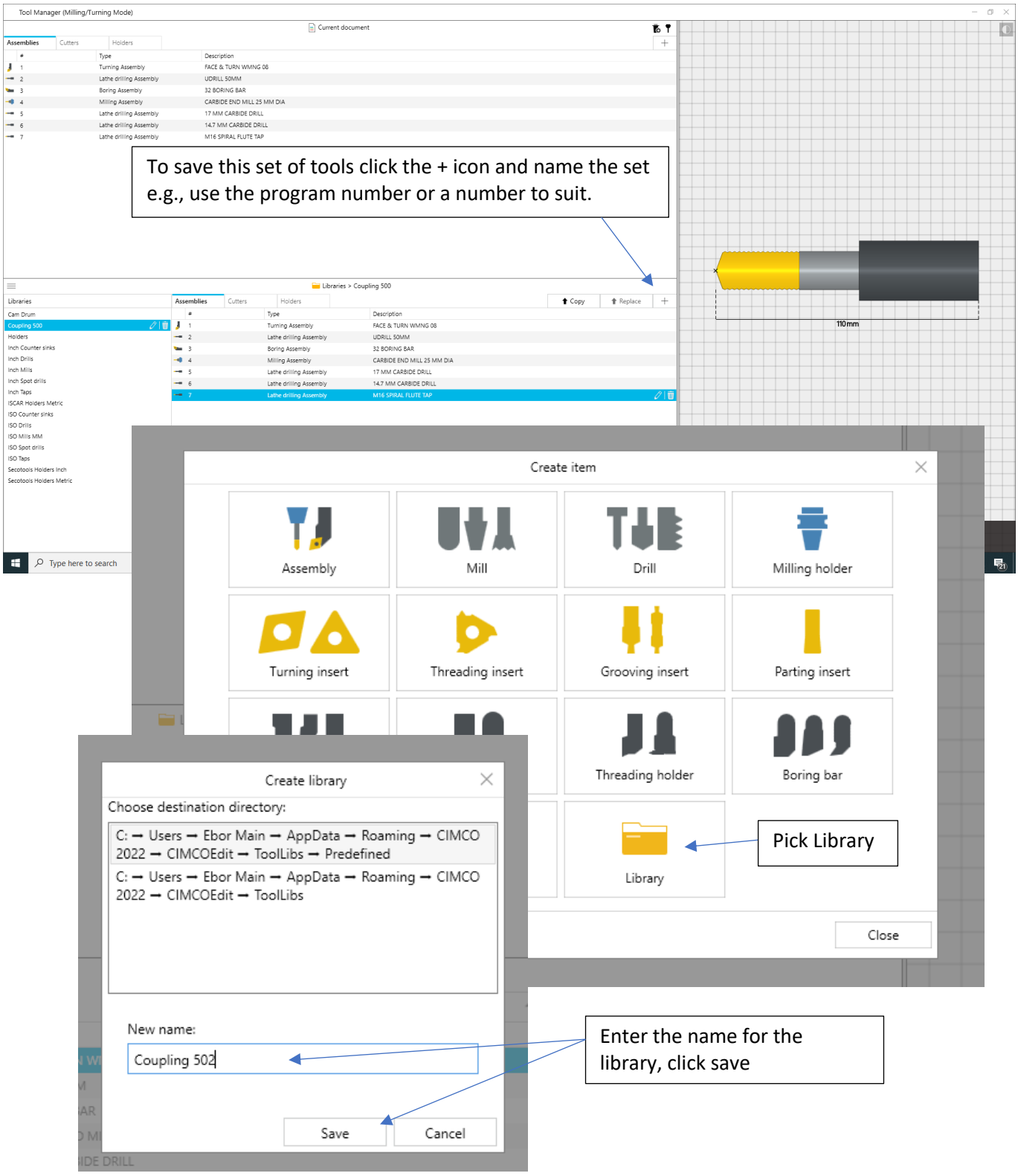


ADDITIONAL TOOL SETUP FEATURES

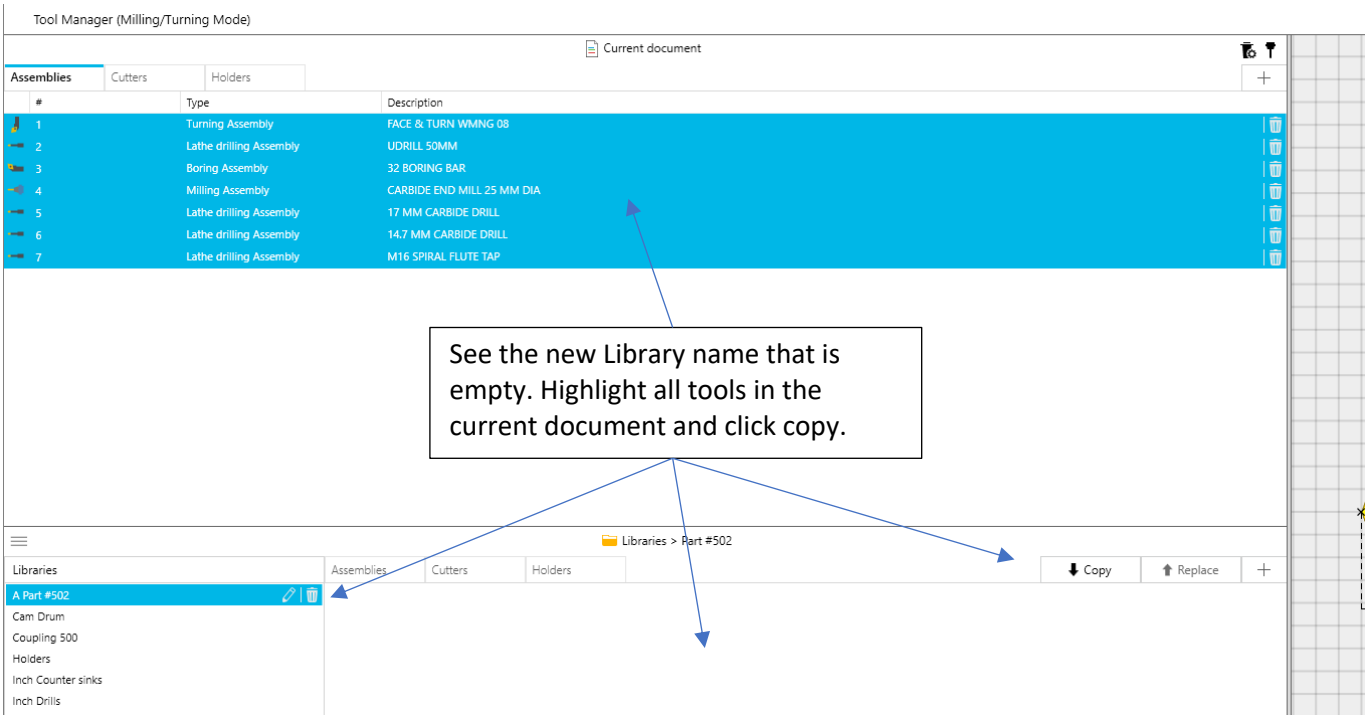


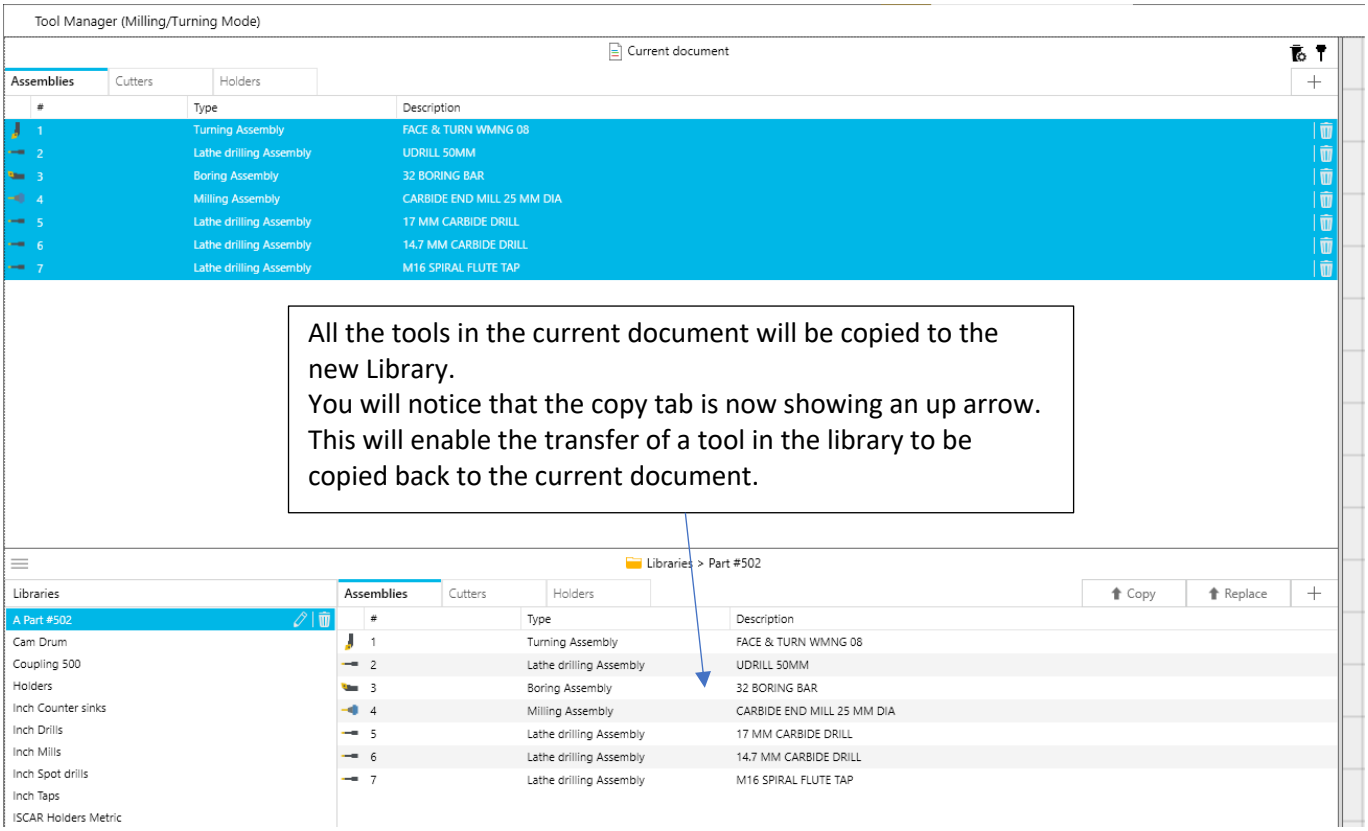
To delete a tool, highlight the tool and pick the bin. To ensure the tool and associated components, (cutters, holders) are also deleted highlight this icon. When greyed out individual cutters and holders can be deleted

A set of tools for a specific part to be machined can be saved in the library.

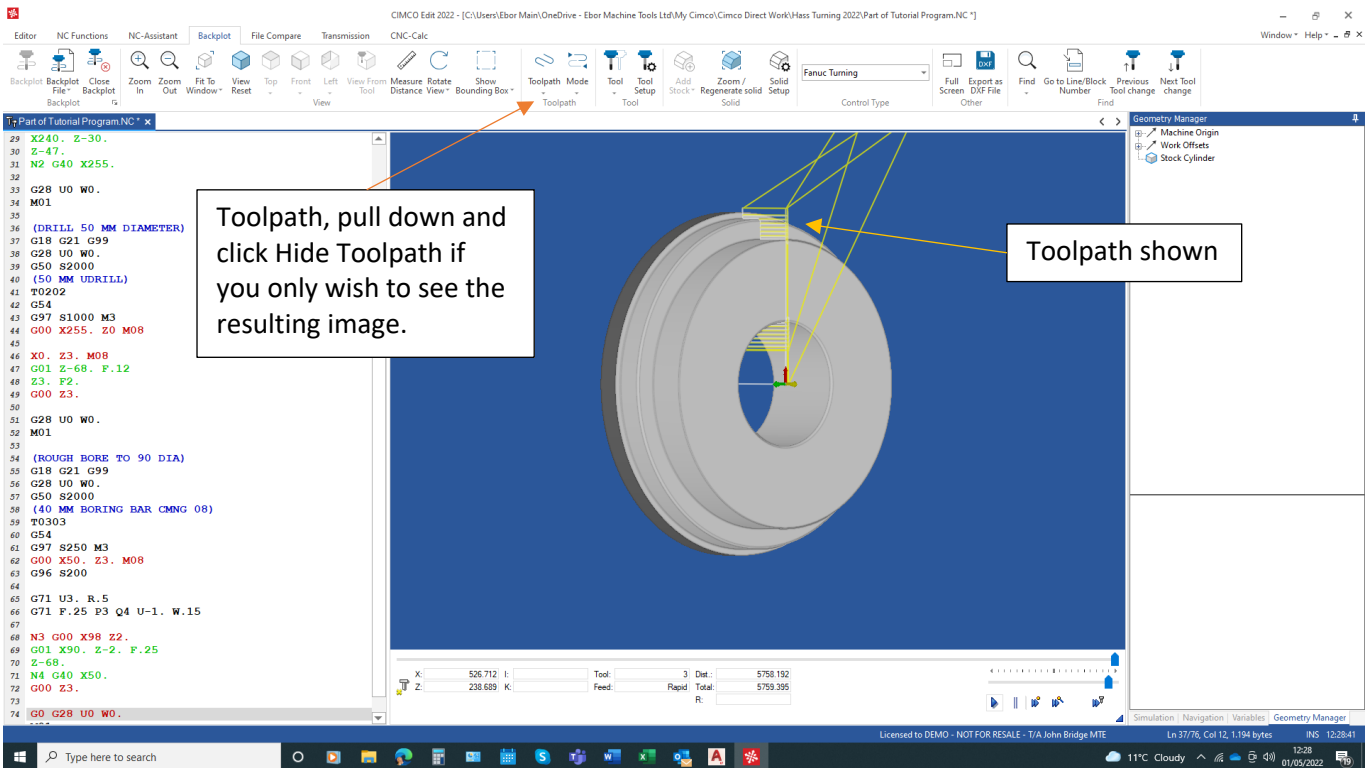


See below the saving of a set of tools to the Library.





We can now test the program in Backplot which will be as below if the program, stock size and tools are correct. By default, Backplot will show the tool path. The tool path can be switched off by clicking the tool path icon in Backplot and selecting “hide toolpath” see below.



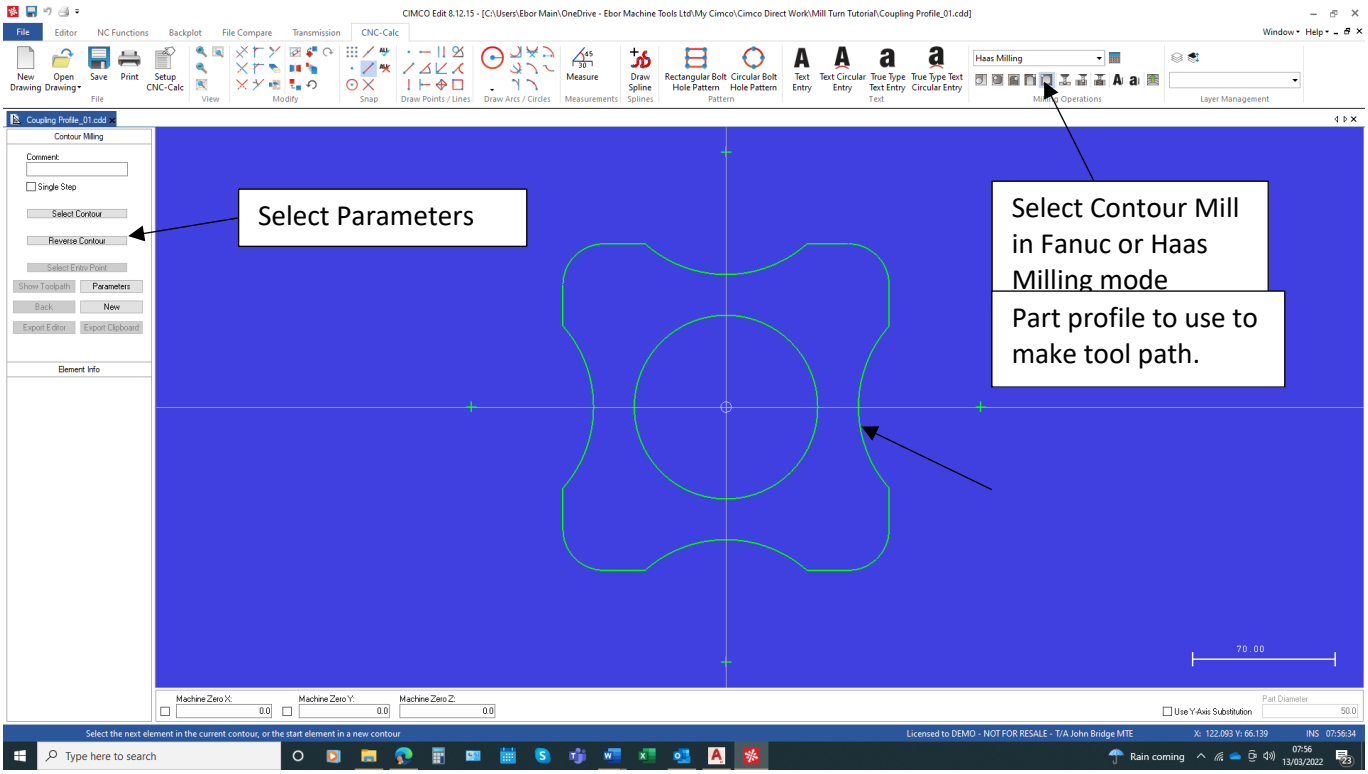
C AXIS MILLING

The G code G112/G113 will enable us to generate the profile in CIMCO CNC-Calc in X, Y coordinates using a milling post processor and the G112 code will convert the generated coordinates to polar in C and X axis commands to machine the profile.

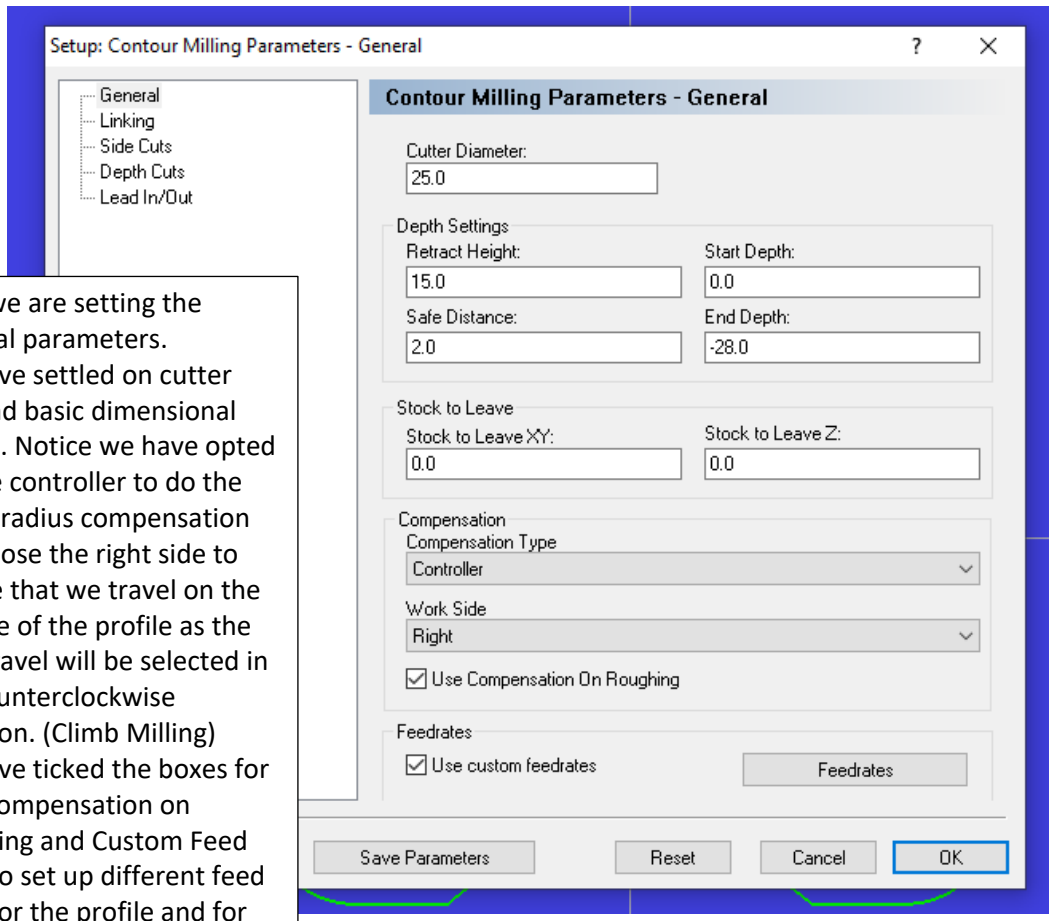
Return to the drawing to generate the Tool path in X, Y.

Create a path drawing from our original drawing by cutting away any lines we do not need for this operation. See screenshot below.

Select Haas Milling in the editor for this example. Select the contour mill icon in CNC-Calc and set parameters for machining the profile, see step by step below starting with the milling profile drawing

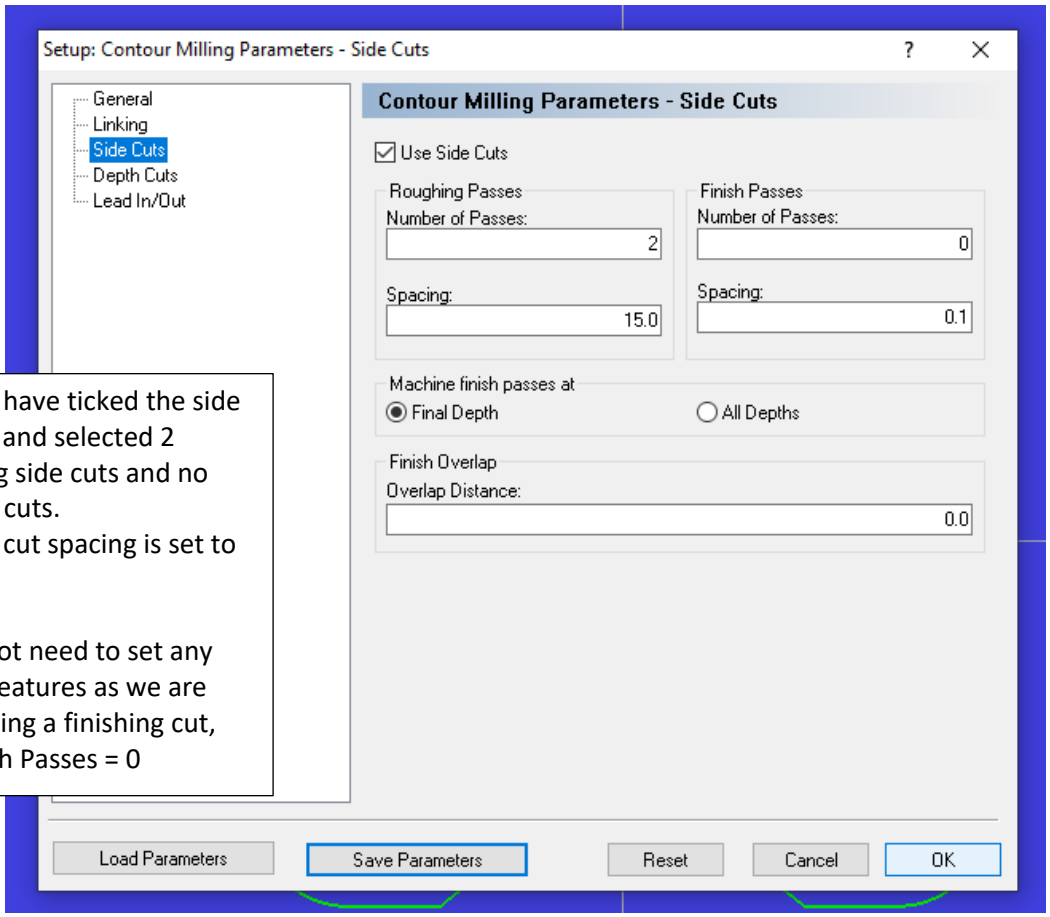


See General Parameter settings. Note as there is only one profile, we do not have to consider the Linking Parameters as this section is for machining multiple profiles.



Here we are setting the General parameters.
We have settled on cutter size and basic dimensional details. Notice we have opted for the controller to do the cutter radius compensation and chose the right side to ensure that we travel on the outside of the profile as the Tool travel will be selected in the counterclockwise direction. (Climb Milling)
We have ticked the boxes for Tool Compensation on Roughing and Custom Feed rates to set up different feed rates for the profile and for plunging in Z etc.

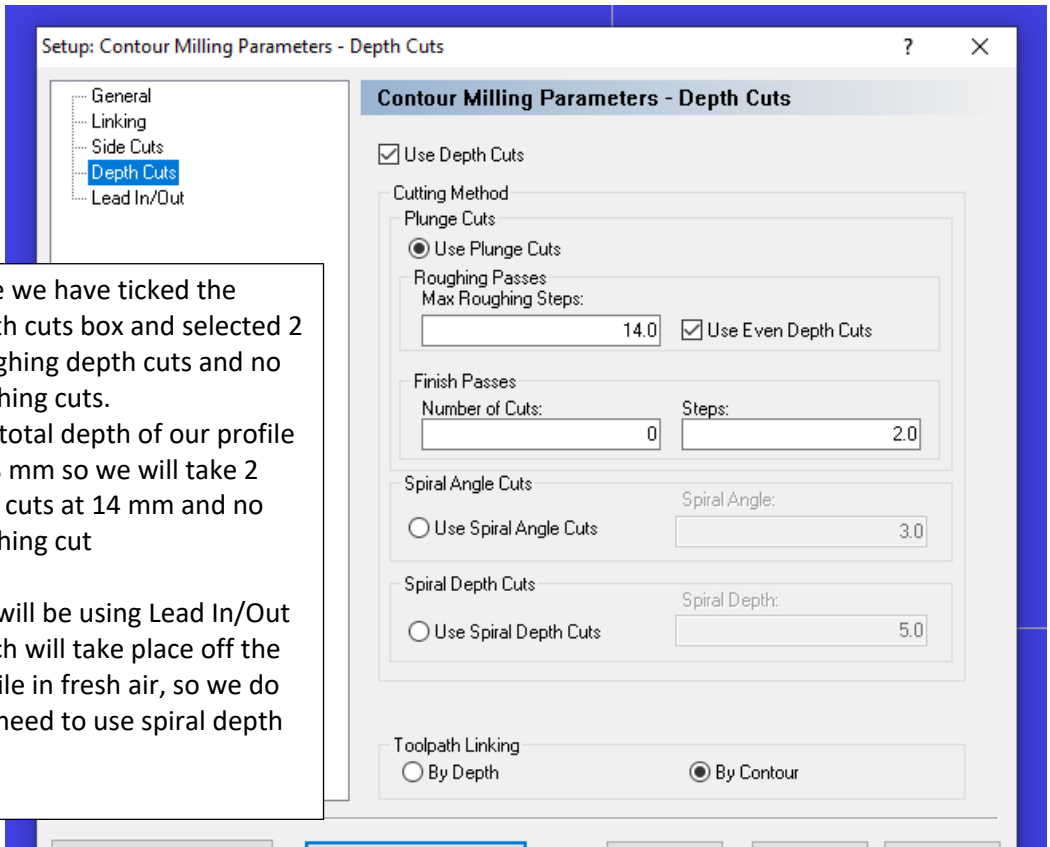
See Side Cuts Parameter settings. As there is a considerable amount of material to remove, we will choose to take side cuts



Here we have ticked the side cuts box and selected 2 Roughing side cuts and no finishing cuts.
Our side cut spacing is set to 15 mm.

We do not need to set any further features as we are not running a finishing cut, see Finish Passes = 0

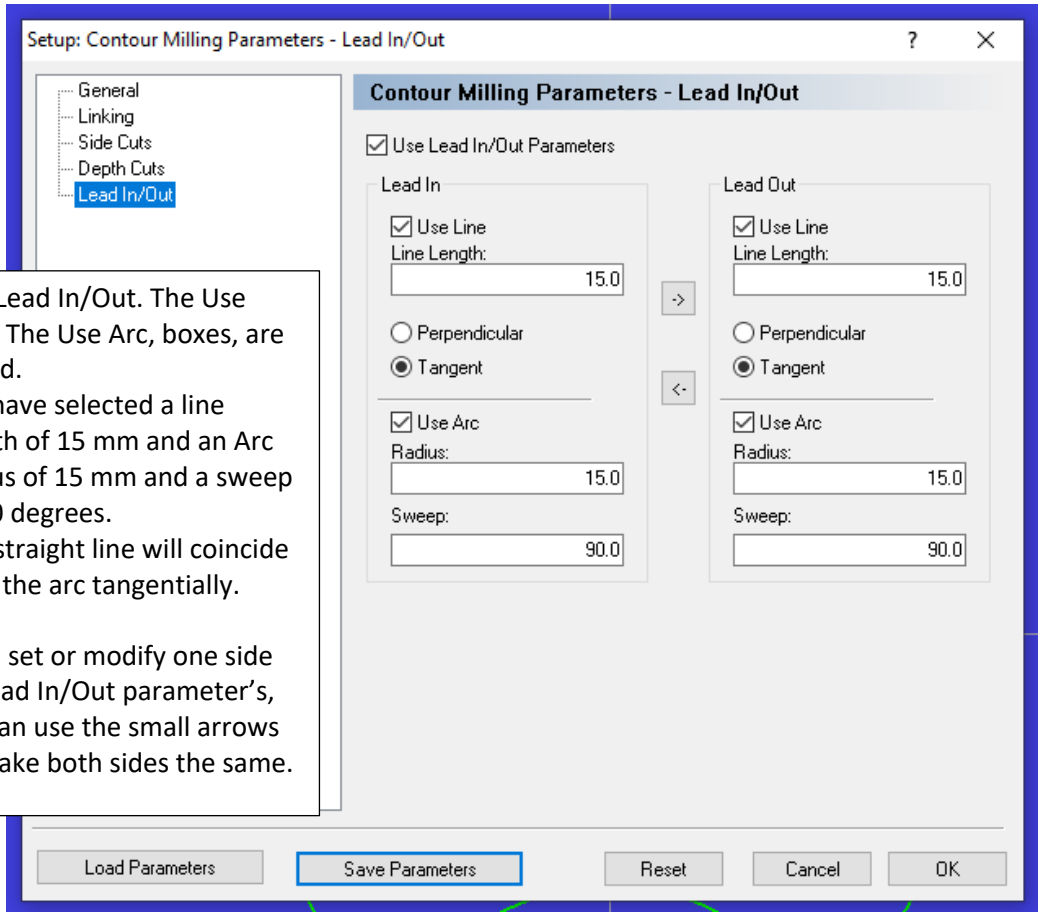
See Depth Cuts Parameter settings. As there is a considerable amount of material to remove, we will choose to take depth cuts



Here we have ticked the depth cuts box and selected 2 Roughing depth cuts and no finishing cuts.
The total depth of our profile is 28 mm so we will take 2 step cuts at 14 mm and no finishing cut

We will be using Lead In/Out which will take place off the profile in fresh air, so we do not need to use spiral depth cuts

See Lead In/Out Parameter settings. We have chosen to lead into the profile with a tangential straight line then into an arc onto the profile and to reverse the path on exiting the profile.

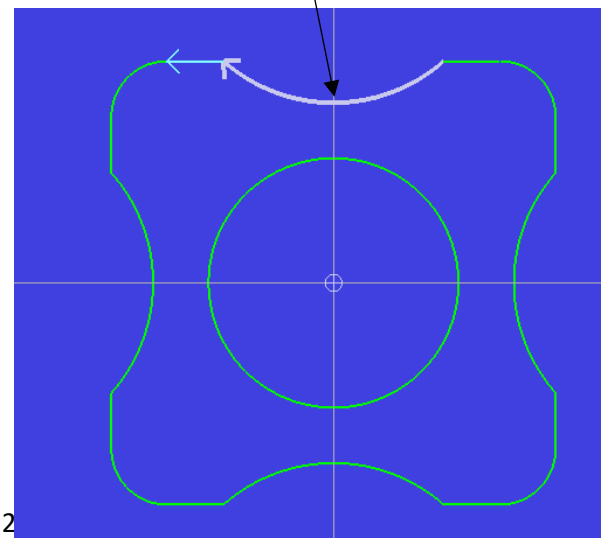


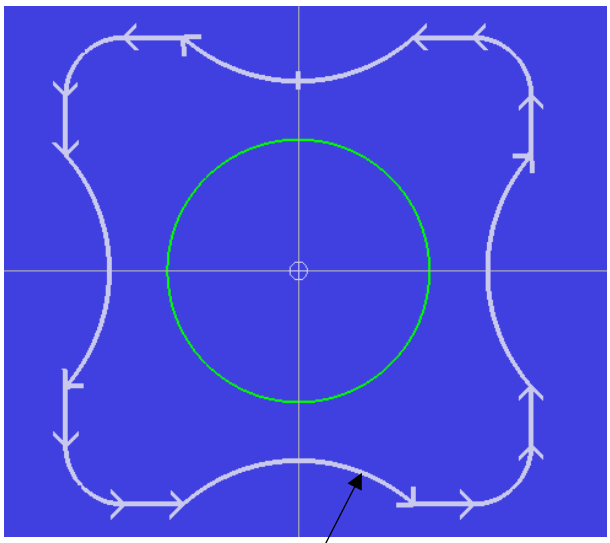
Use Lead In/Out. The Use Line, The Use Arc, boxes, are ticked.
We have selected a line length of 15 mm and an Arc radius of 15 mm and a sweep of 90 degrees.
The straight line will coincide with the arc tangentially.

If we set or modify one side of Lead In/Out parameter's, we can use the small arrows to make both sides the same.

The parameters are set so select the tool path and choose the direction anticlockwise to set a climb milling path. Select the element to start the milling path on and move along the element until the arrow indicates the direction of travel.

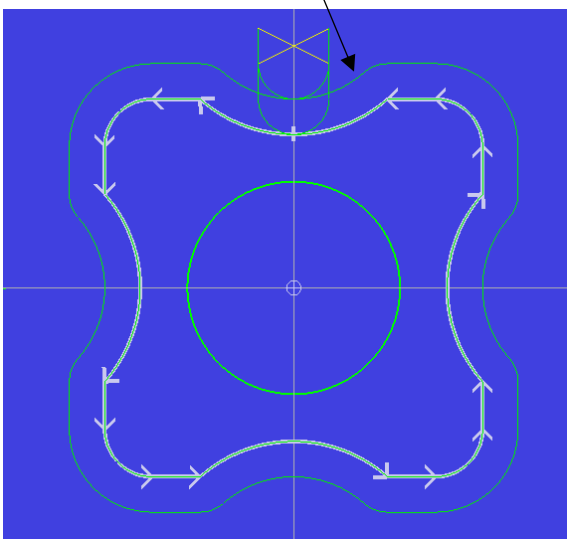
Pick the element to start and move mouse selection to achieve the direction you want



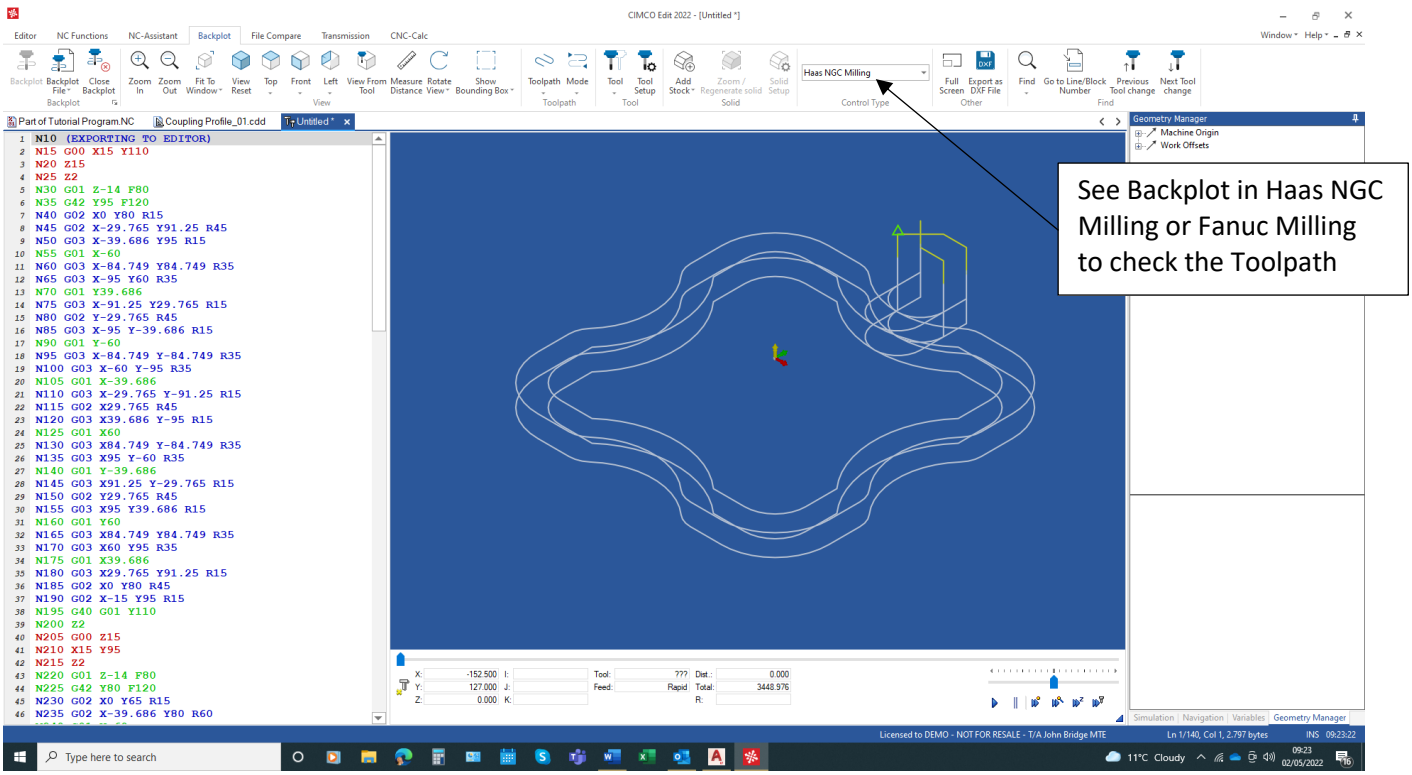


With the Single Step box unticked the tool path will automatically track around the tool path profile.
If it stops at a point, then you will need to investigate the drawing as there may be a fault in element connections or line duplication etc.

Click Show Tool Path, and a visual path image of the tool path will appear. If the path correct as intended then click Export to Editor, otherwise go back to Parameters and modify until the path is acceptable.



The results of the export CNC code will appear on a separate untitled page. After further testing and when correct the CNC code on this page will be pasted into the main program. Evaluate the code first to check the tool path by Backploting. The toolpath generated is reading X, Y coordinated therefore the Backplot evaluation will be with a Milling post processor. We have used Haas Milling and previously set this in the Editor window. The Tool path plotted has two side cuts and two depth cuts and checks out OK. See the resulting Backplot below.



Use the X, Y, milling code to generate the C, X, polar coordinates to create the profile on a Mill-Turn CNC Lathe using the code G112 to switch on the conversion feature and G113 to switch off. See below the code with descriptions of header blocks.

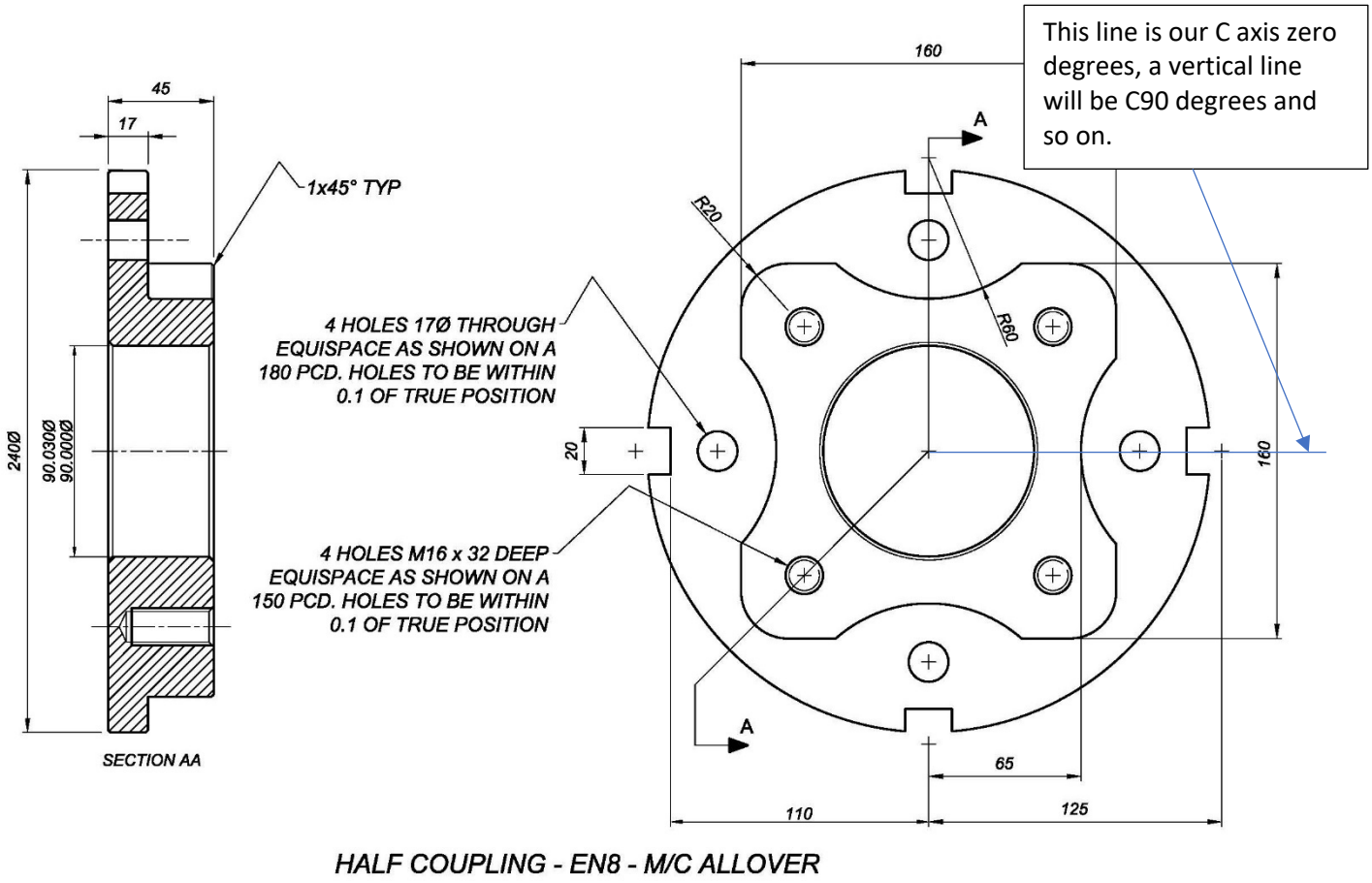
```
(MILL PROFILE)
G21 G40 G98     ;-Safety blocks G21 set mm, G40 cancel CRC, G98 set Feed in mm/min
(CARBIDE END MILL 25 MM DIA)  ;-Tool description
T0606  ;-Tool Call & Offset activate
G0 X250. Z2. M08 ;-Move to position switch on coolant
G97 P1200 M133 ;-G97 Speed in rpm, P1200 = 1200 rpm, M133 = switch on live tool forward
(1ST PASS)
G01 Z-14. F240. ;-Move tool to Z -14.
G17 ;-X-Y interpolation
M154 ;-- engage C axis
G112 ;- Activate Polar transformation from X, Y, to X, C.
G01 X100. Y0. F200
G42 X80.
G02 X95. Y33.541 I45.J0.
G01 Y60.
G03 X60.Y95.I-35. J0.
G01 X33.541
G02 X0.Y80. I-33.541 J30.
G02 X-33.541 Y95. I0. J45.
G01 X-60.
G03 X-95. Y60. I0. J-35.
G01 Y33.541
G02 Y-33.541 I-30. J-33.541
G01 Y-60.
G03 X-60. Y-95. I35. J0.
G01 X-33.541
G02 X33.541 I33.541 J-30.
G01 X60.
G03 X95. Y-60. I0. J35.
G01 Y-33.541
G02 X80. Y0. I30. J33.541
G01 G40 X100.
G01 G42 X65.
G02 X80. Y39.686 I60. J0.
G01 Y60.
G03 X60. Y80. I-20. J0.
```

```
G01 X39.686
G02 X-39.686 I-39.686 J45.
G01 X-60.
G03 X-80. Y60. I0. J-20.
G01 Y39.686
G02 Y-39.686 I-45. J-39.686
G01 Y-60.
G03 X-60. Y-80. I20. J0.
G01 X-39.686
G02 X39.686 I39.686 J-45.
G01 X60.
G03 X80. Y-60. I0. J20.
G01 Y-39.686
G02 X65. Y0. I45. J39.686
G01 G40 X100. Y0.
G1 X250 Z25.
(2ND PASS)
G1 Z-28. F240.
G17
G112
G01 X100.Y0.F200
G42 X80.000
G02 X95. Y33.541 I45.J0.
G01 Y60.
G03 X60.Y95.I-35. J0.
G01 X33.541
G02 X0.Y80. I-33.541 J30.
G02 X-33.541 Y95. I0. J45.
G01 X-60.
G03 X-95. Y60. I0. J-35.
G01 Y33.541
G02 Y-33.541 I-30. J-33.541
G01 Y-60.
G03 X-60. Y-95. I35. J0.
G01 X-33.541
G02 X33.541 I33.541 J-30.
```

```
G01 X60.
G03 X95. Y-60. I0. J35.
G01 Y-33.541
G02 X80. Y0. I30. J33.541
G01 G40 X100.
G01 G42 X65.
G02 X80. Y39.686 I60. J0.
G01 Y60.
G03 X60. Y80. I-20. J0.
G01 X39.686
G02 X-39.686 I-39.686 J45.
G01 X-60.
G03 X-80. Y60. I0. J-20.
G01 Y39.686
G02 Y-39.686 I-45. J-39.686
G01 Y-60.
G03 X-60. Y-80. I20. J0.
G01 X-39.686
G02 X39.686 I39.686 J-45.
G01 X60.
G03 X80. Y-60. I0. J20.
G01 Y-39.686
G02 X65. Y0. I45. J39.686
G01 G40 X100. Y0.
G113 ;- De-activate Polar transformation from X, Y, to X, C.
M155 ;-- disengage C axis
M135 ;-- Switch off live tool
G1 X250 Z25.
G0 G28 U0 W0. ;-move in rapid to safe tool change position.
M01 ;-Option Stop
```

DRILLING & TAPPING OPERATIONS

We can now program using the Editor to conduct the drilling and tapping operations line by line to enter the hole coordinates, then add the drilling and taping canned cycles from NC-Assistant. Let us look again at the drawing.



The coordinates for drilling and tapping can be programmed directly in X, C, and do not need to be transformed from X, Y, coordinates.

The coordinates for the four holes 17 mm diameter.

G00 X180. C0

C90.

C180.

C270.

The coordinates for drilled & tapped holes for M16 thread.

G00 X150. C45.

C135.

C225.

C315.

Using these coordinates, we can work up programs for drilling and tapping the holes.

(DRILL 4 HOLES 17 MM X 40 DEEP)

G18 G21 G40 G99

G50 S1500

G0 G28 U0 W0.

(17 MM CARBIDE DRILL)

T0505

M154

P1000 M133

G00 X180. C0

Z10.

G83 X180. C0 Z-84. R1. Q5. F.12 ;-- Peck drilling cycle

C90.

C180.

C270.

G80

M155

M135

M09

G28 H0

G0 G28 U0 W0

M01

Continue programming the drilling and tapping operation using the coordinates above and canned cycles from the NC-Assistant. See below:

```
(DRILL 4 HOLES 14.7 MM X 40 DEEP)
G18 G21 G40 G99
G0 G28 U0 W0.
(14.7 MM CARBIDE DRILL)
T0606
M154
M19
P1500 M133
G00 X180. C45.
Z30.
G83 X180. C45. Z-40. R2. Q5. F.12;-- Peck drilling cycle
C135.
C225.
C315.
G80
M155
M135
M09
G28 H0
G0 G28 U0 W0
M01

(TAP 4 HOLES M16 X 32 DEEP)
G18 G21 G40 G99
G0 G28 U0 W0.
(M16 SPIRAL FLUTE TAP)
T0707
M154
M19
P1000 M133
G00 X180. C45.
Z30.
G84 X180. C45. Z-32. R10. F2.0 ;-- Tapping cycle
C135.
C225.
C315.
```

```
G80
M155
M135
M09
G28 H0
G0 G28 U0 W0
M01
```

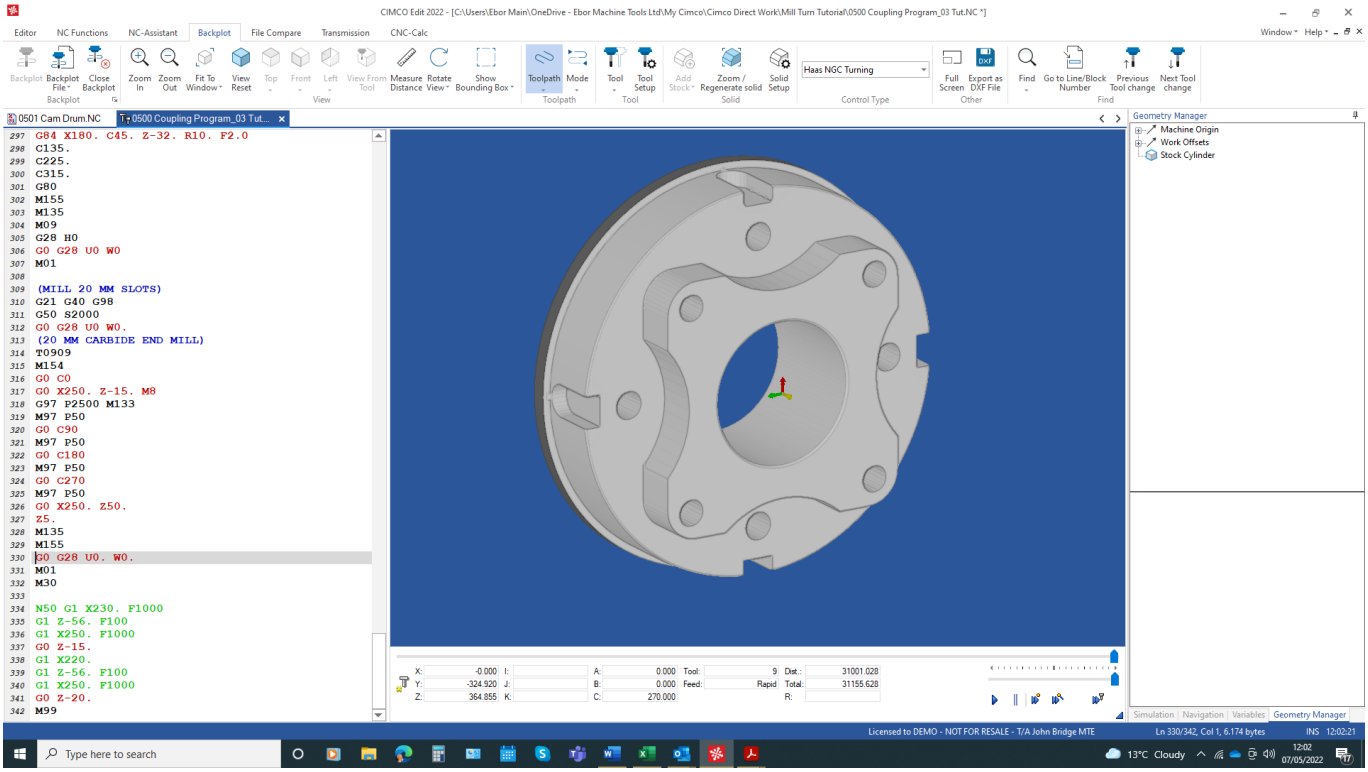
20 MM AXIAL SLOTS IN OD

The last operation is to mill the 20 mm slots in in the outside diameter. This is a simple programming procedure, and we can use direct C, X axis commands to position the tool for milling in the Z axis direction. As there are 4 slots, we will use the subroutine M47 to carry out the machining moves. Using this facility means that the subroutine will be set outside the program after the M30 end of program command, it will be labelled with a block number and ended with an M99. See blow:

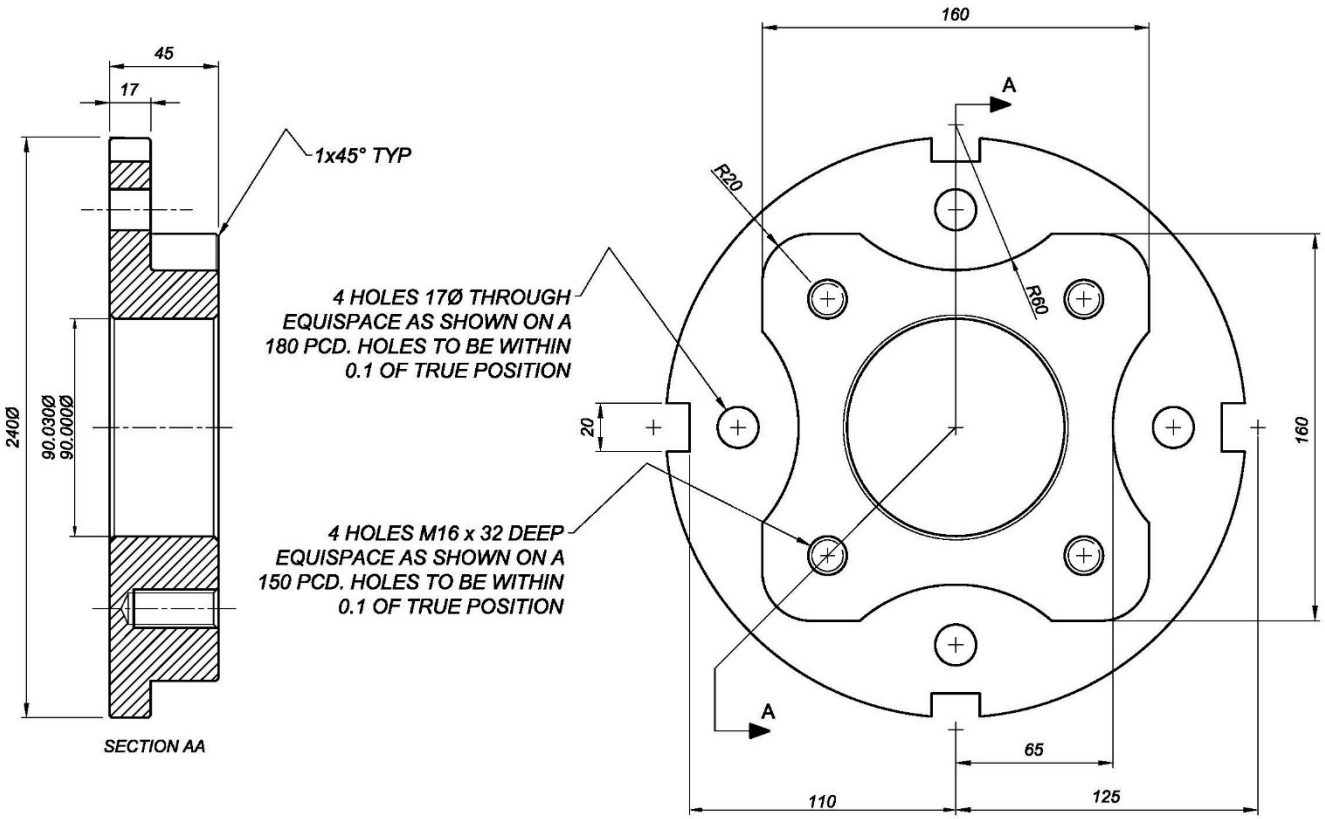
```
(MILL 20 MM SLOTS)
G21 G40 G98
G50 S2000
G0 G28 U0 W0.
(20 MM CARBIDE END MILL)
T0909
M154
G0 C0
G0 X250. Z-15. M8
G97 P2500 M133
M97 P50 ;-- call local subroutine beginning with N50
G0 C90
M97 P50 ;-- call local subroutine beginning with N50
G0 C180
M97 P50 ;-- call local subroutine beginning with N50
G0 C270
M97 P50 ;-- call local subroutine beginning with N50
G0 X250. Z50.
Z5.
M135
M155
```

```
G0 G28 U0. W0.
M01
M30 ;-- end of program and rewind to the start

N50 G1 X230. F1000 ;-- subroutine start
G1 Z-56. F100
G1 X250. F1000
G0 Z-15.
G1 X220.
G1 Z-56. F100
G1 X250. F1000
G0 Z-20.
M99 ;-- subroutine end
```

BACKPLOT TESTING THE WHOLE PROGRAM



Compare Backplot to the drawing and it can be seen that the Machining process complies with the drawing.



HALF COUPLING - EN8 - M/C ALLOVER

THE COMPLETE PROGRAM

```
%
O0501 (MILL TURN TUTORIAL)
( STOCK TURN OD250 ID0 L80 PZ2 )
( WCS ID1 X0.000 Y0.000 Z0.000 )
(OPERATION 1 TURN/MILL COUPLING END)
(----------------------)
( LTOOL 1 "ISO_TURNING" INSERT=C STYLE=L AO0 CR0.8 CPI=T3 FHD4.4 FW32 HL35 HAT80 ICD12.7 US=UM O=OL LS100 SW25 EL12.896 UHW8.66 )
( LTOOL 2 "ISO_LATHE_DRILLING" INSERT=COMMON STYLE=C AO90 BL100 CPI=T8 D50 FL60 US=UM AD50 DMM50 LS50 SD50 SL60 TL0 AT179 )
( LTOOL 3 "ISO_BORING" INSERT=C STYLE=F AO90 CR0.8 CPI=T2 FHD4.4 HL35 HAT80 HW47 ICD12.7 US=UM O=OL DMM40 LS225 EL12.896 UHW32 )
( T 4 "END MILL" HOLDER=9 AO90 BL60 CD0 CR0 CPI=T8 EMCT=FEM D25 FL35 US=UM AD25 SD25 SL40 TL0 )
( LTOOL 5 "ISO_LATHE_DRILLING" INSERT=COMMON STYLE=C AO90 BL80 CPI=T8 D17 FL20 US=UM AD17 DMM25 LS30 SD17 SL25 TL0 AT140 )
( LTOOL 6 "ISO_LATHE_DRILLING" INSERT=COMMON STYLE=C AO90 BL80 CPI=T8 D14.7 FL20 US=UM AD14.7 DMM25 LS30 SD14.7 SL25 TL0 AT140 )
( LTOOL 7 "ISO_LATHE_DRILLING" INSERT=TAPR STYLE=C AO90 BL60 CPI=T8 D16 FL35 US=UM AD16 DMM25 LS50 SD16 SL35 TL0 TP2 AT140 )
( T 9 "END MILL" HOLDER=8 AO0 BL60 CD0 CR0 CPI=T8 EMCT=FEM D20 FL20 US=UM AD20 SD20 SL40 TL0 )
( HOLDER BEGIN 8 "" UM )
( 60, 60, 2 )
( 60, 56, 2 )
( 56, 60, 2 )
( 60, 60, 2 )
( 45, 45, 10 )
( 34, 34, 20 )
( 34, 30, 2 )
( HOLDER END )
( HOLDER BEGIN 9 "" UM )
( 60, 60, 2 )
( 60, 56, 2 )
( 56, 60, 2 )
```

```
( 60, 60, 2 )
( 45, 45, 10 )
( 34, 34, 20 )
( 34, 30, 2 )
( HOLDER END )
G18 G21 G99
G50 S2000
G00 G28 U0 W0.
(FACE & TURN WMNG 08)
T0101
G54 G97 S250 M3
G0 X255. Z0. M08
G96 S180 F.12
G1 X-1.6
Z2.
G0 X250.
G71 P1 Q2 D3. U0.5 W0.05 F0.3
N1 G0 G42 X210.
G1 Z-28. F0.15
G1 X238..
G1 X240. Z-29.
G1 Z-68.
N2 G40 X250.
G0 Z2. M09
G0 G28 U0 W0.
M5
M01

(UDRILL 50MM DIAMETER)
G21 G40 G99
G00 X250. Z100.
(UDRILL 50MM)
T0202
G97 S750 M03
G00 X0. Z3. M08
G01 Z-85. F.08
```

```
G01 Z3. F2.
G00 Z3.
G0 G28 U0 W0.
M01

(ROUGH BORE TO 90 DIA)
G21 G40 G99
G50 S1500
G28 U0 W0.
G00 X250. Z100.
(32 BORING BAR)
T0303
G00 X50. Z3. M08
G96 S200 M3
G71 D3. F.25 I-.5 K.15 P30 Q40 S200 U-1. W.15
G00 Z3.
G0 G28 U0 W0.
M01
GOTO 50
N30 G00 X98 Z2. F.25
G01 X90. Z-2.
Z-85.
N40 G40 X50.
N50 G00 Z3.
G00 G28 U0 W0.
M01

(MILL PROFILE)
(CARBIDE END MILL 25 MM DIA)
T0404
G21 G40 G98
G00 X15.000 Y110. M08
G97 P1200 M133
M154
G17
G112
```

X15. Y110.

Z15.

Z2.

G01 Z-14. F250

G42 Y95. F100

G02 X0. Y80. I-15. J0.

G02 X-29.765 Y91.250 I0. J45.

G03 X-39.686 Y95. I-9.922 J-11.250

G01 X-60.

G03 X-84.749 Y84.749 I0. J-35.

G03 X-95. Y60. I24.749 J-24.749

G01 Y39.686

G03 X-91.250 Y29.765 I15. J0.

G02 Y-29.765 I-33.750 J-29.765

G03 X-95. Y-39.686 I11.250 J-9.922

G01 Y-60.

G03 X-84.749 Y-84.749 I35. J0.

G03 X-60. Y-95. I24.749 J24.749

G01 X-39.686

G03 X-29.765 Y-91.250 I0. J15.

G02 X29.765 I29.765 J-33.750

G03 X39.686 Y-95. I9.922 J11.250

G01 X60.

G03 X84.749 Y-84.749 I0. J35.

G03 X95. Y-60. I-24.749 J24.749

G01 Y-39.686

G03 X91.250 Y-29.765 I-15. J0.

G02 Y29.765 I33.750 J29.765

G03 X95. Y39.686 I-11.250 J9.922

G01 Y60.

G03 X84.749 Y84.749 I-35. J0.

G03 X60. Y95. I-24.749 J-24.749

G01 X39.686

G03 X29.765 Y91.250 I0. J-15.

G02 X0. Y80. I-29.765 J33.750

G02 X-15. Y95. I0. J15.

G40 G01 Y110.

Z2.

G00 Z15.

X15. Y95.

Z2.

G01 Z-14. F250

G42 Y80. F100

G02 X0. Y65. I-15. J0.

G02 X-39.686 Y80. I0. J60.

G01 X-60.

G03 X-74.142 Y74.142 I0. J-20.

G03 X-80. Y60. I14.142 J-14.142

G01 Y39.686

G02 Y-39.686 I-45. J-39.686

G01 Y-60.

G03 X-74.142 Y-74.142 I20. J0.

G03 X-60. Y-80. I14.142 J14.142

G01 X-39.686

G02 X39.686 I39.686 J-45.

G01 X60.

G03 X74.142 Y-74.142 I0. J20.

G03 X80. Y-60. I-14.142 J14.142

G01 Y-39.686

G02 Y39.686 I45. J39.686

G01 Y60.

G03 X74.142 Y74.142 I-20. J0.

G03 X60. Y80. I-14.142 J-14.142

G01 X39.686

G02 X0. Y65. I-39.686 J45.

G02 X-15. Y80. I0. J15.

G40 G01 Y95.

Z2.

G00 Z15.

X15. Y110.

Z2.

G01 Z-28. F250

G42 Y95. F100

G02 X0. Y80. I-15. J0.

G02 X-29.765 Y91.250 I0. J45.

G03 X-39.686 Y95. I-9.922 J-11.250

G01 X-60.

G03 X-84.749 Y84.749 I0. J-35.

G03 X-95. Y60. I24.749 J-24.749

G01 Y39.686

G03 X-91.250 Y29.765 I15. J0.

G02 Y-29.765 I-33.750 J-29.765

G03 X-95. Y-39.686 I11.250 J-9.922

G01 Y-60.

G03 X-84.749 Y-84.749 I35. J0.

G03 X-60. Y-95. I24.749 J24.749

G01 X-39.686

G03 X-29.765 Y-91.250 I0. J15.

G02 X29.765 I29.765 J-33.750

G03 X39.686 Y-95. I9.922 J11.250

G01 X60.

G03 X84.749 Y-84.749 I0. J35.

G03 X95. Y-60. I-24.749 J24.749

G01 Y-39.686

G03 X91.250 Y-29.765 I-15. J0.

G02 Y29.765 I33.750 J29.765

G03 X95. Y39.686 I-11.250 J9.922

G01 Y60.

G03 X84.749 Y84.749 I-35. J0.

G03 X60. Y95. I-24.749 J-24.749

G01 X39.686

G03 X29.765 Y91.250 I0. J-15.

G02 X0. Y80. I-29.765 J33.750

G02 X-15. Y95. I0. J15.

G40 G01 Y110.

Z2.

G00 Z15.

X15. Y95.

Z2.

G01 Z-28. F250

G42 Y80. F100

G02 X0. Y65. I-15. J0.

G02 X-39.686 Y80. I0. J60.

G01 X-60.

G03 X-74.142 Y74.142 I0. J-20.

G03 X-80. Y60. I14.142 J-14.142

G01 Y39.686

G02 Y-39.686 I-45. J-39.686

G01 Y-60.

G03 X-74.142 Y-74.142 I20. J0.

G03 X-60. Y-80. I14.142 J14.142

G01 X-39.686

G02 X39.686 I39.686 J-45.

G01 X60.

G03 X74.142 Y-74.142 I0. J20.

G03 X80. Y-60. I-14.142 J14.142

G01 Y-39.686

G02 Y39.686 I45. J39.686

G01 Y60.

G03 X74.142 Y74.142 I-20. J0.

G03 X60. Y80. I-14.142 J-14.142

G01 X39.686

G02 X0. Y65. I-39.686 J45.

G02 X-15. Y80. I0. J15.

G40 G01 Y95.

G00 Z25.

G113

M155

M135

G00 Z50.

G0 G28 U0 W0.

M01

(DRILL 4 HOLES 17 MM X 40 DEEP)

```
G18 G21 G40 G99
G0 G28 U0 W0.
(17 MM CARBIDE DRILL)
T0505
M154
M19
P1500 M133
G00 X180. C0
Z10.
G83 X180. C0 Z-84. R1. Q5. F.12
C90.
C180.
C270.
G80
M155
M135
M09
G28 H0
G0 G28 U0 W0
M01

(DRILL 4 HOLES 14.7 MM X 40 DEEP)
G18 G21 G40 G99
G0 G28 U0 W0.
(14.7 MM CARBIDE DRILL)
T0606
M154
M19
P1500 M133
G00 X180. C45.
Z30.
G83 X180. C45. Z-40. R2. Q5. F.12
C135.
C225.
C315.
G80
```

```
M155
M135
M09
G28 H0
G0 G28 U0 W0
M01

(TAP 4 HOLES M16 X 32 DEEP)
G18 G21 G40 G99
G0 G28 U0 W0.
(M16 SPIRAL FLUTE TAP)
T0707
M154
M19
P1000 M133
G00 X180. C45.
Z30.
G84 X180. C45. Z-32. R10. F2.0
C135.
C225.
C315.
G80
M155
M135
M09
G28 H0
G0 G28 U0 W0
M01

(MILL 20 MM SLOTS)
G21 G40 G98
G50 S2000
G0 G28 U0 W0.
(20 MM CARBIDE END MILL)
T0909
M154
```

```
G0 C0
G0 X250. Z-15. M8
G97 P2500 M133
M97 P50
G0 C90
M97 P50
G0 C180
M97 P50
G0 C270
M97 P50
G0 X250. Z50.
Z5.
M135
M155
G0 G28 U0. W0.
M01
M30

N50 G1 X230. F1000
G1 Z-56. F100
G1 X250. F1000
G0 Z-15.
G1 X220.
G1 Z-56. F100
G1 X250. F1000
G0 Z-20.
M99
```

This concludes the tutorial for the Coupling Mill/Turn part.