



Siemens 840D Training Course

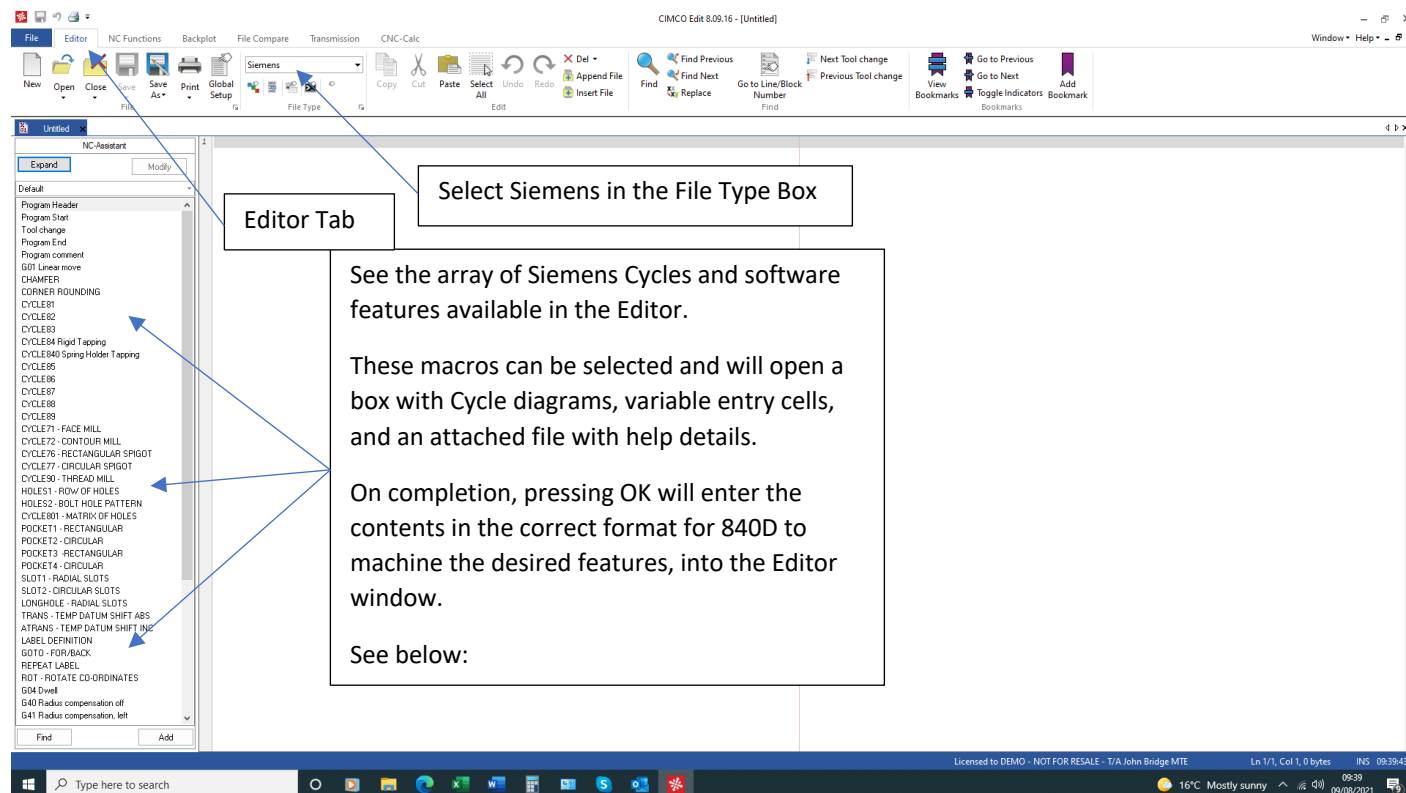


This document is made available as a preliminary version (draft).
Questions and feedback should be sent to support@cimco.com

CIMCO SIEMENS 840D TRAINING

The aim here is to familiarize the trainee using the Cimco Editor to create CNC programs in Siemens ISO and mnemonic format for 840D CNC controls and to test their validity using the Tool Path and Solid Animation graphic Backplotting. This will give the trainee the confidence that the programs can be transferred from the PC to the CNC control for operation on a machine with some assurance that the program will machine the part to the desired dimensions shown on the drawing, without collision on the machine and produce a quality part. This training article assumes some basic knowledge of CNC programming.

Let us first review NC-Assistant in the Editor to see the Siemens Cycles and other software features that are available to us within the Editor.



G01 Linear move	8	---
CHAMFER	9	---
CORNER ROUNDING	10	---
CYCLE81	11	---
CYCLE82	12	G90
CYCLE83	13	G54
CYCLE84 Rigid Tapping	14	SUI
CYCLE840 Spring Holder Tapping	15	T10
CYCLE85	16	G00
CYCLE86	17	S20
CYCLE87	18	Z50
CYCLE88	19	G1
CYCLE89	20	PRG
CYCLE71 - FACE MILL	21	G91
CYCLE72 - CONTOUR MILL	22	G90
CYCLE76 - RECTANGULAR SPIGOT	23	G00
CYCLE77 - CIRCULAR SPIGOT	24	G1
CYCLE90 - THREAD MILL	25	Y10
HOLES1 - ROW OF HOLES	26	X80
HOLES2 - BOLT HOLE PATTERN	27	G3
CYCLE801 - MATRIX OF HOLES	28	G1
POCKET1 - RECTANGULAR	29	G3
POCKET2 - CIRCULAR	30	G1
POCKET3 - RECTANGULAR	31	Y0
POCKET4 - CIRCULAR	32	X21
SLOT1 - RADIAL SLOTS	33	G3
SLOT2 - CIRCULAR SLOTS	34	G1
LONGHOLE - RADIAL SLOTS	35	G3
TRANS - TEMP DATUM SHIFT ABS	36	G1
ATrans - TEMP DATUM SHIFT INC	37	Y50
LABEL DEFINITION	38	G40
GOTO - FOR/BACK	39	ENI
REPEAT LABEL	40	REI
ROT - ROTATE CO-ORDINATES		
G04 Dwell		
G00 Rapid		

CIMCO & SIEMENS 840D

The Cimco Editor provides macros for many of the most used Siemens cycles and programming aids that are resident in the 840D CNC controls and that can be seen in the list in the Cimco NC-Assistant as displayed across the page.

These Siemens cycles and programming aids are supported with Tool Path and Solid Animation Backplot Graphics in the Cimco Editor.

A programmer now has the option of using a Siemens Cycle from the 840D CNC control or using CNC Calc CAD/CAM or using the Cimco / Windows Edit facility to create programs that can be validated by the Backplot Graphics.

Let's see how to use the NC-Assistance on the following page.

Let's look at drilling some holes in a plate.

The screenshot shows the CIMCO Edit 8.09.16 software interface. The top menu bar includes File, Editor, NC Functions, Backplot, File Compare, Transmission, and CNC-Calc. The toolbar contains various icons for file operations, viewing, and tool setup. The main window is divided into several panes:

- NC-Assistant:** Contains a list of macro categories (Program Header, Program Start, Tool change, Program End, Program comment, G01 Linear move, CHAMFER, CORNER ROUNDING, CYCLE81, CYCLE82, CYCLE83, CYCLE84 Rigid Tapping, CYC73 FR40 Spinning Holder Tapping) and a description field.
- Code Editor:** Displays the following code:


```
1 ;-- DRILL 12MM HOLES
2 ;-- 12MM DRILL
3 T D1
4 G00 X0 Y0
5 Z30
6 G00 X0. Y0.
```
- Macro Box (CYCLE81):** A dialog box for defining the drilling cycle parameters:
 - Retract Plane (RTP): 10
 - Reference Plane (RFP): 0
 - Safety Approach Distance (SDIS): 2
 - Drilling Depth (DP): -5
 - Drilling Depth Relative (DPR): (Optional parameter)
- Diagram:** A 3D diagram illustrating the drilling cycle. It shows a drill bit moving down into a workpiece. Key points are labeled: RTP (Retract Plane), RFP+SDIS (Reference Plane + Safety Approach Distance), RFP (Reference Plane), and DP=RFP-DPR (Drilling Depth).

Annotations and callouts:

- "Complete the entry Cells in the macro box. Click OK" points to the OK button in the Macro Box.
- "Preparatory blocks and the cursor set in position for inserting the drilling cycle" points to the code editor.
- "Double click CYCLE81 to open the Macro Box" points to the CYCLE81 macro in the NC-Assistant list.
- "Click Attachment if you need help with completing the entry cells" points to the Attachment button in the Macro Box.

The screenshot shows the software interface after the drilling cycle has been set up. The top toolbar includes icons for Open, Close, Save, Save As, Print, Global Setup, File Type, Copy, Cut, and Paste. The main window displays the following code:

```
1 ;-- DRILL 12MM HOLES
2 ;-- 12MM DRILL
3 T7 D1
4 G00 X0 Y0
5 Z30
6 G00 X0. Y0.
7 MCALL CYCLE81(10,0,2,-5,)
8
```

An annotation points to the MCALL command, stating: "Result is the CYCLE81 drilling sub routine with an MCALL (Modal) command will activate the cycle at the arrival in position of the moves programmed below."

Let's add some holes using the Bolt Hole Pattern feature

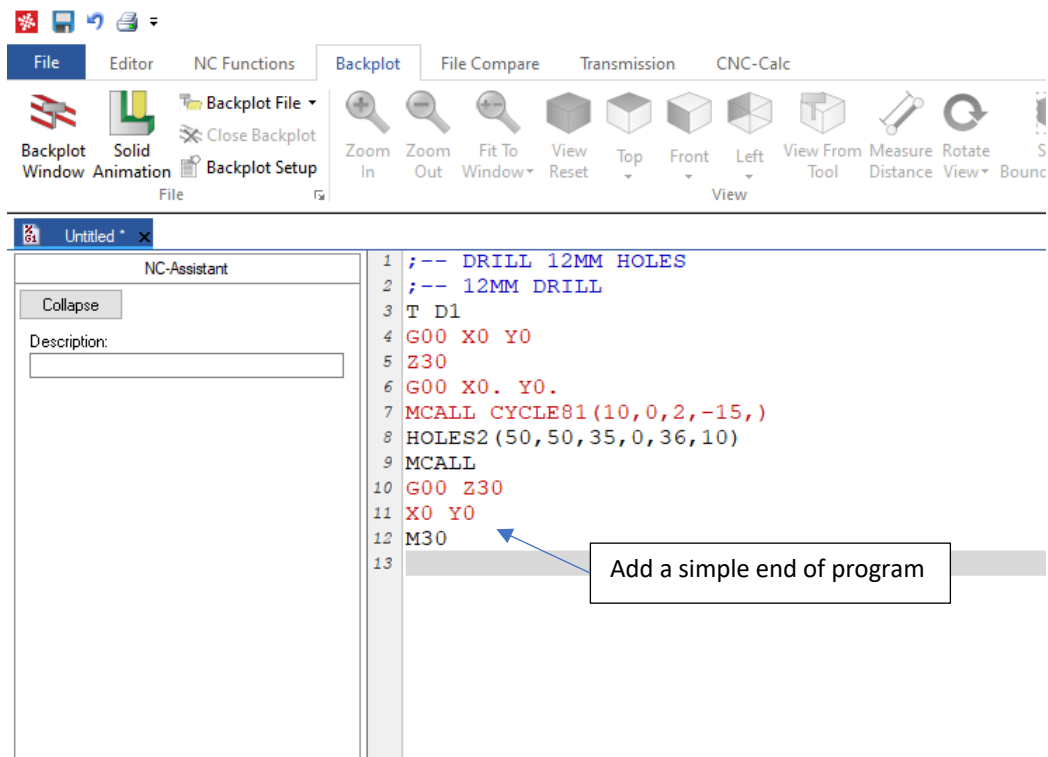
Select HOLES2 to program a Bolt Hole Pattern

Complete the cells to achieve the Bolt Hole Pattern you require

If you need help with HOLES2 click the attached button and the explanation of the cycle will open.

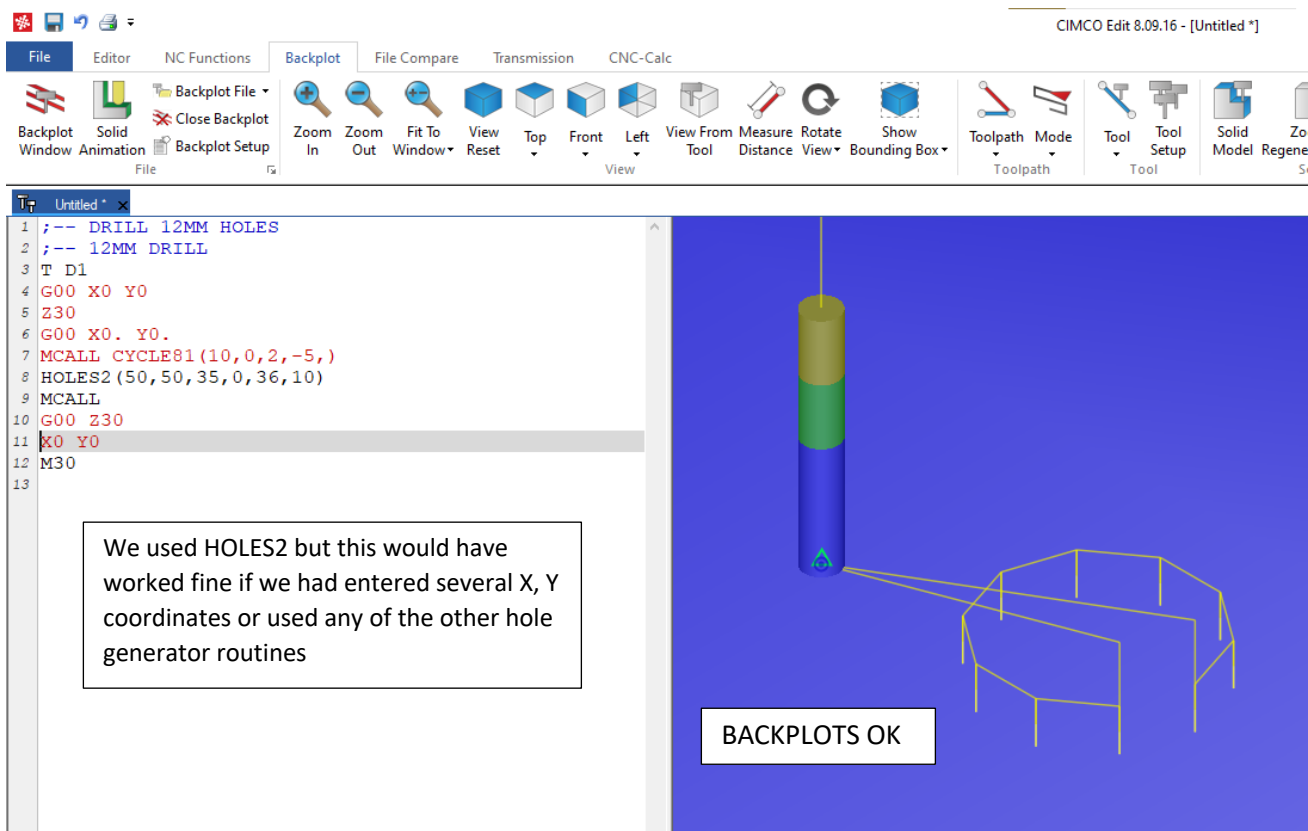
See the HOLES2 cycle followed by MCALL to cancel the Modal Cycle call.

Complete the program



Let's Backplot now to test our program

SEE BELOW:



When we need a programming feature like, Corner Rounding for example, we may need the help to make entries in the cells

Modify: CORNER ROUNDING

Parameters for 'CORNER ROUNDING'

- Radius Between 2 Straight Lines (RND)
- Feed Rate for Radius (FRC)

Optional parameter

ROUNDING BETWEEN 2 STRAIGHT LINES

E.G. G17

FRC = FEED RATE FOR CHAMFER
IF NOT ENTERED THEN
ACTIVE FEED RATE APPLYS

Select Corner Rounding to open the macro box

Click this button to open Rounding Help

See the Help window for technical details of the feature to assist in making the correct entries into the cells.

10.12 Chamfer, rounding (CHF, CHR, RND, RNDM, FRC, FRM)

Contour corners within the active working plane can be executed as roundings or chamfers. For optimum surface quality, a separate feedrate can be programmed for chamfer/rounding. If a feedrate is not programmed, the standard path feedrate F will be applied.

The "Modal rounding" function can be used to round multiple contour corners in the same way one after the other.

Syntax

Chamfer the contour corner:
G... X... Z... CHF/CHR=<value> FRC/FRM=<value>
G... X... Z...

Round the contour corner:
G... X... Z... RND=<value> FRC=<value>
G... X... Z...

Modal rounding:
G... X... Z... RND=<value> FRC=<value>
...
RND=0

Note

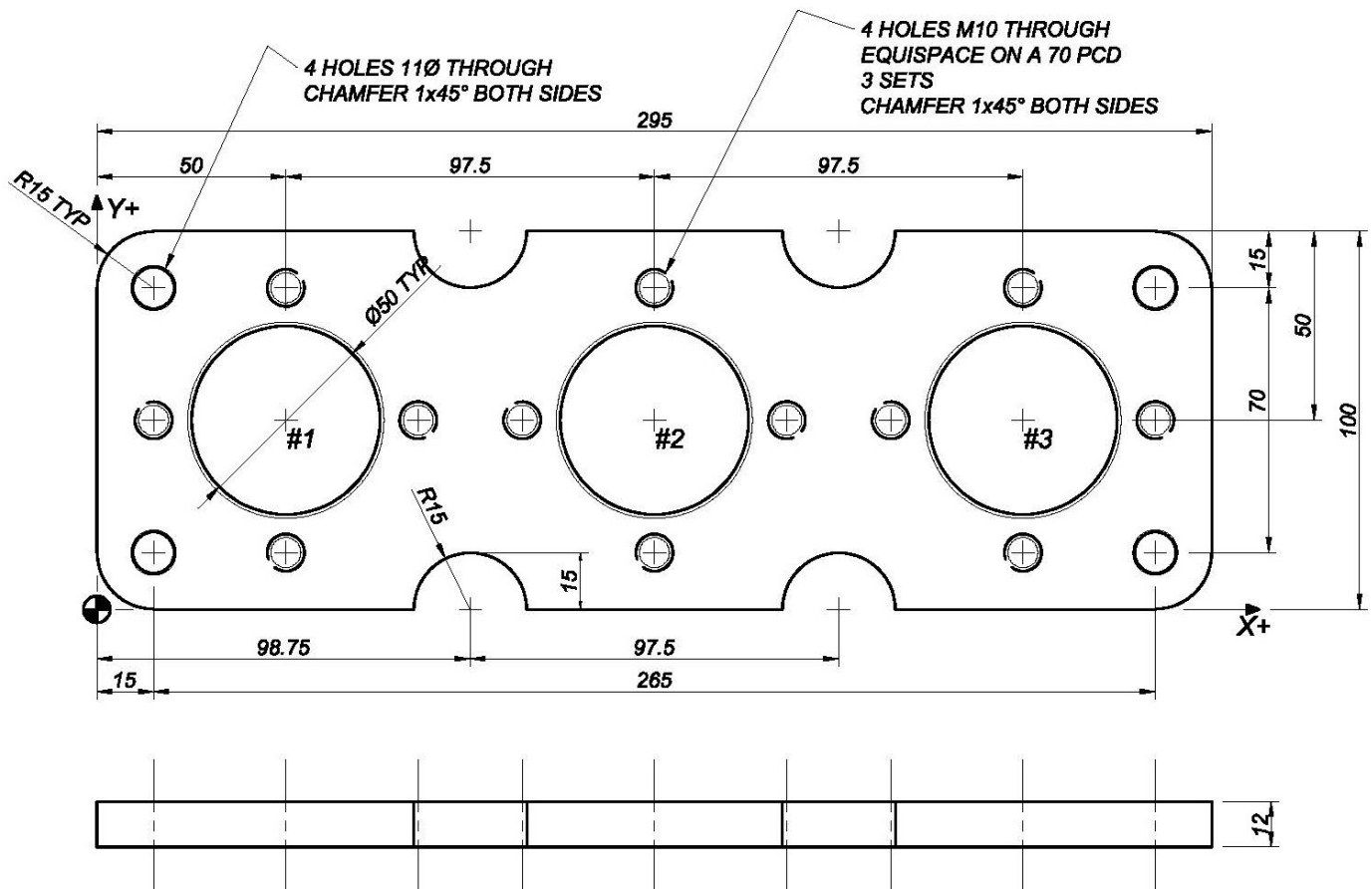
The technology (feedrate, feedrate type, M commands, etc.) for chamfer/rounding is derived from either the previous or the next block dependent on the setting of bit 0 in machine data MD20201 SMC_CHFRND_MODE_MASK (chamfer/rounding behavior). The recommended

ACTIVE FEED RATE APPLYS

Rounding Help opened in resize. Initially, set the size to make it suit your monitor and close the box. It will remember the size of the readable resize box when you open it again.

Example 1 Operation 1

Now we know how to use the Siemens NC-Assistant let us look at a practical program for this Baffle Plate part as below. The rough part will be held in a precision vice to machine the holes and then on a fixture to machine around the periphery. It would be a good idea to print this page so that you have a copy of the drawing to refer to as we proceed to program the Plate. Programming zero is indicated by the checkered circle.



**BAFFLE PLATE - MILD STEEL - FACE MILL TOP SURFACE
PROGRAM No.1 50Ø HOLE AND M10 TAPPED HOLES USE
DATUM SHIFT (TRANS) TO MACHINE HOLES 2 AND 3.
LOCATE ON A FIXTURE AND MACHINE THE OUTSIDE PERIPHERY**

As noted on the drawing we will program the 50 mm hole number one and its tapped holes and use the TRANS feature to make temporary datum shifts to machine the other holes 2 and 3 but first we will mill the top face to make a flat face at Z0.

SEE BELOW:

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

Backplot Window Solid Animation Backplot File Close Backplot Backplot Setup

Zoom In Zoom Out Fit To Window View Reset Top Front Left View From Tool Measure Distance Rotate View Show Bounding

Untitled * x

NC-Assistant

Collapse

Description:

```

1 ;-- FACE MILL SURFACE
2 ;-- USE 50 MM FACE MILL
3 T1 D1 ;TOOL AND OFFSET CALLED
4 G00 X0 Y0 ; MOVE X,Y TO POSITION
5 Z30; MOVE Z TO RETRACTION PLAIN
6
7
8
9
10
11

```

Create the blocks above as preliminary commands described by the comments. Prior to opening the macro CYCLE71 Face Milling. There is a macro for a more comprehensive Program Start in NC-Assistant

Open the CYCLE71 Macro

Insert: CYCLE71 - FACE MILL

Parameters for 'CYCLE71 - FACE MILL'

- ☐ Retract Plane (RTP)
- Reference Plane (RFP)
- Safety Clearance (SDIS)
- Depth (DP)
- Starting Point Abscissa (Absolute) (PA)
- Starting Point Abscissa (Incremental) (PO)
- Length of Rectangle 1st Axis Inc (LENG)
- Width of Rectangle 1st Axis Inc (WID)
- Angle of Rectangle (STA) [0 - 180]
- Max Infeed Depth (MID)
- Max Infeed Width (MIDA)
- Retraction Travel in Cutting Dir (FDP)
- Final Machining Allowance Depth (FALD)
- Feedrate for Surface Machining (FFP1)
- Machining Type *1= Roughing, *2= Finishing (VARI) * See Diagram
- Over Travel in Direction of Plane Infeed (FDP1)

* = Optional parameter

Complete the cells as described, use the Help if required by pressing the Attached Button.

FACE MILLING CYCLE71

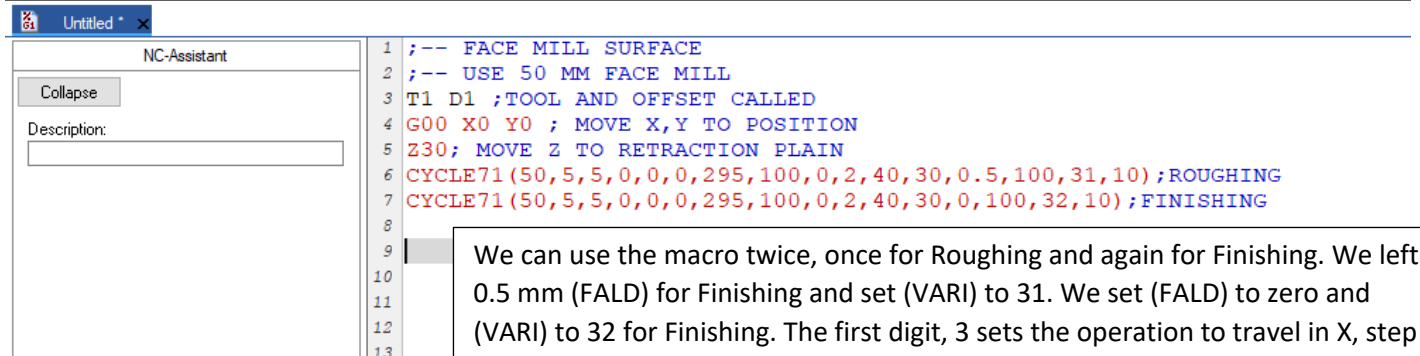
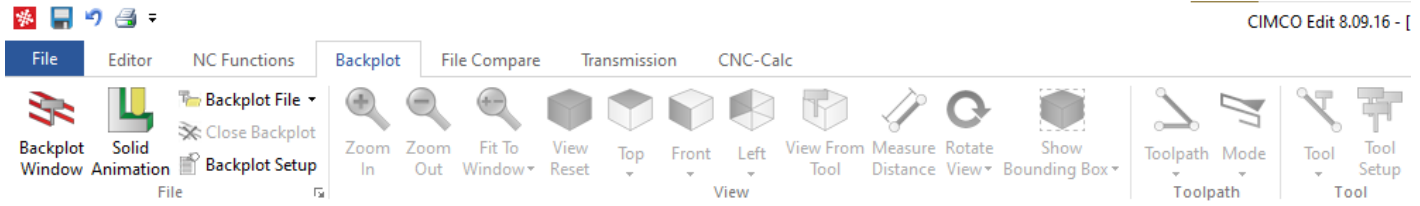
MILLING MOVEMENT WHEN FINISHING IN ALTERNATIVE DIRECTION (MACHINING TYPE 42)

ROUGHING WITH MIDA LARGER THAN THE MILLING RADIUS (MACHINING TYPE 41)

POSSIBLE SOLID MACHINING STRATEGY FOR FACE MILLING
NOTE - *1 = ROUGHING - 2 = FINISHING

Attachment Default Cancel OK

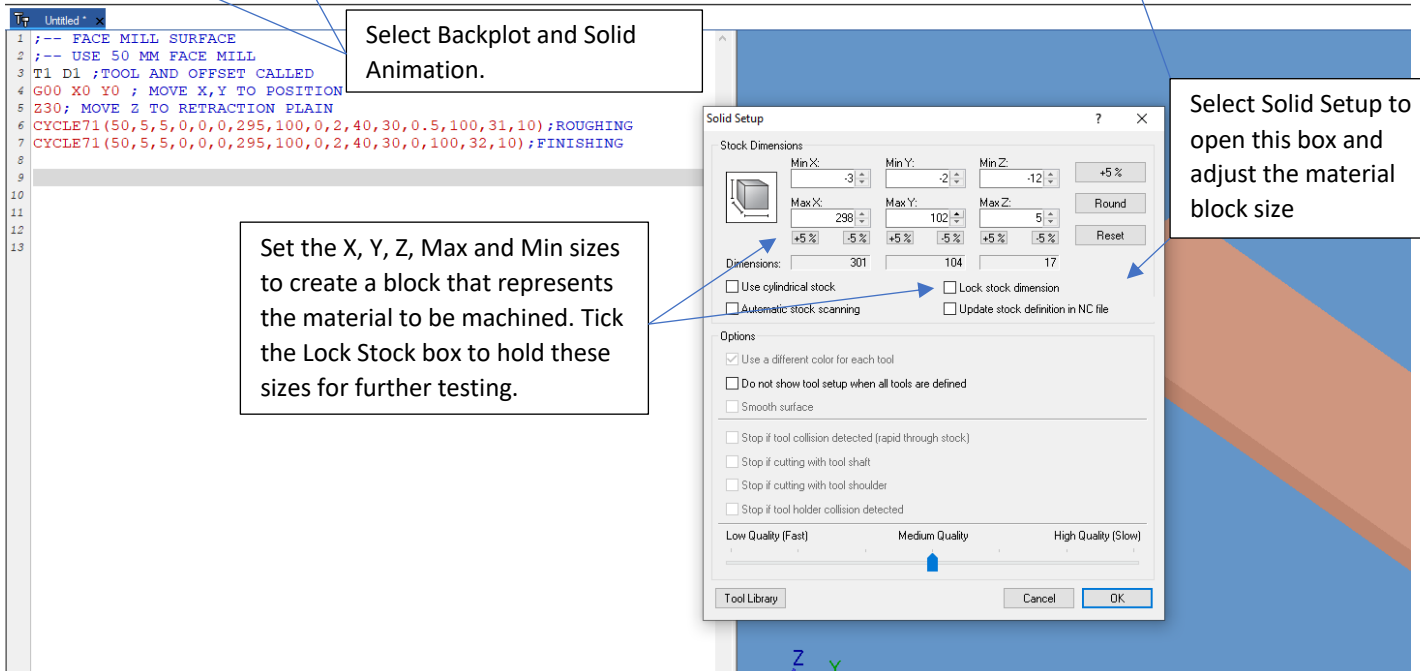
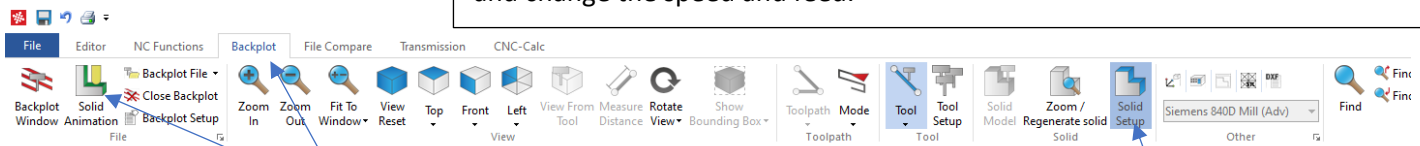
SEE BELOW:



We can use the macro twice, once for Roughing and again for Finishing. We left 0.5 mm (FALD) for Finishing and set (VARI) to 31. We set (FALD) to zero and (VARI) to 32 for Finishing. The first digit, 3 sets the operation to travel in X, step over in Y, if the second digit = 1 sets, Roughing and if the second digit = 2 Finishing.

In later examples we have opted for one cycle only as the difference between roughing and finishing is generally the facility to leave a small amount of material and change the speed and feed.

Now Test in Solid Animation Backplot



Select Backplot and Solid Animation.

Set the X, Y, Z, Max and Min sizes to create a block that represents the material to be machined. Tick the Lock Stock box to hold these sizes for further testing.

Select Solid Setup to open this box and adjust the material block size

Now check the Tooling, SEE BELOW:

The tools will be detected from the program and listed. If the tool description details are incorrect select the correct details from the list and press the "Assign to" button to update the tool details.

When all Tools are correct Click OK

Tool Setup

Type: Face mill Diameter: 50

Description	Type	D	L
End Mill Flat 2MM	End Mill Flat	2	60
End Mill Ball 2MM	End Mill Ball	2	60
End Mill Bull 2MM R0.5	End Mill Bull	2	60
End Mill Flat 3MM	End Mill Flat	3	60
End Mill Ball 3MM	End Mill Ball	3	60
End Mill Bull 3MM R0.8	End Mill Bull	3	60
End Mill Flat 4MM	End Mill Flat	4	60

Buttons: Delete, Modify, Add, Assign to T1*, Load/Save Tool library, Cancel, OK

Save tool information: ☒

Run Backplot:

See the face is not fully machined as there is a small lip left on the edge

So, we will adjust (FDP1) Overtravel in Direction of Plain to push the cutter over some more to clear the material. Make it 20 instead of 10 SEE BELOW:.

Backplot

File Edit NC Functions Backplot File Compare Transmission CNC-Calc

Backplot Window Animation Close Backplot Backplot Setup

Zoom In Zoom Out Fit To Window View Reset Top Front Left View From Measure Rotate Show Bounding Box

Toolpath Mode Tool Setup Solid Model Zoom / Regenerate solid Solid Setup

Siemens 840D Mill (Adv)

Find Find Previous Find Next Go to Line/Block Number Previous Tool ch

File Editor NC Functions Backplot File Compare Transmission CNC-Calc

Backplot Window Solid Animation Backplot File Close Backplot Backplot Setup

Zoom In Zoom Out Fit To Window View Reset Top Front Left View From Tool Measure Distance Rotate View Show Bounding Box Toolpath Mode Tool Tool Setup

Untitled *

NC-Assistant

Collapse

Description:
CYCLE71 - FACE MILL

Retract Plane (RTP):

Reference Plane (RFP):

Safety Clearance (SDIS):

Depth (DP):

Starting Point Abscissa (Absolute) (PA):

Starting Point Abscissa (Ordinate) (PO):

Length of Rectangle 1st Axis Inc (LENG):

Width of Rectangle 1st Axis Inc (WID):

Angle of Rectangle (STA):

Max Infeed Depth (MID):

Max Infeed Width (MIDA):

Retraction Travel in Cutting Dir (FDP):

Final Machining Allowance Depth (FALD):

Feedrate for Surface Machining (FFP1):

Machining Type *1=Roughing, *2=Finishing (VAR1):

Over Travel in Direction of Plane Infeed (FDP1):

Modify

Default

Program Header

Program Start

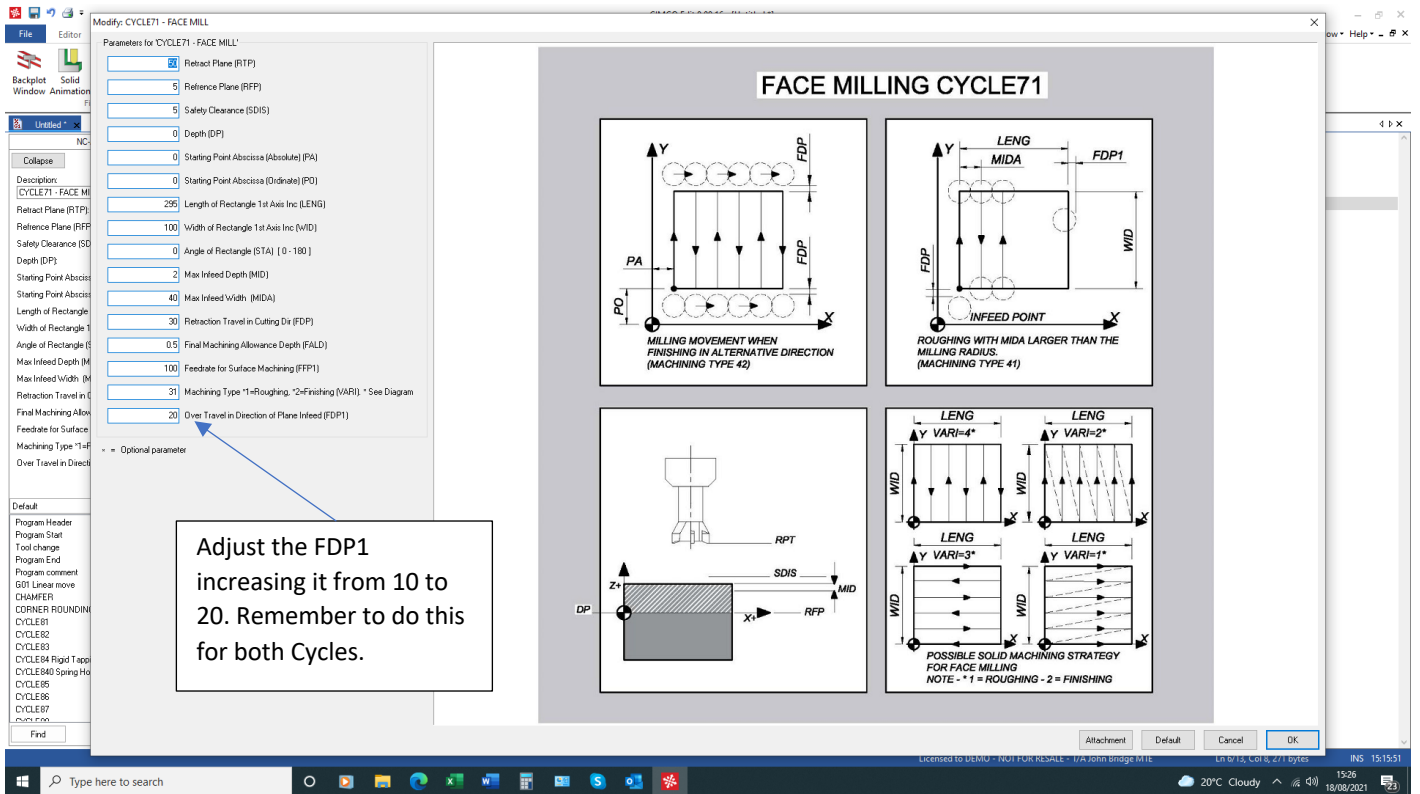
Tool change

```

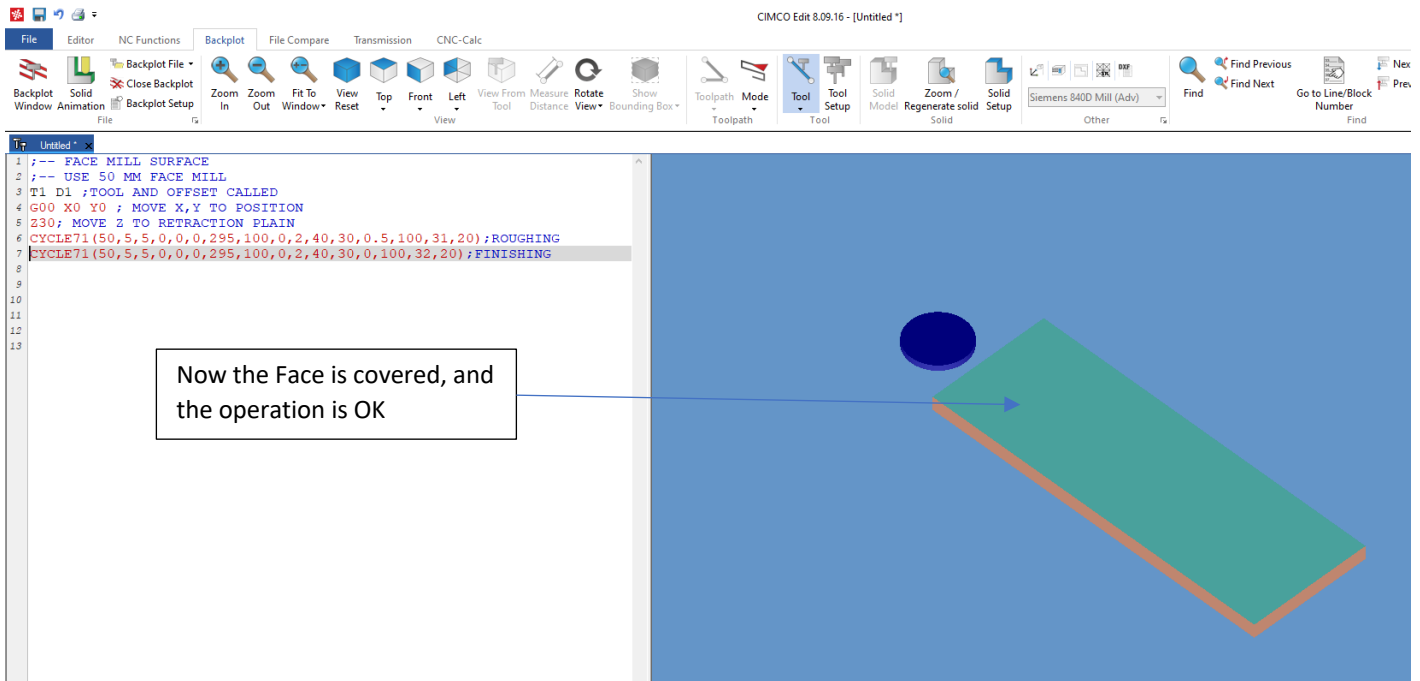
1  ;-- FACE MILL SURFACE
2  ;-- USE 50 MM FACE MILL
3  T1 D1 ;TOOL AND OFFSET CALLED
4  G00 X0 Y0 ; MOVE X,Y TO POSITION
5  Z30; MOVE Z TO RETRACTION PLAIN
6  CYCLE71(50,5,5,0,0,0,295,100,0,2,40,30,0.5,100,31,20);ROUGHING
7  CYCLE71(50,5,5,0,0,0,295,100,0,2,40,30,0,100,32,20);FINISHING
8
9
10
11
12
13

```

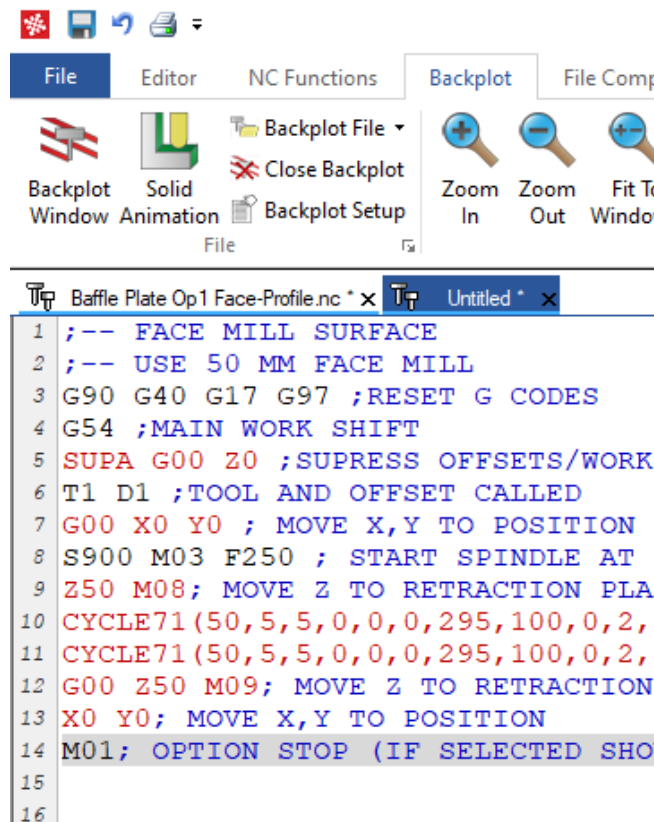
To make the changes highlight the line with the CYLCE71 and the details will appear in the top window. Click Modify



Re-run Backplot



So, our first operation is working in a basic form. We need to add the other blocks to make the programs work on the machine SEE BELOW:



```

1  ;-- FACE MILL SURFACE
2  ;-- USE 50 MM FACE MILL
3  G90 G40 G17 G97 ;RESET G CODES
4  G54 ;MAIN WORK SHIFT
5  SUPA G00 Z0 ;SUPRESS OFFSETS/WORK SHIFT MOVE Z UP TO MACHINE ZERO
6  T1 D1 ;TOOL AND OFFSET CALLED
7  G00 X0 Y0 ; MOVE X,Y TO POSITION
8  S900 M03 F250 ; START SPINDLE AT 900 RPM
9  Z50 M08; MOVE Z TO RETRACTION PLAIN TURN ON COOLANT
10 CYCLE71(50,5,5,0,0,0,295,100,0,2,40,30,0.5,100,31,20);ROUGHING
11 CYCLE71(50,5,5,0,0,0,295,100,0,2,40,30,0,100,32,20);FINISHING
12 G00 Z50 M09; MOVE Z TO RETRACTION PLAIN, SWITCH OFF COOLANT
13 X0 Y0; MOVE X,Y TO POSITION
14 M01; OPTION STOP (IF SELECTED SHOULD TURN OF THE SPINDLE)
15
16

```

See commented details at the end of each block. They explain the additional blocks required to make up the program to operate other functions to create the planned operation. The machine tool builder may demand other instructions to make his machine function the way he wants depending on the devices provided on the machine. The programmer must study the machine tool builder's manual to look for any bespoke additional programming codes

A good CNC programmer will always seek to plan his move sequencing to avoid collisions on the machine. i.e., move Z up to clear the job before moving X, Y etc. See the SUPA block which is programming in the machine co-ordinate system not the work co-ordinate system. This is a one-shot block and is not modal, so work shift G54 will be active again after this move has completed.

Next, we can look at programming the machining of the 50 mm holes using one of the 4 pocket milling routines in NC-Assistant for Siemens 840D CNC controls. But first, we should now save the program we have so far so that we don't lose it.

All CNC programs are made up of a section (a small program complete) for each tool and within, the sections are broken down to 3 parts, a header, cutting data and completed with a trailer. So, with this in mind we can copy the Header from our first Face milling operation and edit it for our 50 mm hole operation.

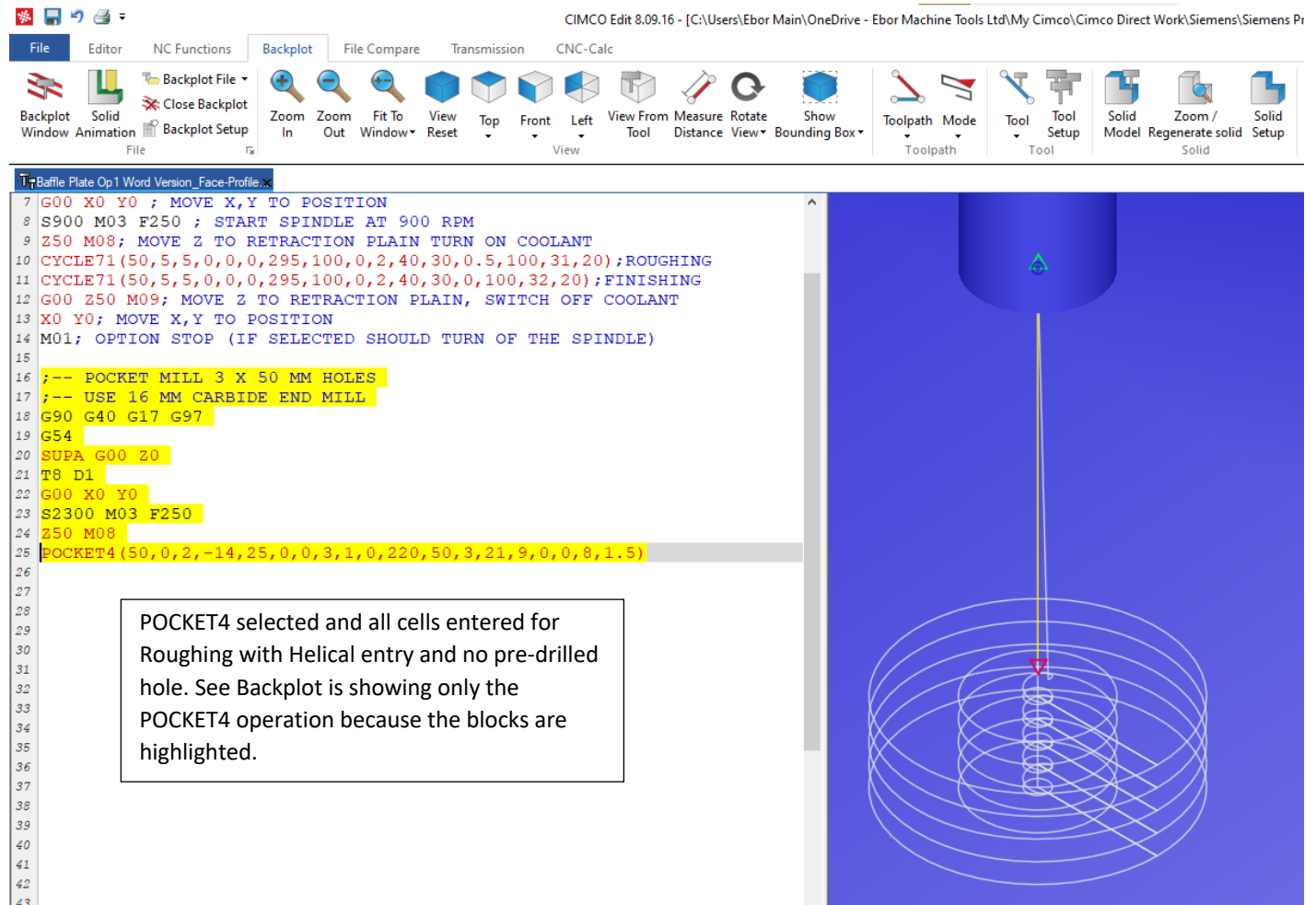
```

10 CYCLE71(50,5,5,0,0,0,295,100,0,2,40,30,0.5,100,31,20);ROUGHING
11 CYCLE71(50,5,5,0,0,0,295,100,0,2,40,30,0,100,32,20);FINISHING
12 G00 Z50 M09; MOVE Z TO RETRACTION PLAIN, SWITCH OFF COOLANT
13 X0 Y0; MOVE X,Y TO POSITION
14 M01; OPTION STOP (IF SELECTED SHOULD TURN OF THE SPINDLE)
15
16 ;-- POCKET MILL 3 X 50 MM HOLES
17 ;-- USE 16 MM CARBIDE END MILL
18 G90 G40 G17 G97
19 G54
20 SUPA G00 Z0
21 T8 D1
22 G00 X0 Y0
23 S2300 M03 F250
24 Z50 M08

```

New header copied from above and edited to suit our next operation to machine to the 50 mm holes.

We will now use POCKET4 to machine the 50 mm hole, with a 16 mm Carbide End Mill that will cut over the centre



The screenshot shows the CIMCO Edit 8.09.16 interface. The left pane displays a CNC program with the following code:

```

7 G00 X0 Y0 ; MOVE X,Y TO POSITION
8 S900 M03 F250 ; START SPINDLE AT 900 RPM
9 Z50 M08; MOVE Z TO RETRACTION PLAIN TURN ON COOLANT
10 CYCLE71(50,5,5,0,0,0,295,100,0,2,40,30,0.5,100,31,20);ROUGHING
11 CYCLE71(50,5,5,0,0,0,295,100,0,2,40,30,0,100,32,20);FINISHING
12 G00 Z50 M09; MOVE Z TO RETRACTION PLAIN, SWITCH OFF COOLANT
13 X0 Y0; MOVE X,Y TO POSITION
14 M01; OPTION STOP (IF SELECTED SHOULD TURN OF THE SPINDLE)
15
16 ;-- POCKET MILL 3 X 50 MM HOLES
17 ;-- USE 16 MM CARBIDE END MILL
18 G90 G40 G17 G97
19 G54
20 SUPA G00 Z0
21 T8 D1
22 G00 X0 Y0
23 S2300 M03 F250
24 Z50 M08
25 POCKET4(50,0,2,-14,25,0,0,3,1,0,220,50,3,21,9,0,0,8,1.5)

```

A text box highlights lines 16-17, stating: "POCKET4 selected and all cells entered for Roughing with Helical entry and no pre-drilled hole. See Backplot is showing only the POCKET4 operation because the blocks are highlighted."

The right pane shows a 3D backplot of the pocket mill operation. A blue cylindrical tool is shown cutting a helical path into a blue workpiece. The toolpath is visualized as a series of concentric circles and a central vertical line.

Now we will use the same cycle for Finishing by removing the FAL finishing allowance, by setting the VARI to finishing, by setting the AP1 to the pocket radius – 2 mm to avoid machining the whole bottom face area.

SEE BELOW:

```

16 ;-- POCKET MILL 3 X 50 MM HOLES
17 ;-- USE 16 MM CARBIDE END MILL
18 G90 G40 G17 G97
19 G54
20 SUPA G00 Z0
21 T8 D1
22 G00 X0 Y0
23 S2300 M03 F250
24 Z50 M08
25 POCKET4(50,0,2,-14,25,0,0,3,1,0,220,50,3,21,9,0,0,8,1.5);ROUGHING
26 POCKET4(50,0,2,-14,25,0,0,0,0,0,220,50,3,12,,,,,);FINISHING
27

```


So, we have a working pocket milling routine for roughing and finishing and we need to machine three holes. There will be several possible solutions, but we are going to use Label and Endlabel to be able to repeat the Pocket routines and we will use the TRANS and ATRANS to make temporary datum shifts to machine holes #1, #2, #3. SEE BELOW:

Create a new line above the start of the blocks you want to repeat later.

Open the LABEL DEFINITION box. A label can be any text you wish to use. We will use POCKET. Enter pocket into the cell. Case doesn't matter as it will automatically be changed to upper case

Insert: LABEL DEFINITION

Parameters for 'LABEL DEFINITION'

* Label Name

* ENDLABEL: End of Label

* = Optional parameter

Default Cancel OK

ENDLABEL is used to define the end of the blocks you wish to repeat. It will appear in the cell when the box is open. You have the choice of deleting this and only the Label Name cell will be inserted on the line or leaving it and both cells text will be on the same line. If you take the latter course, then you can cut and paste ENDLABEL to the correct line

```

5 SUPA G00 Z0
6 T1 D1 ;TOOL
7 G00 X0 Y0 ;
8 S900 M03 F2
9 Z50 M08; MC
10 CYCLE71(50,
11 CYCLE71(50,
12 G00 Z50 M09; MOVE Z TO RETRACTION PLAIN, SWITCH OFF COOLANT
13 X0 Y0; MOVE X,Y TO POSITION
14 M01; OPTION STOP (IF SELECTED)
15
16 ;-- POCKET MILL 3 X 50 MM HOLES
17 ;-- USE 16 MM CARBIDE END MILL
18 G90 G40 G17 G97
19 G54
20 SUPA G00 Z0
21 T8 D1
22 G00 X0 Y0
23 S2300 M03 F250
24 Z50 M08
25
26 POCKET4(50,0,2,-14,25,0,0,3,1,0,220,50,3,21,9,0,0,8,1.5);ROUGHING
27 POCKET4(50,0,2,-14,25,0,0,0,0,0,220,50,3,12,,,,);FINISHING
28
29
30

```

Note both start and finish labels are on the same line. Note the colon suffix denoting that this is a label.

Now cut and paste to get the labels into the correct position.

```

13 X0 Y0; MOVE X,Y TO POSITION
14 M01; OPTION STOP (IF SELECTED SHOULD TURN OF THE SPINDLE)
15
16 ;-- POCKET MILL 3 X 50 MM HOLES
17 ;-- USE 16 MM CARBIDE END MILL
18 G90 G40 G17 G97
19 G54
20 SUPA G00 Z0
21 T8 D1
22 G00 X0 Y0
23 S2300 M03 F250
24 Z50 M08
25 POCKET: ENDLABEL:
26
27 POCKET4(50,0,2,-14,25,0,0,3,1,0,220,50,3,21,9,0,0,8,1.5);ROUGHING
28 POCKET4(50,0,2,-14,25,0,0,0,0,0,220,50,3,12,,,,);FINISHING
29
30
31

```



```

22 G00 X0 Y0
23 S2300 M03 F250
24 Z50 M08
25 POCKET:
26 POCKET4 (50,0,2,-14,25,0,0,3,1,0,220,50,3,21,9,0,0,8,1.5);ROUGHING
27 POCKET4 (50,0,2,-14,25,0,0,0,0,0,220,50,3,12,,,,);FINISHING
28 ENDLABEL:
29
30

```

The Labels are now set correctly

So, we have set the Labels and now to create the 3-hole machining using the TRANS, ATRANS, and REPEAT commands to organise the program.

To machine hole #1 we will need to move the programming zero to a position X50 Y50 and this can be achieved with TRANS.

The screenshot shows the CIMCO Edit 8.09.16 interface. The main window displays a CNC program with the following code:

```

5 SUPA G00 Z0 ;SUPPRESS OFFSETS/WORK
6 T1 D1 ;TOOL AND OFFSET CALLED
7 G00 X0 Y0 ; MOVE X,Y TO POSITION
8 S900 M03 F250 ; START SPINDLE AND
9 Z50 M08; MOVE Z TO RETRACTION P
10 CYCLE71 (50,5,5,0,0,0,295,100,0,
11 CYCLE71 (50,5,5,0,0,0,295,100,0,
12 G00 Z50 M09; MOVE Z TO RETRACTIO
13 X0 Y0; MOVE X,Y TO POSITION
14 M01; OPTION STOP (IF SELECTED S
15
16 ;-- POCKET MILL 3 X 50 MM HOLES
17 ;-- USE 16 MM CARBIDE END MILL
18 G90 G40 G17 G97
19 G54
20 SUPA G00 Z0
21 T8 D1
22 G00 X0 Y0
23 S2300 M03 F250
24 Z50 M08
25
26 POCKET:
27 POCKET4 (50,0,2,-14,25,0,0,3,1,0,220,50,3,21,9,0,0,8,1.5);ROUGHING
28 POCKET4 (50,0,2,-14,25,0,0,0,0,0,220,50,3,12,,,,);FINISHING
29 ENDLABEL:
30
31

```

The 'Insert: TRANS - TEMP DATUM SHIFT ABS' dialog box is open, showing the following parameters:

- X Axis Tempory Datum Shift ABS-TRANS: 50
- Y Axis Tempory Datum Shift ABS-TRANS: 50
- Z Axis Tempory Datum Shift ABS-TRANS: (empty)

The diagram on the right shows a coordinate system with X+ and Y+ axes. It indicates the position of the tool (G54) and the datum shift points (POINT 1 - N30 TRANS X20 Y20, POINT 2 - N50 ATRANS X15 Y15, CANCEL - N100 TRANS).

Open the TRANS box and entre 50, 50 into the X and Y cells. Press OK

```

22 G00 X0 Y0
23 S2300 M03 F250
24 Z50 M08
25 TRANS X50 Y50
26 POCKET:
27 POCKET4 (50,0,2,-14,25,0,0,3,1,0,220,50,3,21,9,0,0,8,1.5);ROUGHING
28 POCKET4 (50,0,2,-14,25,0,0,0,0,0,220,50,3,12,,,,);FINISHING
29 ENDLABEL:
30
31

```

Result

So now we have created the temporary absolute work shift to machine hole #1. So, we will now look at hole #2 and #3 to achieve to this with ATRANS temporary incremental work shift

The screenshot shows the CIMCO Edit 8.09.16 interface. The main window displays a CNC program with the following code:

```

13 X0 Y0; MOVE X,Y TO P
14 M01; OPTION STOP (IF
15
16 ;-- POCKET MILL 3 X
17 ;-- USE 16 MM CARBID
18 G90 G40 G17 G97
19 G54
20 SUPA G00 Z0
21 T8 D1
22 G00 X0 Y0
23 S2300 M03 F250
24 Z50 M08
25 TRANS X50 Y50
26 POCKET:
27 POCKET4 (50,0,2,-14,2
28 POCKET4 (50,0,2,-14,2
29 ENDLABEL:
30
31
32
33

```

The 'Insert: ATRANS - TEMP DATUM SHIFT INC' dialog box is open, showing the 'Parameters for ATRANS - TEMP DATUM SHIFT INC' section. The X-axis temporary datum shift is set to 97.5. The Y and Z axes are set to 0. The diagram on the right shows a coordinate system with points 1 and 2, and dimensions 15, 20, and 15. The text 'TRANS / ATRANS TEMP DATUM SHIFTS' is displayed. The 'POINT 1 - N30 TRANS X20 Y20' and 'POINT 2 - N50 ATRANS X15 Y15' are noted, along with 'CANCEL - N100 TRANS'.

A callout box points to the X-axis input field in the dialog, stating: "Open the ATRANS box and entre 97.5 (See the Baffle Plate drawing) into the X cell. Press OK".

The 'Result' section shows the updated program code:

```

22 G00 X0 Y0
23 S2300 M03 F250
24 Z50 M08
25 TRANS X50 Y50
26 POCKET:
27 POCKET4 (50,0,2,-14,25,0,0,3,1,0,220,50,3,21,9,0,0,8,1.5);ROUGHING
28 POCKET4 (50,0,2,-14,25,0,0,0,0,0,220,50,3,12,,,,);FINISHING
29 ENDLABEL:
30 ATRANS X97.5
31
32
33

```

SEE BELOW:

Now we are in position for #2 holes we will machine it with the REPEAT command.

Open the REPEAT box and entre the Label you wish to repeat.

Note: whatever case you use will be converted to upper case

Insert: REPEAT LABEL

Parameters for 'REPEAT LABEL'

Label Name

* = Optional parameter

Default Cancel OK

Result, this will repeat the blocks between the labels and produce hole #2. We can now copy the last two blocks to get hole #3

```

5 SUPA G00 Z0 ;SUPRESS OFFSET
6 T1 D1 ;TOOL AND OFFSET CALL
7 G00 X0 Y0 ; MOVE X,Y TO POS
8 S900 M03 F250 ; START SPIND
9 Z50 M08; MOVE Z TO RETRACTI
10 CYCLE71(50,5,5,0,0,0,295,10
11 CYCLE71(50,5,5,0,0,0,295,10
12 G00 Z50 M09; MOVE Z TO RETR
13 X0 Y0; MOVE X,Y TO POSITION
14 M01; OPTION STOP (IF SELECTED SHOULD TURN OF THE SPINDLE)
15
16 ;-- POCKET MILL 3 X 50 MM HOLES
17 ;-- USE 16 MM CARBIDE END MILL
18 G90 G40 G17 G97
19 G54
20 SUPA G00 Z0
21 T8 D1
22 G00 X0 Y0
23 S2300 M03 F250
24 Z50 M08
25 TRANS X50 Y50
26 POCKET:
27 POCKET4(50,0,2,-14,25,0,0,3,1,0,220,50,3,21,9,0,0,8,1.5);ROUGHING
28 POCKET4(50,0,2,-14,25,0,0,0,0,0,220,50,3,12,,,,);FINISHING
29 ENDLABEL:
30 ATRANS X97.5
31 REPEAT POCKET
32
33
34

```

SEE BELOW:

```

28 POCKET4 (50,0,2,-14,25,0,0,0,0,0,2
29 ENDLABEL:
30 ATRANS X97.5
31 REPEAT POCKET ;-- HOLE #2
32 ATRANS X97.5
33 REPEAT POCKET ;-- HOLE #3
34 TRANS ;-- THIS COMMAND WILL CANCEL ALL TEMP WORK SHIFT & RETURN TO G54
35
36

```

Result of copied blocks to get hole #3 and after added the TRANS command alone in a block to cancel all temporary work shifts. See comments added as well.

We can end this operation now by moving the tool to a safe position etc.

```

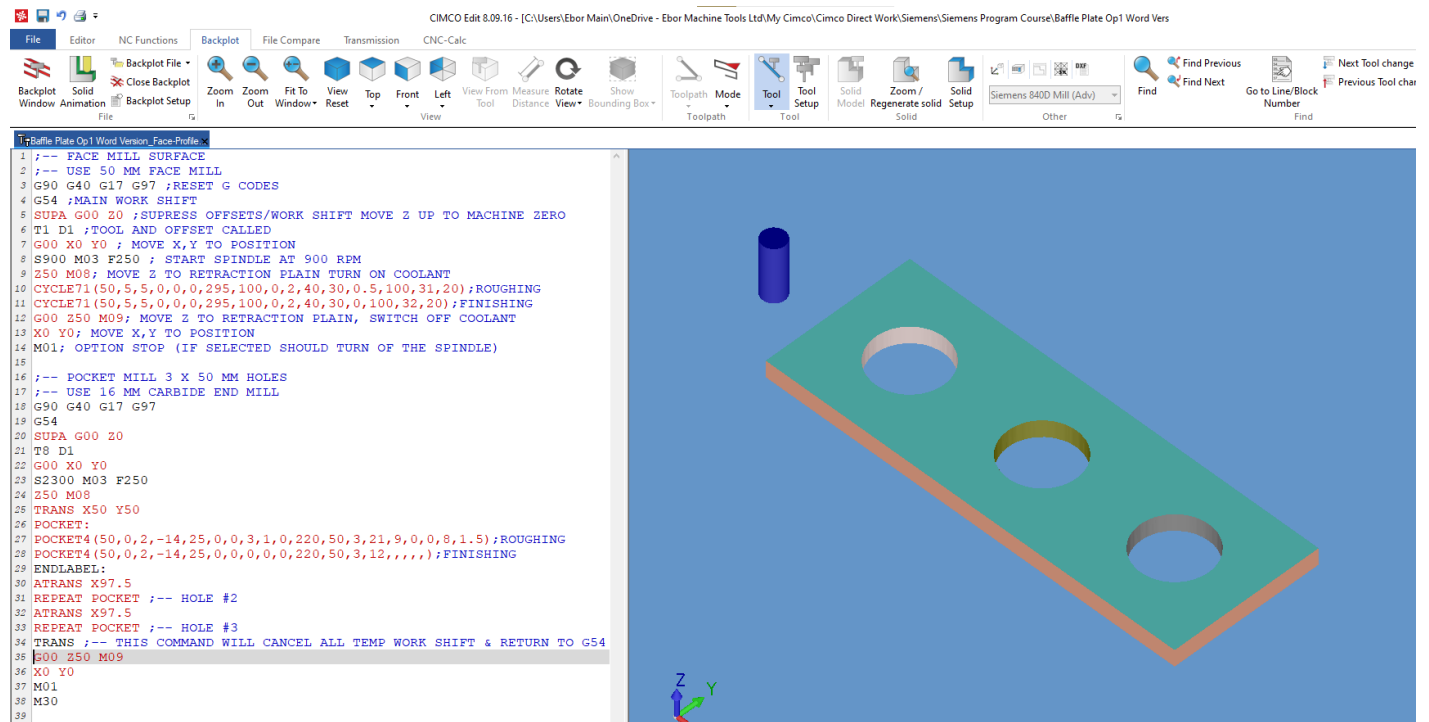
16 ;-- POCKET MILL 3 X 50 MM HOLES
17 ;-- USE 16 MM CARBIDE END MILL
18 G90 G40 G17 G97
19 G54
20 SUPA G00 Z0
21 T8 D1
22 G00 X0 Y0
23 S2300 M03 F250
24 Z50 M08
25 TRANS X50 Y50
26 POCKET:
27 POCKET4 (50,0,2,-14,25,0,0,3,1,0,220,50,3,21,9,0,0,8,1.5) ;ROUGHING
28 POCKET4 (50,0,2,-14,25,0,0,0,0,0,220,50,3,12,,,,,) ;FINISHING
29 ENDLABEL:
30 ATRANS X97.5
31 REPEAT POCKET ;-- HOLE #2
32 ATRANS X97.5
33 REPEAT POCKET ;-- HOLE #3
34 TRANS ;-- THIS COMMAND WILL CANCEL ALL TEMP WORK SHIFT & RETURN TO G54
35 G00 Z50 M09
36 X0 Y0
37 M01
38 M30
39

```

See complete operation for the three holes.
Nowc we can test with the Solid Animation
Backplot

SEE BELOW:

Test in Solid Animation Backplot, both operations Facing and Hole generation are looking OK.



Now we can consider the drilling and tapping operations on the holes. We will centre drill, drill, then tap the threads. We will use the same program style that as we used on the 50 mm holes so we can copy that section and edit the cycles etc.

SEE BELOW:

38
39 ;-- CENTRE DRILL ALL HOLES
40 ;-- USE T2 CENTRE DRILL
41 G90 G40 G17 G97
42 G54
43 SUPA G00 Z0
44 T2 D1
45 G00 X0 Y0
46 S1000 M03 F80
47 Z50 M08
48 TRANS X50 Y50
49 CDRILL:
50 MCALL CYCLE81(10,0,2,-5,);-- CENTRE DRILL M10 HOLES
51 HOLES2(0,0,35,0,90,4);-- BOLT HOLE PATTERN HOLE #1
52 MCALL
53 ENDLABEL:
54 ATRANS X97.5
55 REPEAT CDRILL ;-- HOLE #2
56 ATRANS X97.5
57 REPEAT CDRILL ;-- HOLE #3
58 TRANS ;-- THIS COMMAND WILL CANCEL ALL TEMP WORK SHIFT & RETURN TO G54
59 MCALL CYCLE81(10,0,2,-5,);-- CENTRE DRILL 4 X 11 MM HOLES
60 X15 Y15
61 X280 Y15
62 X280 Y85
63 X15 Y85
64 MCALL
65 G00 Z50 M09
66 X0 Y0
67 M01
68

Modify

er Tapping

Add

Copy the previous operation, edit the header comments, edit in a new Tool and speeds / feeds, change the cycles to drilling, add the hole pattern, then add the 4 - 11 mm holes with their coordinates to complete the centre drilling.

The drill cycle and hole pattern macros can be found in the NC-Assistant list.

SEE BELOW:

To create the drilling operation, we can copy and paste the centre drilling operation and again edit the details. But now leave out the 4 – 11 mm holes.

SEE BELOW:

Modify

default
 program Header
 program Start
 tool change
 program End
 program comment
 D1 Linear move
 HAMFER
 ORNER ROUNDING
 YCLE81
 YCLE82
 YCLE83
 YCLE84 Rigid Tapping
 YCLE840 Spring Holder Tapping
 YCLE85
 YCLE86
 YC1 FR7

```

62 X280 Y85
63 X15 Y85
64 MCALL
65 G00 Z50 M09
66 X0 Y0
67 M01
68
69 ;-- DRILL M10 HOLES
70 ;-- USE T22 8.5 MM CARBIDE DRILL
71 G90 G40 G17 G97
72 G54
73 SUPA G00 Z0
74 T22 D1
75 G00 X0 Y0
76 S3000 M03 F300
77 Z50 M08
78 TRANS X50 Y50
79 DRILL:
80 MCALL CYCLE81(10,0,2,-15,);-- DRILL 8.5 MM HOLES
81 HOLES2(0,0,35,0,90,4);-- BOLT HOLE PATTERN HOLE #1
82 MCALL
83 ENDLABEL:
84 ATRANS X97.5
85 REPEAT DRILL ;-- HOLE #2
86 ATRANS X97.5
87 REPEAT DRILL ;-- HOLE #3
88 TRANS ;-- THIS COMMAND WILL CANCEL ALL TEMP WORK SHIFT & RETURN TO G54
89 G00 Z50 M09
90 X0 Y0
91 M01
92
93
94

```

oles.

We will now drill the 4 – 11 mm holes and will copy the centre drilling operation cut out all the programming that we do not need.

X0 Y0

```

65 G00 Z50 M09
66 X0 Y0
67 M01
68
69 ;-- DRILL 4 HOLES 11 MM THROUGH
70 ;-- USE T27 11 MM CARBIDE DRILL
71 G90 G40 G17 G97
72 G54
73 SUPA G00 Z0
74 T27 D1
75 G00 X0 Y0
76 S3000 M03 F300
77 Z50 M08
78 MCALL CYCLE81(10,0,2,-15,);-- DRILL 11 MM HOLES
79 X15 Y15
80 X280 Y15
81 X280 Y85
82 X15 Y85
83 MCALL
84 G00 Z50 M09
85 X0 Y0
86 M01
87

```

See the new Header comments, See the new Tool and speeds /feeds and the new cycle. All the rest remains the same as we are visiting the same holes.

Note that co-ordinates are modal and do not need to be repeated on the following line if the axis is not moved. But when programming hole co-ordinates it can be useful to have both X and Y to be able to move directly to a hole position after reading the Cycle to revisit a particular hole.

Finally, we need to Tap the M10 holes using Rigid Tapping CYCLE84 if we have a machine that has a spindle encoder otherwise, we will need to use a spring-loaded tool holder and CYCLE840. Copy the M10 drilling operation and edit as below:

```

109 X0 Y0
110 M01
111
112 ;-- TAP M10 HOLES
113 ;-- USE T43 M10 SPIRAL POINT TAP
114 ;-- USE RIGID TAPPING CYCLE84
115 G90 G40 G17 G97
116 G54
117 SUPA G00 Z0
118 T43 D1
119 G00 X0 Y0
120 S200 M03
121 Z50 M08
122 TRANS X50 Y50
123 TAP:
124 MCALL CYCLE84(30,0,5,-18,,0,3,,1.5,0,200,400,3,0,0,1,,);-- TAP M10 THROUGH
125 HOLES2(0,0,35,0,90,4);-- BOLT HOLE PATTERN HOLE #1
126 MCALL
127 ENDLABEL:
128 ATRANS X97.5
129 REPEAT TAP ;-- HOLE #2
130 ATRANS X97.5
131 REPEAT TAP ;-- HOLE #3
132 TRANS ;-- THIS COMMAND WILL CANCEL ALL TEMP WORK SHIFT & RETURN TO G54
133 G00 Z50 M09
134 SX0 Y0
135 M01
136
137

```

See the new Header comments, See the new Tool and speeds /feeds and the new cycle. All the rest remains the same as we are visiting the same holes.


We can now test our whole programming with Backplot. It is advisable to test each individual operation as we went along by high lighting the operation and running Backplot. It can be easier to snag programming when we focus in on a particular operation to ensure it is OK before continuing as isolating several programming errors can be tricky.

Remember, check the solid set up first to see that you have the Stock Dimensions set to represent the Stock material size. Tick “lock stock dimension” so that the stock will remain set for repeated Backplot testing on this part. If you look at the Bolster Plate drawing again you will see the stock dimensions are in line with the block that this plate may be made from.

SEE BELOW:

Solid Setup ? X

Stock Dimensions


 Min X: Min Y: Min Z:
 Max X: Max Y: Max Z:

 Dimensions:

☐ Use cylindrical stock ☒ Lock stock dimension
☐ Automatic stock scanning ☐ Update stock definition in NC file

Options

☒ Use a different color for each tool
☐ Do not show tool setup when all tools are defined
☐ Smooth surface

☐ Stop if tool collision detected (rapid through stock)
☐ Stop if cutting with tool shaft
☐ Stop if cutting with tool shoulder
☐ Stop if tool holder collision detected

Low Quality (Fast) Medium Quality High Quality (Slow)

Check that all the tooling detected in the program is defined correctly

IFT & RETURN

,1,,);-- TAP

Tool Setup

T1 Face mill 50MM Type: Face mill Diameter: 50

T2 Center Drill 3MM

T8 End Mill Flat 16MM

T22 Drill 8.5MM

Description	Type	D	L
End Mill Flat 2MM	End Mill Flat	2	60
End Mill Ball 2MM	End Mill Ball	2	60
End Mill Bull 2MM R0.5	End Mill Bull	2	60
End Mill Flat 3MM	End Mill Flat	3	60
End Mill Ball 3MM	End Mill Ball	3	60
End Mill Bull 3MM R0.8	End Mill Bull	3	60
End Mill Flat 4MM	End Mill Flat	4	60

Delete Modify Add Assign to 'T1'

☒ Save tool information Load/Save Tool library Cancel OK

SEE BELOW:

Test with Solid Animation Backplot:

CIMCO Edit 8.09.16 - [C:\Users\Ebor Main\OneDrive - Ebor Machine Tools Ltd\My Cimco\Cimco Direct Work\Siemens\Siemens Program Course\Baffle Plate Op1 Word Vers

File Editor NC Functions Backplot File Compare Transmission CNC-Calc

Backplot Window Animation Solid Close Backplot Backplot Setup

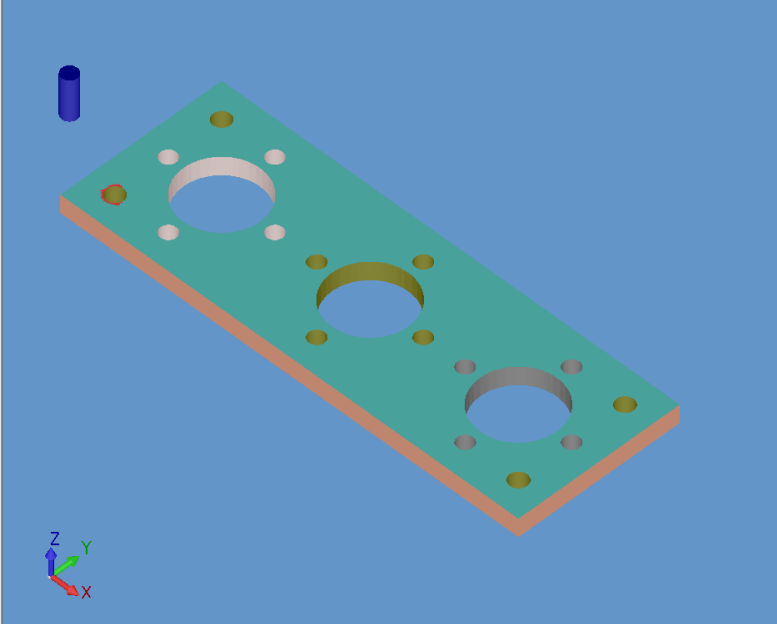
Zoom In Zoom Out Fit To Window View Reset Top Front Left View From Tool Measure Rotate View Show Bounding Box

Toolpath Mode Tool Tool Setup Solid Model Regenerate solid Solid Setup Siemens 840D Mill (Adv)

Find Find Previous Find Next Go to Line/Block Number Previous Tool ch

Untitled Drawing x1 Baffle Plate Op1 Word Version_Face-Profile...

```
95 S3000 M03 F300
96 Z50 M08
97 TRANS X50 Y50
98 DRILL:
99 MCALL CYCLE81(10,0,2,-15,);-- DRILL 8.5 MM HOLES
100 HOLES2(0,0,35,0,90,4);-- BOLT HOLE PATTERN HOLE #1
101 MCALL
102 ENDLABEL:
103 ATRANS X97.5
104 REPEAT DRILL ;-- HOLE #2
105 ATRANS X97.5
106 REPEAT DRILL ;-- HOLE #3
107 TRANS ;-- THIS COMMAND WILL CANCEL ALL TEMP WORK SHIFT & RETURN TO G54
108 G00 Z50 M09
109 X0 Y0
110 M01
111
112 ;-- TAP M10 HOLES
113 ;-- USE T43 M10 SPIRAL POINT TAP
114 ;-- USE RIGID TAPPING CYCLE84
115 G90 G40 G17 G97
116 G54
117 SUPA G00 Z0
118 T43 D1
119 G00 X0 Y0
120 S200 M03
121 Z50 M08
122 TRANS X50 Y50
123 TAP:
124 MCALL CYCLE84(30,0,5,-18,,0,3,,1.5,0,200,400,3,0,0,1,,);-- TAP M10 TH
125 HOLES2(0,0,35,0,90,4);-- BOLT HOLE PATTERN HOLE #1
126 MCALL
127 ENDLABEL:
128 ATRANS X97.5
129 REPEAT TAP ;-- HOLE #2
130 ATRANS X97.5
131 REPEAT TAP ;-- HOLE #3
132 TRANS ;-- THIS COMMAND WILL CANCEL ALL TEMP WORK SHIFT & RETURN TO G54
133 G00 Z50 M09
134 EX0 Y0
135 M01
136
137
```



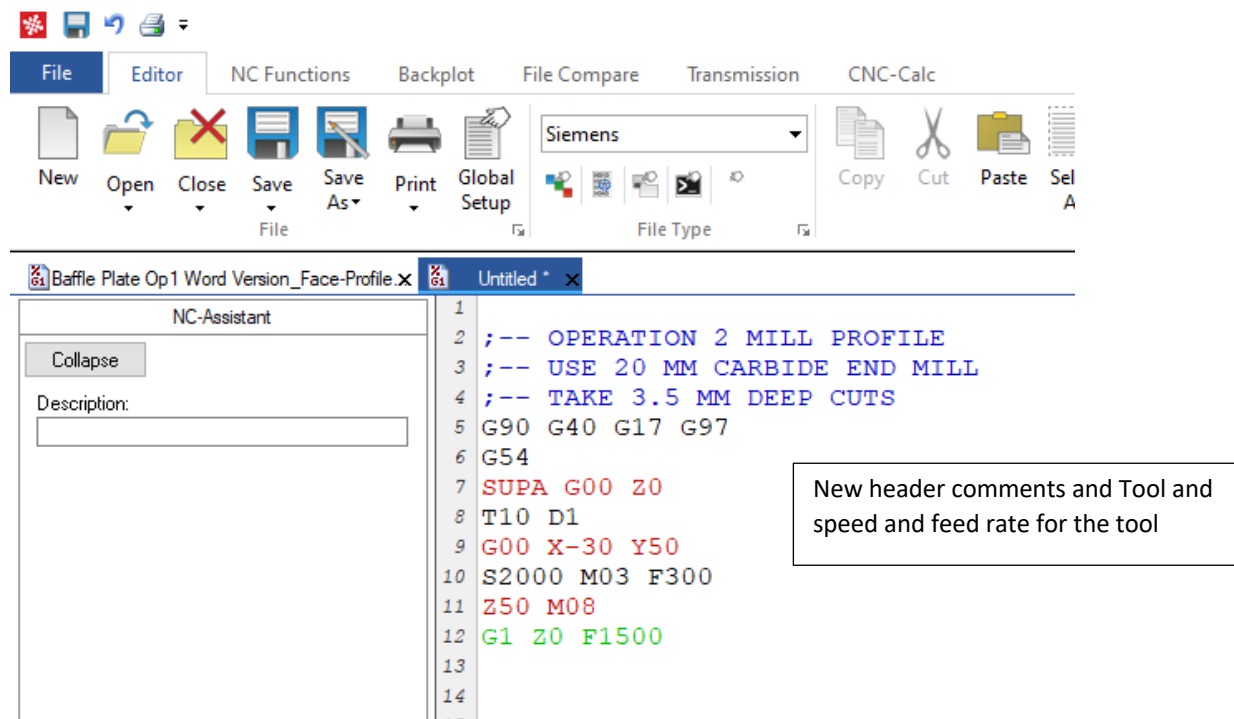
Example 1 Operation 2

Now that we have operation 1 tested, we can move onto operation 2. So, the part can be located on the jig and clamped. The jig will enable the cutter to be programmed to produce the outside profile.

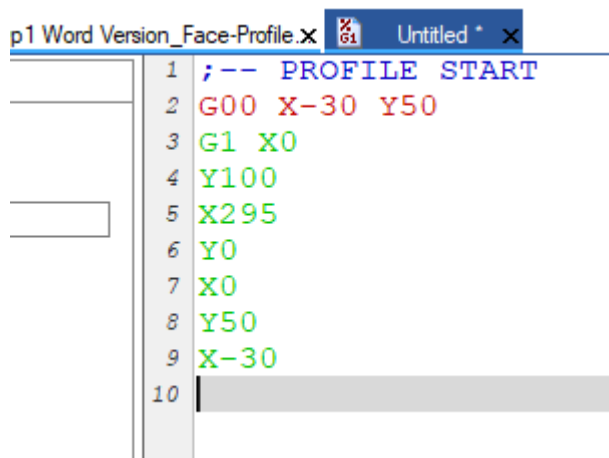
To be able to test our profile with the other operations we will continue to program operation 2 on the page of the Editor. Later we may cut this away into a separate file later when we look at program management.

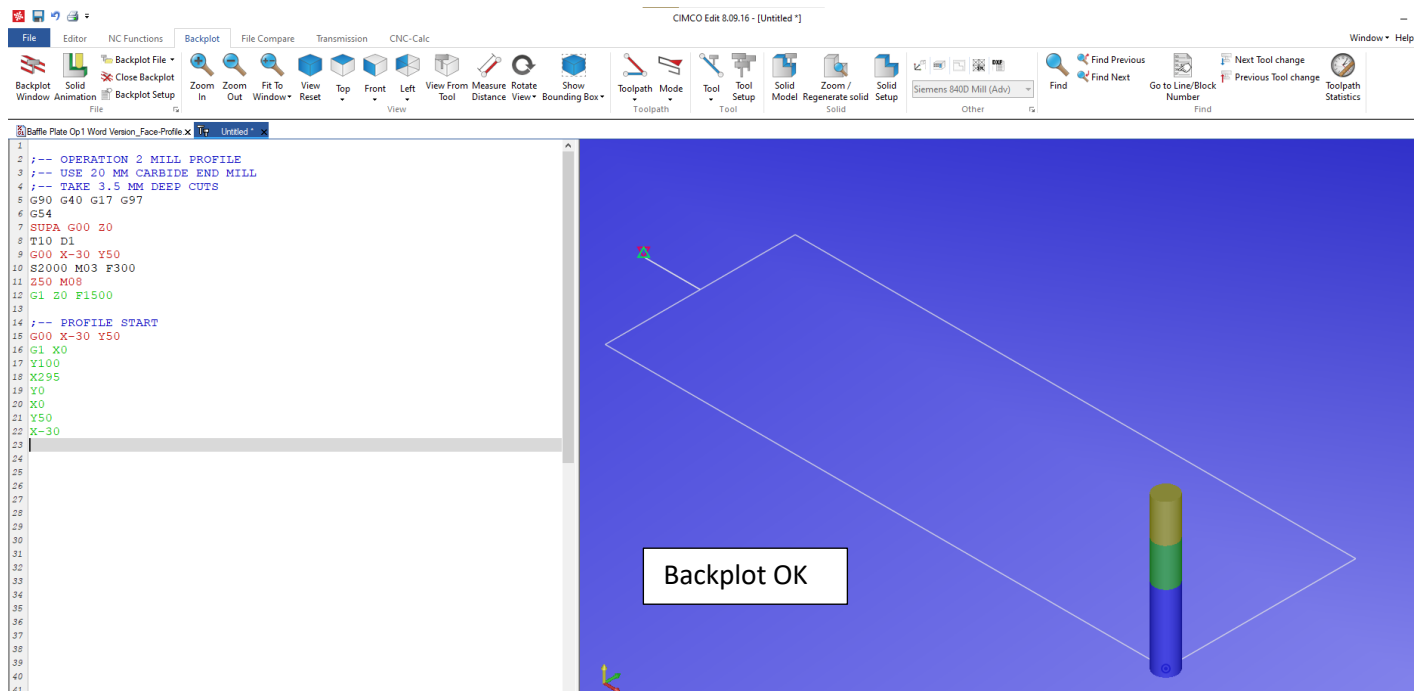
We could use CNC-Calc to draw the profile and to generate a tool path, but we will program this profile in the editor to practice our basic programming skills but also use some of the simple Siemens 840D mnemonic commands to make it easier.

We will create a header by copy and paste and edit the details

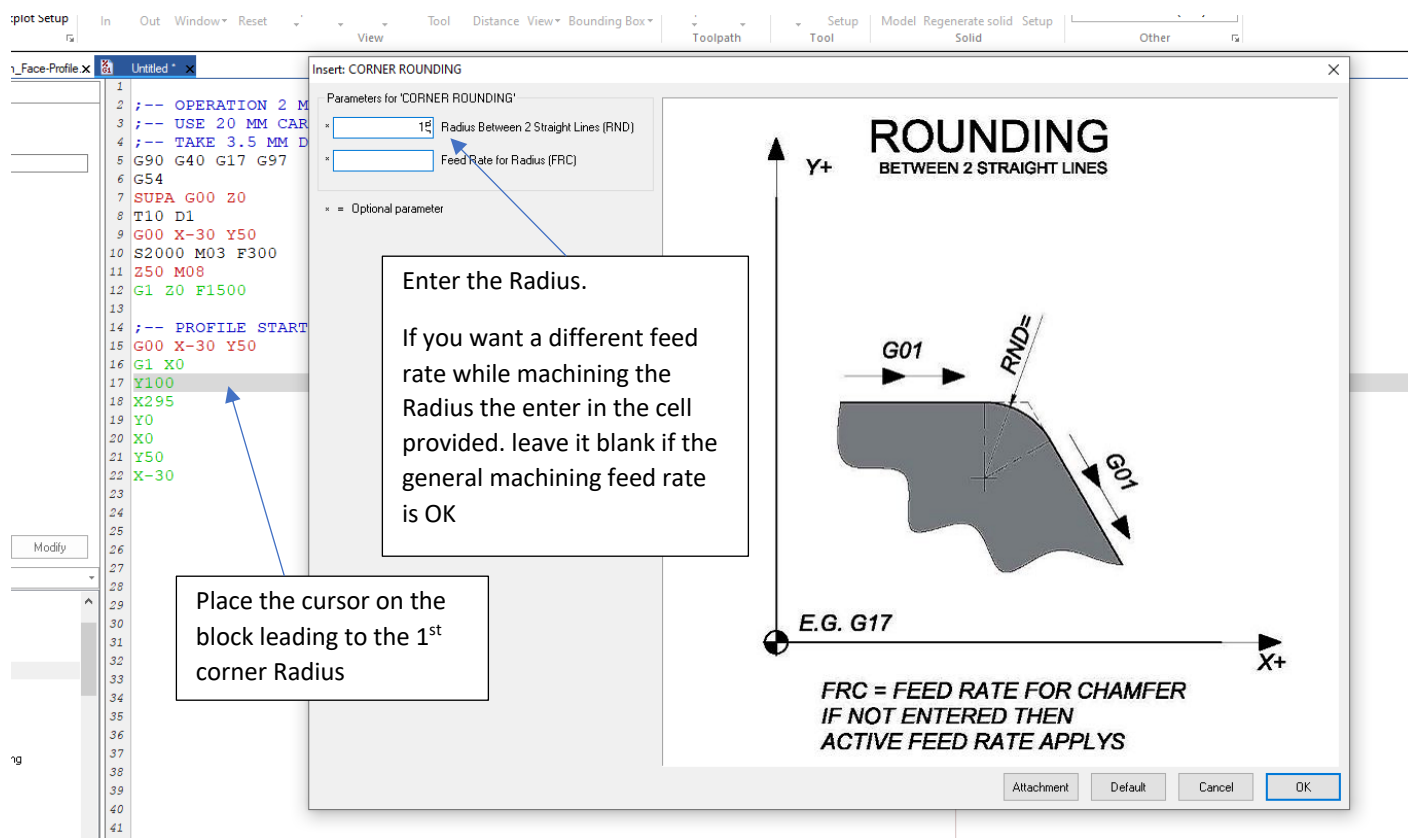


We will program the basic rectangle and test with tool path Backplot. SEE BELOW:





We can now use Corner Rounding to program the 15 mm radius at each corner.



```

11 Z50 M08
12 G1 Z0 F1500
13
14 ;-- PROFILE START
15 G00 X-30 Y50
16 G1 X0
17 Y100 RND=15
18 X295
19 Y0
20 X0
21 Y50
22 X-30

```

Result, see here.

Now copy and paste for the other 3 corners

```

12 G1 Z0 F1500
13
14 ;-- PROFILE START
15 G00 X-30 Y50
16 G1 X0
17 Y100 RND=15
18 X295 RND=15
19 Y0 RND=15
20 X0 RND=15
21 Y50
22 X-30
23

```

Now copy and paste for the other 3 corners

Test with Tool Pathe Backplot

hhh

The screenshot displays the CIMCO Edit 8.09.16 interface. The left pane shows a CNC program with the following code:

```

1  ;-- OPERATION 2 MILL PROFILE
2  ;-- USE 20 MM CARBIDE END MILL
3  ;-- TAKE 3.5 MM DEEP CUTS
4  G90 G40 G17 G97
5  G54
6  SUPA G00 Z0
7  T10 D1
8  G00 X-30 Y50
9  S2000 M03 F300
10 Z50 M08
11 G1 Z0 F1500
12
13
14 ;-- PROFILE START
15 G00 X-30 Y50
16 G1 X0
17 Y100 RND=15
18 X295 RND=15
19 Y0 RND=15
20 X0 RND=15
21 Y50
22 X-30

```

The right pane shows a 3D backplot of the part. A white toolpath is visible on a blue part. A text box "Backplot OK" is overlaid on the plot. The status bar at the bottom shows the following information:

X	-30.000	I	Tool	777	Dist.	0.000
Y	50.000	J	Feed	Flanged	Total	624.248
Z	250.000	K	R			

the semi-circle notches now need to be programmed

13
14 ;-- PROFILE START
15 G00 X-30 Y50
16 G1 X0
17 Y100 RND=15
18
19
20
21 X295 RND=15
22 Y0 RND=15
23
24
25
26
27 X0 RND=15
28 Y50
29 X-30
30
31

Modify

Open a space in the blocks where the 15 mm Radius notches will go. Program the start and finish positions of the notches using a calculator and the drawing.

7

SUPA G00 Z0

8

T10 D1

9

G00 X-30 Y50

10

S2000 M03 F300

11

Z50 M08

12

G1 Z0 F1500

13

14

;-- PROFILE START

15

G00 X-30 Y50

16

G1 X0

17

Y100 RND=15

18

X83.75

19

20

21

X295 RND=15

22

Y0 RND=15

23

24

25

26

27

X0 RND=15

28

Y50

29

X-30

30

31

32

33

34

35

36

37

38

Calculator

Standard

98.75 - 15 =

83.75

MC MR M+ M- MS M*

% CE C <X>

1/x x² √x ÷

7 8 9 ×

4 5 6 -

1 2 3 +

+/- 0 . =

Continue to put in the start and finish positions


```

12 G1 Z0 F1500
13
14 ;-- PROFILE START
15 G00 X-30 Y50
16 G1 X0
17 Y100 RND=15
18 X83.75 ;START
19 X113.75 ;FIN
20 X181.25 ;START
21 X211.25 ;FIN
22 X295 RND=15
23 Y0 RND=15
24 X211.25 ;START
25 X181.25 ;FIN
26 X113.75 ;START
27 X83.75 ;FIN
28 X0 RND=15
29 Y50
30 X-30
31

```

Now we can add the circular interpolation data

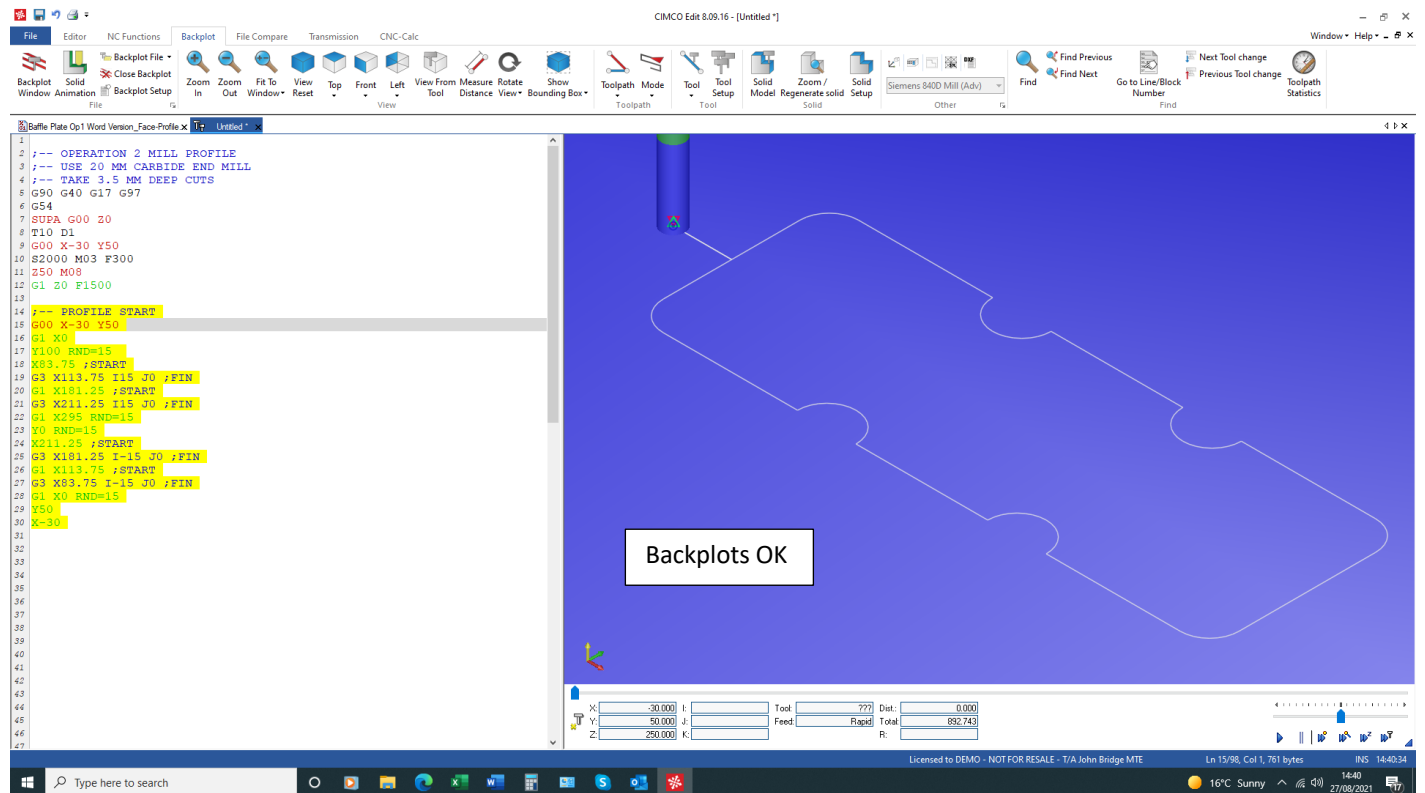
```

12 G1 Z0 F1500
13
14 ;-- PROFILE START
15 G00 X-30 Y50
16 G1 X0
17 Y100 RND=15
18 X83.75 ;START
19 G3 X113.75 I15 J0 ;FIN
20 G1 X181.25 ;START
21 G3 X211.25 I15 J0 ;FIN
22 G1 X295 RND=15
23 Y0 RND=15
24 X211.25 ;START
25 G3 X181.25 I-15 J0 ;FIN
26 G1 X113.75 ;START
27 G3 X83.75 I-15 J0 ;FIN
28 G1 X0 RND=15
29 Y50
30 X-30
31

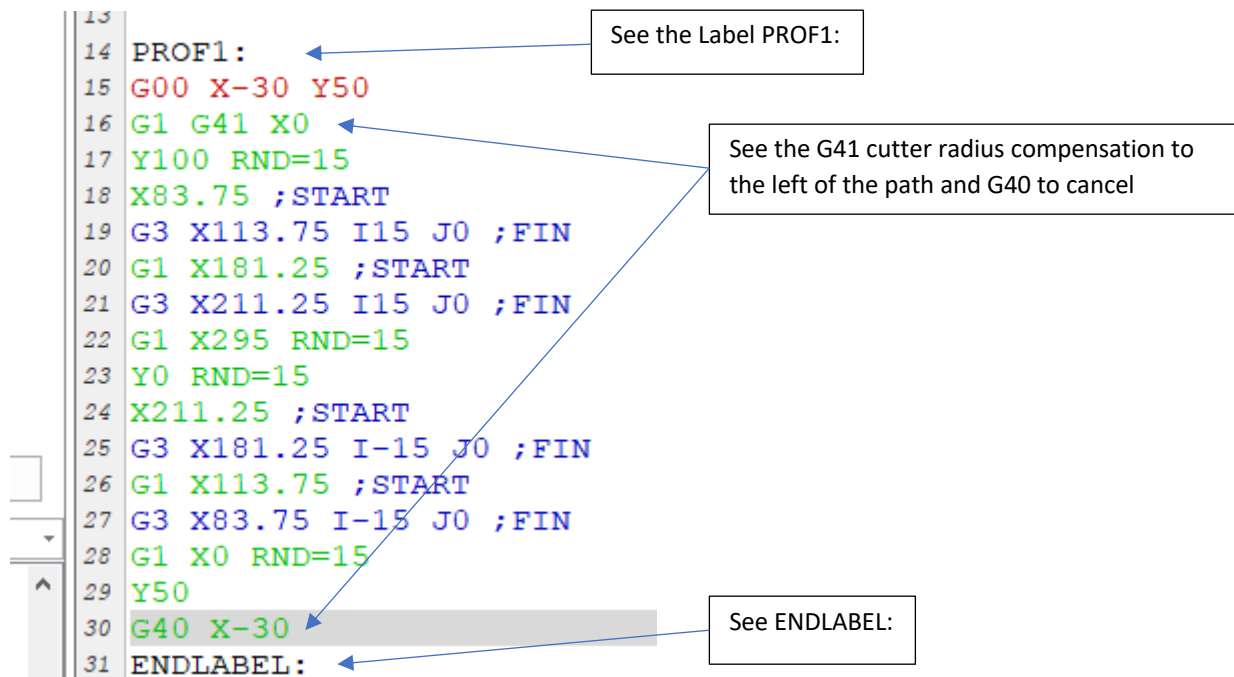
```

Add the G3 for CCW direction and the I and J for arc centre from the start position for the circular interpolation blocks, then on the next line go back to G1 linear interpolation. Notice on the top edge the I = +15 to the arc centre from the start position and on the bottom edge the I = -15 to the arc center from the start position.

Test with Tool path Backplot



Now we can look at how to use the proven profile. First, and as we have used the drawing dimension to establish the profile, we will need to activate tool radius compensation to achieve the correct sizes on the finished part. We will take 3 cuts at 3.5 mm deep to machine the profile.



Now add the incremental depth move and the REPEAT command

```
12 G1 Z0 F1500
13 PROF1:
14 G91 Z-3.5 F1000
15 G90
16 G00 X-30 Y50
17 G1 G41 X0 F100
18 Y100 RND=15
19 X83.75 ;START
20 G3 X113.75 I15 J0 ;FIN
21 G1 X181.25 ;START
22 G3 X211.25 I15 J0 ;FIN
23 G1 X295 RND=15
24 Y0 RND=15
25 X211.25 ;START
26 G3 X181.25 I-15 J0 ;FIN
27 G1 X113.75 ;START
28 G3 X83.75 I-15 J0 ;FIN
29 G1 X0 RND=15
30 Y50
31 G40 X-30
32 ENDLABEL:
33 REPEAT PROF1 P=3
34
```

See G91 incremental move in Z and on the next line G90 back to absolute.

See REPEAT command with the label PROF1 and the P=3 that will repeat the profile a totals of 3 more times, so a total of 4.

Now we can test with Tool Path Backplot

Backplots OK

This completes the programming operations.

Program Management

We have now programmed all the operations with their separate headers and trialing blocks so now we need to make them into program files that can be uploaded to the CNC control for running the machine to produce the part.

If we look at the original aims and the practical fixturing that has been stated before, then the job breaks down into two operations.

Operation 1 is to hold the block in a vice and machine to top face, the 50 mm holes and all the drilling and tapping.

Operation 2 is to hold in a fixture and profile the outside of the plate.

We need to have a proper main header that will place the files in the correct folder on the Siemens 840D control at the top of each program. This header is very format sensitive to ensure the program will upload and be in the correct folder.

The screenshot shows the CIMCO Edit 8.09.16 software interface. The main window displays a CNC program for 'Baffle Plate Op1 Word Version_Face-Profile'. The program text includes comments for 'OPERATION 1' and 'POCKET MILL 3 X 50 MM HOLES', along with various G-code commands like G90, G54, SUPA, T1, G00, S900, M03, F250, M08, M09, and M01. A callout box points to the top of the program with the text 'Place the cursor at the top of the program'. Another callout box points to the 'Program Header' option in the left-hand menu with the text 'Click on Program Header and the box will open'. The 'Insert: Program Header' dialog box is open, showing fields for Program No (101), Work Piece Folder (WKS), Job Folder (JB-PRECISION), Part Description (BAFFLE PLATE), Drawing No (12345), Drawing Issue (A), Operation, and Programmer (JOE SMITH). A callout box points to the 'OK' button with the text 'Complete the cells and click OK to create the Program Haeder'. The dialog box also includes a legend for optional parameters and buttons for Default, Cancel, and OK.

The header will be created SEE BELOW:

CIMCO Edit 8.09.16 - [C:\Users\Ebr...

File Editor NC Functions Backplot File Compare Transmission CNC-Calc

New Open Close Save Save As Print Global Setup

File File Type Edit

Siemens

Copy Cut Paste Select All Undo

Baffle Plate Op1 Word Version_Face-Profile.x 100_MPF.nc x

NC-Assistant

Collapse

Description:

1 %_N_0100_MPF

2 ;\$PATH=/_N_WKS_DIR/_N_JB-PRECISION_WPD

3 ;DESCRIPTION-BAFFLE PLATE

4 ;DRAWING NO-12345 ISSUE-A

5 ;OPERATION-MILL & DRILL & TAP

6 ;PROGRAMMER-JOE SMITH

7 ;DATE/TIME-30/08/2021 19:57:49

8 ;-----

9

13

14 ;-- OPERATION

15 ;-- FACE MILL

16 ;-- USE 50 MM

All other Header Fields can be configured to suit the company system and will be used in Cimco NC-Base database.

See the Program Number, the Directory, the folder, where program number 100 will reside.

The syntax of this line (block) will access the CNC control and place the program file in the correct folder and is very sensitive and must be formatted as shown here in every detail!!

The only variables are "100" the Program number, "WKS" the directory, "JB-PRECISION" the folder name.

Header for Operation 1

Save Save As Print Global Setup

File File Type Edit

Copy Cut Paste Select All

Version_Face-Profile.x 100_MPF.nc x

start

1 %_N_0100_MPF

2 ;\$PATH=/_N_WKS_DIR/_N_JB-PRECISION_WPD

3 ;DESCRIPTION-BAFFLE PLATE

4 ;DRAWING NO-12345 ISSUE-A

5 ;OPERATION-MILL FACE, DRILL & TAP

6 ;PROGRAMMER-JOE SMITH

7 ;DATE/TIME-30/08/2021 19:57:49

8 ;-----

9 ;-- FACE MILL SURFACE

10 ;-- USE 50 MM FACE MILL

11 G90 G40 G17 G97 ;RESET G CODES

12 G54 ;ZERO WORK COORD

DRILL & TAP

Header for Operation 2

141	G00 Z50 M09
142	SX0 Y0
143	M01
144	
145	
146	%_N_0101_MPF
147	;/_N_WKS_DIR/_N_JB-PRECISION_WPD
148	;DESCRIPTION-BAFFLE PLATE
149	;DRAWING NO-12345 ISSUE-A
150	;OPERATION-MILL PROFILE
151	;PROGRAMMER-JOE SMITH
152	;DATE/TIME-31/08/2021 08:43:24
153	;-----
154	;-- OPERATION 2 MILL PROFILE
155	;-- USE 20 MM CARBIDE END MILL
156	;-- TAKE 3.5 MM DEEP CUTS
157	G90 G40 G17 G97
158	G54

Modify

Now cut and paste Operation 2 to a new page and save both programs in the correct folder on your PC for uploading to the CNC control.

The use of block numbers is optional in an 840D control. It can be useful to have block numbers particularly when searching for a program mid start otherwise you will need to search for some text or other.

Under the tab NC Functions see how to set up your block numbering style. SEE BELOW:

CIMCO Edit 8.09.16 - [C:\Users\Ebor Main\OneDrive - Ebor Machine Tools Ltd\My Cimco\Cimco Direct Work\Siemens\Siemens Program Course\Baffle Plate Op2 Prog101.n

File Editor NC Functions Backplot File Compare Transmission CNC-Calc

Block Numbers Advanced renumbering Remove Mark/Delete Range Block Skips X<>Y Spaces Strings Auto insert Upper/Lowercase Adjust Feedrate Adjust Spindle Speed Simple Math Functions Address Adjustments Toolpath Make Tool Statistics List

Siemens Default Program Header Macros

NC-Function Assistant

Expand Modify

Default

Program End
Program comment
G01 Linear move
CHAMFER
CORNER ROL
CYCLE81
CYCLE82
CYCLE83
CYCLE84 Rigid Tapping
CYCLE840 Spring Holder Tapping
CYCLE85
CYCLE86
CYCLE87
CYCLE88
CYCLE89
CYCLE71 - FACE MILL
CYCLE72 - CONTOUR MILL
CYCLE76 - RECTANGULAR SPIGOT
CYCLE77 - CIRCULAR SPIGOT
CYCLE90 - THREAD MILL
HOLES1 - ROW OF HOLES
HOLES2 - BOLT HOLE PATTERN
CYCLE801 - MATRIX OF HOLES
POCKET1 - RECTANGULAR
POCKET2 - CIRCULAR
POCKET3 - RECTANGULAR
POCKET4 - CIRCULAR
SLOT1 - RADIAL SLOTS
SLOT2 - CIRCULAR SLOTS
LONGHOLE - RADIAL SLOTS
TRANS - TEMP DATUM SHIFT ABS
ATrans - TEMP DATUM SHIFT INC
LABEL DEFINITION
GOTO - FOR/BACK
REPEAT LABEL
ROT - ROTATE CO-ORDINATES

NC Functions

```

1 % N_0101 MPF
2 ;=/ N_WKS_DIR/ N_JB-PRECISION WPD
3 ;DESCRIPTION-BAFFLE PLAT Setup: Block numbers - Siemens
4 ;DRAWING NO-12345 ISSUE-
5 ;OPERATION-MILL PROFILE
6 ;PROGRAMMER-JOE SMITH
7 ;DATE/TIME-31/08/2021 08
8 ;-----
9 ;-- OPERATION 2 MILL PRO
10 ;-- USE 20 MM CARBIDE EN
11 ;-- TAKE 3.5 MM DEEP CUT
12 G90 G40 G17 G97
13 G54
14 SUPA G00 Z0
15 T10 D1
16 G00 X-30 Y50
17 S2000 M03 F300
18 Z50 M08
19 G1 Z0 F1500
20 PROF1:
21 G91 Z-3.5 F1000
22 G90
23 G00 X-30 Y50
24 G1 G41 X0 F100
25 Y100 RND=15
26 X83.75 ;START POSITION
27 G3 X113.75 I15 J0 ;FIN P
28 G1 X181.25 ;START POSITI
29 G3 X211.25 I15 J0 ;FIN P
30 G1 X295 RND=15
31 Y0 RND=15
32 X211.25 ;START POSITION
33 G3 X181.25 I-15 J0 ;FIN
34 G1 X113.75 ;START POSITI
35 G3 X83.75 I-15 J0 ;FIN P
36 G1 X0 RND=15
37 Y50
38 G40 X-30
39 RNDT.ABRT.

```

Block numbers - Siemens

Renum settings:

- ☐ Auto block numbering
- ☐ Skip lines without block numbers
- ☐ Renum empty lines
- ☐ Disable standard renumbering
- ☐ Show quick setup
- ☐ Disable undo prompt for large files.
- ☒ Auto indent lines

Block format

Format: 01 Start with: 10 Restart every: Skip lines starting with: %:

Interval: 10 Start from line: 1 Skip: 0 Skip lines containing:

Precede lines with: N Alternate block number character: Restart on lines starting with:

Spaces after block number: 1 From line: To line: Start renumbering from line containing:

Start at next block

Select Block Numbering

OK to save

Set cells to suit your style of block numbering

Help Default Cancel OK

SEE BELOW:

NC-Assistant	1	%_N_0101_MPF
band	2	;=/_N_WKS_DIR/_N_JB-PRECISION_WPD
Modify	3	;DESCRIPTION-BAFFLE PLATE
lt	4	;DRAWING NO-12345 ISSUE-A
am End	5	;OPERATION-MILL PROFILE
am comment	6	;PROGRAMMER-JOE SMITH
.linear move	7	;DATE/TIME-31/08/2021 08:43:24
AFER	8	;-----
VER ROUNDING	9	;-- OPERATION 2 MILL PROFILE
E81	10	;-- USE 20 MM CARBIDE END MILL
E82	11	;-- TAKE 3.5 MM DEEP CUTS
E83	12	N10 G90 G40 G17 G97
E84 Rigid Tapping	13	N20 G54
E840 Spring Holder Tapping	14	N30 SUPA G00 Z0
E85	15	N40 T10 D1
E86	16	N50 G00 X-30 Y50
E87	17	N60 S2000 M03 F300
E88	18	N70 Z50 M08
E89	19	N80 G1 Z0 F1500
E71 - FACE MILL	20	N90 PROF1:
E72 - CONTOUR MILL	21	N100 G91 Z-3.5 F1000
E76 - RECTANGULAR SPIGOT	22	N110 G90
E77 - CIRCULAR SPIGOT	23	N120 G00 X-30 Y50
E90 - THREAD MILL	24	N130 G1 G41 X0 F100
:S1 - ROW OF HOLES	25	N140 Y100 RND=15
:S2 - BOLT HOLE PATTERN	26	N150 X83.75 ;START POSITION
E801 - MATRIX OF HOLES	27	N160 G3 X113.75 I15 J0 ;FIN POSITION
:ET1 - RECTANGULAR	28	N170 G1 X181.25 ;START POSITION
:ET2 - CIRCULAR	29	N180 G3 X211.25 I15 J0 ;FIN POSITION
:ET3 - RECTANGULAR	30	N190 G1 X295 RND=15
:ET4 - CIRCULAR	31	N200 Y0 RND=15
`1 - RADIAL SLOTS	32	N210 X211.25 ;START POSITION
`2 - CIRCULAR SLOTS	33	N220 G3 X181.25 I-15 J0 ;FIN POSITION
:HOLE - RADIAL SLOTS	34	N230 G1 X113.75 ;START POSITION
:S - TEMP DATUM SHIFT ABS	35	N240 G3 X83.75 I-15 J0 ;FIN POSITION
:NS - TEMP DATUM SHIFT INC	36	N250 G1 X0 RND=15
L DEFINITION	37	N260 Y50
J - FOR/BACK	38	N270 G40 X-30
:AT LABEL	39	N280 ENDLABEL:
- ROTATE CO-ORDINATES	40	N290 REPEAT PROF1 P=3
owell	41	N300 G00 Z50
radius compensation off	42	N310 X0 Y0
radius compensation, left	43	N320 M01
radius compensation, right	44	N330 M30
Machine coordinate system setting	45	
absolute coordinates mode		

See Block numbers added

See below the programs for the two operations with some comments:

OPERATION 1

```
%_N_0100_MPF
;$PATH=/_N_WKS_DIR/_N_JB-PRECISION_WPD
;DESCRIPTION-BAFFLE PLATE
;DRAWING NO-12345 ISSUE-A
;OPERATION-MILL FACE, DRILL & TAP
;PROGRAMMER-JOE SMITH
;DATE/TIME-30/08/2021 19:57:49
;-----
;-- FACE MILL SURFACE
;-- USE 50 MM FACE MILL
G90 G40 G17 G97 ;RESET G CODES
G54 ;MAIN WORK SHIFT
SUPA G00 Z0 ;SUPRESS OFFSETS/WORK SHIFT MOVE Z UP TO MACHINE ZERO
T1 D1 ;TOOL AND OFFSET ACTIVATED
G00 X0 Y0 ; MOVE X,Y TO POSITION
S900 M03 F250 ; START SPINDLE AT 900 RPM
Z50 M08; MOVE Z TO RETRACTION PLAIN, TURN ON COOLANT
CYCLE71(50,5,5,0,0,0,295,100,0,2,40,30,0.5,100,31,20);ROUGHING
CYCLE71(50,5,5,0,0,0,295,100,0,2,40,30,0,100,32,20);FINISHING
G00 Z50 M09; MOVE Z TO RETRACTION PLAIN, SWITCH OFF COOLANT
X0 Y0; MOVE X,Y TO POSITION
M01; OPTION STOP (IF SELECTED SHOULD TURN OF THE SPINDLE)

;-- POCKET MILL 3 X 50 MM HOLES
;-- USE 16 MM CARBIDE END MILL
G90 G40 G17 G97
```

G54

SUPA G00 Z0

T8 D1

G00 X0 Y0

S2300 M03 F250

Z50 M08

TRANS X50 Y50 ;-- TEMPORARY WORK SHIFT TO HOLE #1 ABSOLUTE

POCKET: ; START LABEL

POCKET4(50,0,2,-14,25,0,0,3,1,0,220,50,3,21,9,0,0,8,1.5);ROUGHING

POCKET4(50,0,2,-14,25,0,0,0,0,0,220,50,3,12,,,,,);FINISHING

ENDLABEL: ; END LABEL

ATRANS X97.5 ;-- TEMPORARY WORK SHIFT TO HOLE #1 INCREMENTAL

REPEAT POCKET ;-- HOLE #2

ATRANS X97.5 ;-- TEMPORARY WORK SHIFT TO HOLE #1 INCREMENTAL

REPEAT POCKET ;-- HOLE #3

TRANS ;-- CANCEL ALL TEMP WORK SHIFT & RETURN TO G54

G00 Z50 M09

X0 Y0

M01

;-- CENTRE DRILL ALL HOLES

;-- USE T2 CENTRE DRILL

G90 G40 G17 G97

G54

SUPA G00 Z0

T2 D1

G00 X0 Y0

S1000 M03 F80

Z50 M08

TRANS X50 Y50

CDRILL:

MCALL CYCLE81(10,0,2,-5,);-- CENTRE DRILL M10 HOLES

HOLES2(0,0,35,0,90,4);-- BOLT HOLE PATTERN HOLE #1

MCALL

ENDLABEL:

ATRANS X97.5

REPEAT CDRILL ;-- HOLE #2

ATRANS X97.5

REPEAT CDRILL ;-- HOLE #3

TRANS ;-- THIS COMMAND WILL CANCEL ALL TEMP WORK SHIFT & RETURN TO G54

MCALL CYCLE81(10,0,2,-5);-- CENTRE DRILL 4 X 11 MM HOLES

X15 Y15

X280 Y15

X280 Y85

X15 Y85

MCALL

G00 Z50 M09

X0 Y0

M01

;-- DRILL 4 HOLES 11 MM THROUGH

;-- USE T27 11 MM CARBIDE DRILL

G90 G40 G17 G97

G54

SUPA G00 Z0

T27 D1

G00 X0 Y0

S3000 M03 F300

Z50 M08

MCALL CYCLE81(10,0,2,-15,);-- DRILL 11 MM HOLES

X15 Y15

X280 Y15

X280 Y85

X15 Y85

MCALL

G00 Z50 M09

X0 Y0

M01

;-- DRILL M10 HOLES

;-- USE T22 8.5 MM CARBIDE DRILL

G90 G40 G17 G97

G54

SUPA G00 Z0

T22 D1

G00 X0 Y0

S3000 M03 F300

Z50 M08

TRANS X50 Y50

DRILL:

MCALL CYCLE81(10,0,2,-15,);-- DRILL 8.5 MM HOLES

HOLES2(0,0,35,0,90,4);-- BOLT HOLE PATTERN HOLE #1

MCALL

ENDLABEL:

ATRANS X97.5

REPEAT DRILL ;-- HOLE #2

ATRANS X97.5

REPEAT DRILL ;-- HOLE #3

TRANS ;-- THIS COMMAND WILL CANCEL ALL TEMP WORK SHIFT & RETURN TO G54

G00 Z50 M09

X0 Y0

M01

;-- TAP M10 HOLES

;-- USE T43 M10 SPIRAL POINT TAP

;-- USE RIGID TAPPING CYCLE84

G90 G40 G17 G97

G54

SUPA G00 Z0

T43 D1

G00 X0 Y0

S200 M03

Z50 M08

TRANS X50 Y50

TAP:

MCALL CYCLE84(30,0,5,-18,,0,3,,1.5,0,200,400,3,0,0,1,,);-- TAP M10 THROUGH

HOLES2(0,0,35,0,90,4);-- BOLT HOLE PATTERN HOLE #1

MCALL

ENDLABEL:

ATRANS X97.5

REPEAT TAP ;-- HOLE #2

ATRANS X97.5

REPEAT TAP ;-- HOLE #3

TRANS ;-- THIS COMMAND WILL CANCEL ALL TEMP WORK SHIFT & RETURN TO G54

G00 Z50 M09

SX0 Y0

45

M01

M30

OPERATION 2

%_N_0101_MPF

;\$PATH=/_N_WKS_DIR/_N_JB-PRECISION_WPD

;DESCRIPTION-BAFFLE PLATE

;DRAWING NO-12345 ISSUE-A

;OPERATION-MILL PROFILE

;PROGRAMMER-JOE SMITH

;DATE/TIME-31/08/2021 08:43:24

;-----

;-- OPERATION 2 MILL PROFILE

;-- USE 20 MM CARBIDE END MILL

;-- TAKE 3.5 MM DEEP CUTS

G90 G40 G17 G97

G54

SUPA G00 Z0

T10 D1

G00 X-30 Y50

S2000 M03 F300

Z50 M08

G1 Z0 F1500

PROF1:

G91 Z-3.5 F1000

G90

G00 X-30 Y50

G1 G41 X0 F100

Y100 RND=15

X83.75 ;START POSITION

G3 X113.75 I15 J0 ;FIN POSITION

G1 X181.25 ;START POSITION

G3 X211.25 I15 J0 ;FIN POSITION

G1 X295 RND=15

Y0 RND=15

X211.25 ;START POSITION

G3 X181.25 I-15 J0 ;FIN POSITION

G1 X113.75 ;START POSITION

G3 X83.75 I-15 J0 ;FIN POSITION

G1 X0 RND=15

Y50

G40 X-30

ENDLABEL:

REPEAT PROF1 P=3

G00 Z50

X0 Y0

M01

M30