Mastercam® to Mazatrol® Post-Processor Tutorial

Introduction

The following tutorial instructs the user in the approach to programming that allows a *Mastercam®* file with it's associated toolpaths to output the desired *Mazatrol®* code.

It is not the intention of this tutorial to teach the use of *Mastercam®* or the *Mazatrol®* conversational system. It is assumed that the user of this product has been instructed in the use of the former items. We provide in addition to this tutorial both a help file accessible when in the Mazatrol Menu by clicking on **Help** and a **Mazak for Mastercam** Manual - For Mastercam instruction please contact your local Mastercam reseller. For mazatrol instruction please refer to your *Mazak/ Mazatrol* Programming Manuals or contact your local *Mazak* representative.

Section 1. Programming a Mill Part
## Section 2. Programming a Lathe Part

Note: This text was compiled using Version 8.0.8 of the Mazatrol Product – some dialogs presented may have changed or you may be using either an earlier or later version of the software.

Section 1 - Mill

1. Creating simple face and contour toolpaths

Exercise 1 - Opening the part file

1. Choose **Main Menu, File, Get**
2. Navigate to the folder with the tutorial parts.
3. Select **Mazak_1_Mill.mc9**; then choose **Open.**
4. Choose, **Main Menu, Toolpaths, Job Setup**
5. Enter settings as shown.

This setting will be set as INITIAL Z
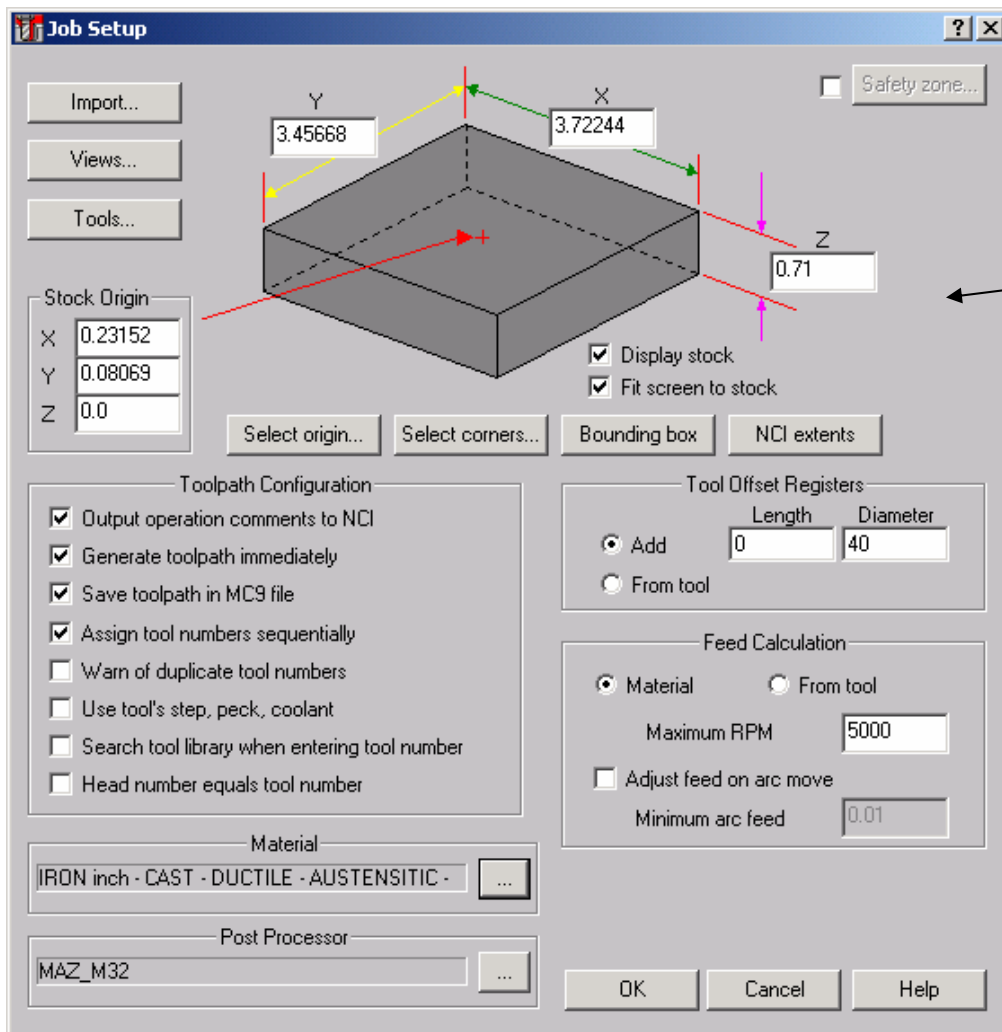
Note: Job Setup settings will affect the first line of the mazatrol PNR and MAT i.e. the material selected will be output and the Z depth of the material will be output as INITIAL-Z see below:

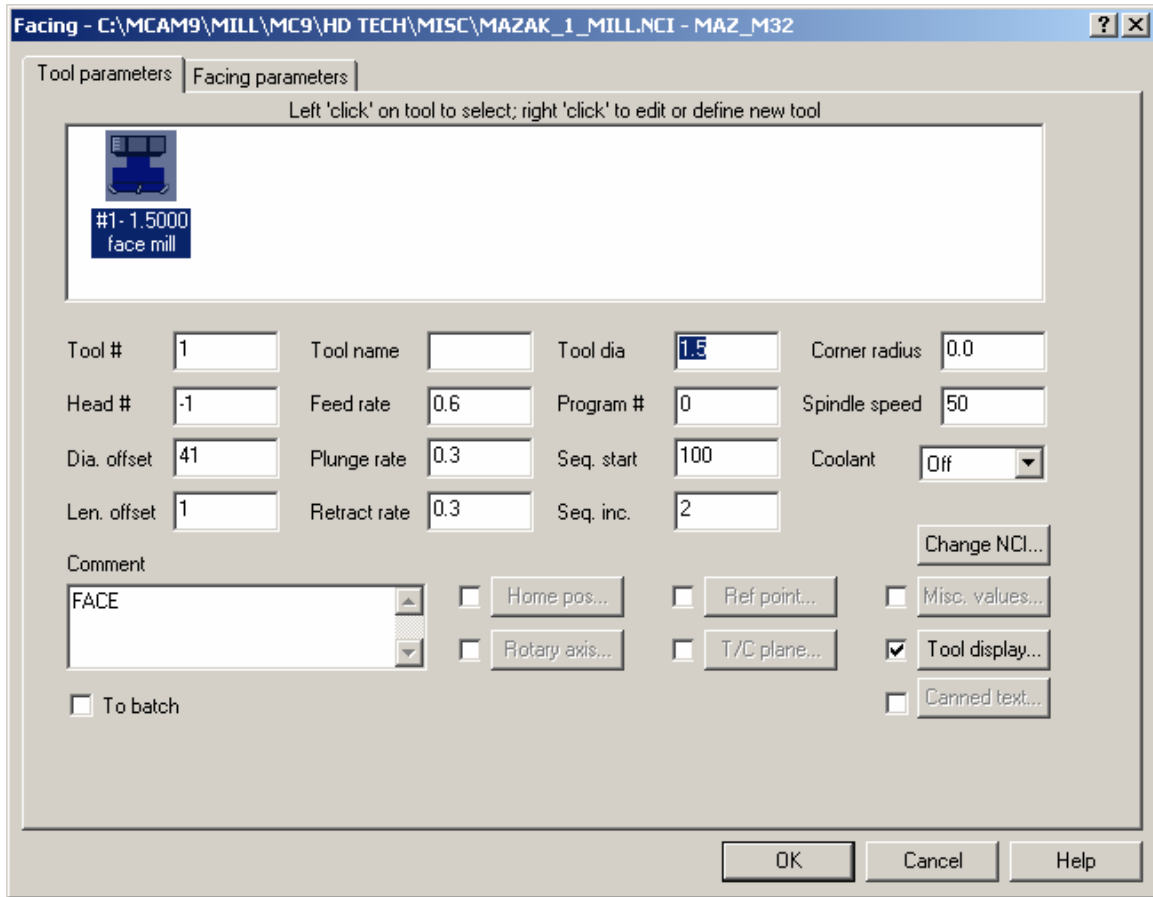| PNR	MAT	INITIAL-Z	ATC MODE	MULTI MODE	MULTI FLG	PITCH-X	PITCH-Y
0	IRON	0.7100	1	OFF			

The other settings will have to be manually entered by the user if desired either using the editor (if available) or at the control. Also the values for federate and spindle speed that are set in the mastercam parameter pages will also output to the Mazatrol code.

Exercise 2 - Creating Facing Toolpath for outside profile

1. Choose **Main Menu, Toolpaths, Face**
2. Select outside profile as shown using chain
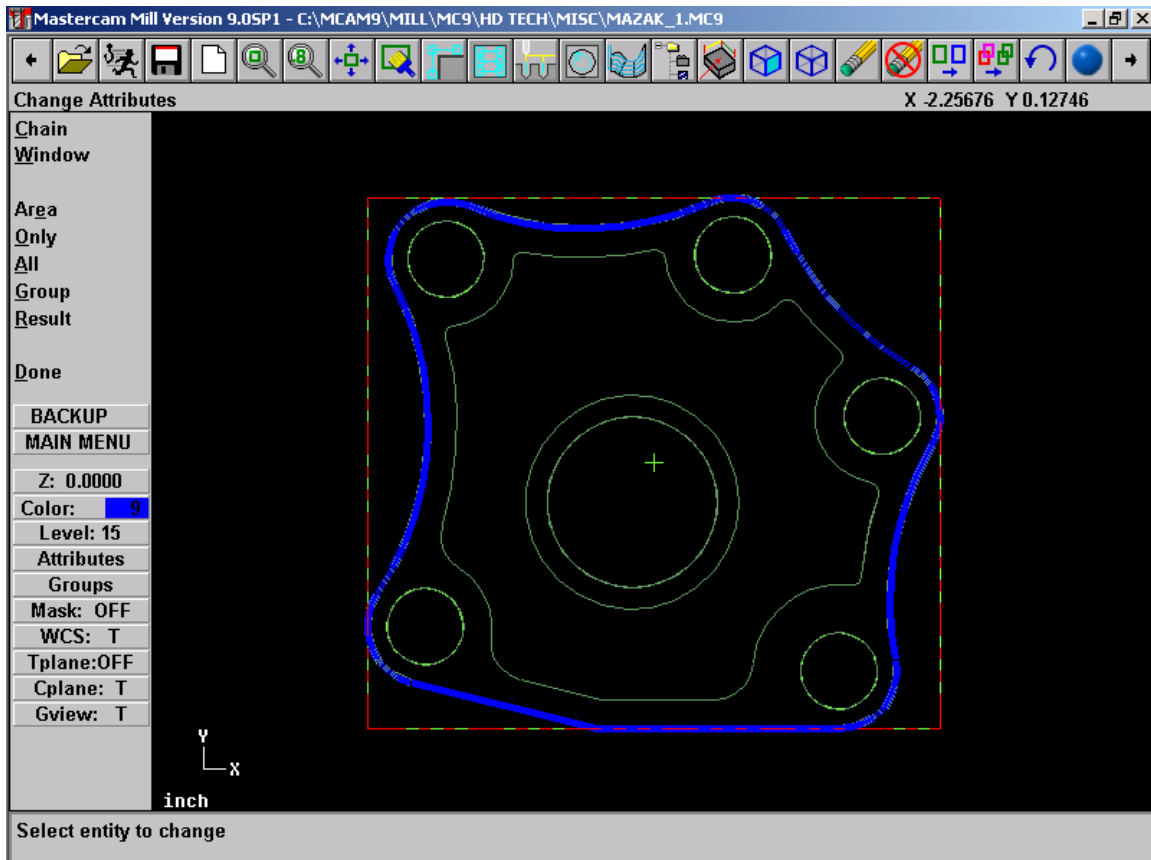
2. Select **Done**
3. Select or Create a 1.5"Dia Face Mill as shown.

4. Click on **Misc. Values** button and set Face Machining to Face as shown below

5. Click **OK** when done.
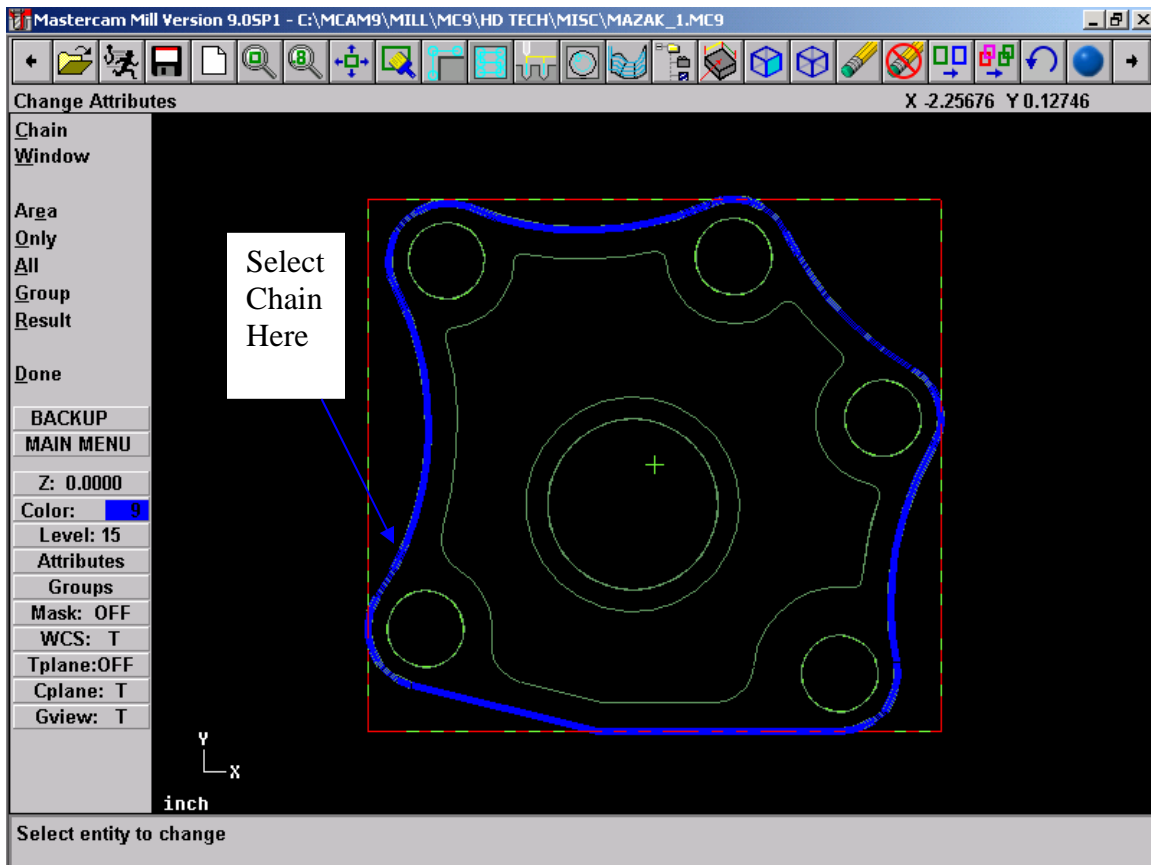6. Click on **Facing Parameters Tab** and set Values as shown;
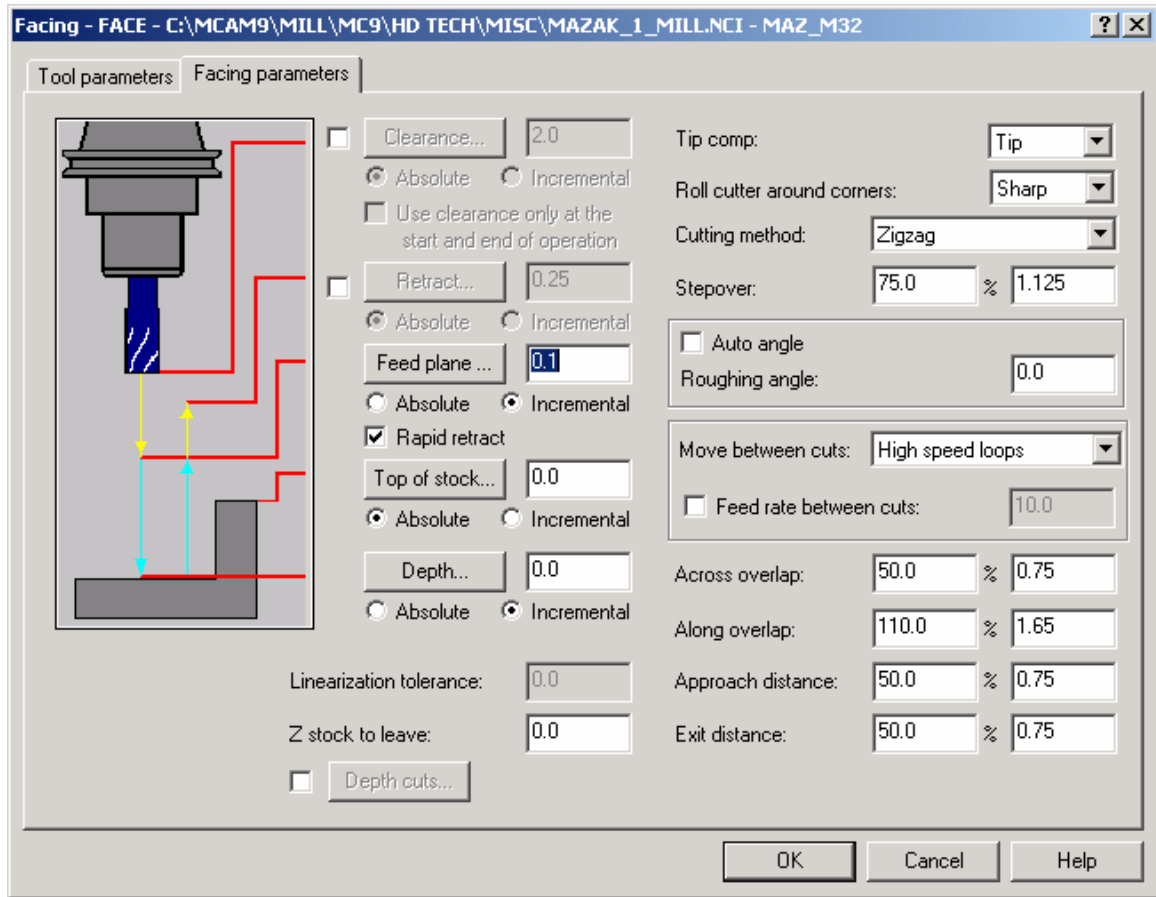
7. Click on **OK** when completed.

Exercise 3 - Creating Contour Toolpath for outside profile

1.   Choose **Main Menu, Toolpaths, Contour**
2.   Select   outside profile as shown using chain

3.   Select **Done**
4.   Select 0.5" Dia Flat end Mill as shown.
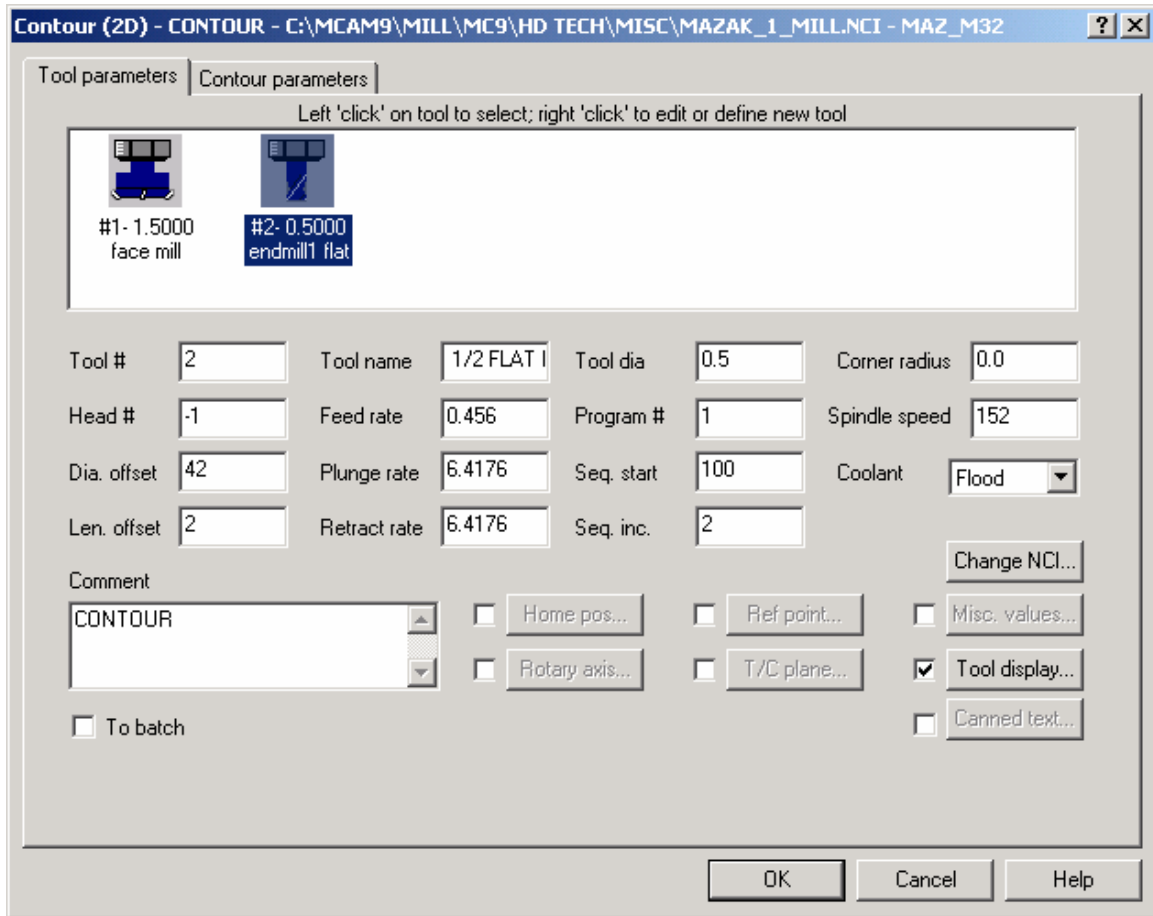
5. Click on **Misc. Values** button and modify settings as shown below
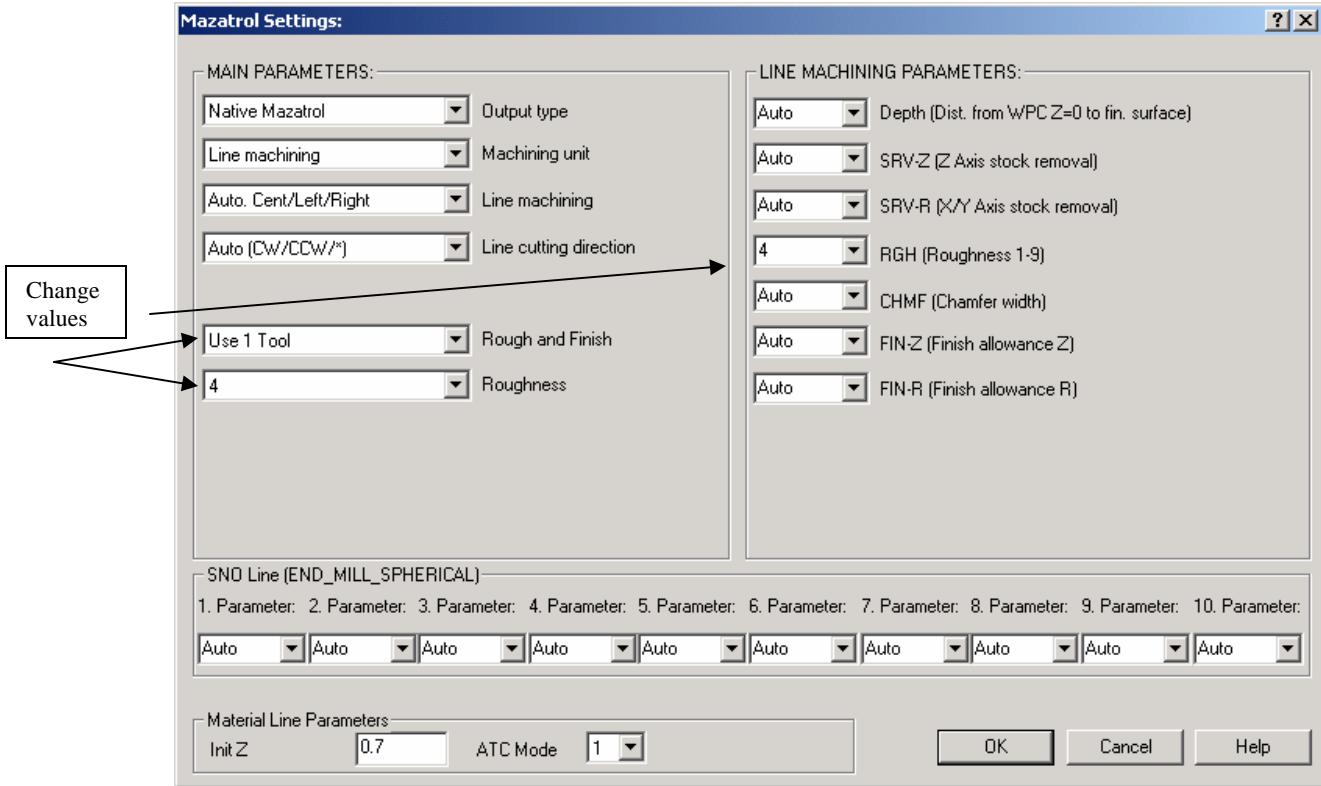
Note: As you may notice – the Misc. Values dialog box allows every setting in the mazatrol SNO line and UNIT (UNO) line to be set by the user and override the automatically set values output by the post-processor. This will be shown in more detail in the next chapter.

*Note: Another advantage of using the Mazatrol Post-Processor is that we can output lead-in and lead-out values from mastercam. In the previous settings we have computer compensation with left direction. Therefore only use LINE-CTR so that correct accuracy is maintained. You can of c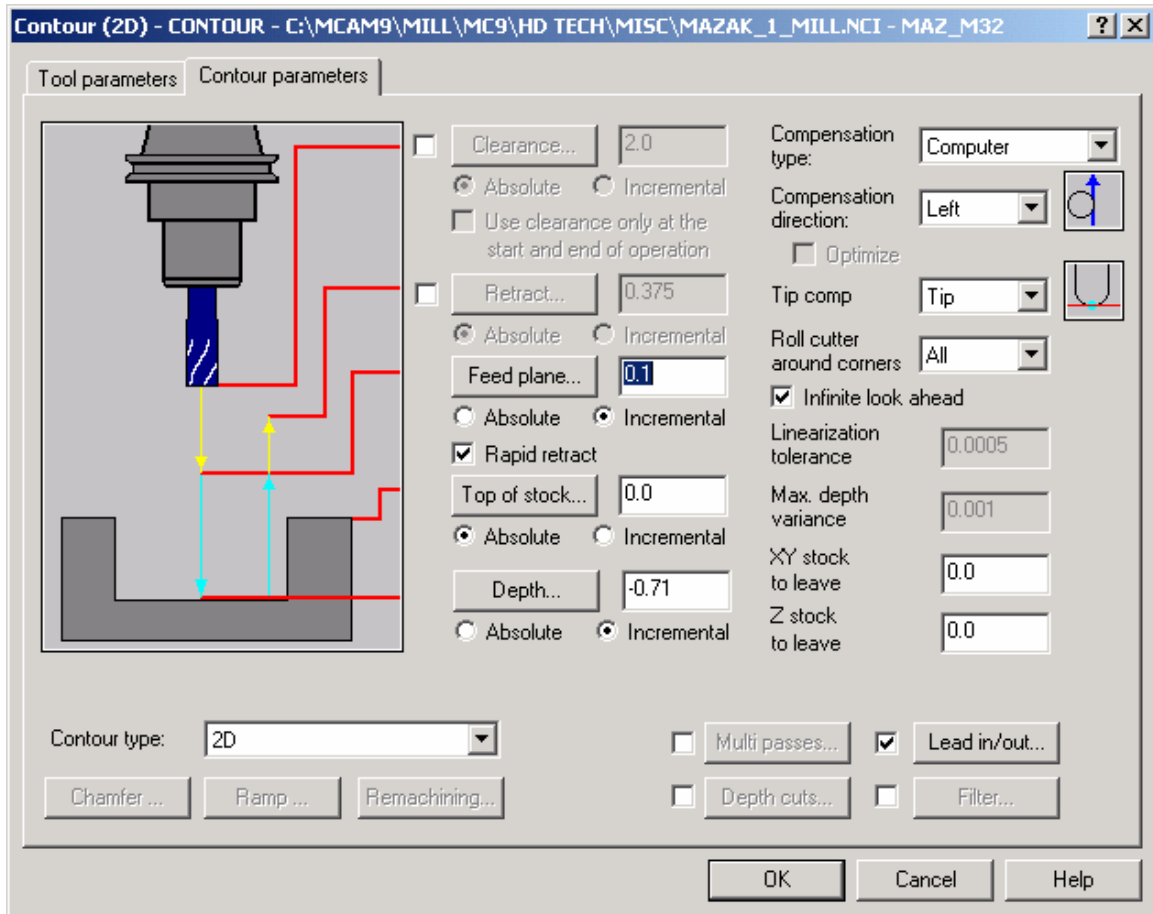ourse also use other type of compensation such as LINE-LFT and LINE-RGT but in those cases it would be safer to set Compensation to **Control** so that the Control picks up the tool radius and compensates accordingly.*

6. Select **Done**. This should return you to the operations manager. Select **Post** Modify settings as shown below. (In this example we are using the M32 post-processor shown as MAZ_32.PST. Yours may vary but all the Mazatrol Post-Processors will have the format of MAZ_XXX.PST)
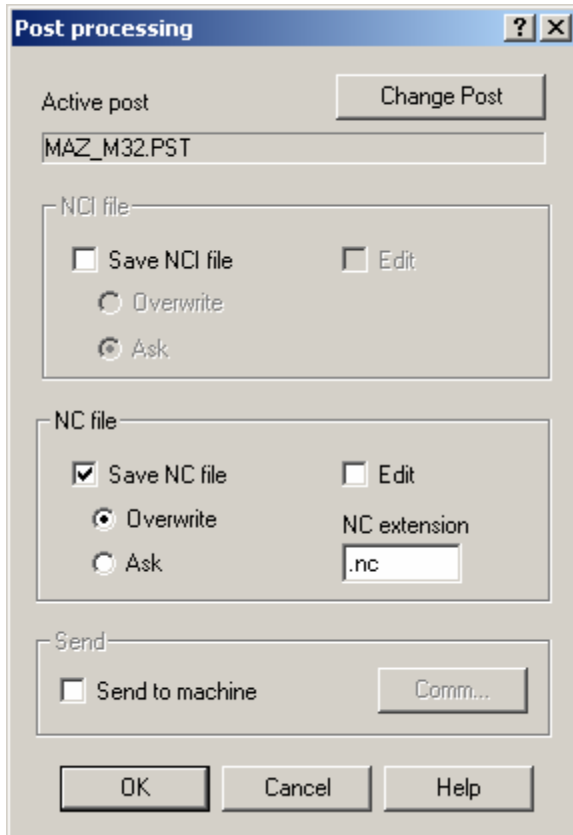
7. Select **OK**. The file name dialog should then appear as shown below:

Note: We do not need to create an NC file but Mastercam needs to have this setting so that the post-processor can function

8. Click **Save**.

The **Mazak Menu** will then appear in place of the **Mastercam Main Menu**
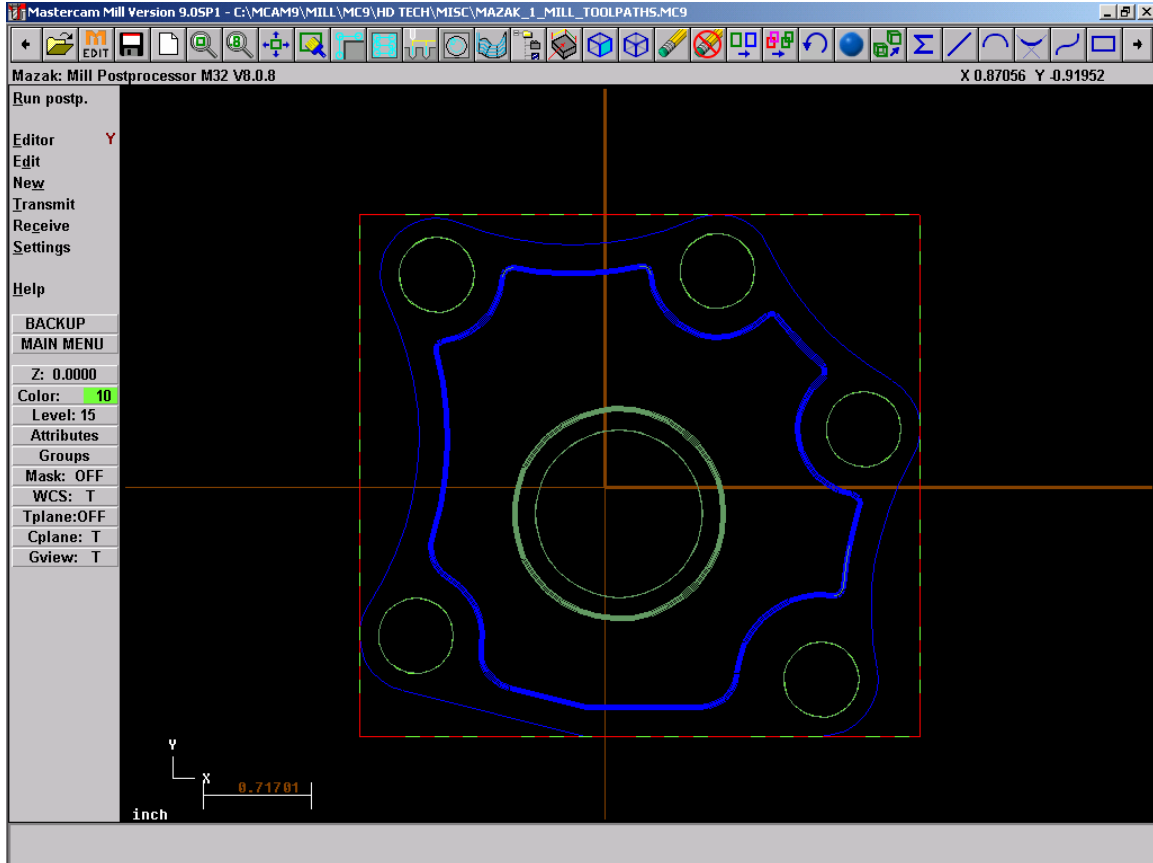
10. From this menu select **Run postp.** to run the Mazatrol Post.

11. Select a number between 1 and 9999 and hit **OK**. This will be the program number for your Mazatrol output file.

You should then see output as shown below (output below is shown as a Notepad window – if you have purchased the Editor and you have the **Editor** set to **Y**es in the Mazatrol Menu the output will open up in the Mazatrol Editor)

```
0001.MED - Notepad                                                      _|□|×|
File  Edit  Format  Help
0001,            ,0000,0000,0000                                              ▲
PNR MAT       INITIAL-Z ATC MODE     MULTI MODE  MULTI FLG    PITCH-X      PITCH-Y
0    IRON      0.7100    1            OFF
-----------------|------------------------------------------------------------
UNO UNO       DEPTH     SRV-Z       SRV-R      BTM   WAL  FIN-Z       FIN-R
1    FACE-MIL  0.0000    .001        *         1     *    0           0
SNO SNO    NOM.  NO.   APRCH-X   APRCH-Y  TYPE ZFD DEP-Z  WID-R C-SP FR     M   M
1    F-MILL  1.50      ?         ?        XB1  G01 0.0010 1.05 50    0.600 3   8
FIG PTN       X         Y         R/0      I         J          P          CNR
1    LINE      -3.0925   -1.6475
2    LINE      2.6553    -1.6475
3    LINE      2.6677    -1.6475
4    CCW       2.6677    -0.7834   0.4321   2.6677    -1.2155
5    LINE      -2.5141   -0.7834
6    CW        -2.5141   0.0807    0.4321   -2.5141   -0.3514
7    LINE      2.9560    0.0807
8    CCW       2.9560    0.9448    0.4320   2.9560    0.5128
9    LINE      -2.3055   0.9448
10   LINE      -2.3279   0.9448
11   CW        -2.3279   1.8089    0.4320   -2.3279   1.3769
12   LINE      2.7487    1.8089
------------------------------------------------------------------------------
UNO UNO       DEPTH     SRV-Z       SRV-R      RGH   CHMF        FIN-Z       FIN-R
2    LINE-CTR  0.7100    0.7100      0.25       4     *           0           0
SNO SNO    NOM.  NO.   APRCH-X   APRCH-Y  TYPE ZFD DEP-Z  WID-R C-SP FR     M   M
1    E-MILL  0.50 E    ?         ?        *    G01 0.7100 *    152   0.456 3   8
FIG PTN       X         Y         R/0      I         J          P          CNR
1    LINE      -2.1030   -0.0352
2    LINE      -1.8035   -0.0524
3    CCW       -1.4868   0.2299    0.3000   -1.7863   0.2471
4    CCW       -1.6745   1.1243    1.7501   -3.2340   0.3301
5    CW        -0.8804   1.9861    0.6250   -1.1176   1.4079                   ▼
◄                                                                         ►  //
```
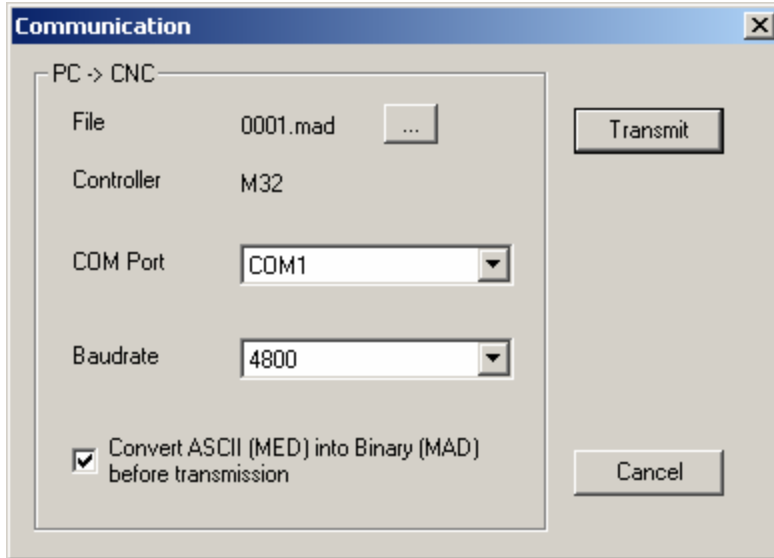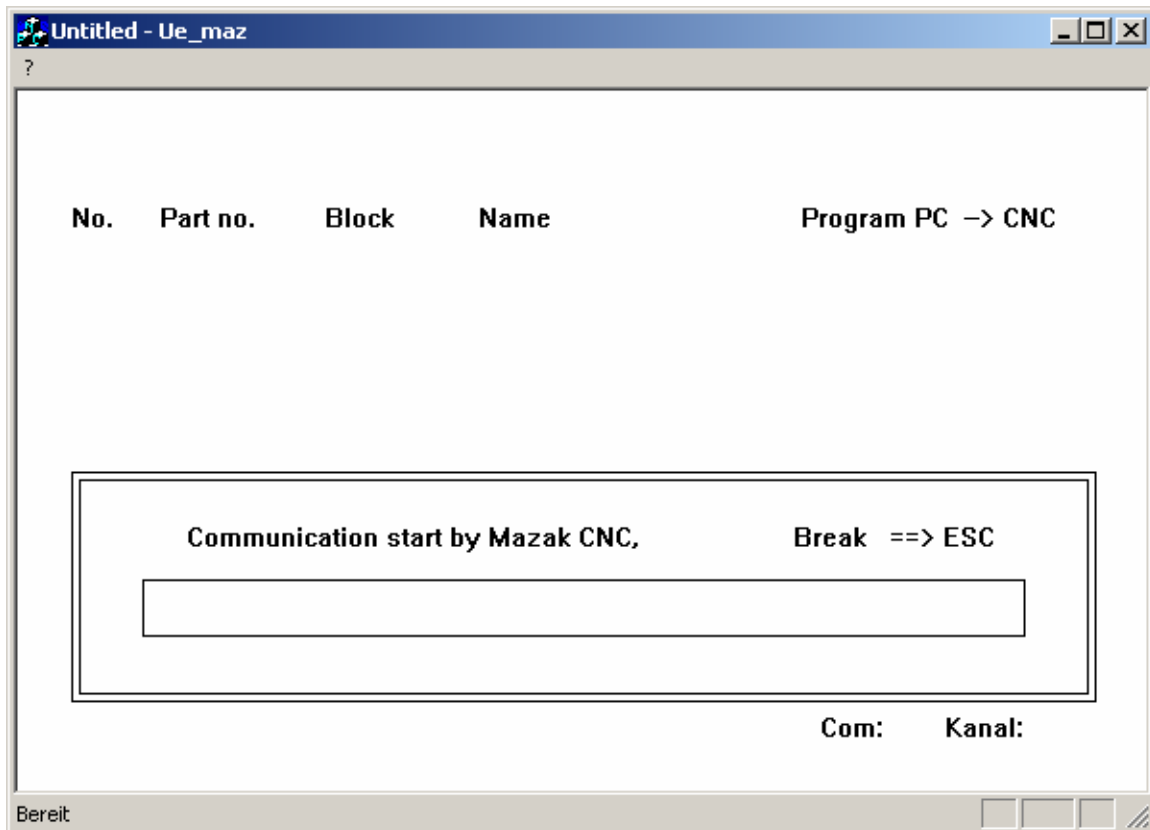
12. Close this window.

We will then send this program to the controller

13. From the Mazatrol Menu select **Transmit.**

Communication ✕

PC -> CNC

| File | 0001.mad | ... |
| Controller | M32 | |

COM Port [COM1 ▼]

Baudrate [4800 ▼]

☑ Convert ASCII (MED) into Binary (MAD)
before transmission

[Transmit]

[Cancel]

15. If the settings are correct and you are using the Built in DNC click **Transmit.**

Untitled - Ue_maz _ ☐ ✕

?

No.   Part no.   Block   Name   Program PC → CNC

Communication start by Mazak CNC,   Break ==> ESC

Com: Kanal:

Bereit

This is the progress bar.

To complete the download complete the following steps at
The **Mazak Controller**.

➢ PROGRAM-LIST or INDEX
➢ DATA IN/OUT
➢ CMT-NC
➢ INPUT
➢ ENTER THE PROGRAM NUMBER AND SELECT INPUT
➢ HIT START

You should then see the file being downloaded by a blue bar filling the progress bar shown above.

Congratulations! You have created your first mastercam to mazatrol program.

16. Hit esc once the Progress Bar is completed.

17. Hit esc to get back to Mastercam Main Menu.

Save File as Mazak_1_Mill_1.mc9

2. Adding Pocketing and Drill Toolpaths

Exercise 1 - Creating Pocket Toolpath

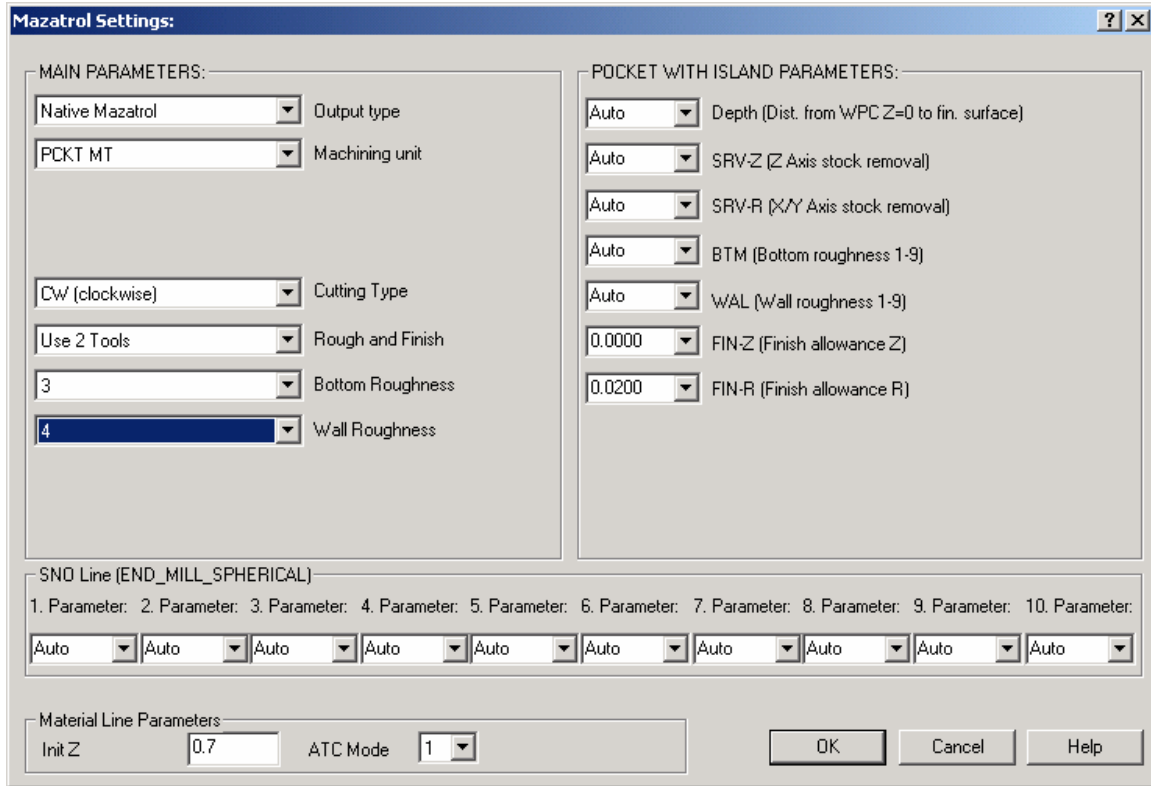We will re-open the file we had previously created to add some more toolpaths

1. Choose **Main Menu, File, Get**
2. Navigate to the folder with the tutorial parts.
3. Select **Mazak_1_Mill_1.mc9**; then choose **Open.**
4. Choose **Main Menu, Toolpaths, Pocket**
5. Chain outside profile shown in Blue and Inside Island as shown in Green
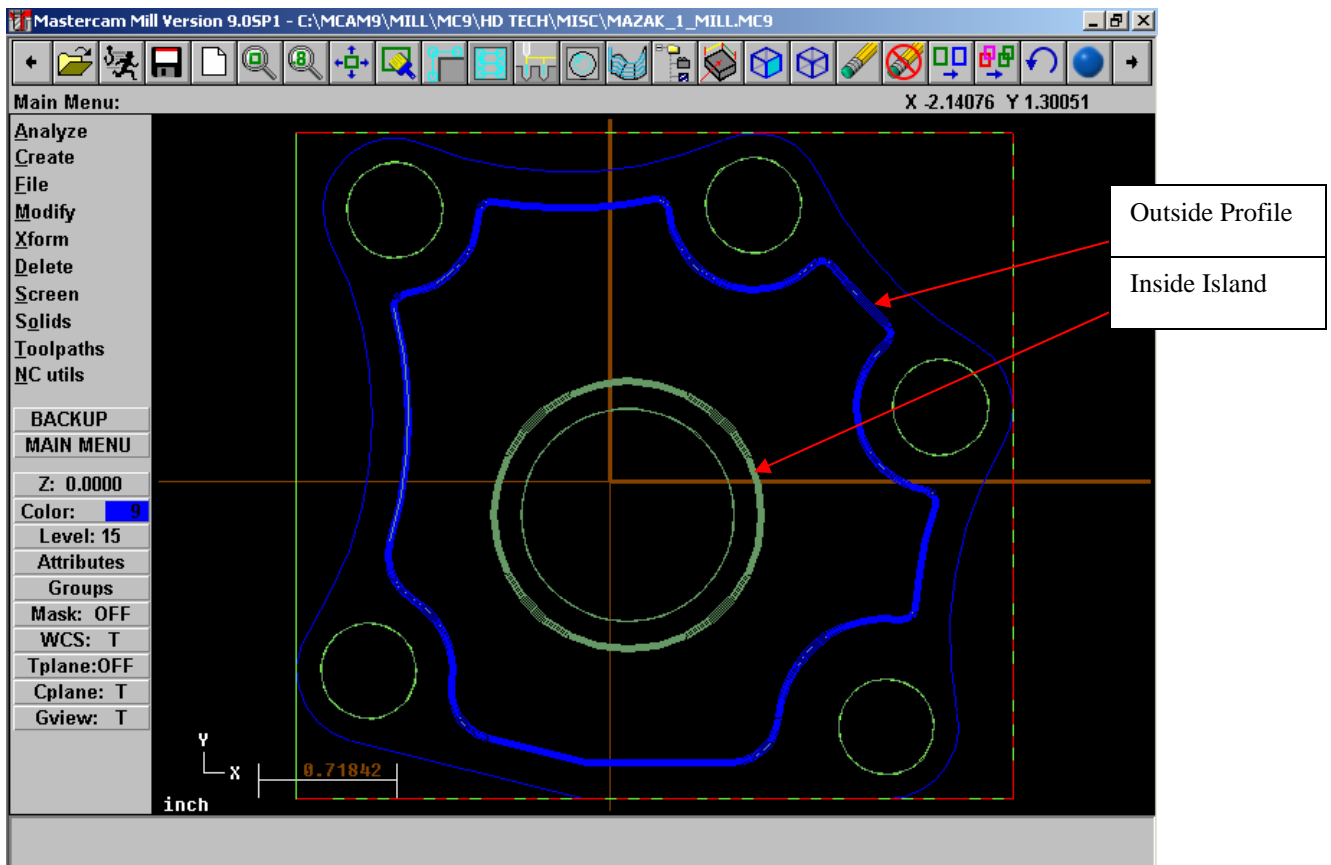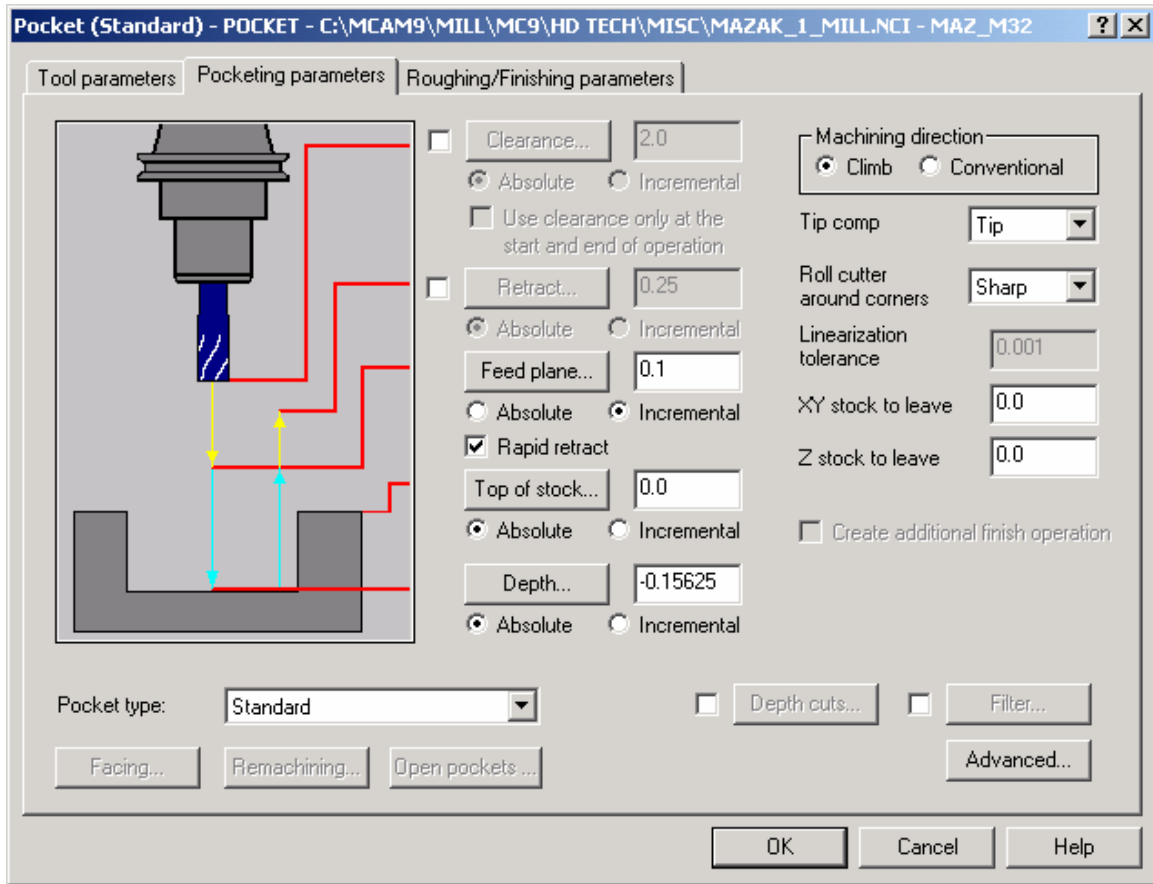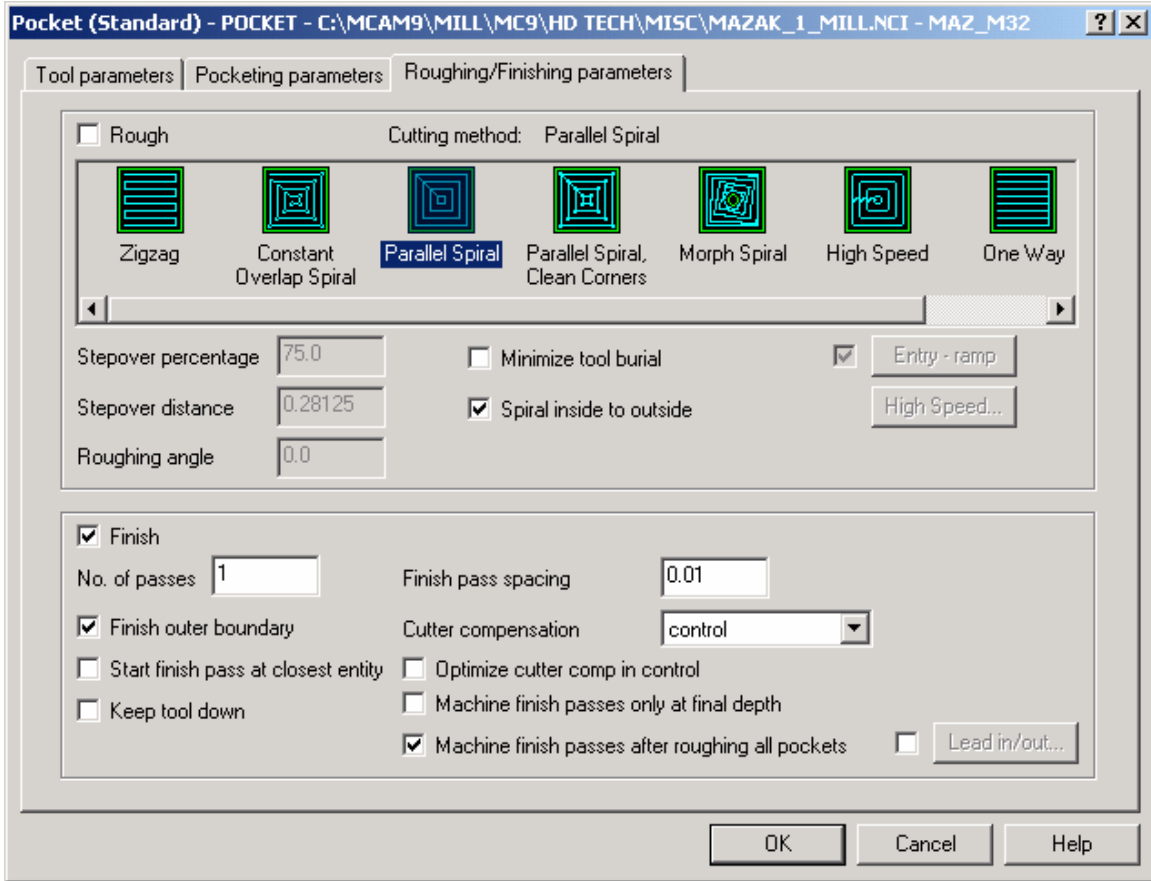

6. Select **Done**
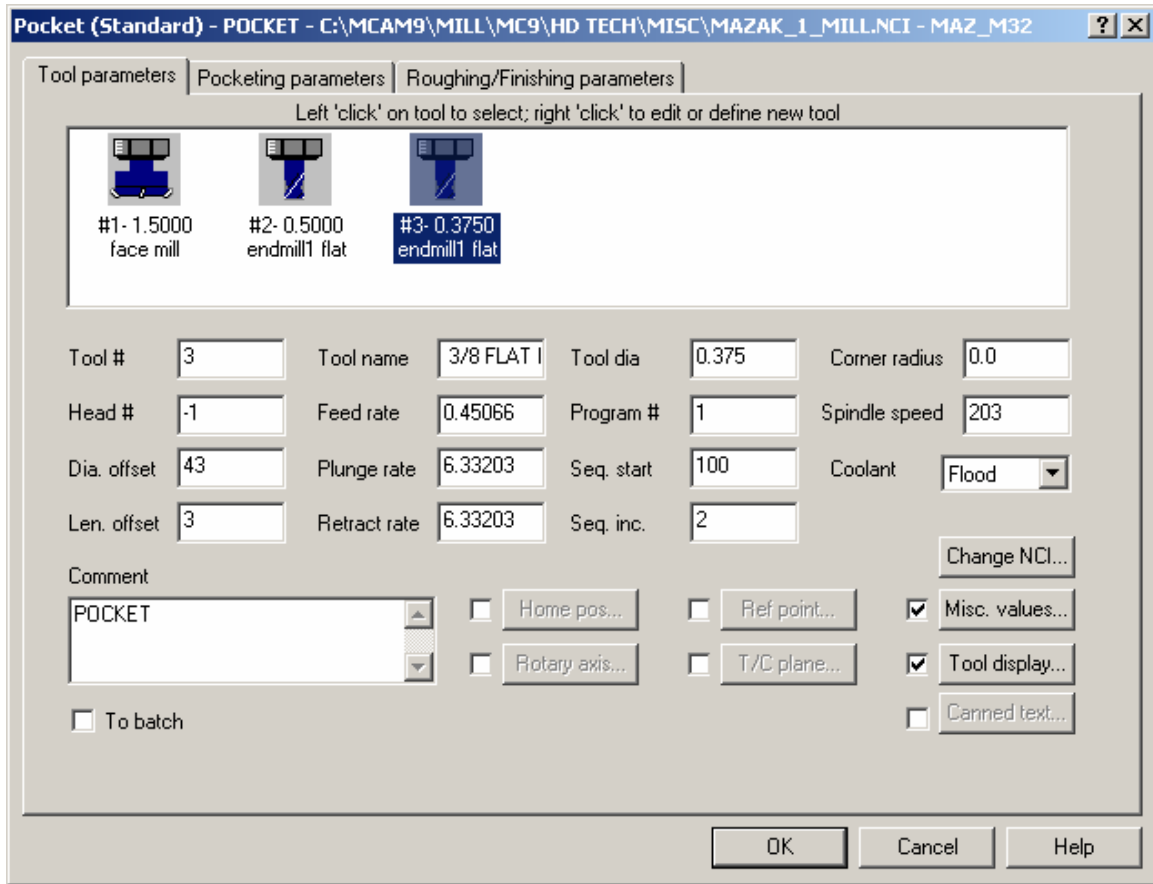
7. Set **Tool Parameters** as shown

8. Set **Misc. Values** as shown:

9. Set **Pocketing Parameters** and **Roughing/Finishing Parameters** as shown below:

Pocket (Standard) - POCKET - C:\MCAM9\MILL\MC9\HD TECH\MISC\MAZAK_1_MILL.NCI - MAZ_M32

Tool parameters | Pocketing parameters | Roughing/Finishing parameters

☐ Clearance... | 2.0
○ Absolute ○ Incremental
☐ Use clearance only at the start and end of operation

☐ Retract... | 0.25
○ Absolute ○ Incremental

Feed plane... | 0.1
○ Absolute ● Incremental
☑ Rapid retract

Top of stock... | 0.0
● Absolute ○ Incremental

Depth... | -0.15625
● Absolute ○ Incremental

Machining direction
● Climb ○ Conventional

Tip comp | Tip

Roll cutter around corners | Sharp

Linearization tolerance | 0.001

XY stock to leave | 0.0

Z stock to leave | 0.0

☐ Create additional finish operation

Pocket type: | Standard

☐ Depth cuts... ☐ Filter...

Facing... | Remachining... | Open pockets ...

Advanced...

OK | Cancel | Help

Note: It is best not to use Depth Cuts when machining pockets. If depth cuts are used unnecessarily long code is output. It is best if you set the value SRV-Z within the misc. values dialog.

*Note: To have the option of either using one tool or two tools for roughing and finishing we can set this at the Rough and Finish pull down menu in the **Misc. Values** dialog box (this option is also available for contour machining equivalent to LINE machining in Mazatrol). We have also set specific Bottom finishes and Wall finishes. **In the mastercam toolpaths it is not possible to create or activate many of these types of conversational language settings therefore in many cases the only access to these parameters will be through the misc. values pages as shown above.***

Sample output below when this is processed.

```
--------------------------------------------------------------------------------
UNO UNO     DEPTH    SRV-Z    SRV-R    BTM  WAL  FIN-Z    FIN-R
1  PCKT.MT  0.0912   0.0912   *     1   1   0      0
SNO SNO   NOM.  NO.  APRCH-X  APRCH-Y  TYPE ZFD DEP-Z  WID-R C-SP FR   M  M
1  E-MILL  0.38 E        ?          ?      CW      G01 0.0912    0.27 203  0.450 3 8
2  E-MILL  0.38 E              ?            ?      CW      G01         0.27 203  0.450 3 8
```

## Exercise 2 - Creating Drill Toolpaths with Multiple Tools

Select the following:

1. **Main Menu**
2. **Toolpaths**
3. **Drill**
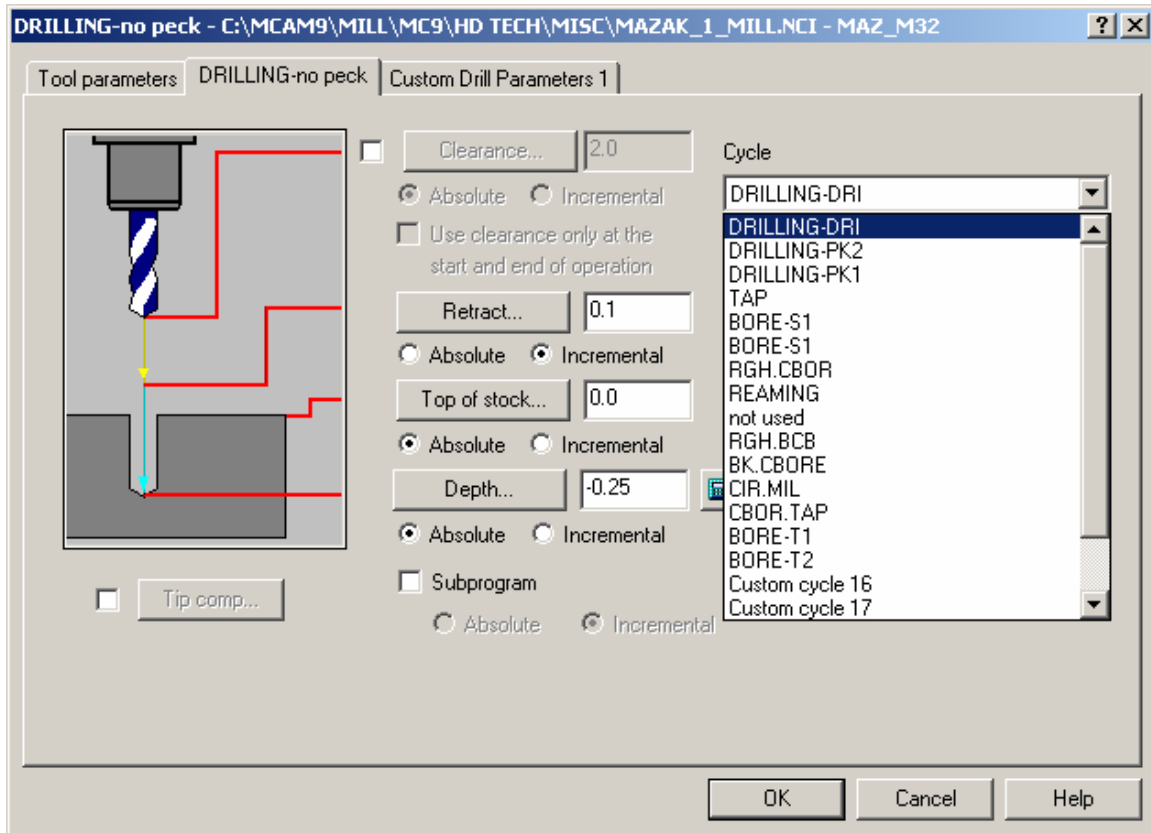4. The five (5) x 0.5"dia circles
5. **Done**
6. **Done**

Set Program # to 10001
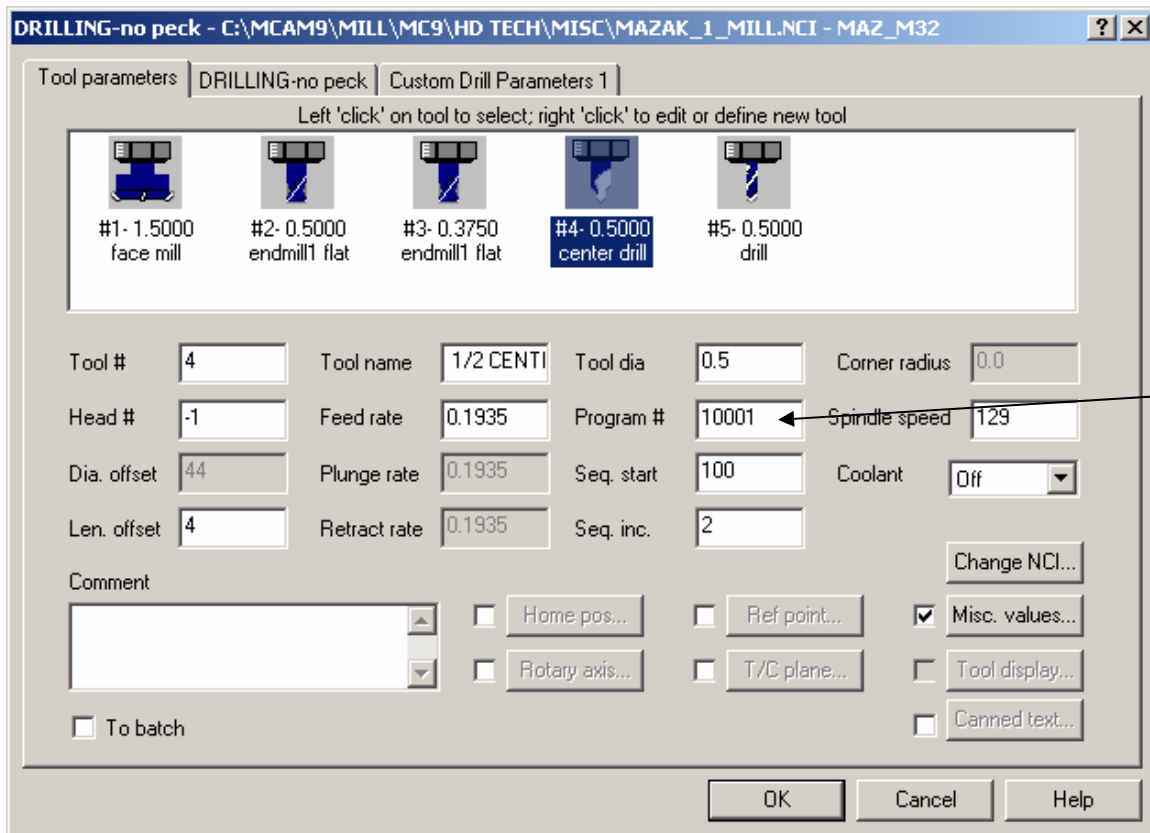
Select 0.5"center drill as shown

In order for all the tools to be captured and appear at the top of the drill line set the Program # to the value shown. (Values of 10001 –10099 may be used to group common tools together for this type of operation)
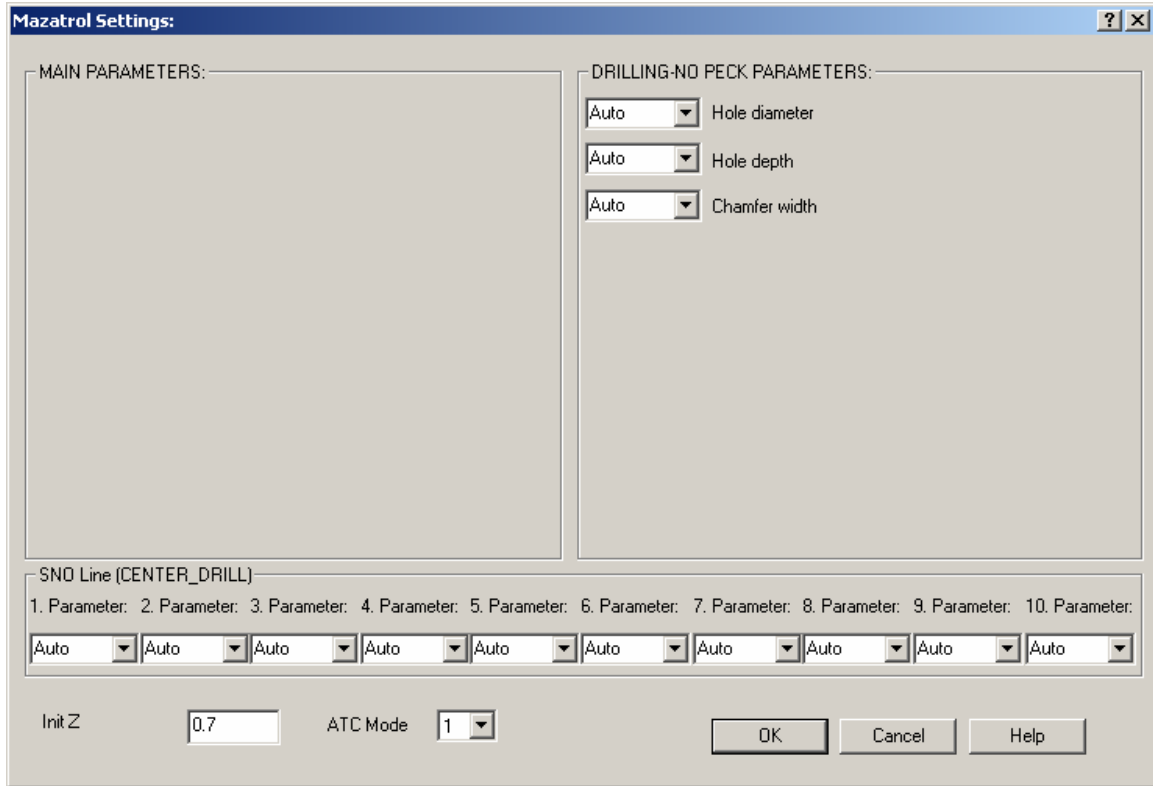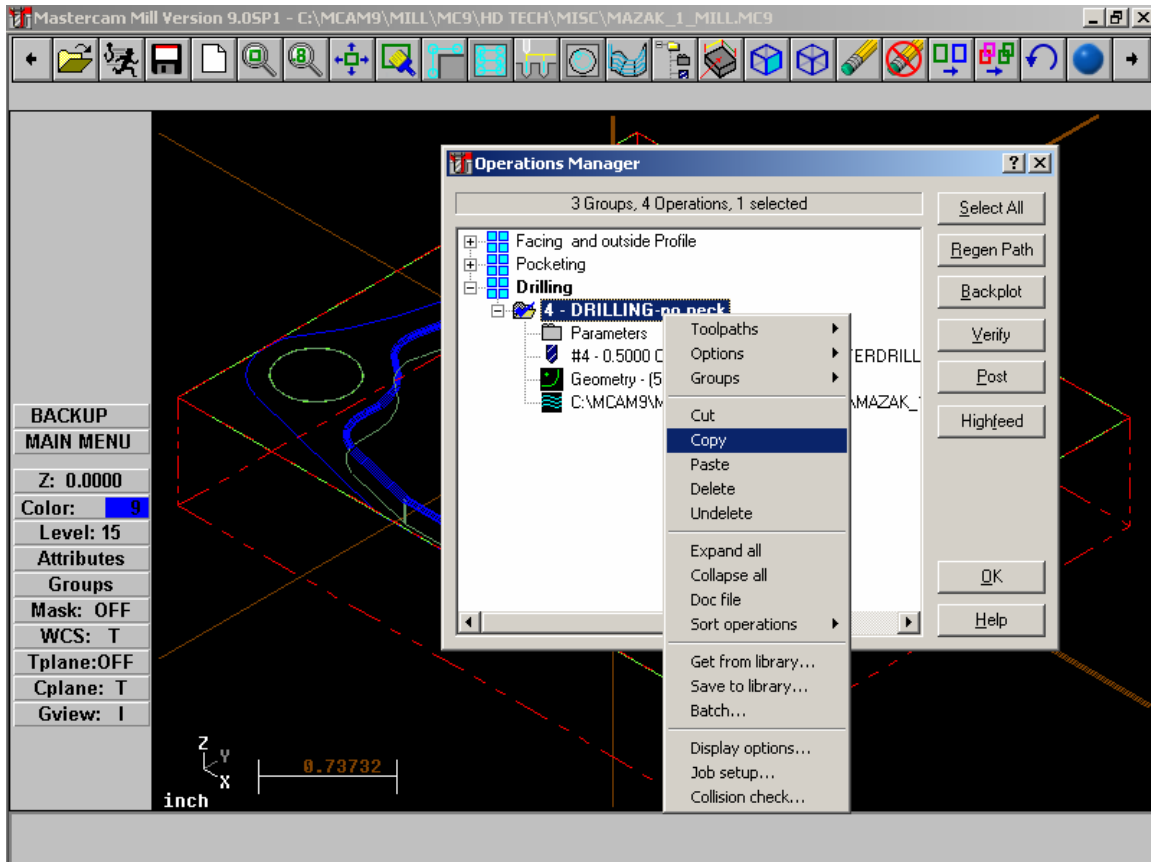
7. **Misc. Values** Leave settings on Auto as shown

8. Go to **Drilling** and set **DRILLING – DRI** as shown

Note: All the Drill Cycles available to Mazatrol are accessible via Drill Cycle Menu as shown above.
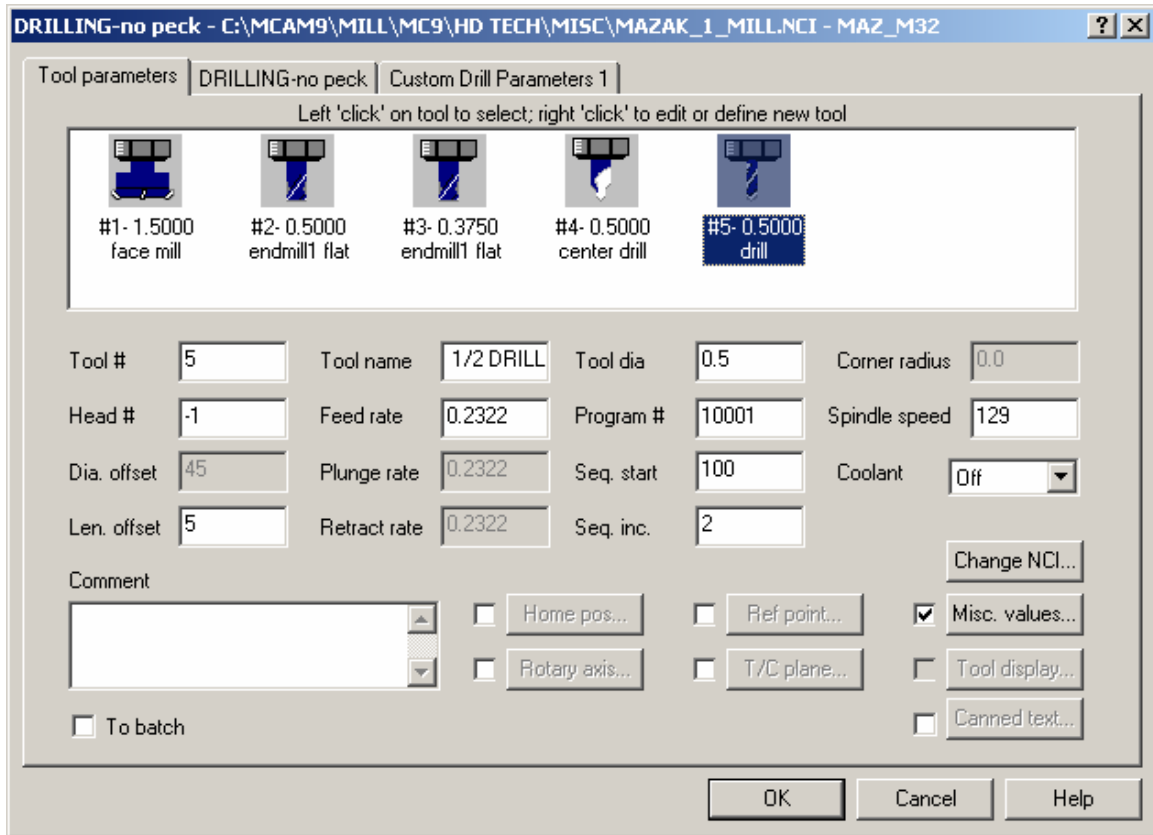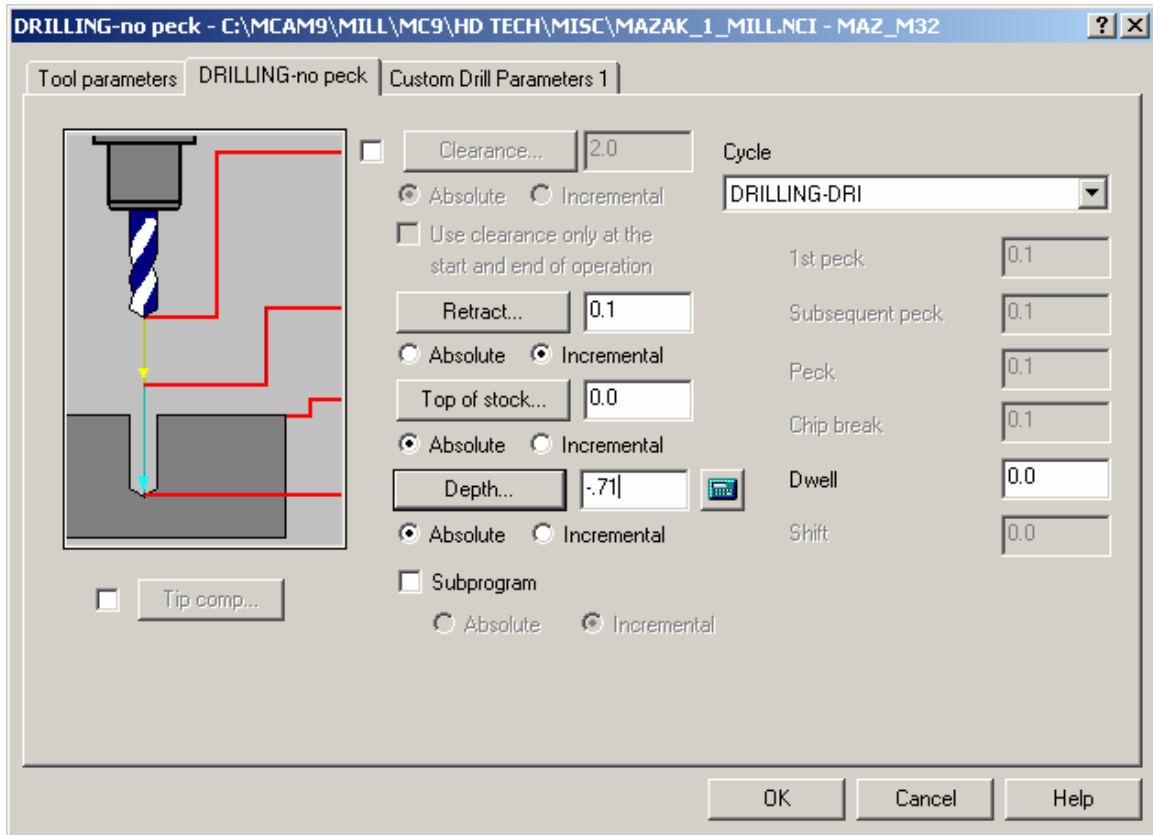
9.**OK**

Now we will copy the previous operation. Therefore the only changes we need to make will be the tool we want to use and the drilling depth. All the other values will stay the same.
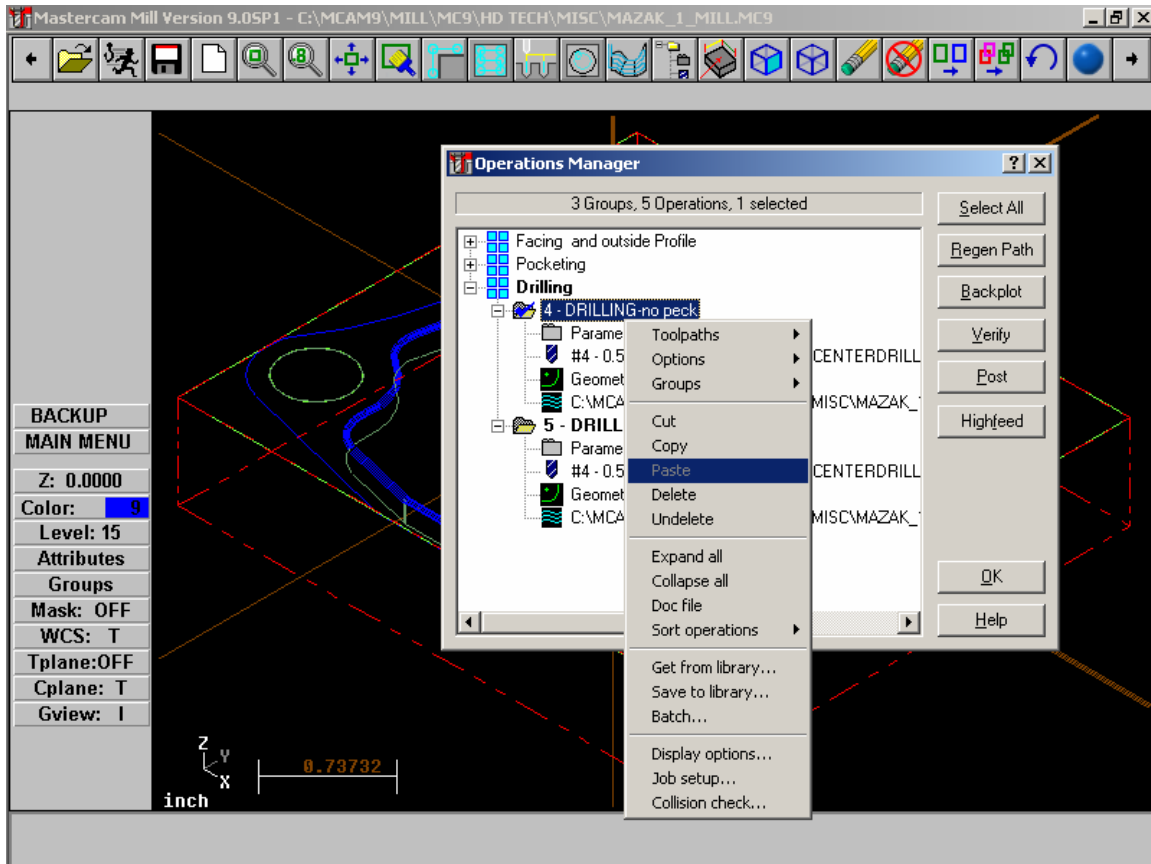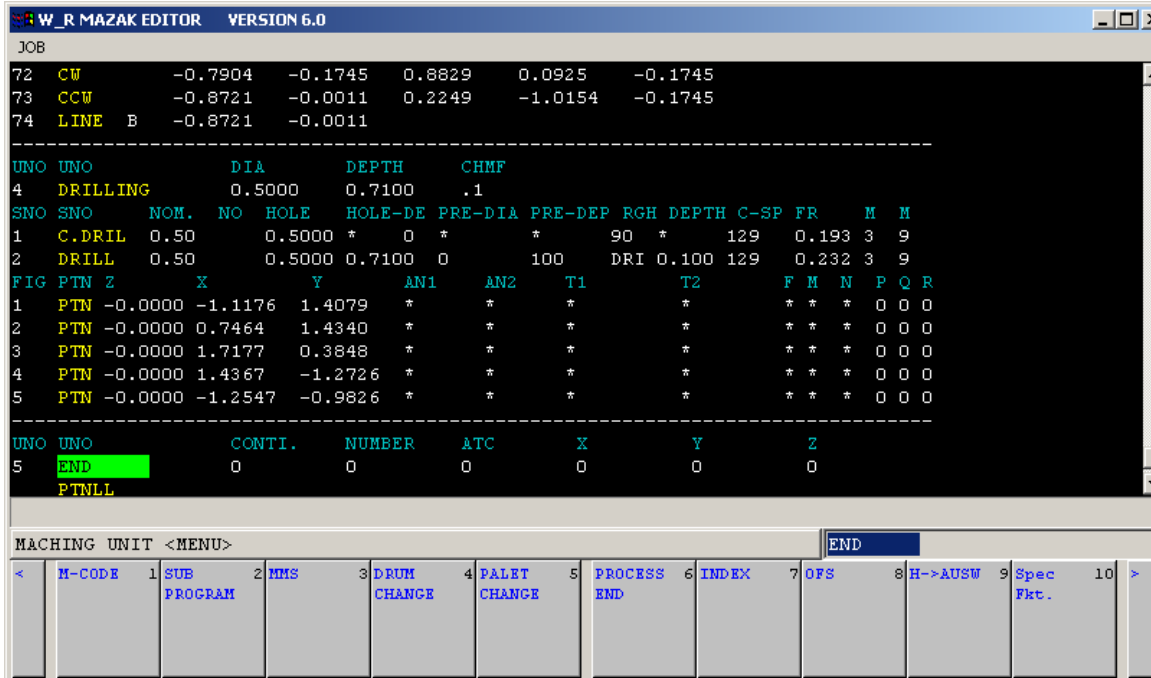
11. Paste new operation

12. Select 0.5" tool as shown

13. Set **Depth** as shown.

14. **OK**.

After posting the output will appear as shown below.

15. **Save File**

16. **Post File** and view output.

3. Modifying a previously programmed part

Exercise 1 - Opening Part File

In this exercise the object is to modify an existing part previously programmed perhaps for another type of control such as a Fanuc – or perhaps a situation where the programmer wishes to get all the toolpaths built before adapting the output for Mazatrol.

1. Choose **Main Menu, File, Get**
2. Navigate to the folder with the tutorial parts.
3. Select **Mazak_2_Mill.mc9**; then choose **Open.**
4. Go to **Operations Manager** you should see dialog as below
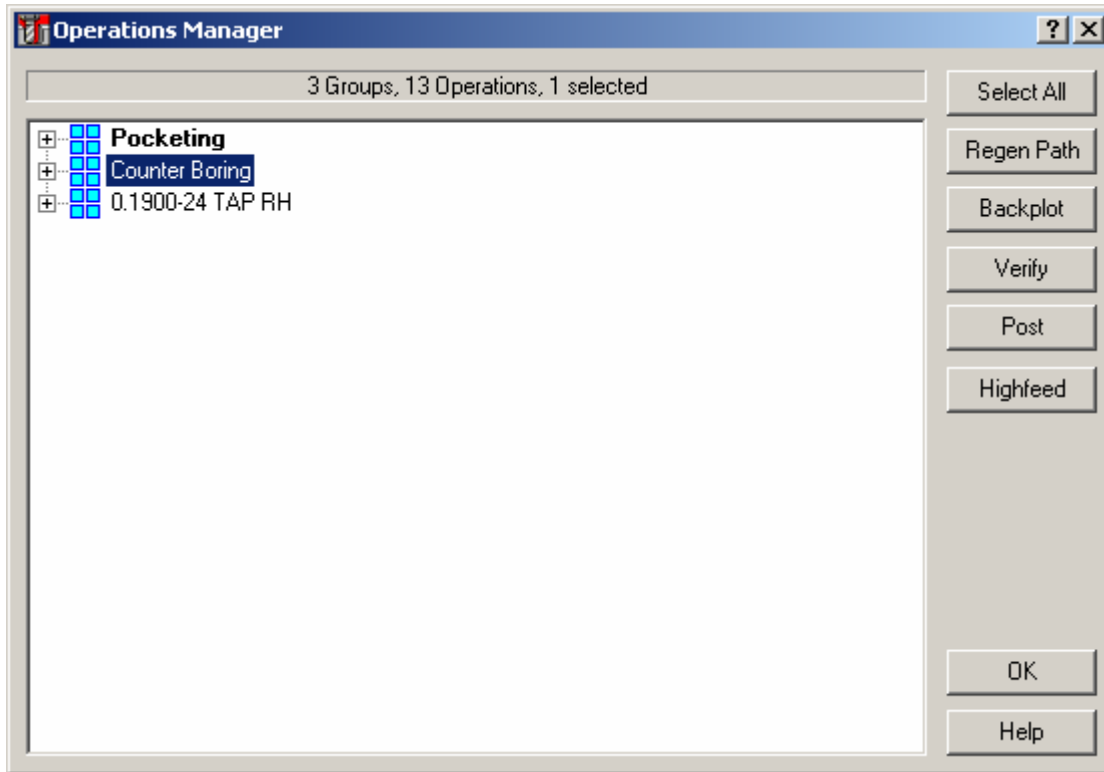
In this file we have created a part in the using pocketing that would be very difficult to program in Mazatrol because the pocket has multiple islands. We have also used a tool that is too big to complete the machining of the pocket and then taken advantage of Mastercam's Pocket Remachining routine. As the part already has defined stock go ahead and run verify out of the Operations Manager to see the current toolpaths.

Exercise 2 - Line-Center output for Pockets

We have two options in this case. We could program all the pockets using line-center and modifying settings as shown below – this would take advantage of mastercam's many different type of pocketing strategies available when setting the Roughing component or we could program separate areas of the part using either Mazatrol's Pocket or Pocket MT. In this section we will program output as Line center.
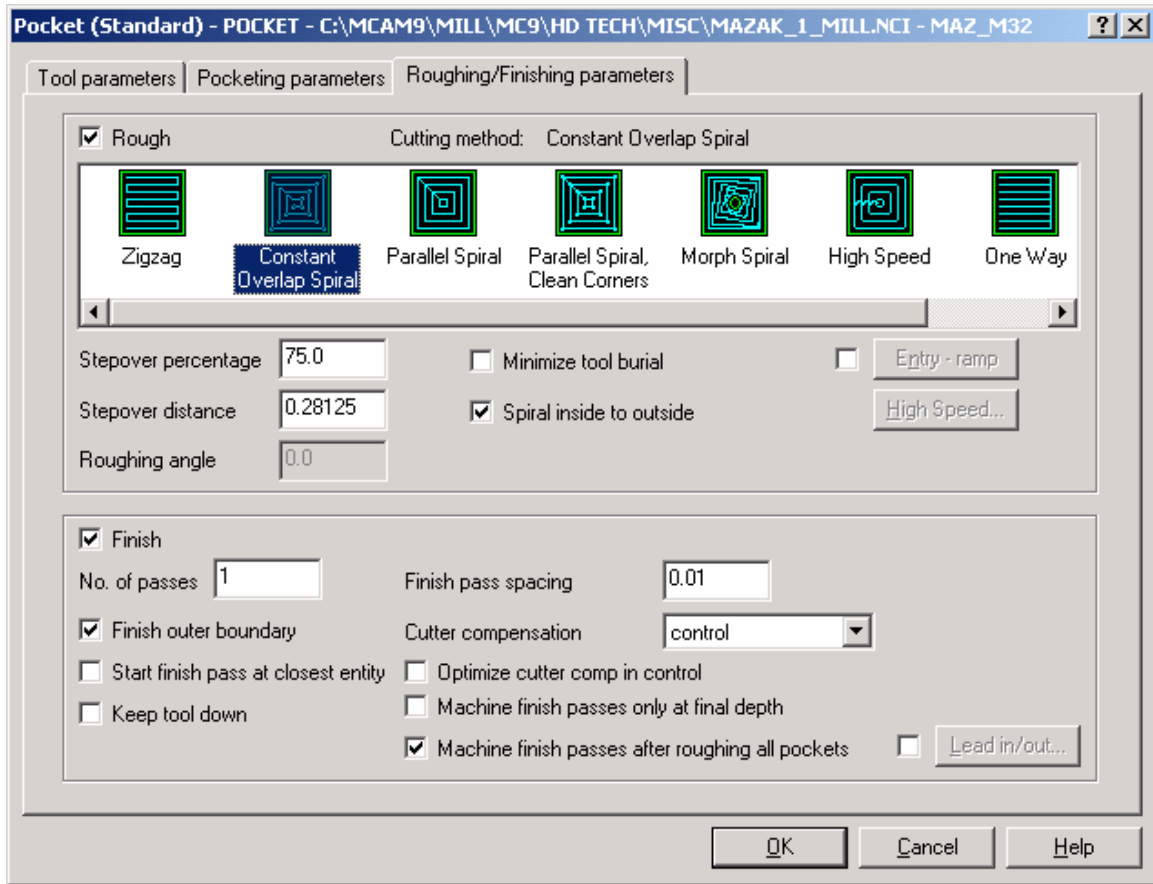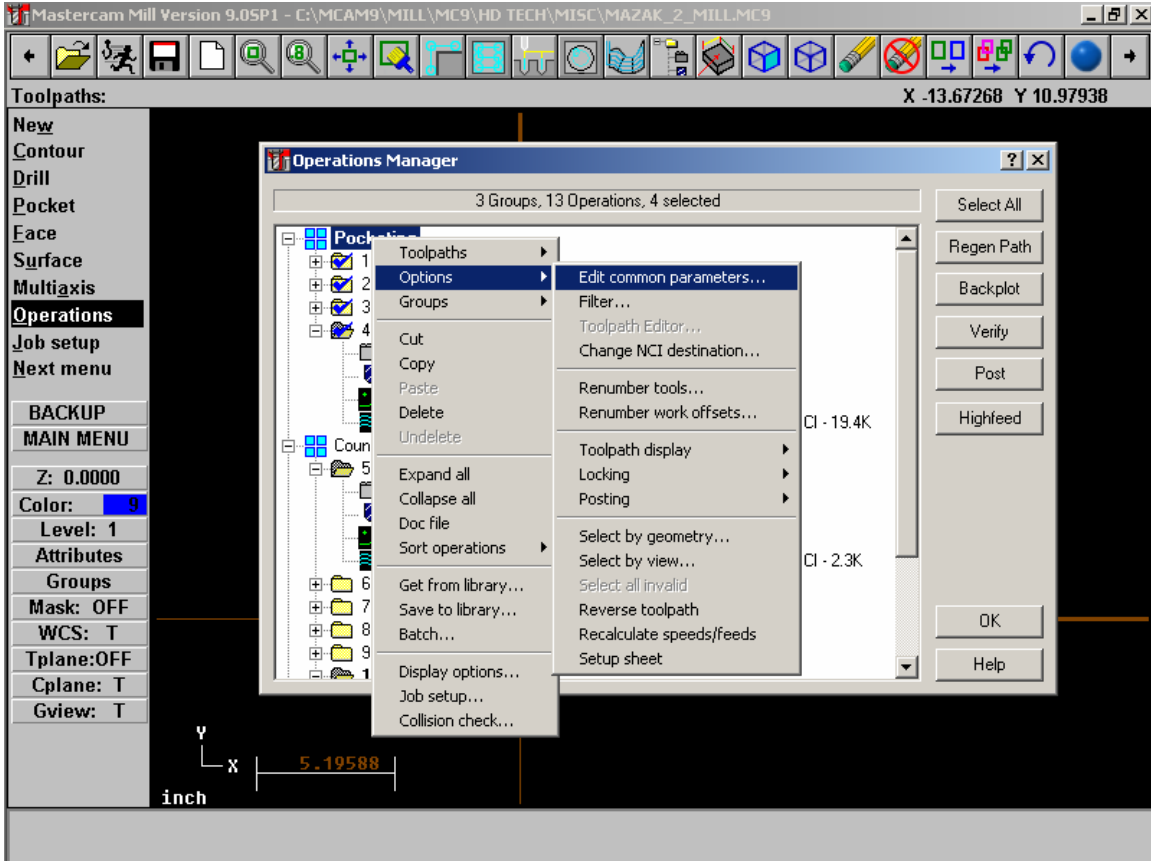
1. Fill in settings as shown below:
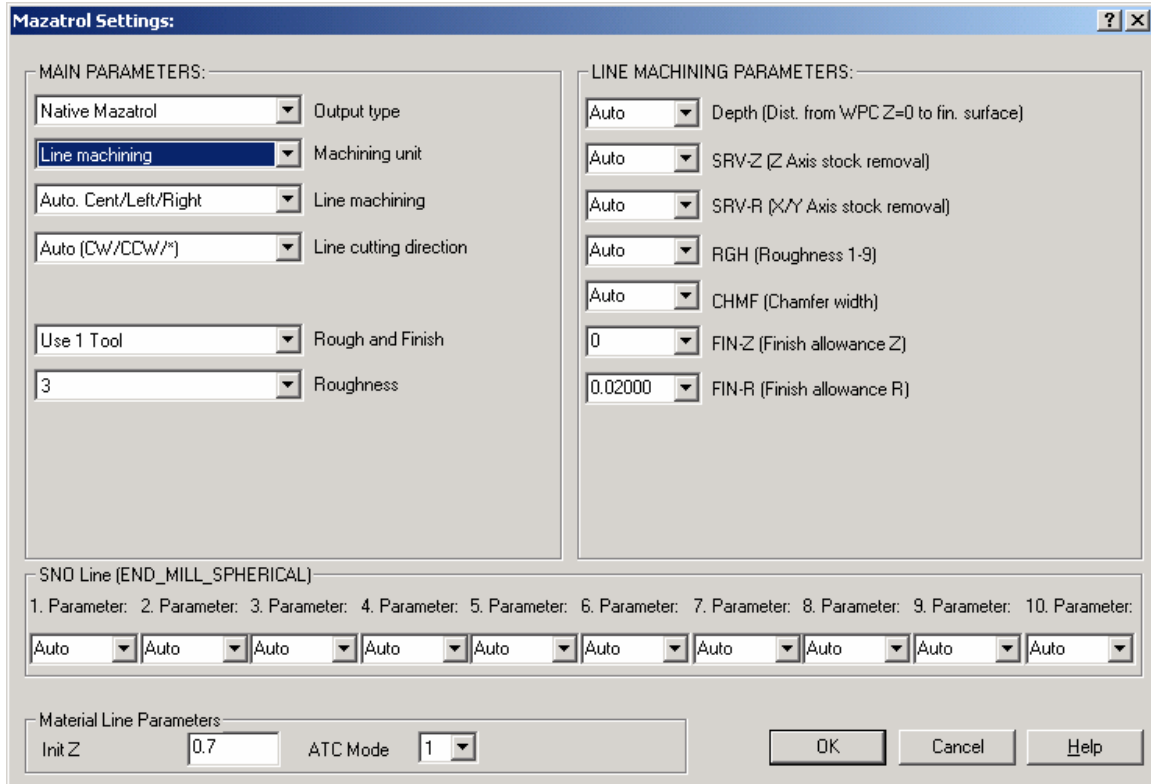
2. Set all other pocket toolpaths programmed likewise using **Edit Common Toolpath** parameters and go to Misc. Values button
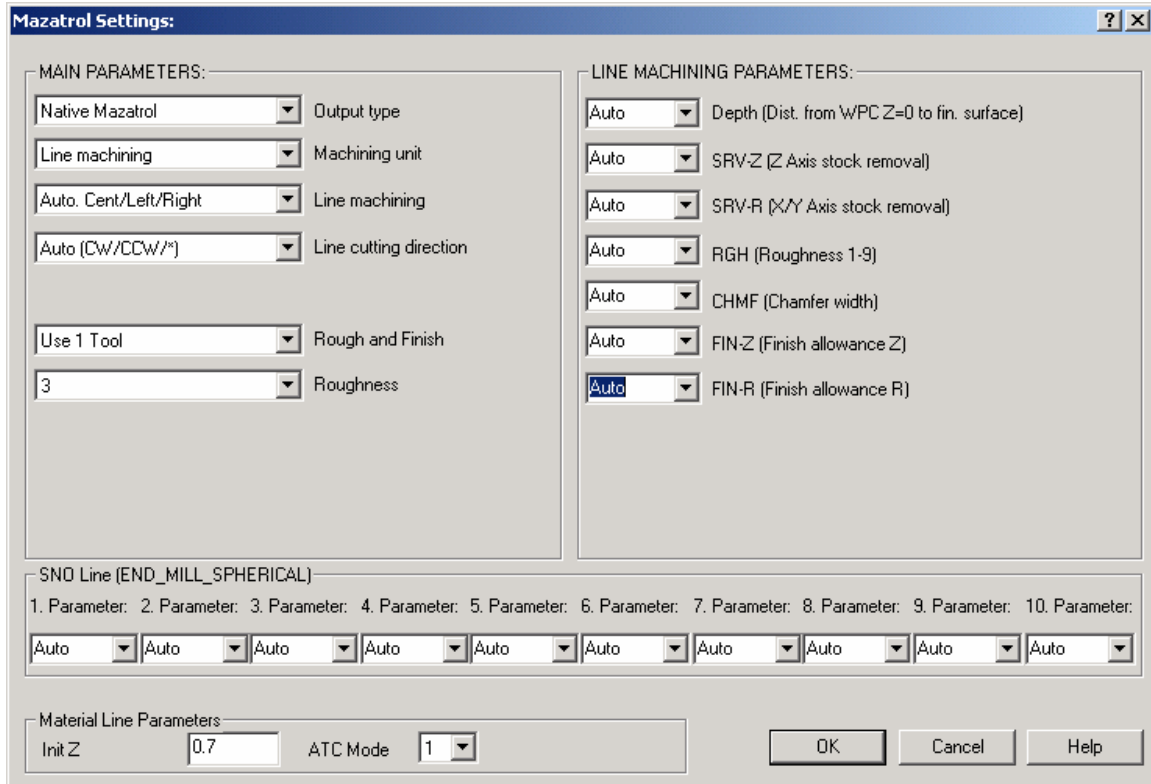
A section of the Mazatrol output will be as below:

```
PNR MAT     INITIAL-Z ATC MODE   MULTI MODE  MULTI FLG  PITCH-X     PITCH-Y
0  ALUMINUM 1.0     0         OFF
-------------------------------------------------------------------------------
UNO UNO     DEPTH    SRV-Z    SRV-R    RGH  CHMF    FIN-Z    FIN-R
1  LINE-CTR 0.3000   0.3000   0.25    3   *     0      0
SNO SNO    NOM.  NO.  APRCH-X   APRCH-Y  TYPE ZFD DEP-Z  WID-R C-SP FR    M  M
1  E-MILL 0.50     ?      ?      *  G01 0.3000 *   1069  6.417 3 9
FIG PTN      X        Y        R/0     I      J       P      CNR
1   LINE     8.7198   9.2517
2   CCW      8.7600   9.4741   0.6350   8.1250   9.4741
3   LINE     8.7600   10.0514
```

There will be times when you may wish to modify the settings that are automatically calculated for those parameters on both the UNO (unit Line) and SNO (Tool Cutting Definition Line) this will be done as shown below. Again you will need to access the Misc. Values Button.

For example above we will change the output for SRV-Z and SRV-R to values shown below:
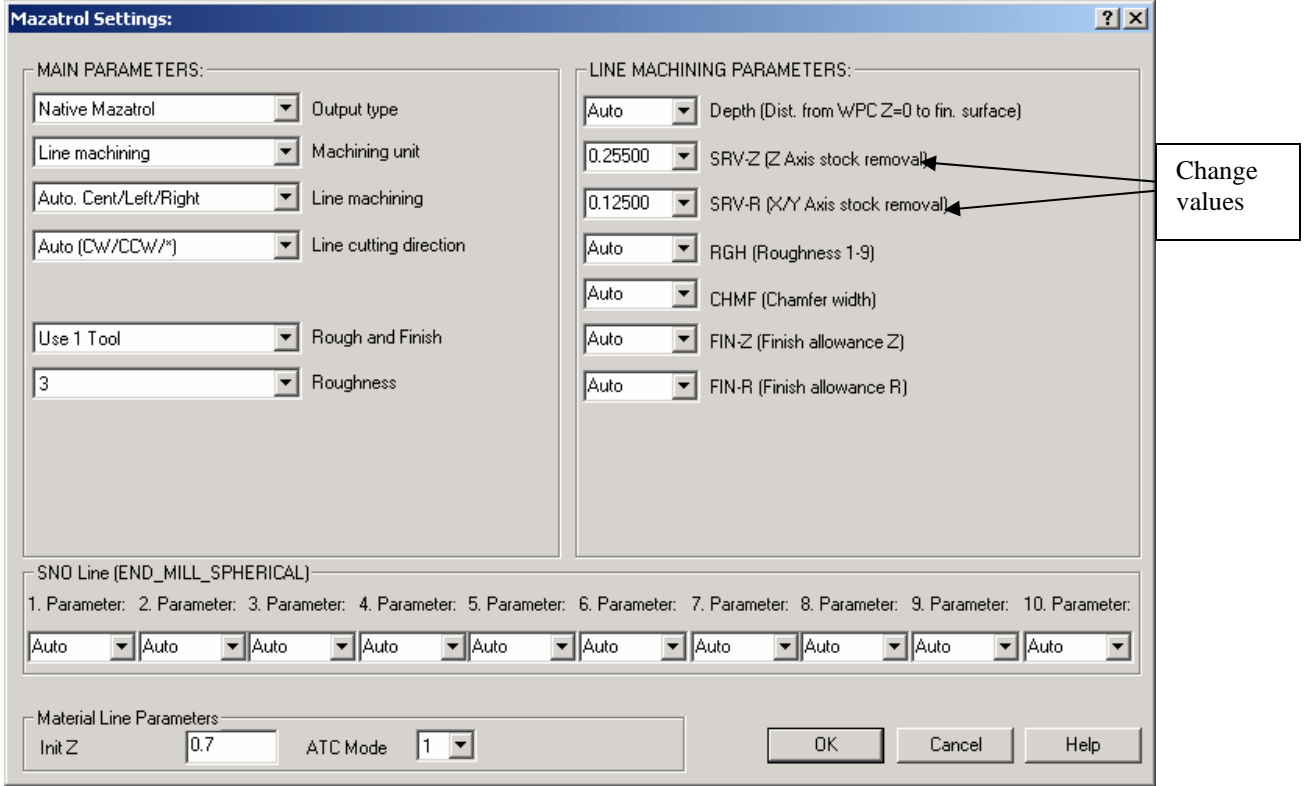
A section of the Mazatrol output will be as below:
As you see the settings are output and shown in bold text below:

```
PNR MAT     INITIAL-Z ATC MODE    MULTI MODE  MULTI FLG   PITCH-X     PITCH-Y
0   ALUMINUM 1.0     0        OFF
--------------------------------------------------------------------------------
UNO UNO     DEPTH    SRV-Z    SRV-R    RGH CHMF     FIN-Z    FIN-R
1   LINE-CTR  0.3000   .255    .125    3   *     0      0
SNO SNO    NOM. NO.  APRCH-X   APRCH-Y  TYPE ZFD DEP-Z  WID-R C-SP FR    M  M
1   E-MILL  0.50    ?      ?      *  G01 0.2550 *   1069 6.417 3 9
FIG PTN      X        Y        R/0     I      J       P      CNR
1   LINE     8.7198   9.2517
2   CCW      8.7600   9.4741   0.6350  8.1250   9.4741
3   LINE     8.7600   10.0514
4   CCW      8.2298   10.6777  0.6350  8.1250   10.0514
5   LINE     9.0401   11.1456
6   CCW      9.7400   10.7415  0.4601  9.5000   11.1340
7   LINE     9.7400   10.3717
8   CCW      9.2801   9.5752   0.4601  9.5000   9.9793
9   LINE     8.7198   9.2517
10  CCW      8.7577   9.4200   0.6350  8.1250   9.4741
11  LINE     8.9961   10.4767
12  LINE     8.8260   10.7017
13  LINE     8.9987   10.5517
14  LINE     8.7198   9.2517
--------------------------------------------------------------------------------
UNO UNO     DEPTH    SRV-Z    SRV-R    RGH CHMF     FIN-Z    FIN-R
2   LINE-CTR  0.6000   .255    .125    3   *     0      0       SNO SNO   NOM. NO.  APRCH-X  APRCH-Y  TYPE ZFD
DEP-Z  WID-R C-SP FR    M  M
1   E-MILL  0.50    ?      ?      *  G01 0.2550 *   1069 6.417 3 9
FIG PTN      X        Y        R/0     I      J       P      CN
```

Exercise 2 - Mazatrol Style Pocket output for Pockets

In order to use the Mazatrol Pocket Styles we have to disable Mastercam's Pocket Roughing routines. The Mazatrol's Pocketing styles will be based upon the **Parameters that are set within the controller itself**. We will set the Mastercam Parameter Pages as Follows for all the pocket toolpaths:
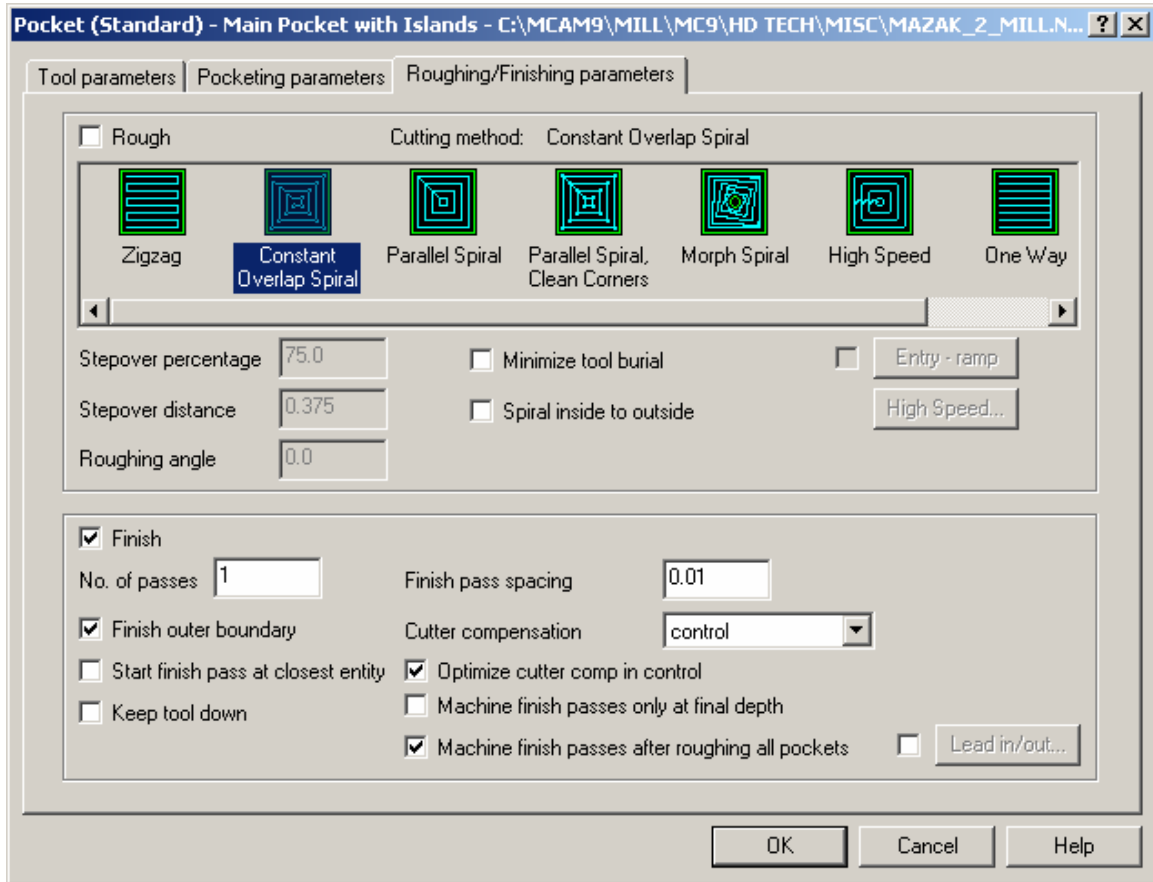

Set **Misc. Values** as below:
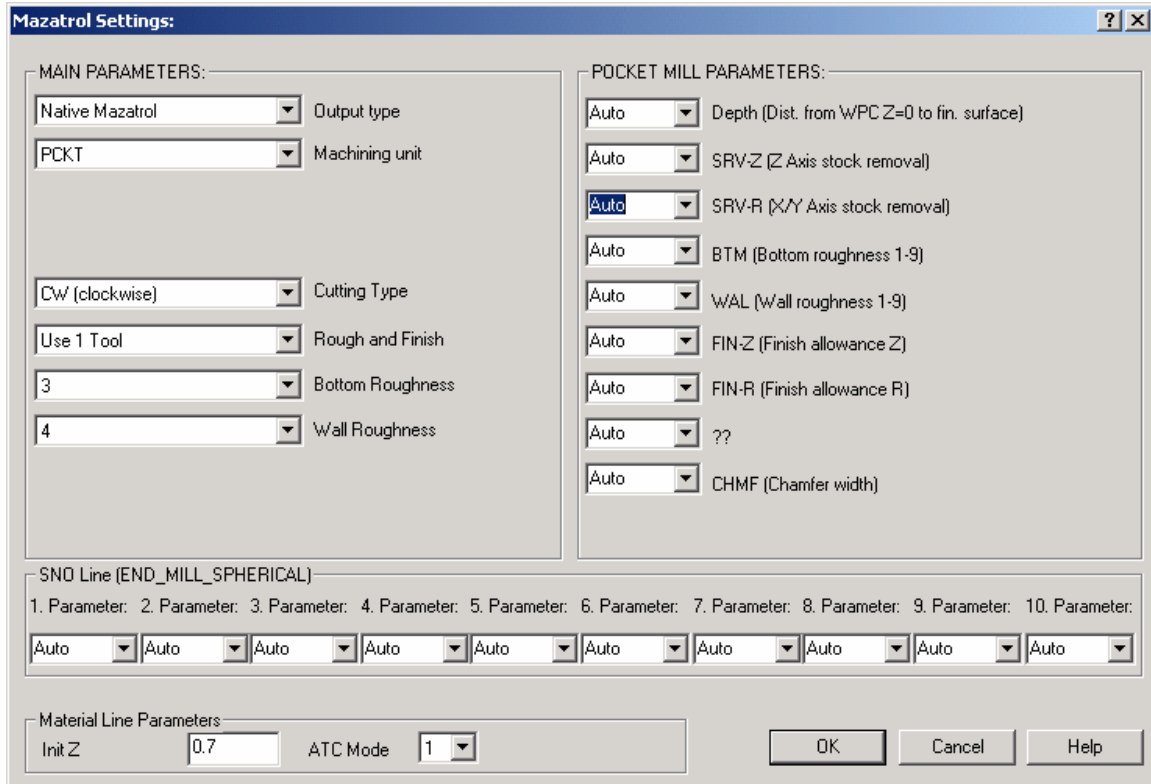
Exercise 3 - Modifying Drill Cycles in Counter Boring Group

In this section we will modify the Toolpath Group labeled as Counter Boring. If we were to post this section each one of the tools would be in a separate UNO section a with a drill cycle defined by what is shown currently in the Operations Manager. Therefore we need to Group these operations together and also we need to make sure that the drill cycle type is consistent. In this case we will set it to Mazatrol's RGH CBOR.
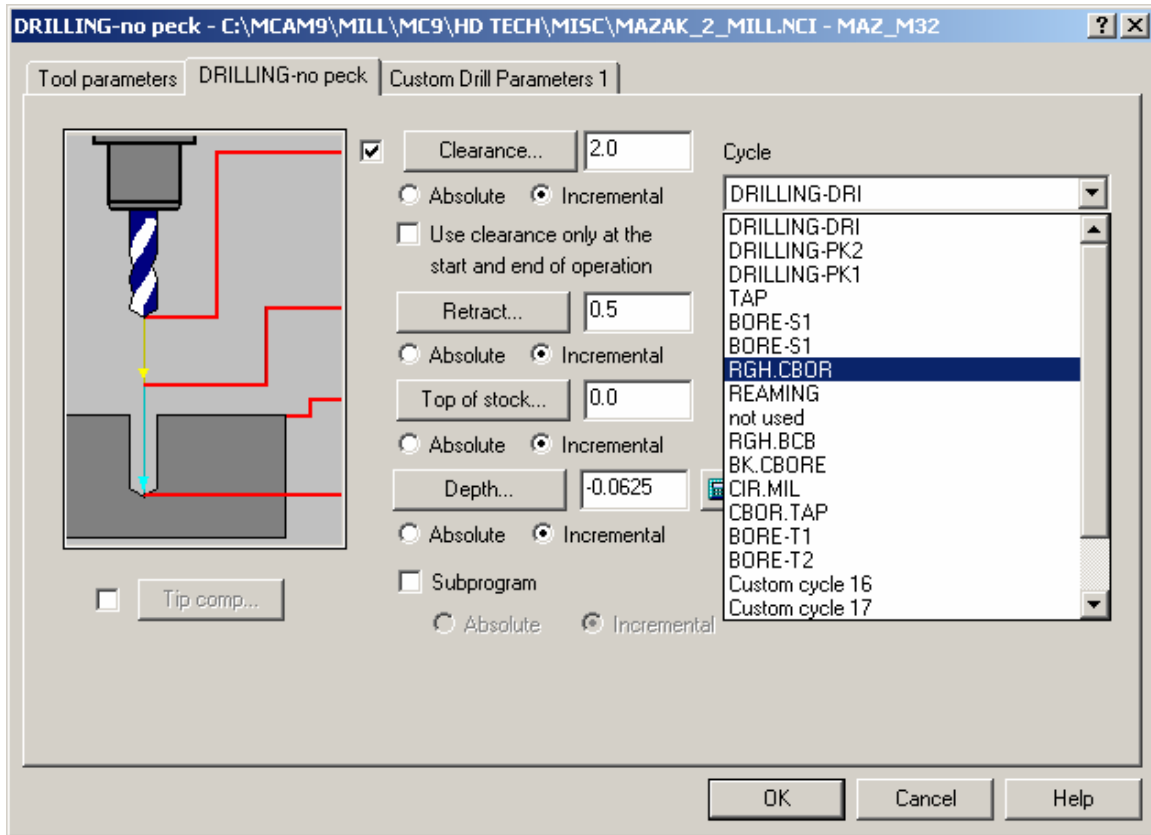
We need to do the following:

1. Using **EDIT COMMON PARAMETERS** highlight the Counter Boring Group in the **Operations Manager** set the Program # as follows:
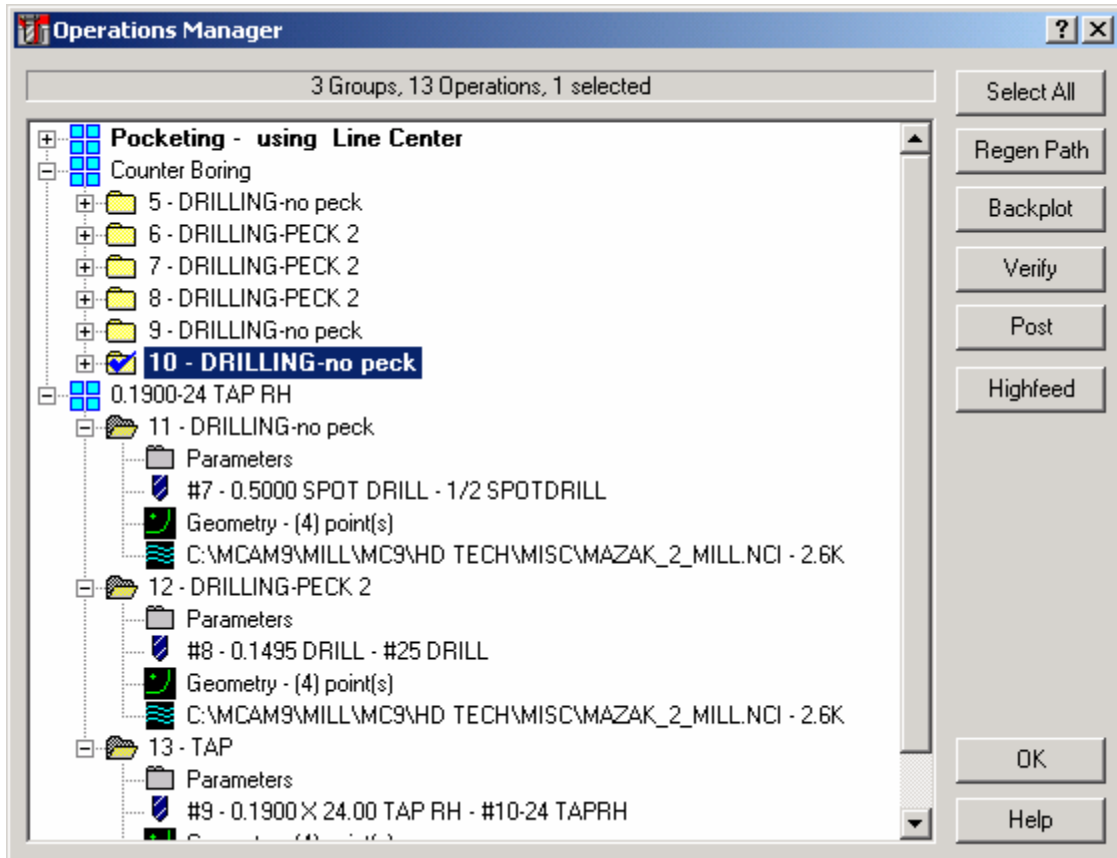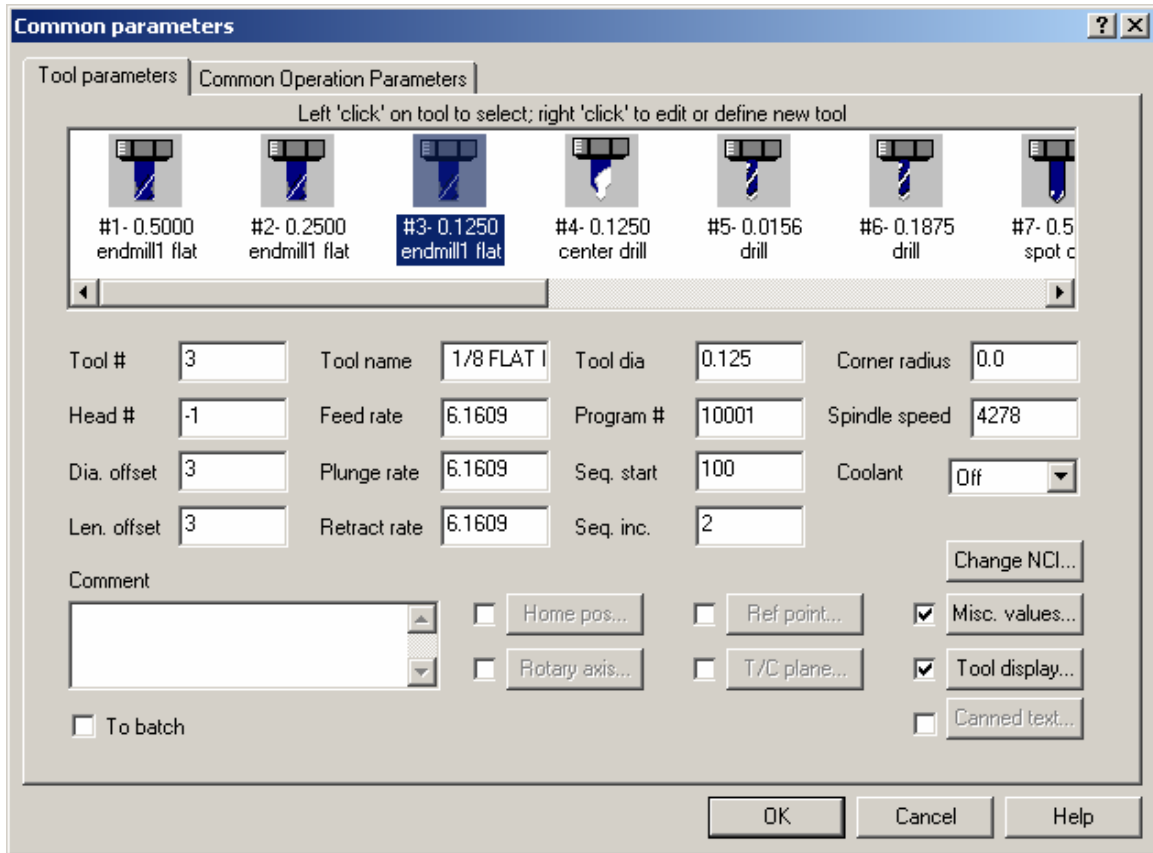
2. We now need to make sure that for all the operations in this group the drill cycles are set as follows:

*Note: We have used 6 tools in the previous section - the Mazatrol will allow this many tools for this type of cycle - but the number of tools used by the mazatrol when manually programming at the control is based upon internal calculations which reference **Built-In** Parameters.*

Operations manager should then look as below
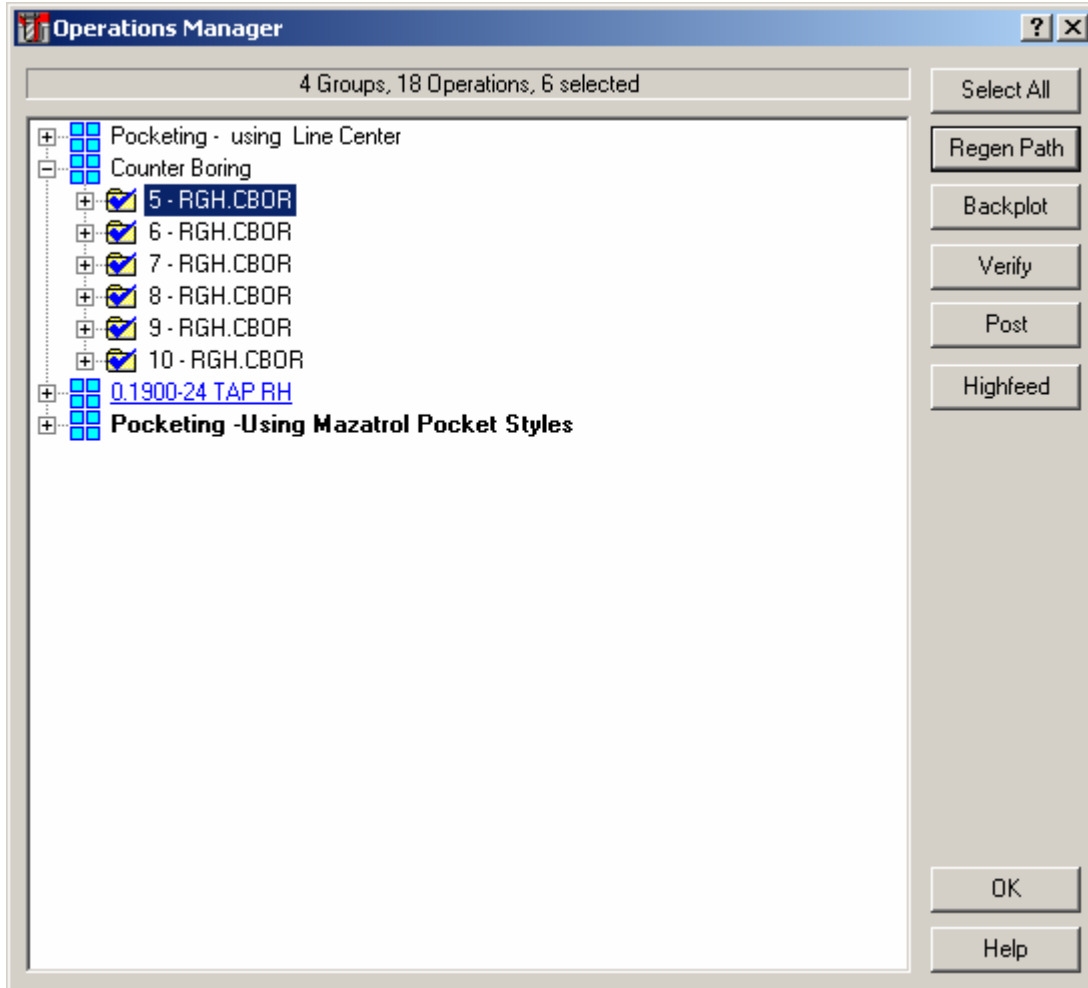
Exercise 4 - Modifying Drill Cycles in Tapping Group

As in the previous exercise we will modify the three operations grouped as **.1900-24 TAP RH** so that the output will be more efficient and readable as Mazatrol format. In addition we will add an operation to create a chamfer before the final tapping operation.

We need to do the following:

1. Using **EDIT COMMON PARAMETERS** highlight the Tapping Group in the **Operations Manager** set the Program # as follows: