

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

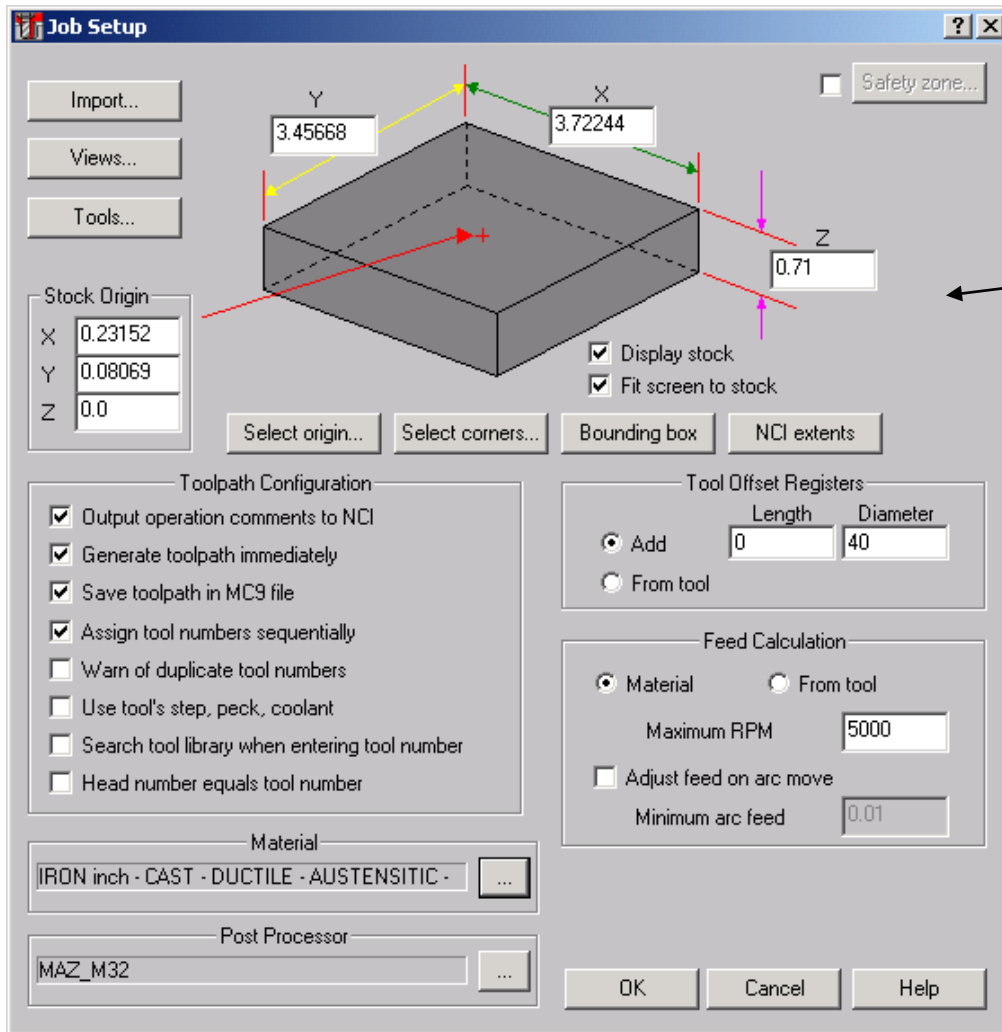
Note: This text was compiled using Version 8.0.8 of the Mazatrol Product – some dialogs presented may have changed or you may be using either an earlier or later version of the software.

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.



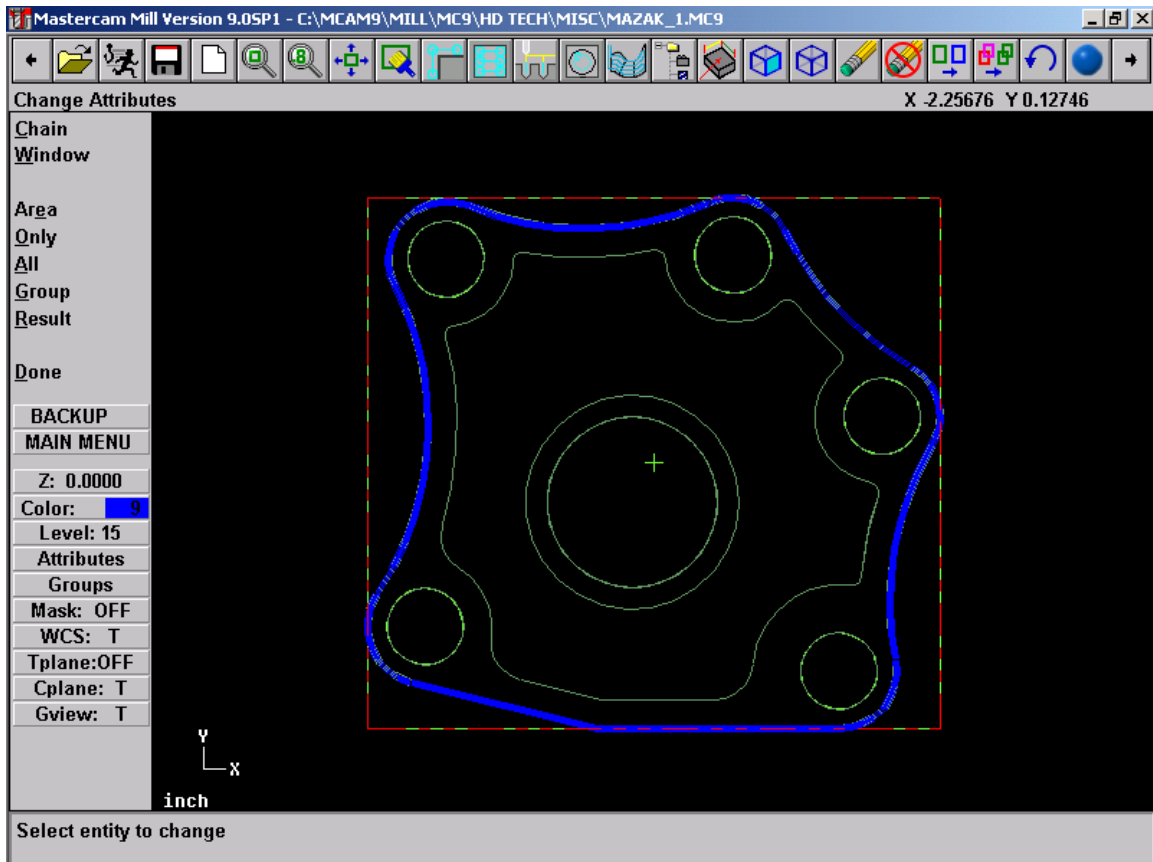
Note: Job Setup settings will affect the first line of the mazatrol PNR and MAT i.e. the material selected will be output and the Z depth of the material will be output as INITIAL-Z see below:

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

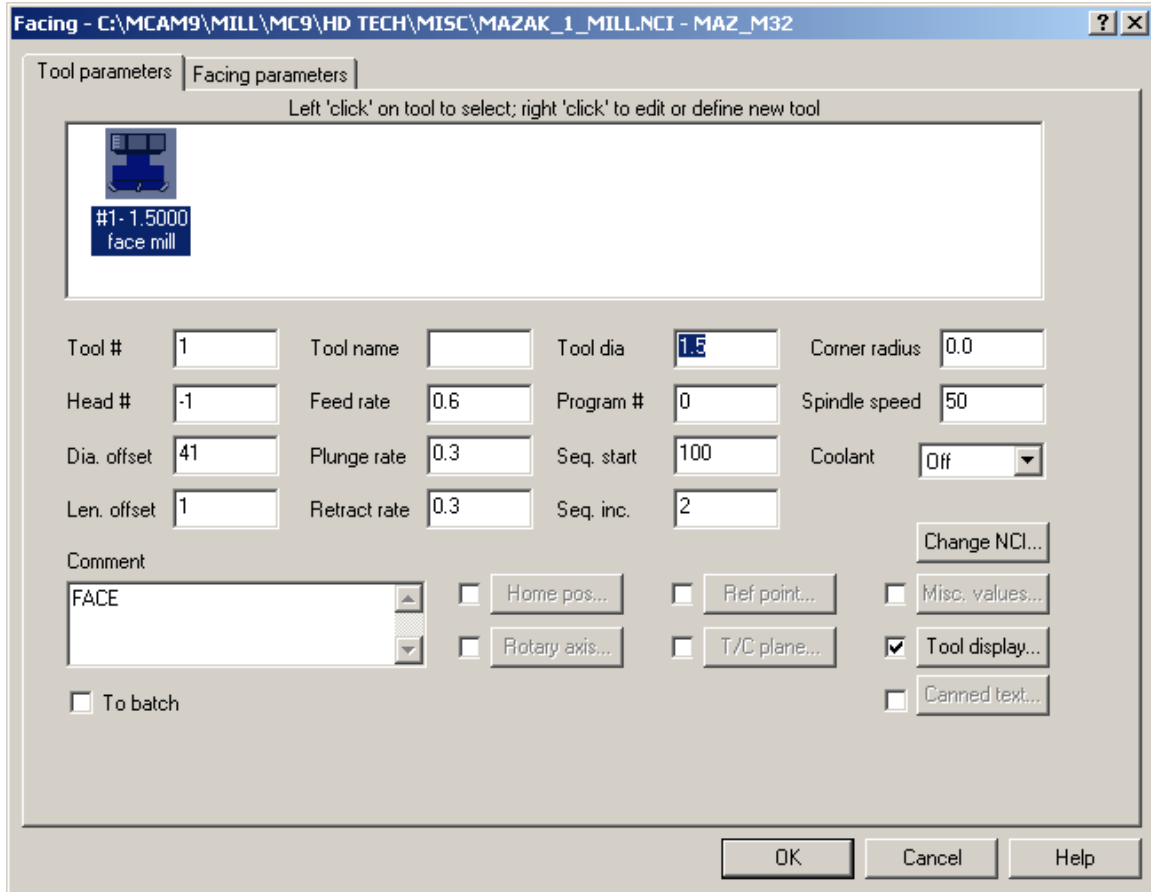
The other settings will have to be manually entered by the user if desired either using the editor (if available) or at the control. Also the values for federate and spindle speed that are set in the mastercam parameter pages will also output to the Mazatrol code.

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

Mastercam to Mazatrol Post-Processor Tutorial 11/1/2018

Mazatrol Settings:

MAIN PARAMETERS:

Native Mazatrol Output type
Face Machining unit

Cutting Type
Rough and Finish
Bottom Roughness

FACE MILL PARAMETERS:

Auto Depth (Dist. from WPC Z=0 to fin. surface)
Auto SRV-Z (Z Axis stock removal)
Auto BTM (Bottom roughness 1-9)
Auto FIN-Z (Finish allowance Z)
Auto FIN-R (Finish allowance R)

SNO Line (SLOT_MILL)

1. Parameter: 2. Parameter: 3. Parameter: 4. Parameter: 5. Parameter: 6. Parameter: 7. Parameter: 8. Parameter: 9. Parameter: 10. Parameter:

Auto Auto Auto Auto Auto Auto Auto Auto Auto Auto

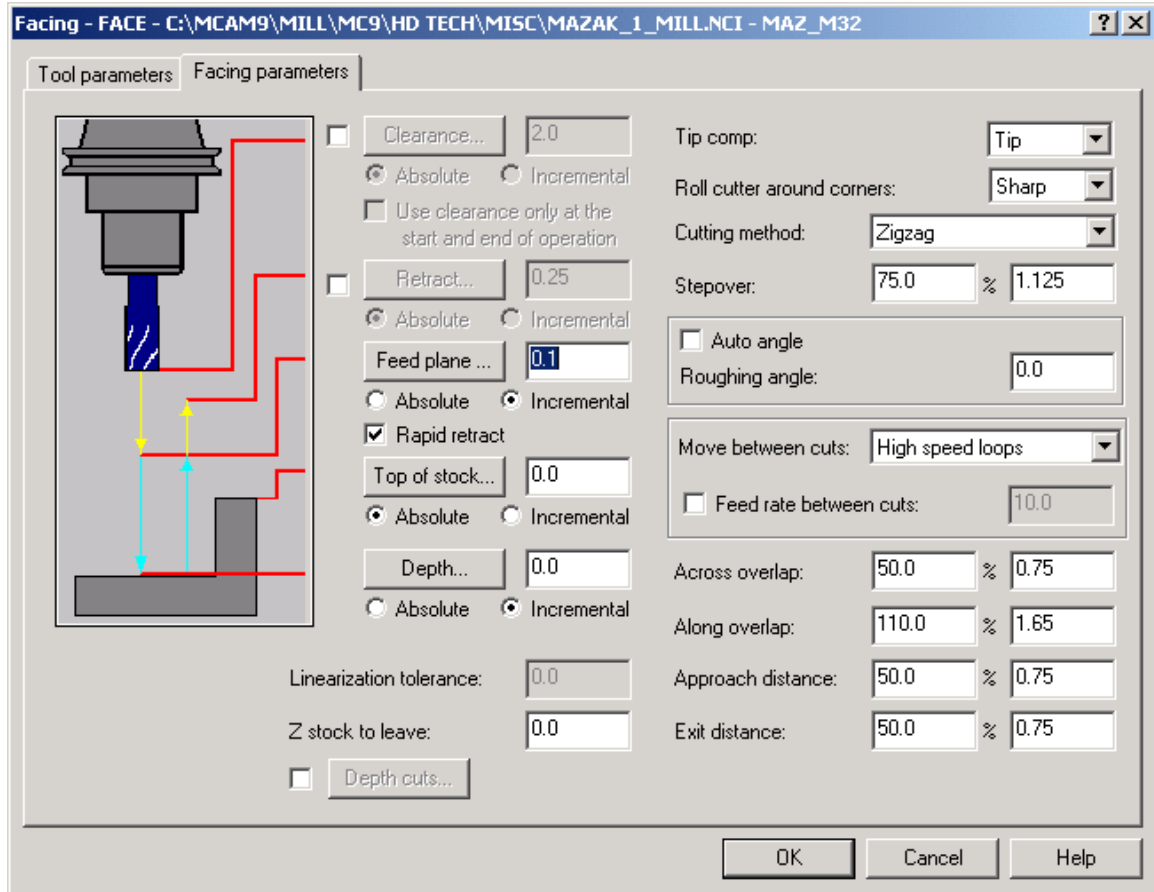
Material Line Parameters

Init Z 0.7 ATC Mode 1

OK Cancel Help

5. Click **OK** when done.

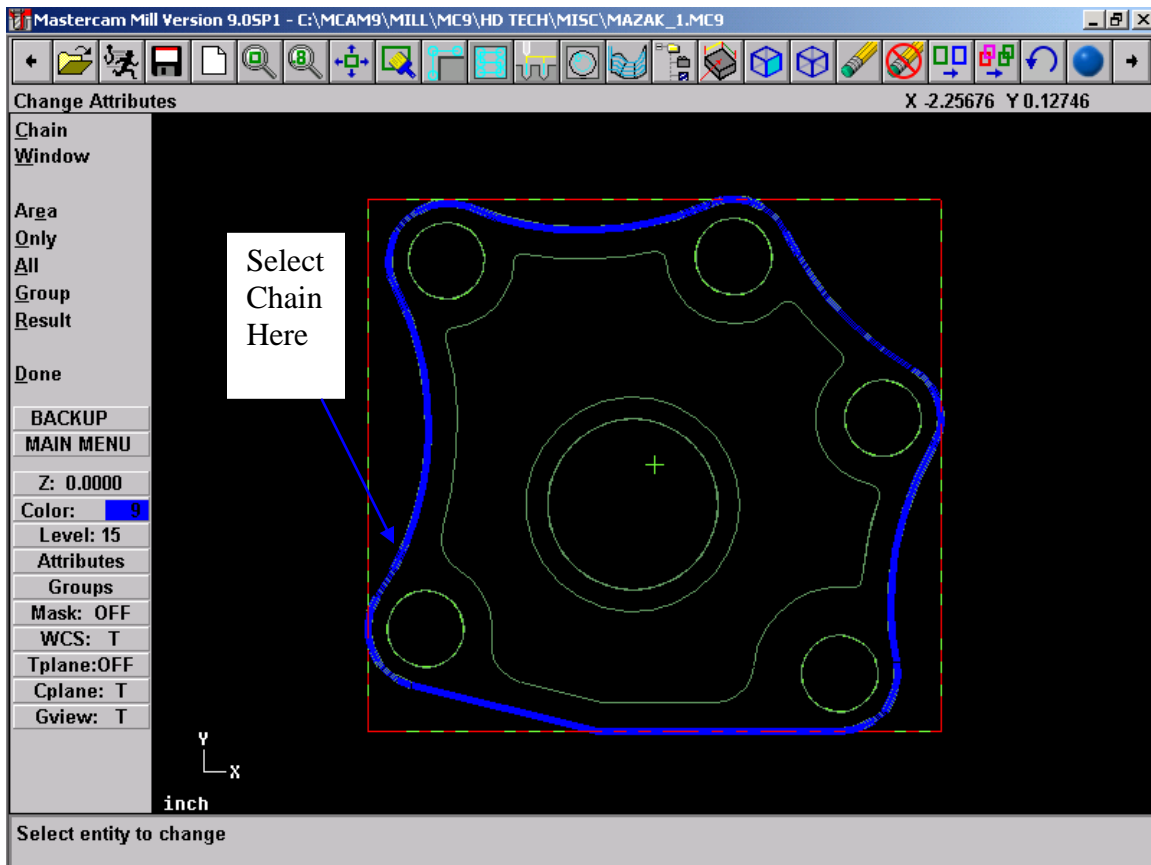
6. Click on **Facing Parameters Tab** and set Values as shown;



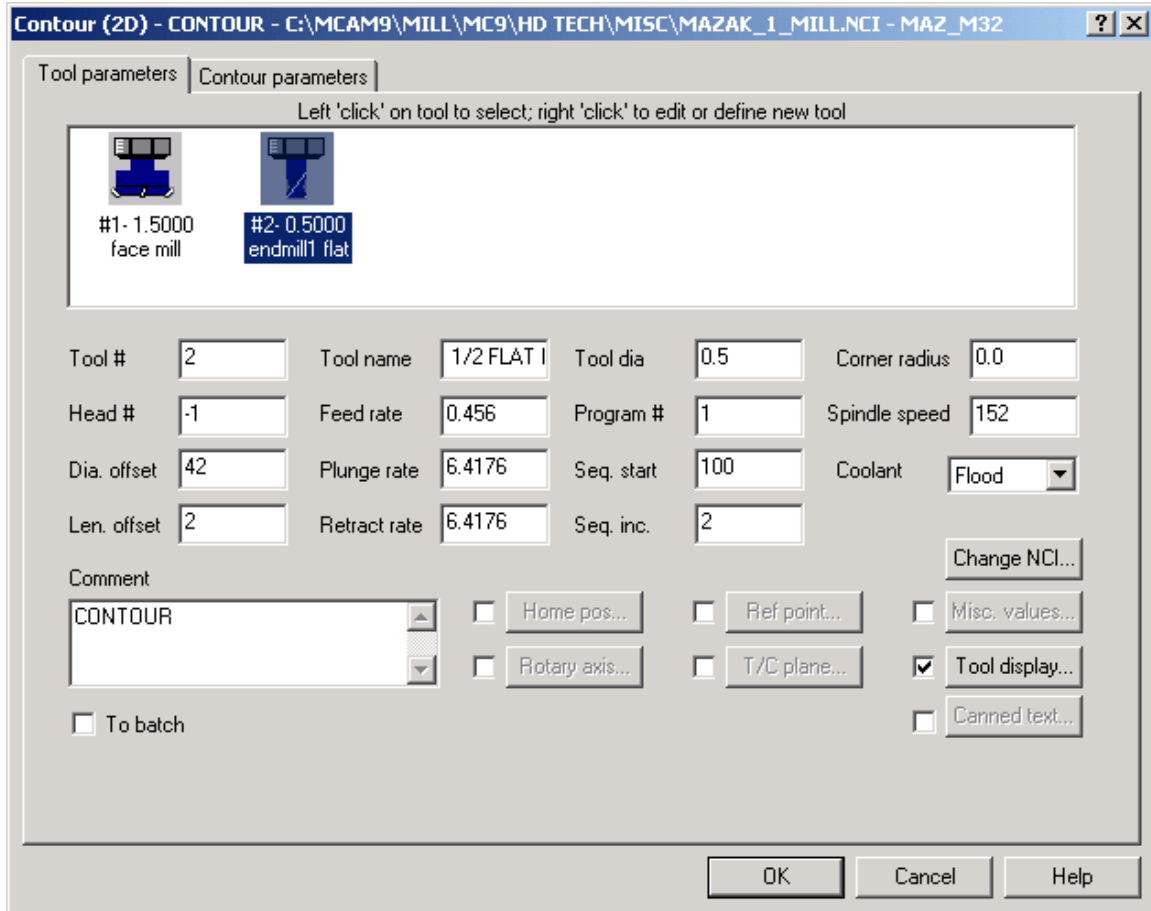
7. Click on **OK** when completed.

Exercise 3 - Creating Contour Toolpath for outside profile

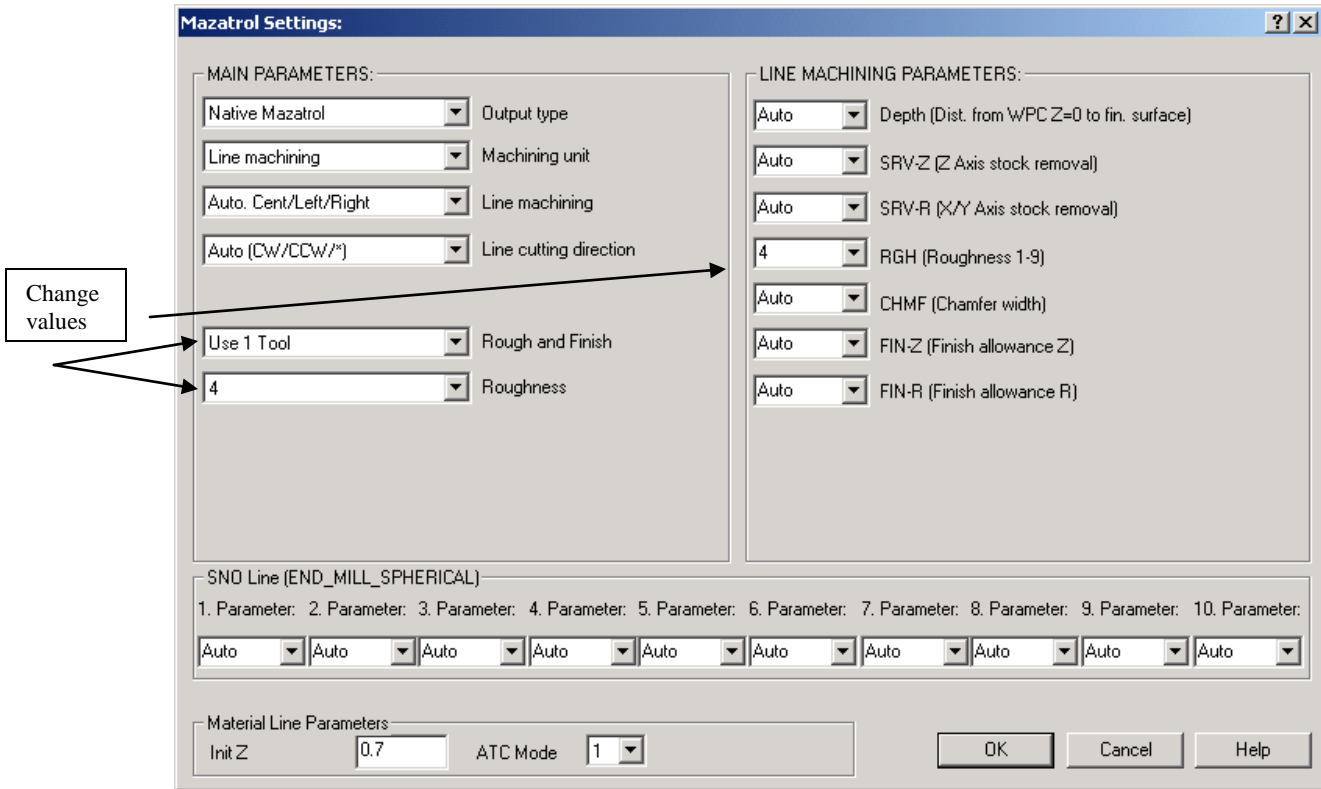
1. Choose **Main Menu, Toolpaths, Contour**
2. Select outside profile as shown using chain



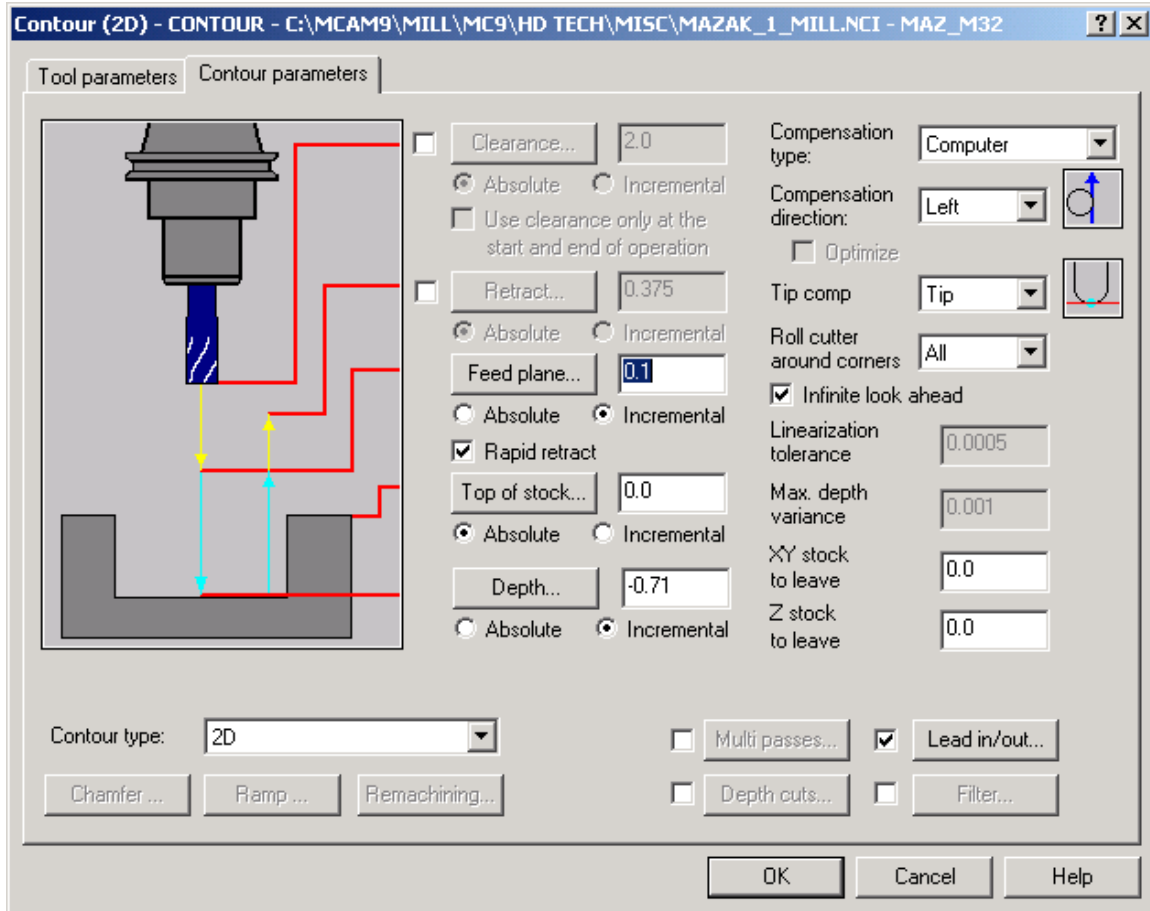
3. Select **Done**
4. Select 0.5" Dia Flat end Mill as shown.



5. Click on **Misc. Values** button and modify settings as shown below

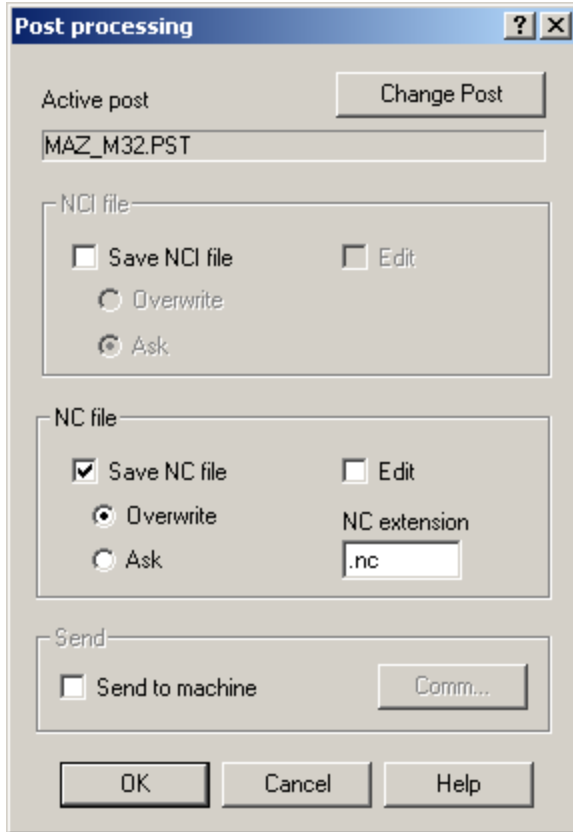


Note: As you may notice – the Misc. Values dialog box allows every setting in the mazatrol SNO line and UNIT (UNO) line to be set by the user and override the automatically set values output by the post-processor. This will be shown in more detail in the next chapter.



*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 course 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)

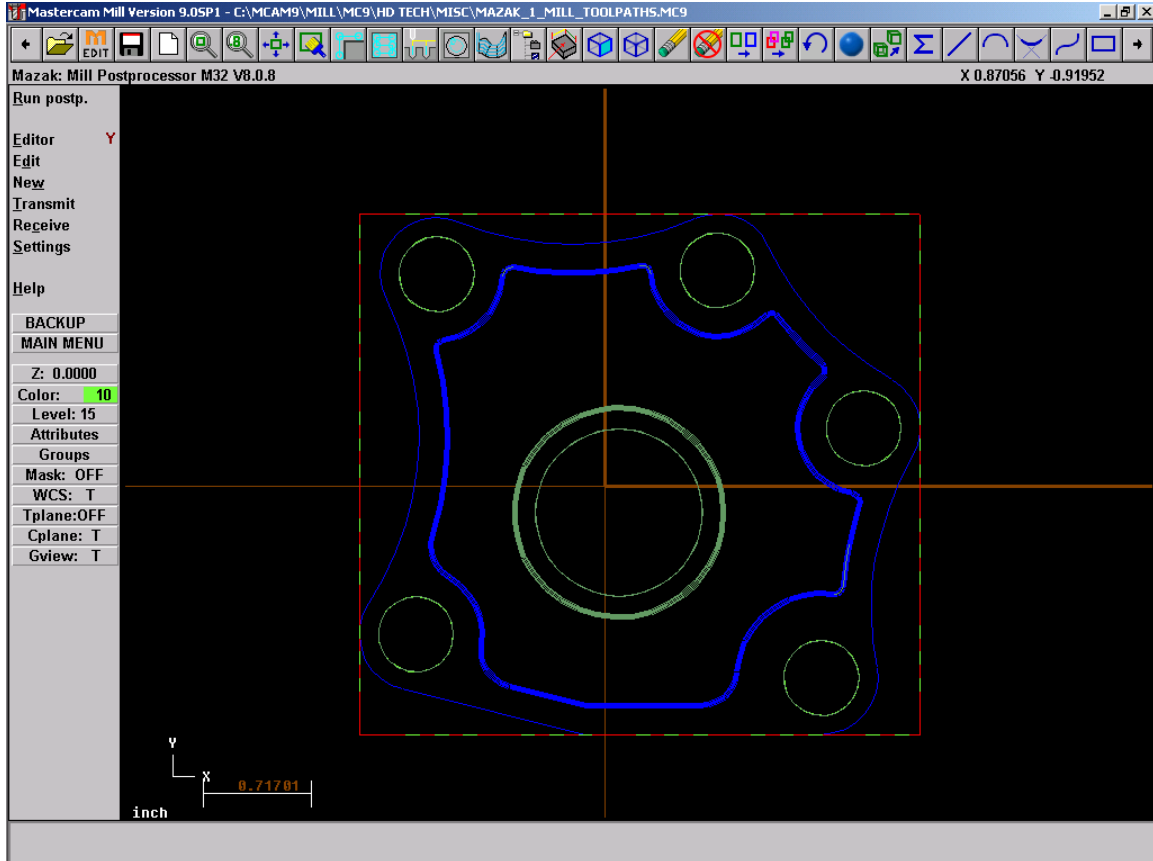


7. Select **OK**. The file name dialog should then appear as shown below:

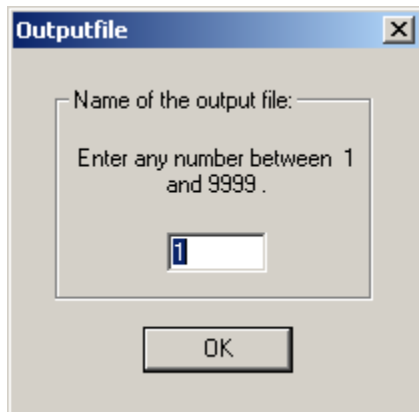
Note: We do not need to create an NC file but Mastercam needs to have this setting so that the post-processor can function

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 **Yes** in the Mazatrol Menu the output will open up in the Mazatrol Editor)

```

0001, ,0000,0000,0000
PNR MAT INITIAL-Z ATC MODE MULTI MODE MULTI FLG PITCH-X PITCH-Y
0 IRON 0.7100 1 OFF

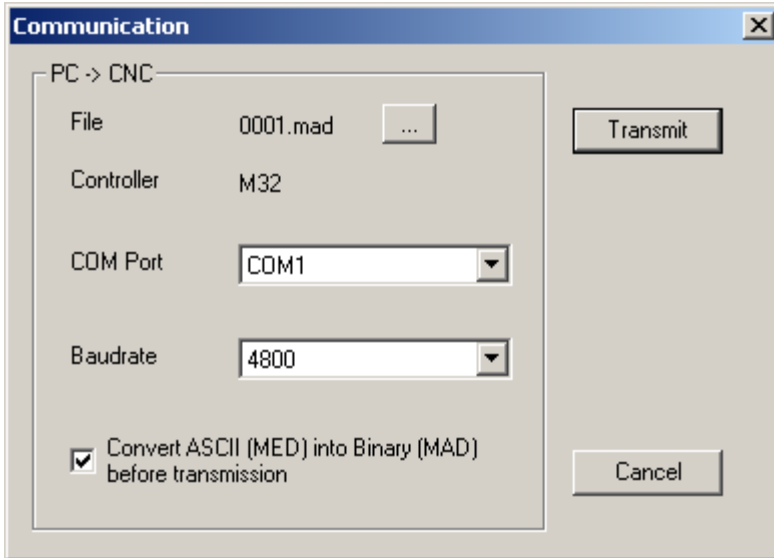
-----
UNO UNO DEPTH SRV-Z SRV-R BTM WAL FIN-Z FIN-R
1 FACE-MIL 0.0000 .001 * 1 * 0 0
SNO SNO NOM. NO. APRCH-X APRCH-Y TYPE ZFD DEP-Z WID-R C-SP FR M M
1 F-MILL 1.50 ? ? XB1 G01 0.0010 1.05 50 0.600 3 8
FIG PTN X Y R/0 I J P CNR
1 LINE -3.0925 -1.6475
2 LINE 2.6553 -1.6475
3 LINE 2.6677 -1.6475
4 CCW 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
    
```

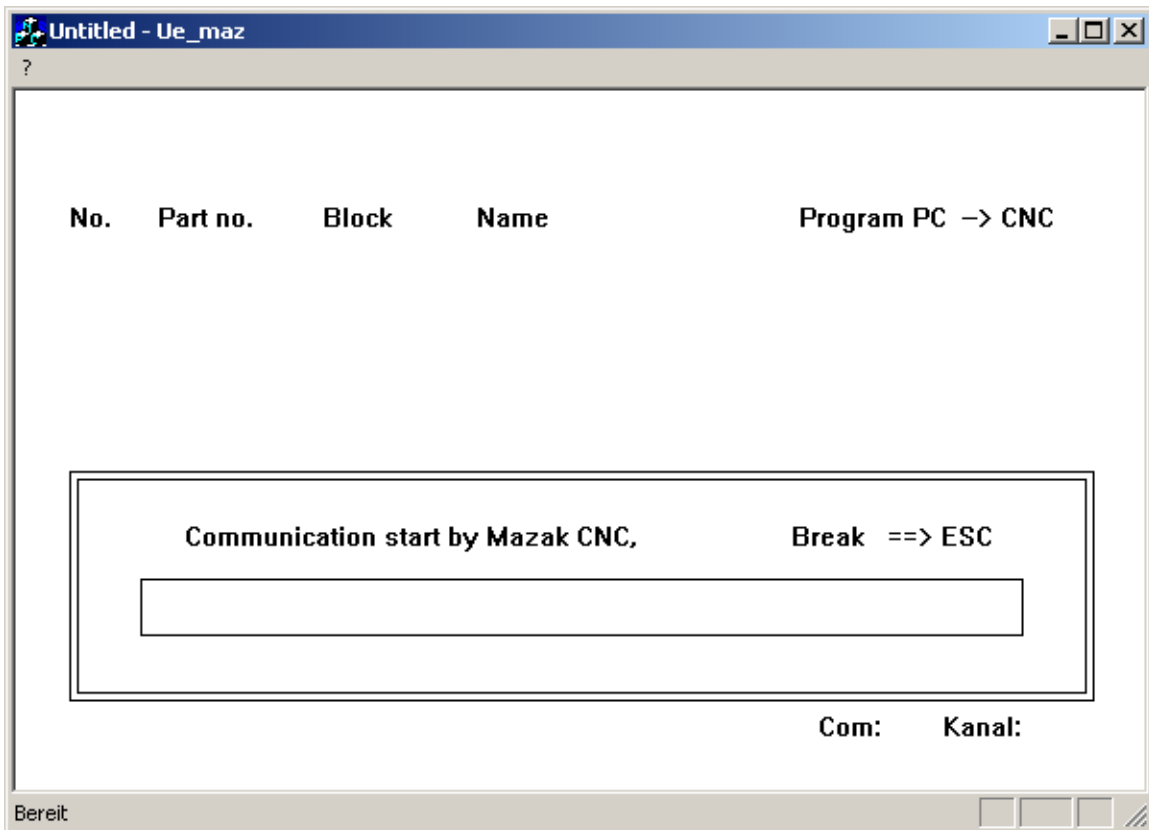
12. Close this window.

We will then send this program to the controller

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



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



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.

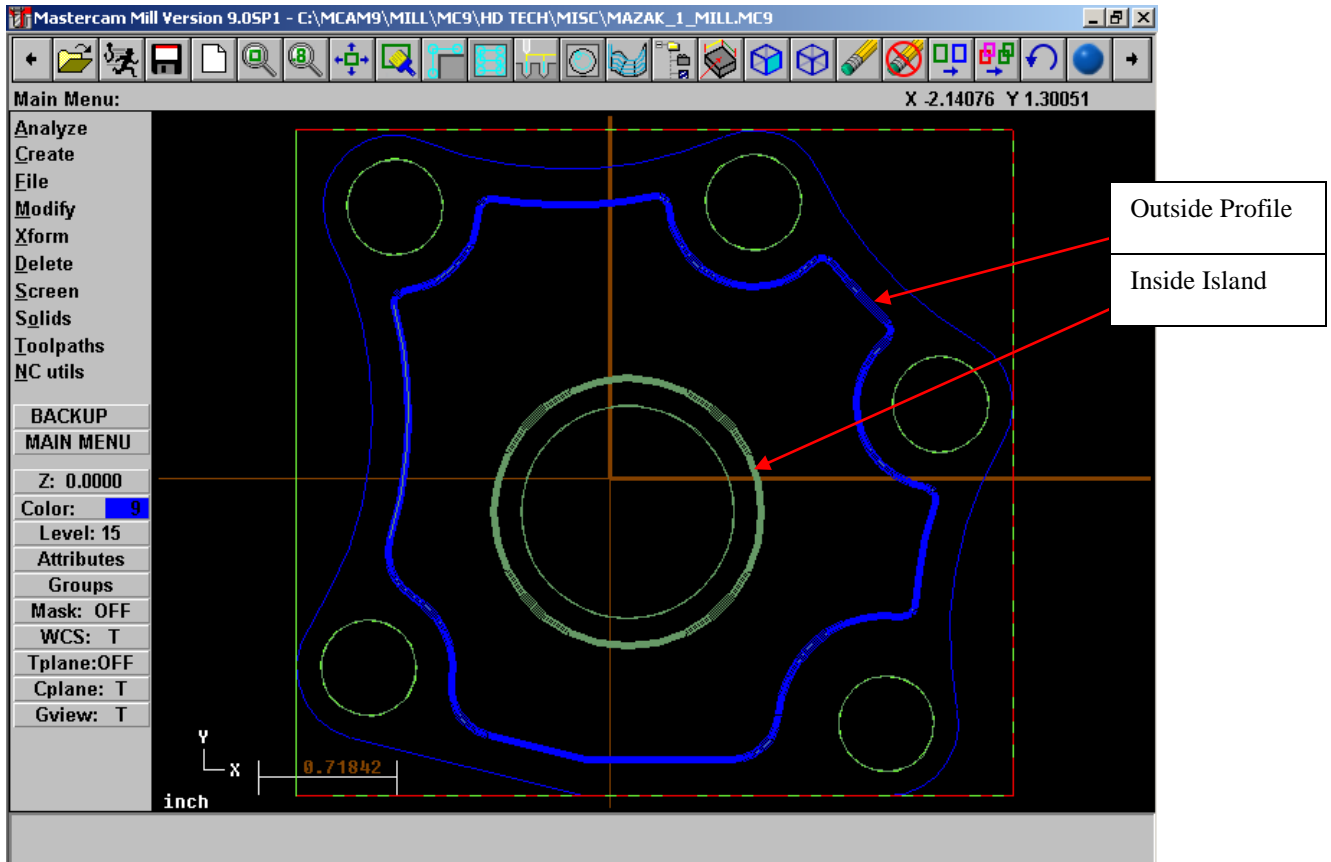
Save File as Mazak_1_Mill_1.mc9

2. Adding Pocketing and Drill Toolpaths

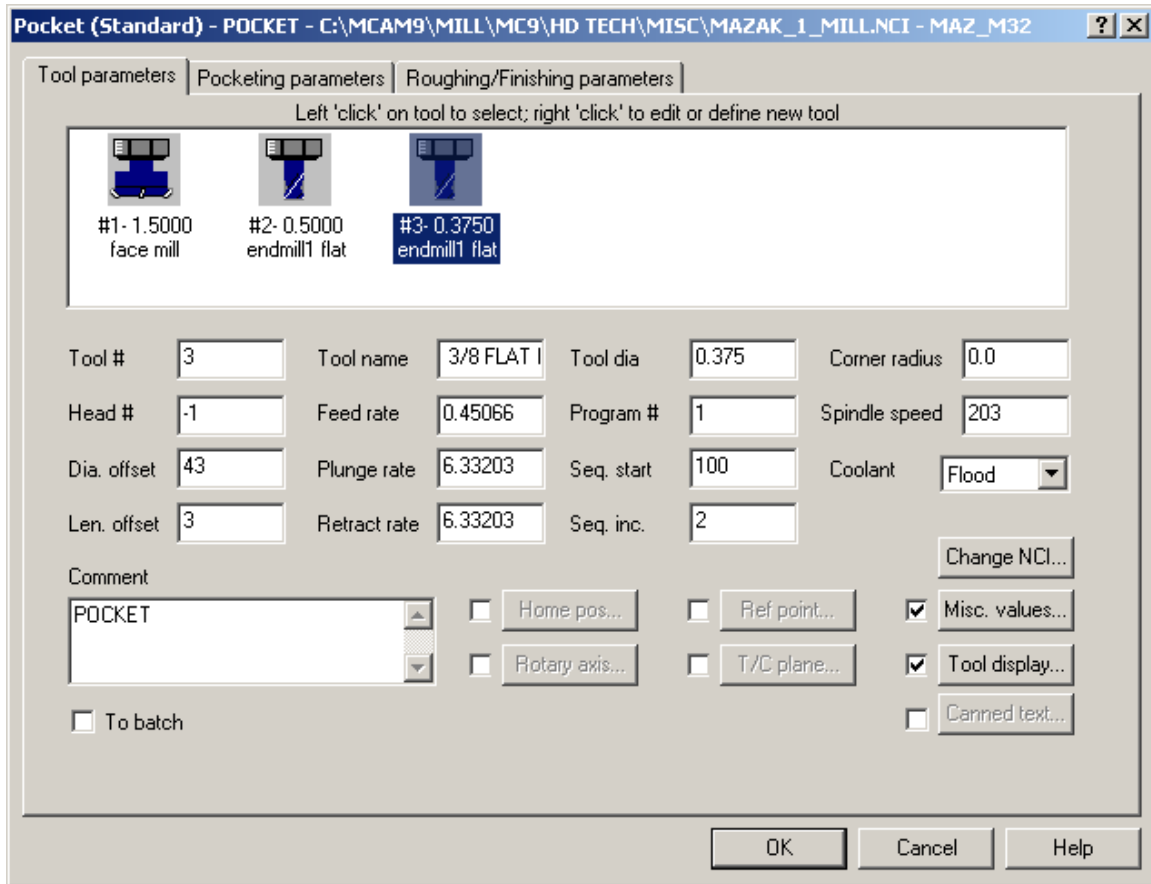
Exercise 1 - Creating Pocket Toolpath

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

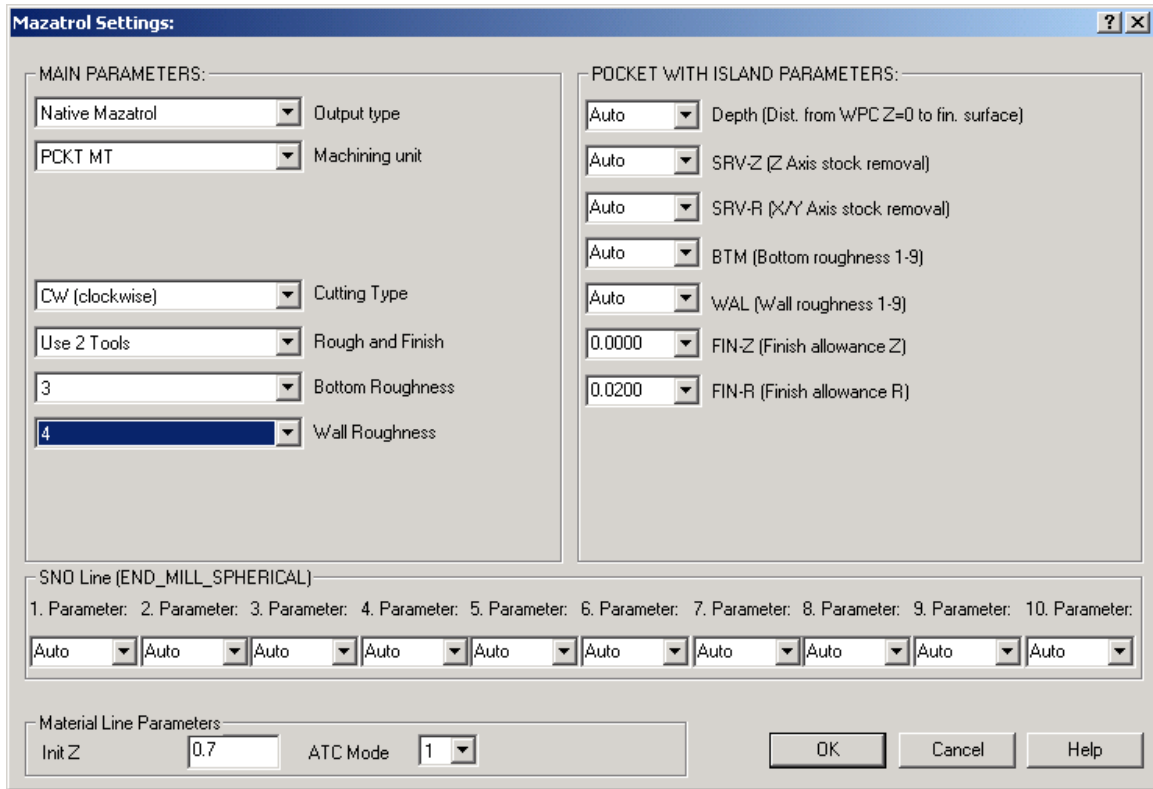
1. Choose **Main Menu, File, Get**
2. Navigate to the folder with the tutorial parts.
3. Select **Mazak_1_Mill_1.mc9**; then choose **Open**.
4. Choose **Main Menu, Toolpaths, Pocket**
5. Chain outside profile shown in Blue and Inside Island as shown in Green



6. Select **Done**

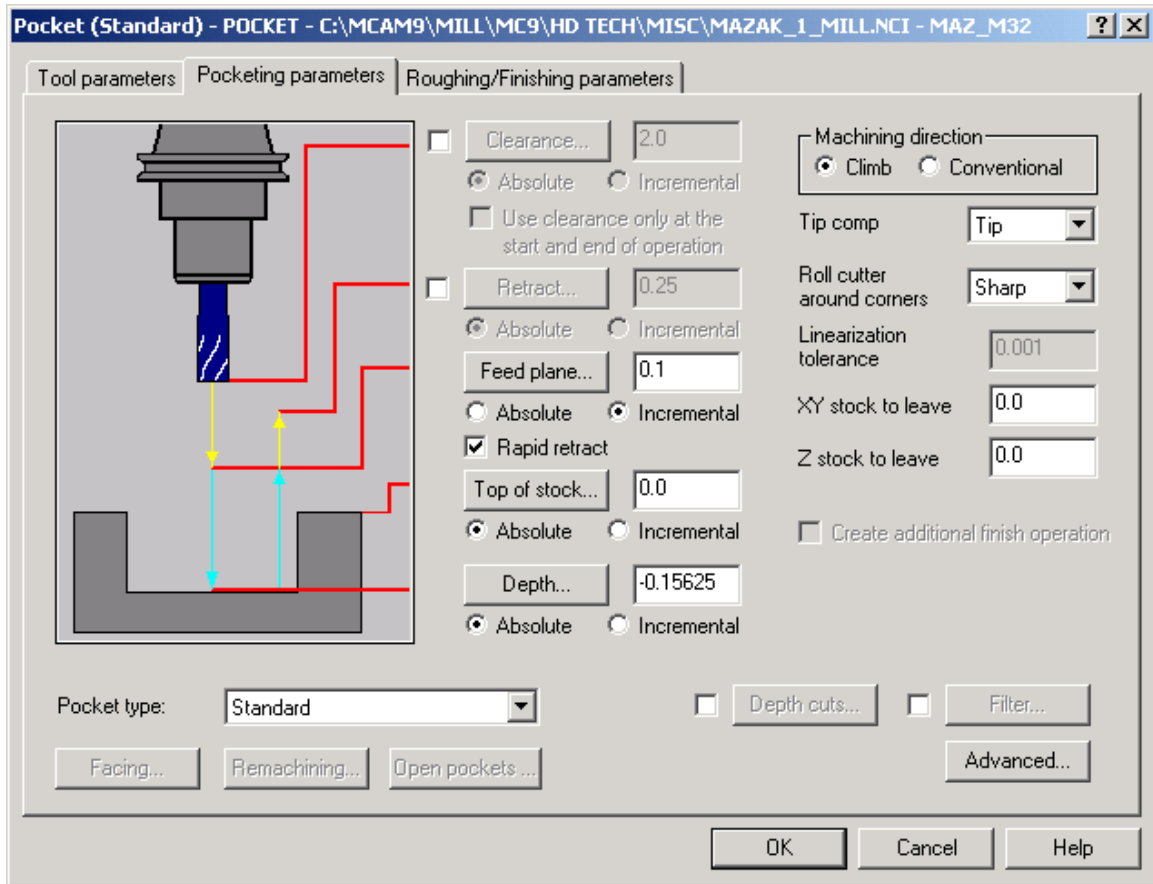


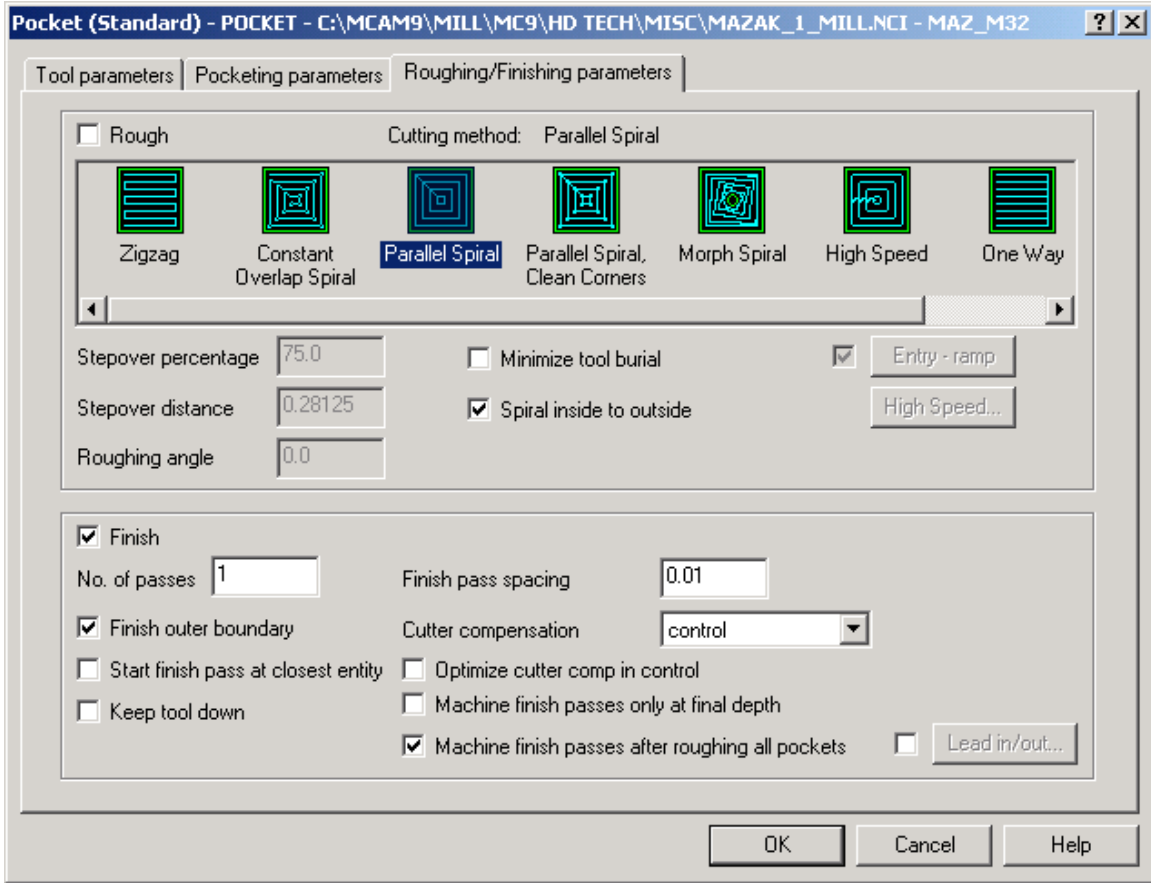
7. Set **Tool Parameters** as shown



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

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





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.

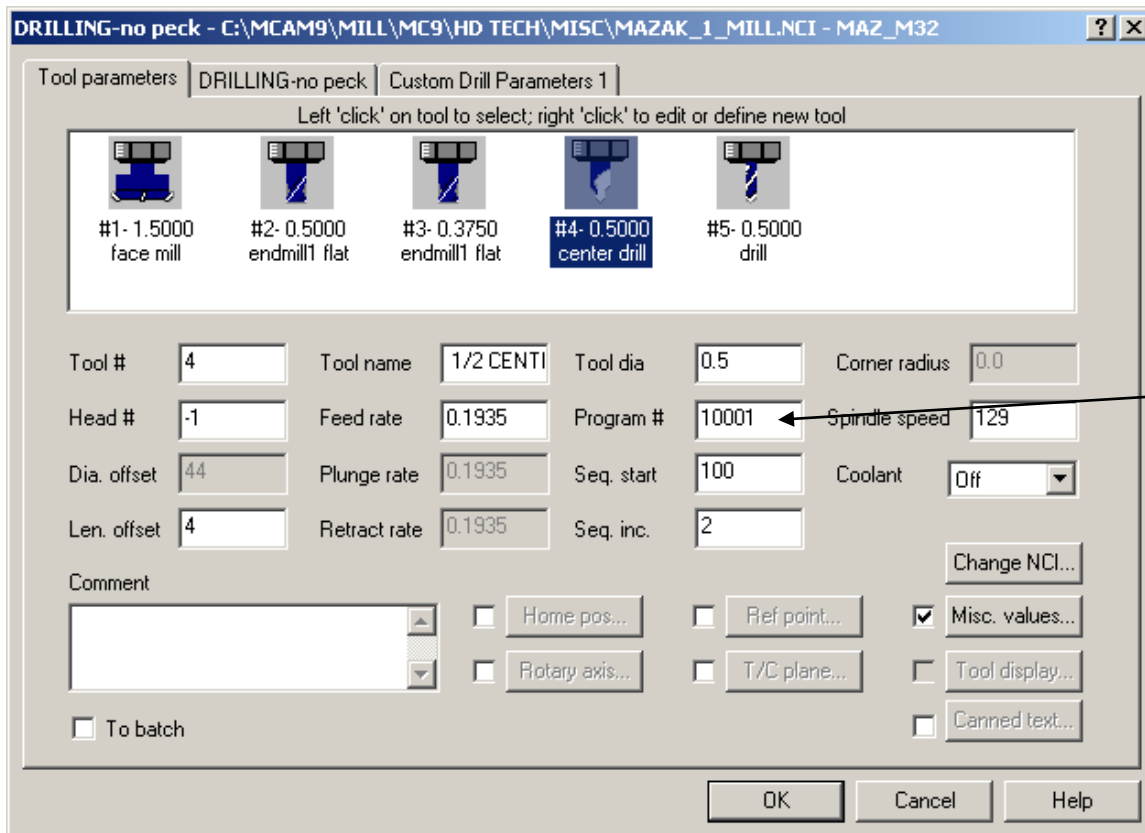
```

-----
UNO UNO  DEPTH  SRV-Z  SRV-R  BTM  WAL  FIN-Z  FIN-R
1  PKT.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

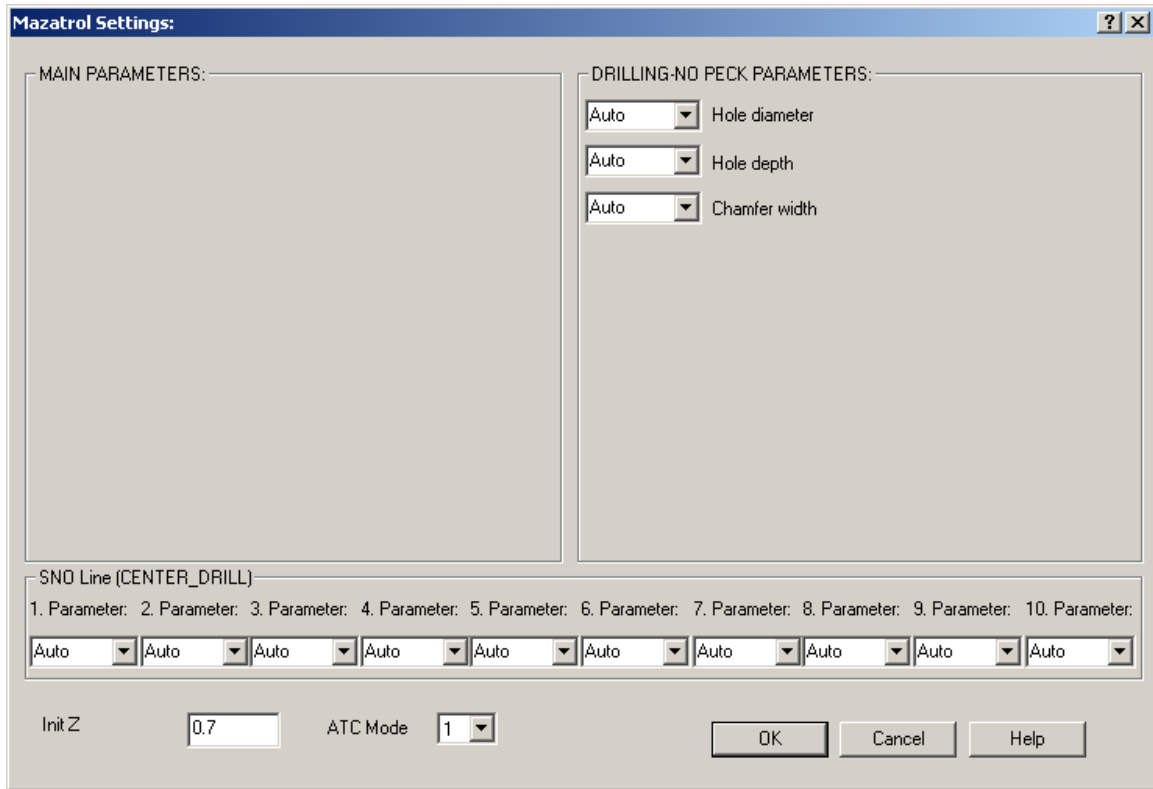
Select the following:

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

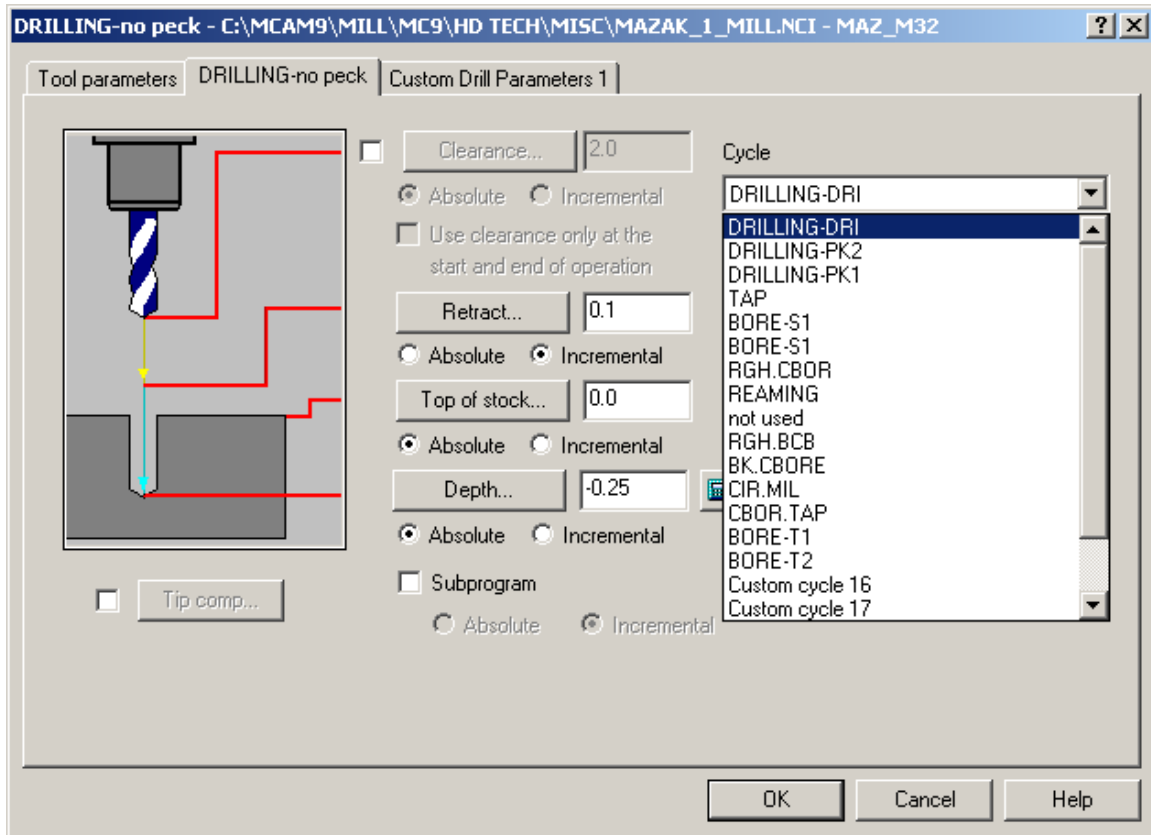


Select 0.5”center drill as shown

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)



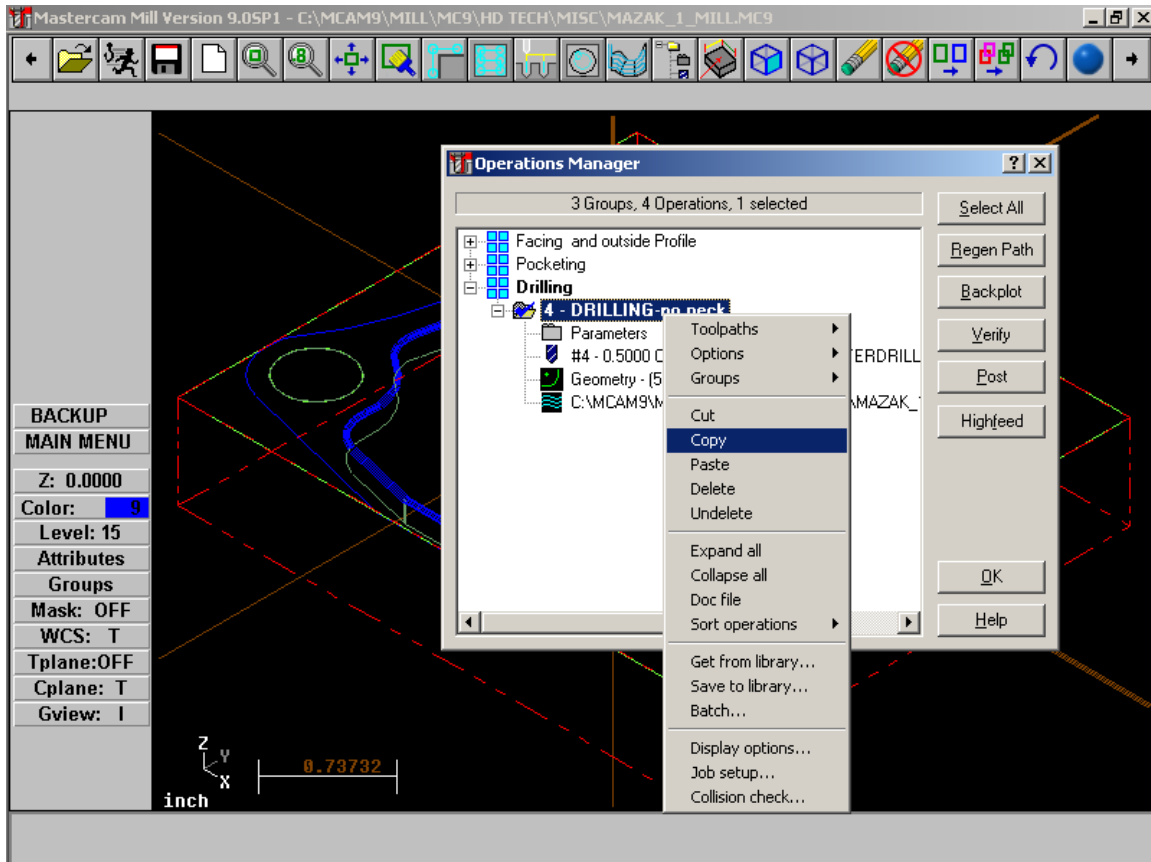
7. **Misc. Values** Leave settings on Auto as shown



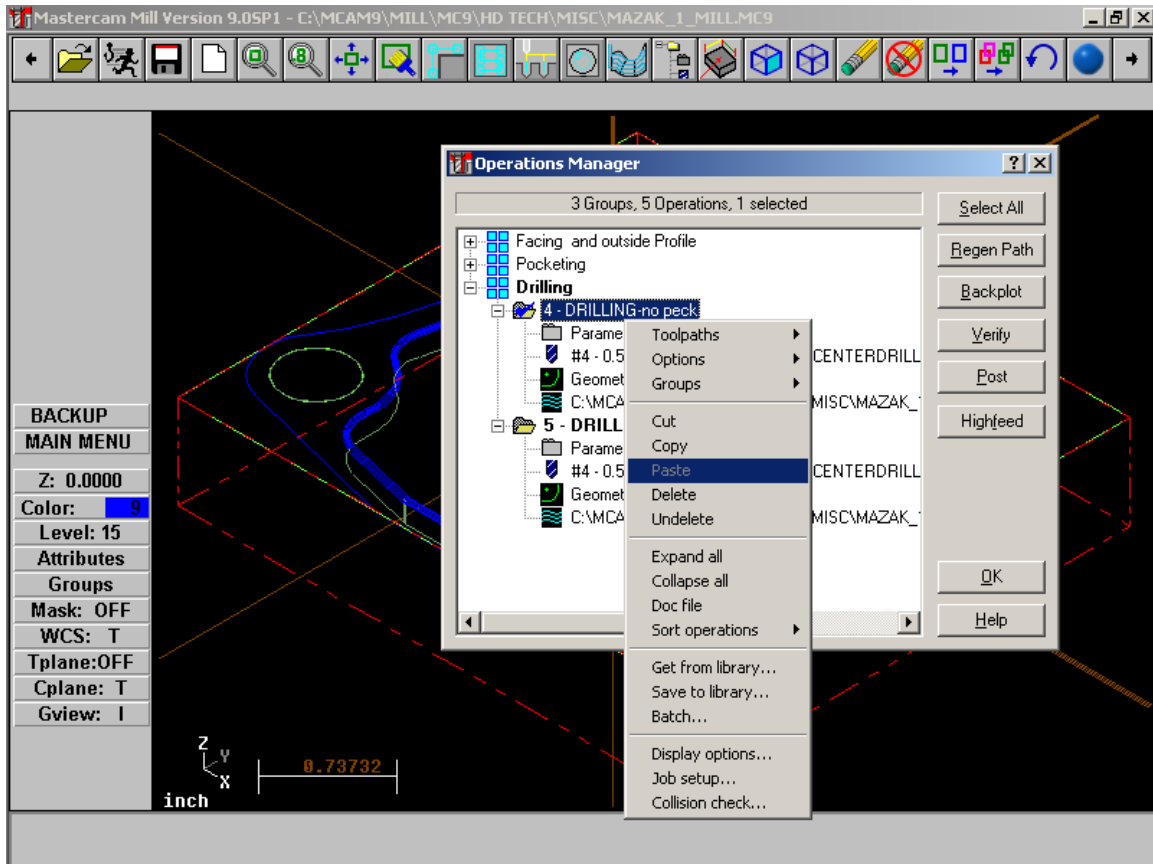
8. Go to **Drilling** and set **DRILLING – DRI** as shown

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

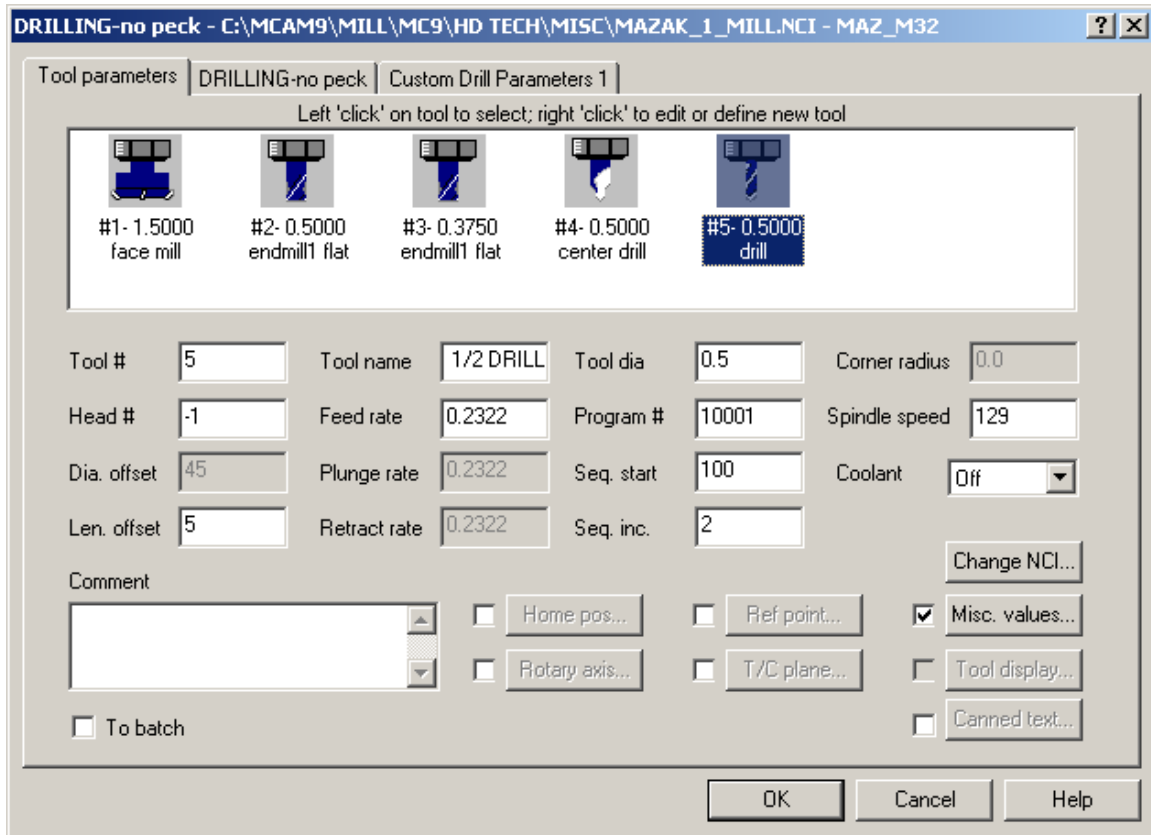
9.OK



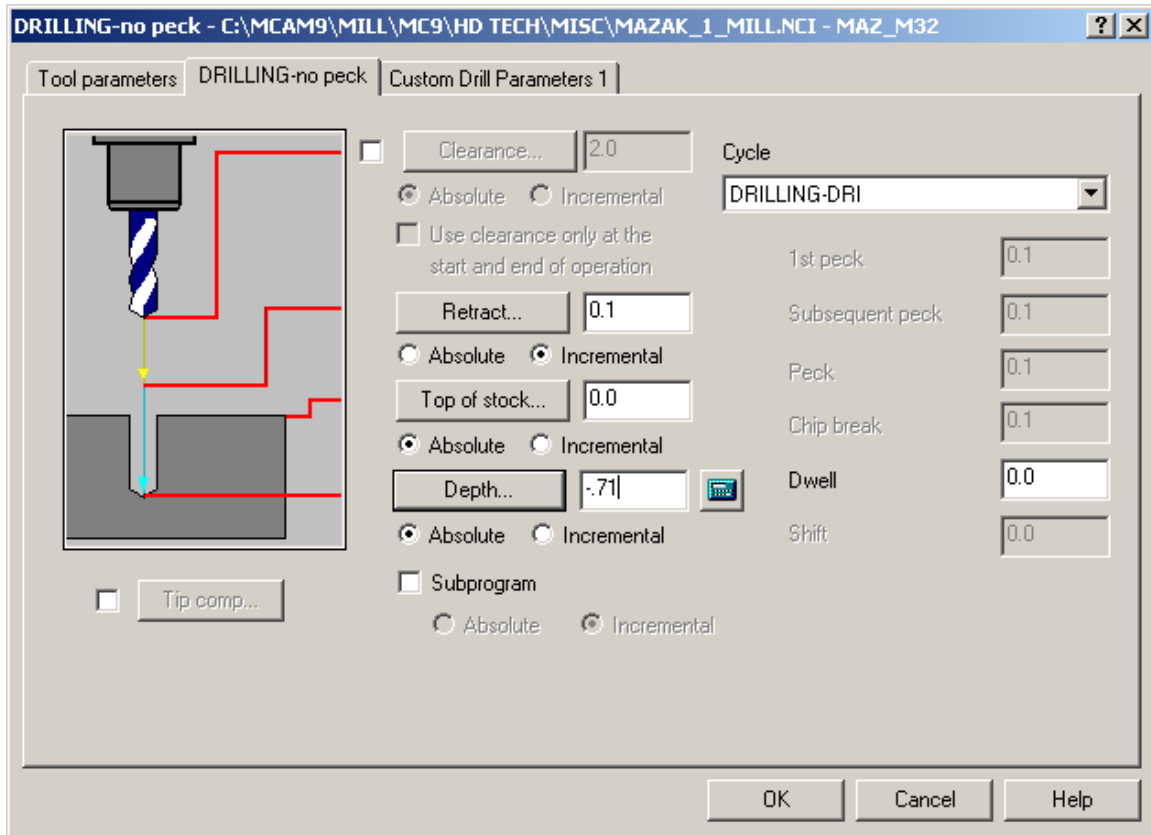
Now we will copy the previous operation. Therefore the only changes we need to make will be the tool we want to use and the drilling depth. All the other values will stay the same.



11. Paste new operation



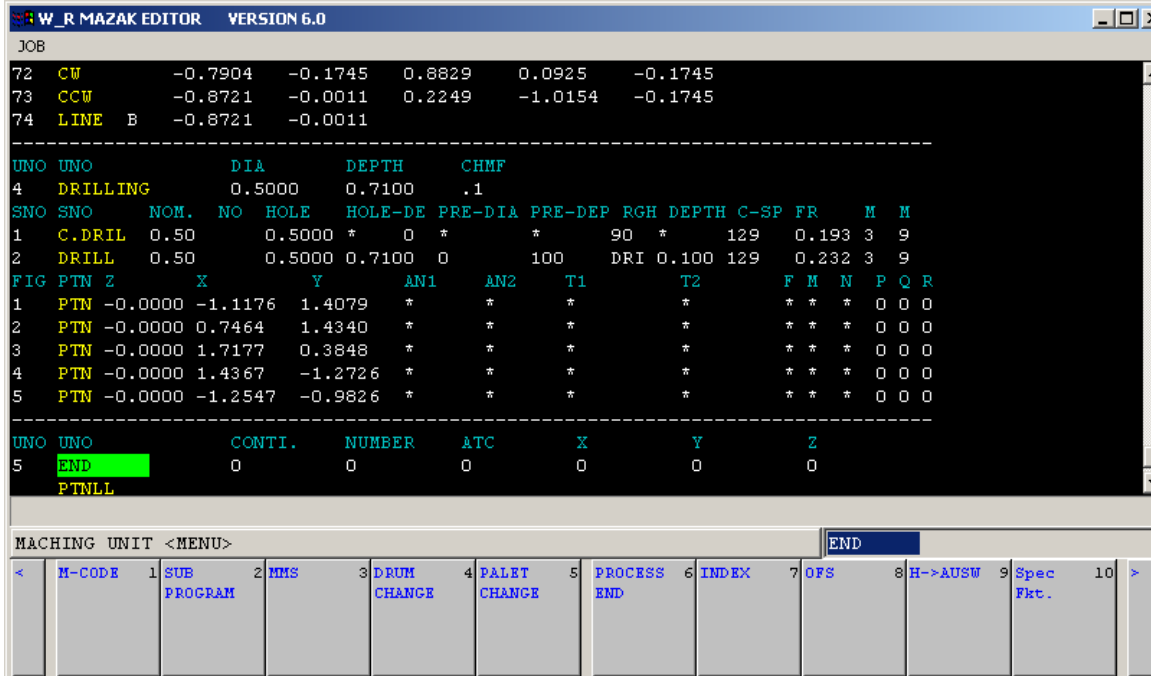
12. Select 0.5” tool as shown



13. Set **Depth** as shown.

14. **OK.**

After posting the output will appear as shown below.



15. Save File

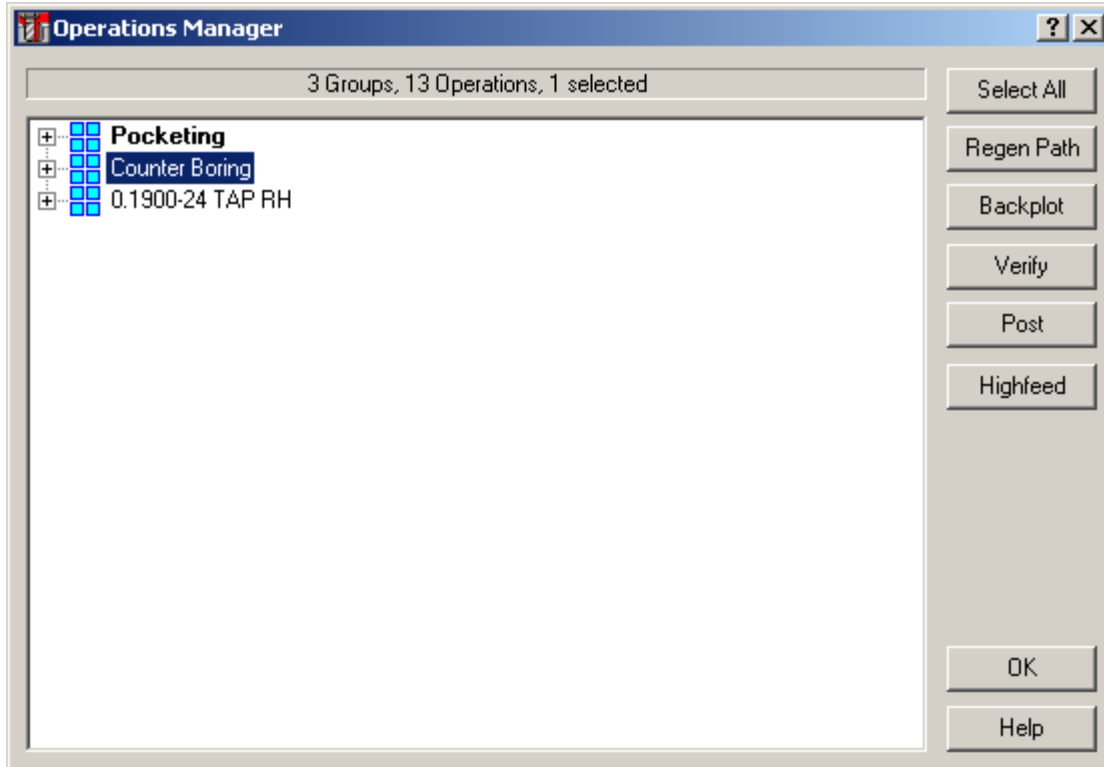
16. Post File and view output.

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.

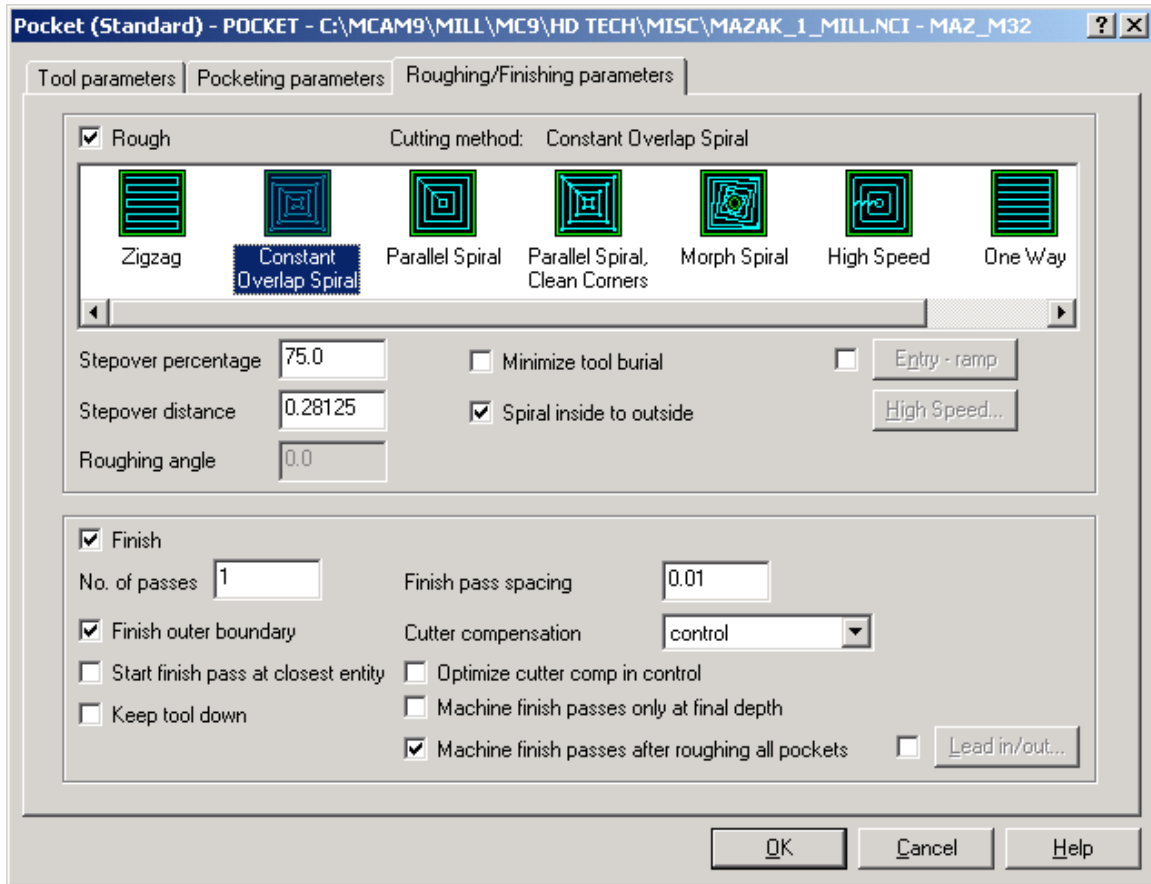
1. Choose **Main Menu, File, Get**
2. Navigate to the folder with the tutorial parts.
3. Select **Mazak_2_Mill.mc9**; then choose **Open**.
4. Go to **Operations Manager** you should see dialog as below



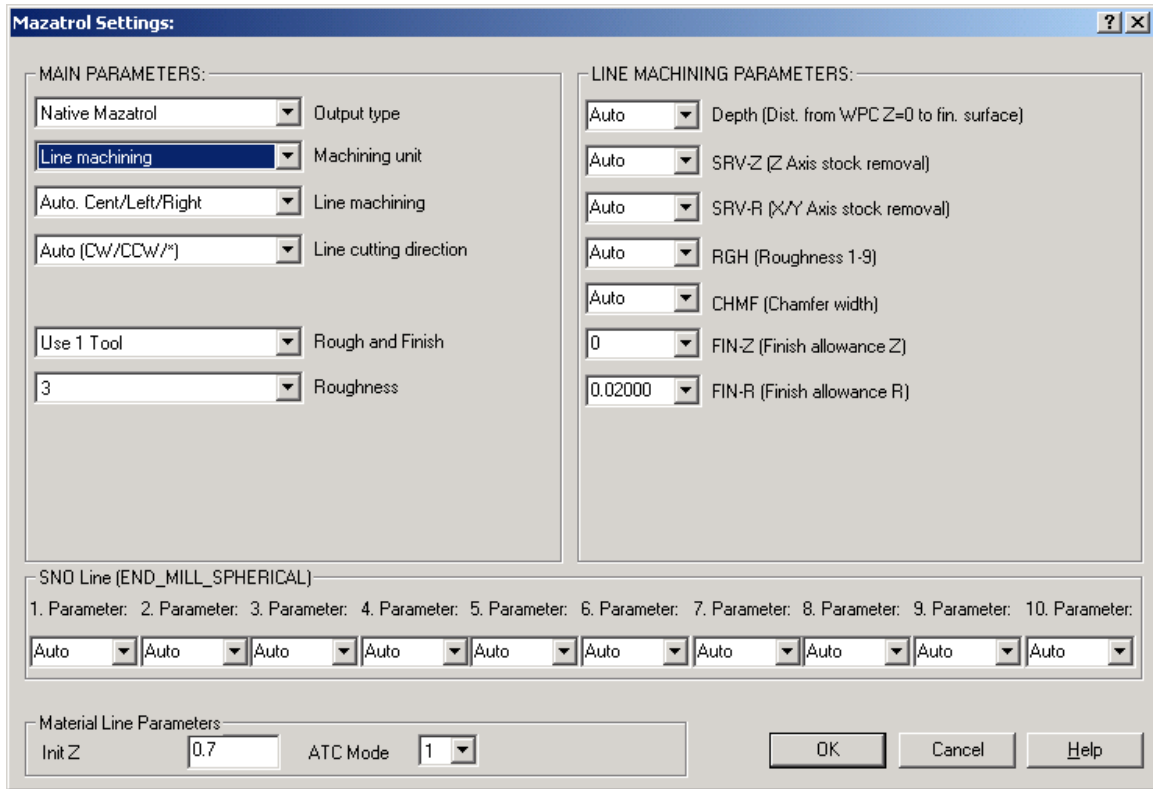
In this file we have created a part in the using pocketing that would be very difficult to program in Mazatrol because the pocket has multiple islands. We have also used a tool that is too big to complete the machining of the pocket and then taken advantage of Mastercam's Pocket Remachining routine. 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.

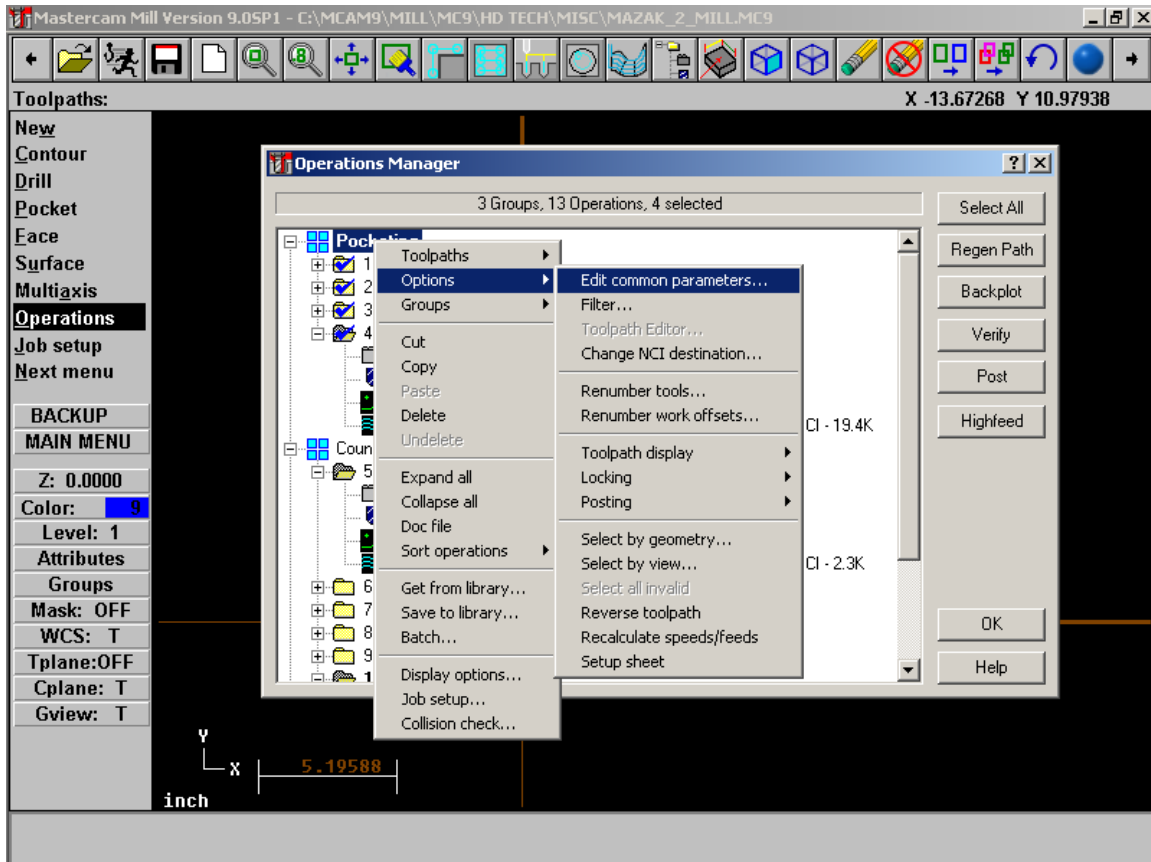


1. Fill in settings as shown below:



2. Set all other pocket toolpaths programmed likewise using **Edit Common Toolpath** parameters and go to Misc. Values button

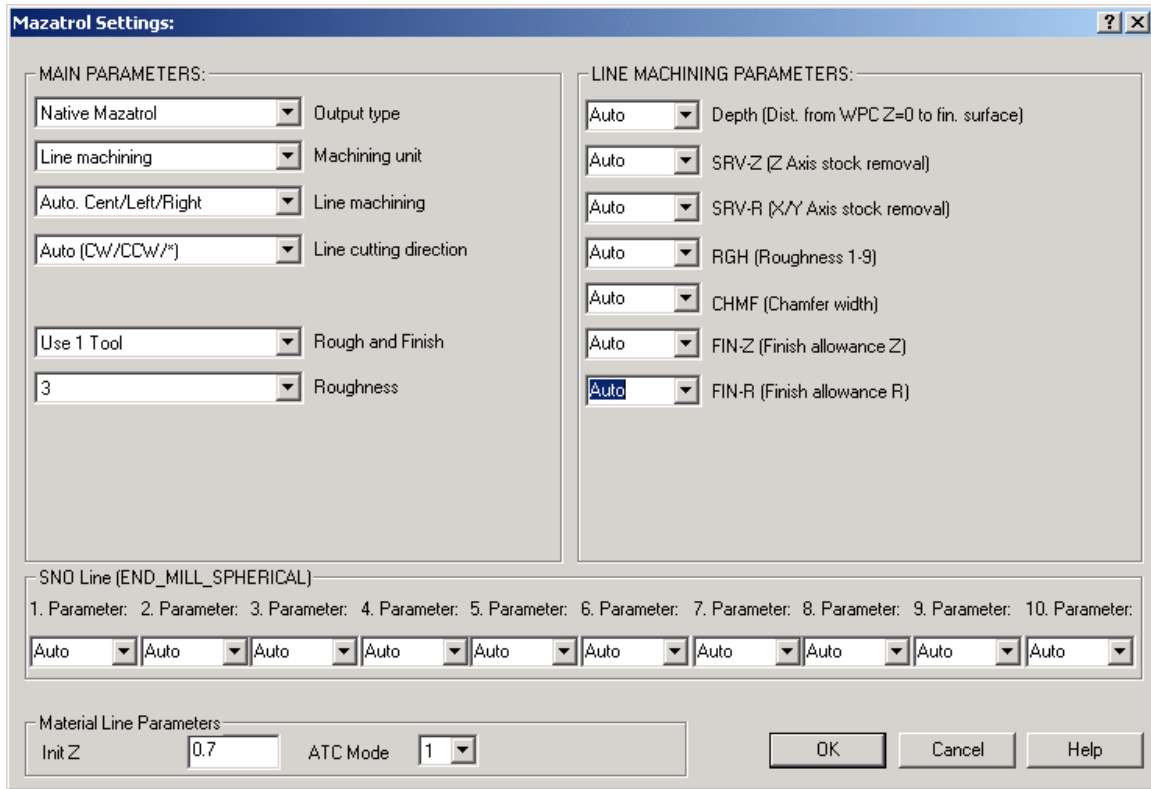
Mastercam to Mazatrol Post-Processor Tutorial 11/1/2018



A section of the Mazatrol output will be as below:

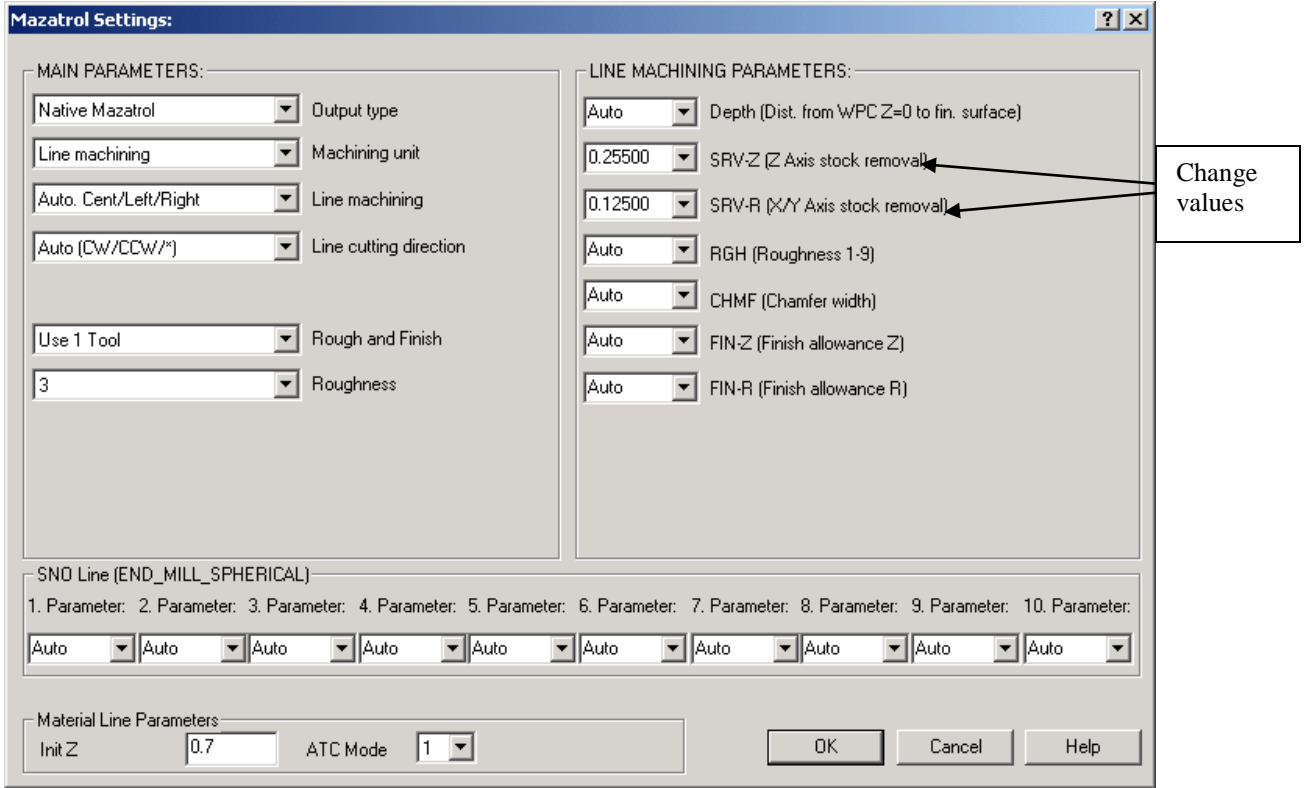
```

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/O 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:

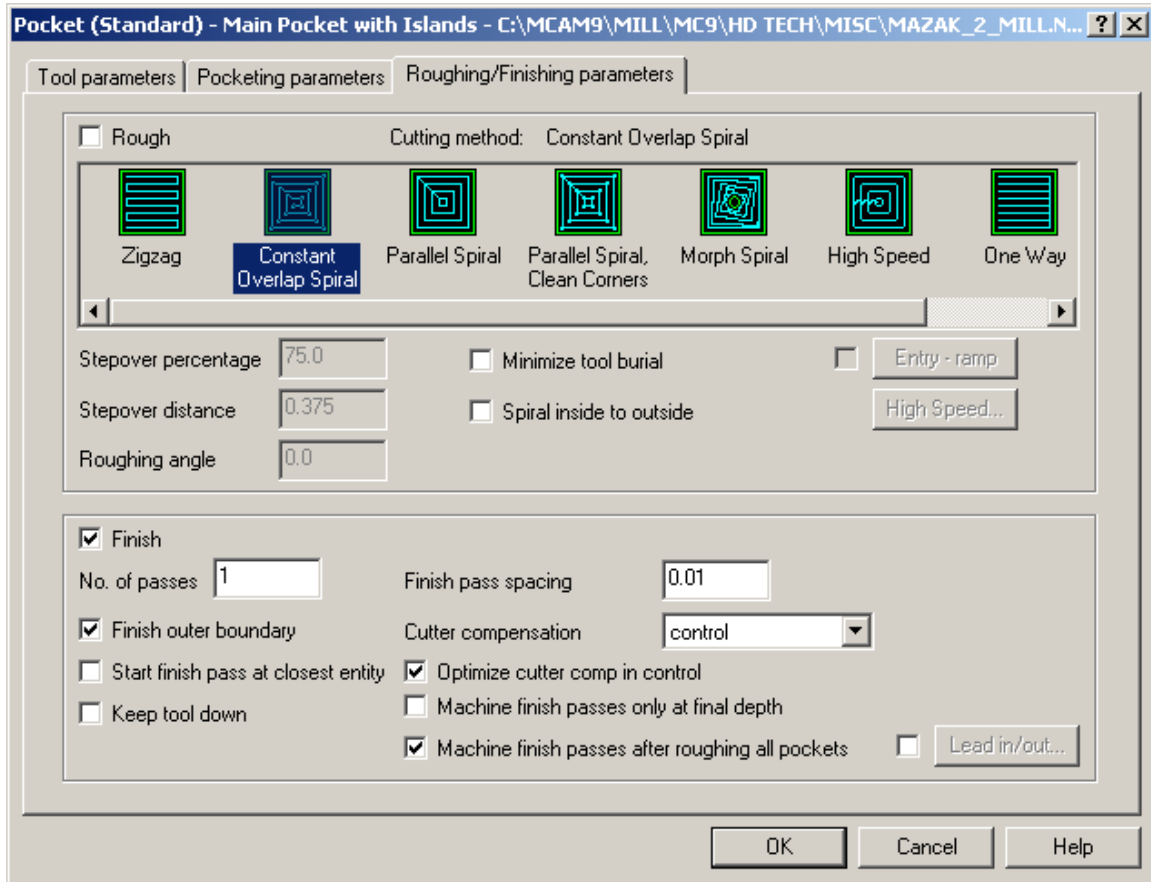
```
PNR MAT INITIAL-Z ATC MODE MULTI MODE MULTI FLG PITCH-X PITCH-Y
0 ALUMINUM 1.0 0 OFF
```

```
-----
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/O 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
```

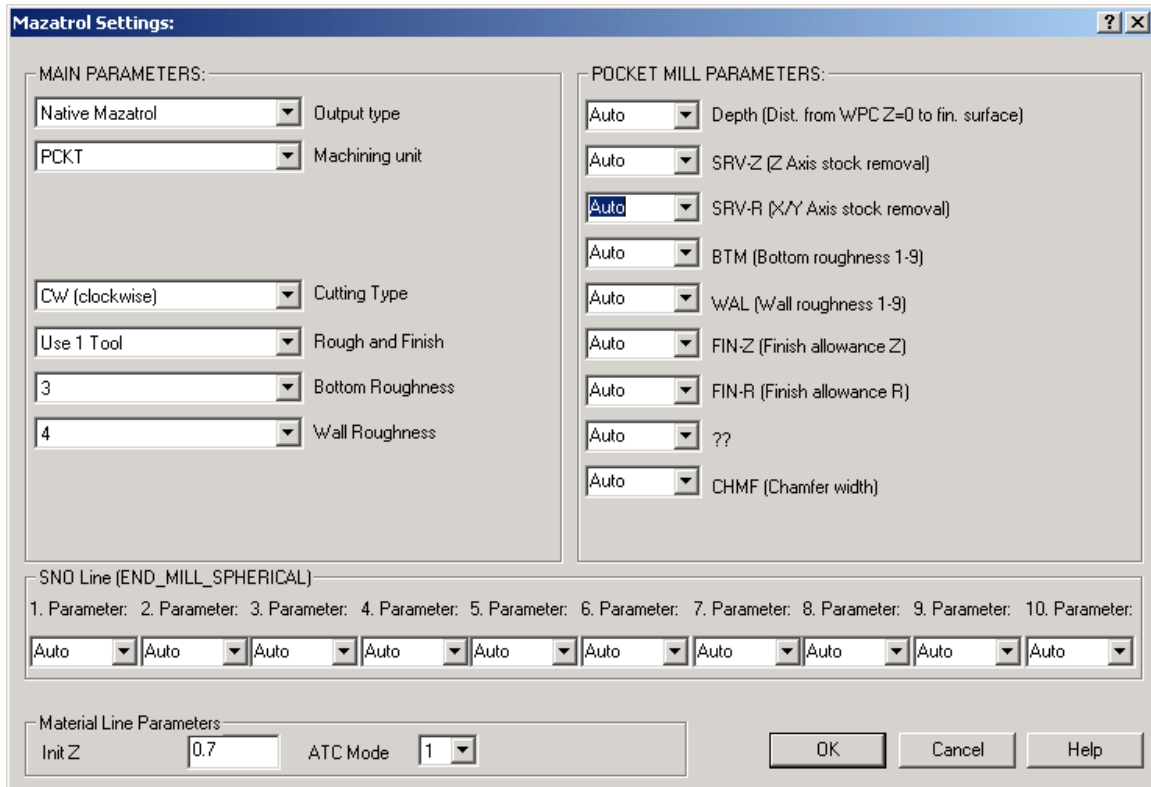
```
-----
UNO UNO DEPTH SRV-Z SRV-R RGH CHMF FIN-Z FIN-R
2 LINE-CTR 0.6000 .255 .125 3 * 0 0 SNO SNO NOM. NO. APRCH-X APRCH-Y TYPE ZFD
DEP-Z WID-R C-SP FR M M
1 E-MILL 0.50 ? ? * G01 0.2550 * 1069 6.417 3 9
FIG PTN X Y R/O I J P CNR
```

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:

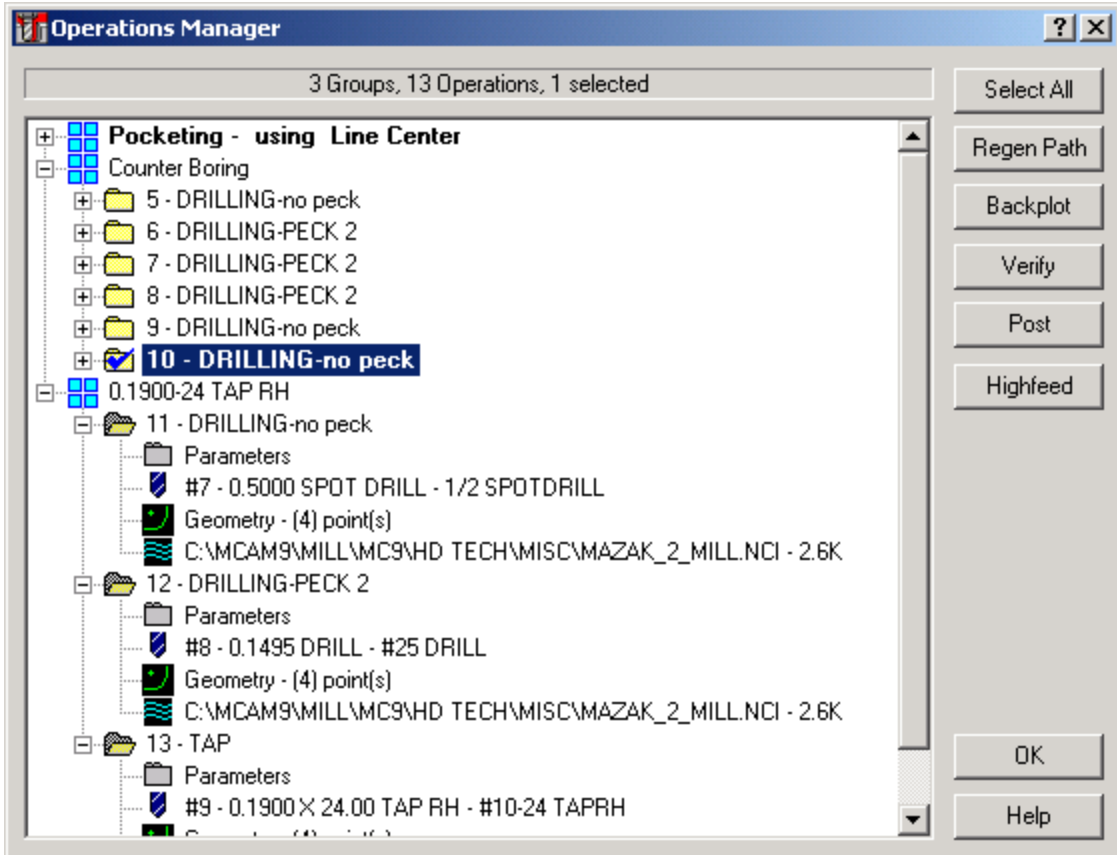


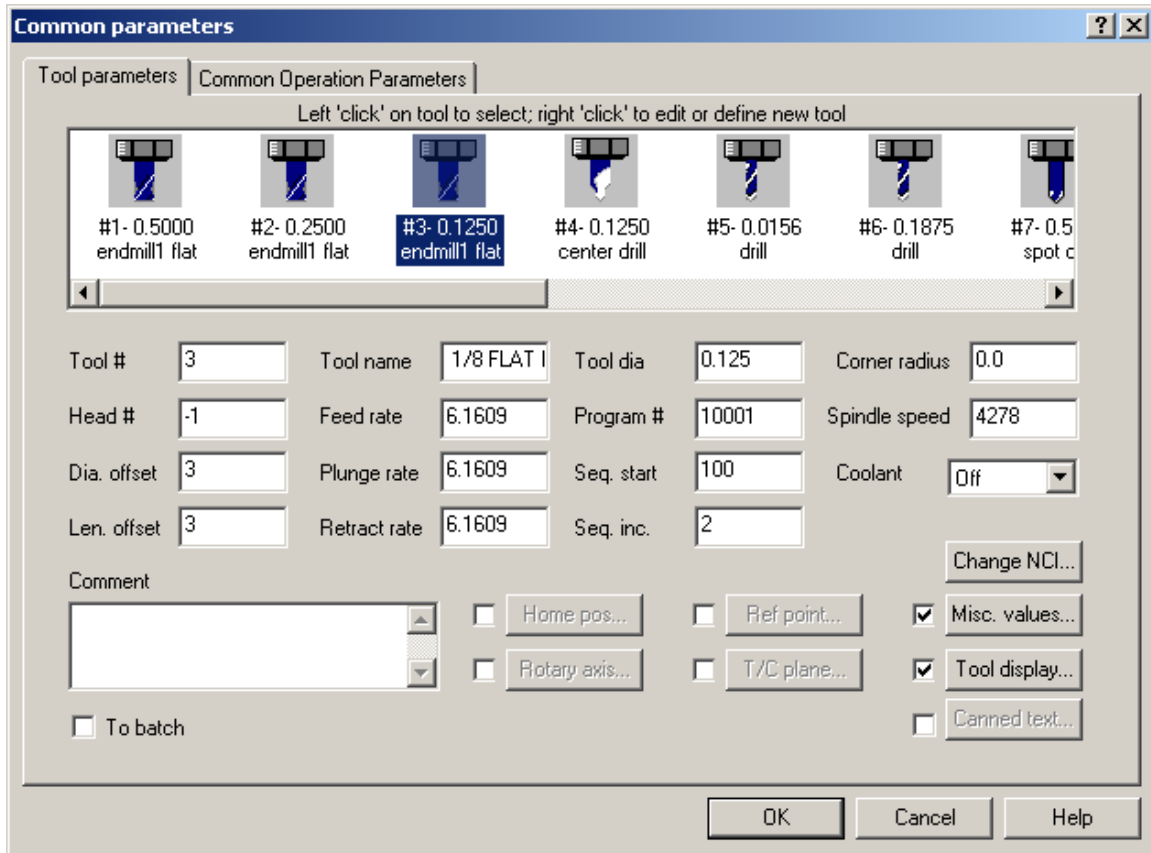
Exercise 3 - Modifying Drill Cycles in Counter Boring Group

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.

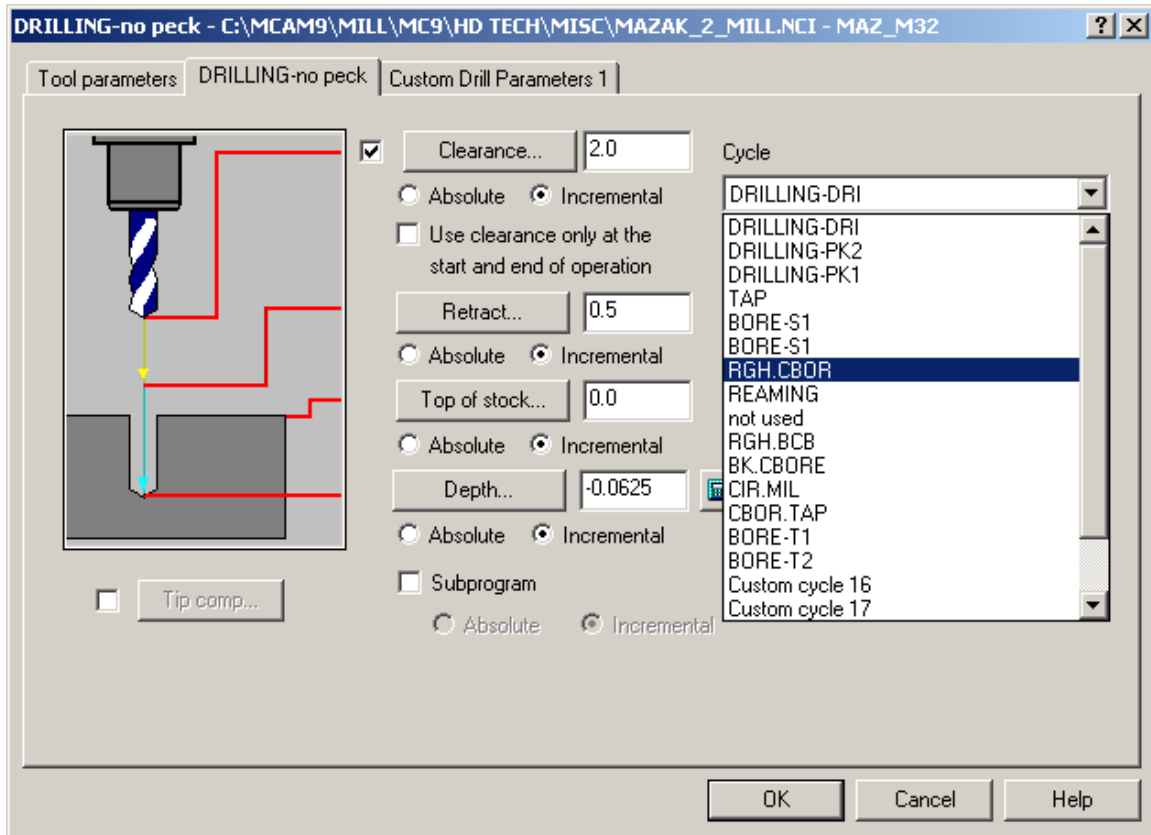
We need to do the following:

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



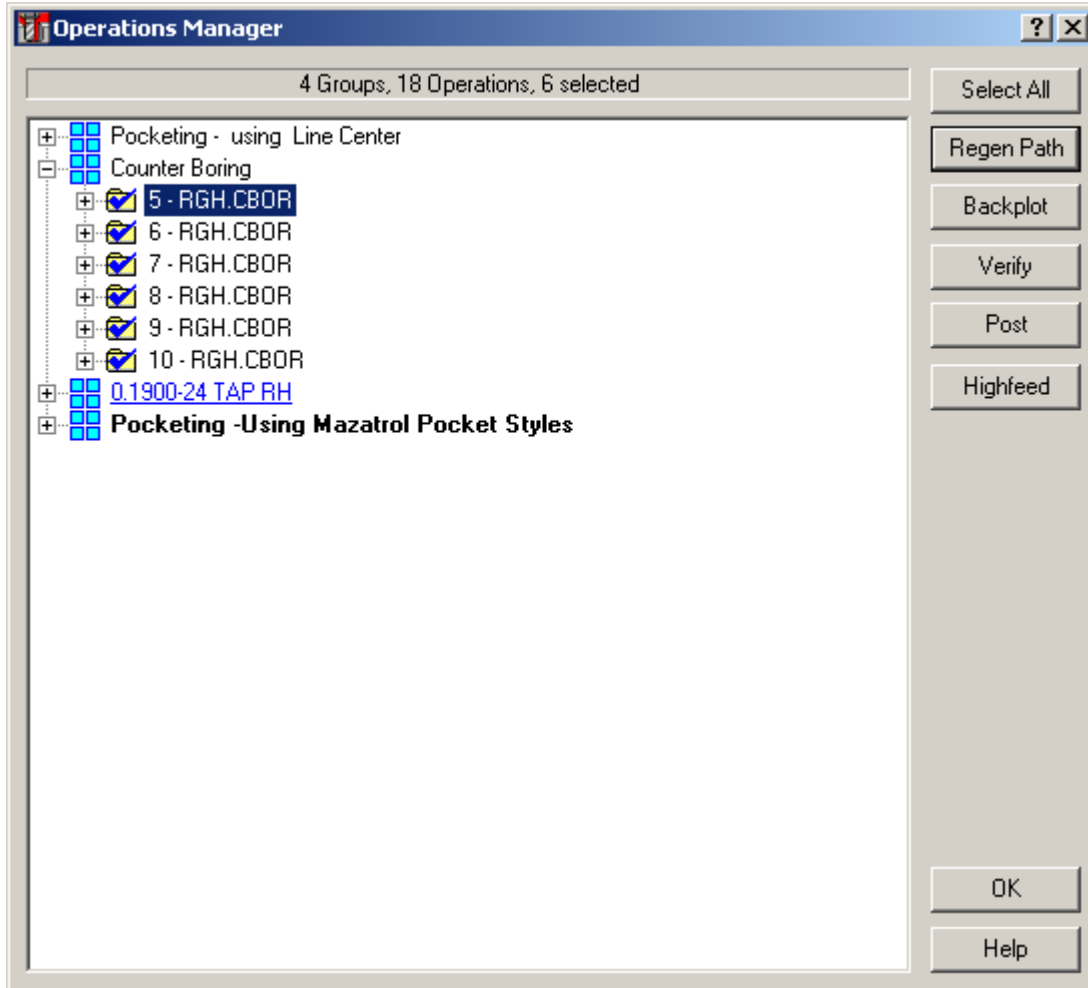


2. We now need to make sure that for all the operations in this group the drill cycles are set as follows:



*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

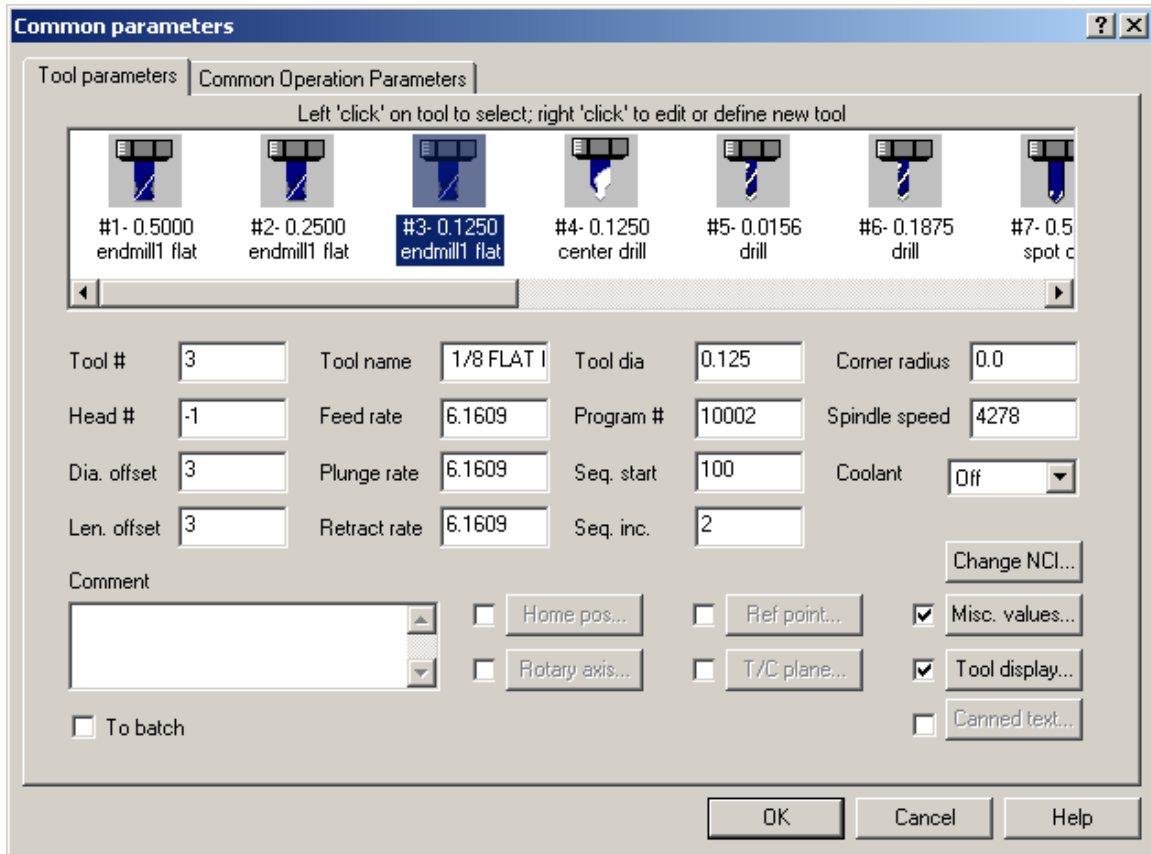


Exercise 4 - Modifying Drill Cycles in Tapping Group

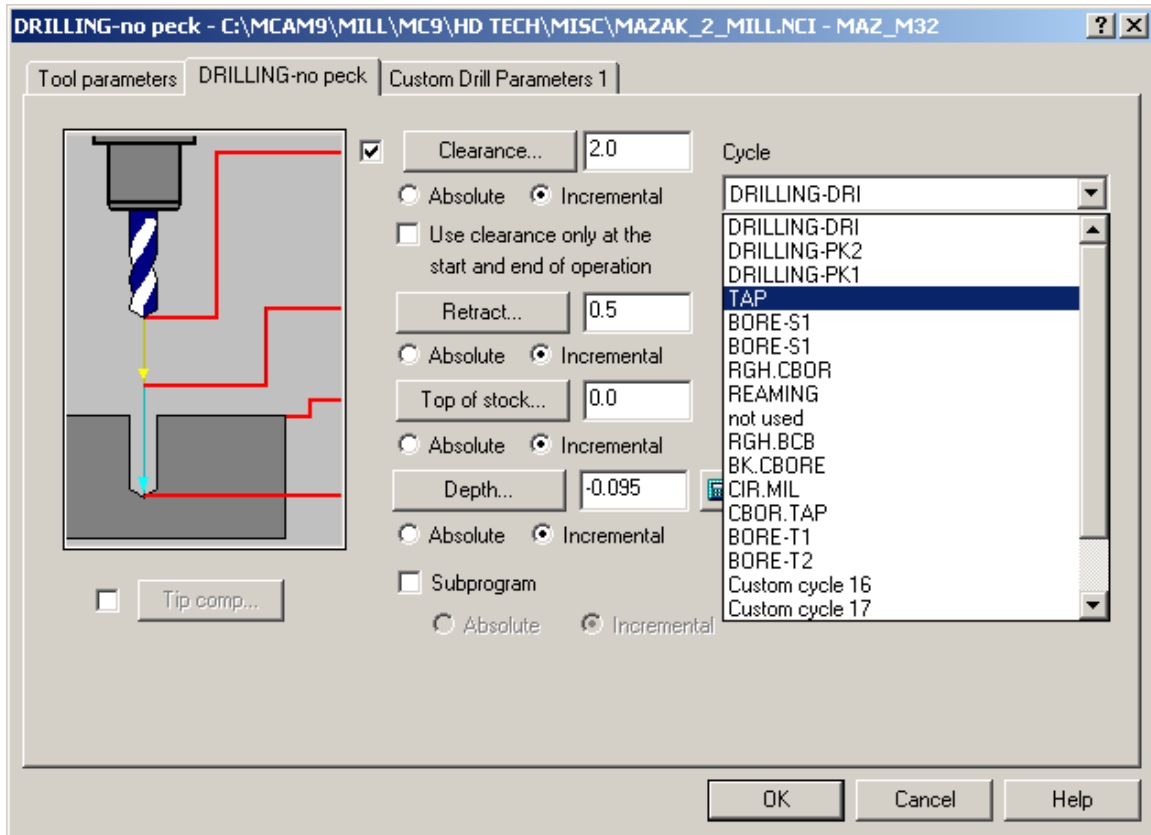
As in the previous exercise we will modify the three operations grouped as **.1900-24 TAP RH** so that the output will be more efficient and readable as Mazatrol format. In addition we will add an operation to create a chamfer before the final tapping operation.

We need to do the following:

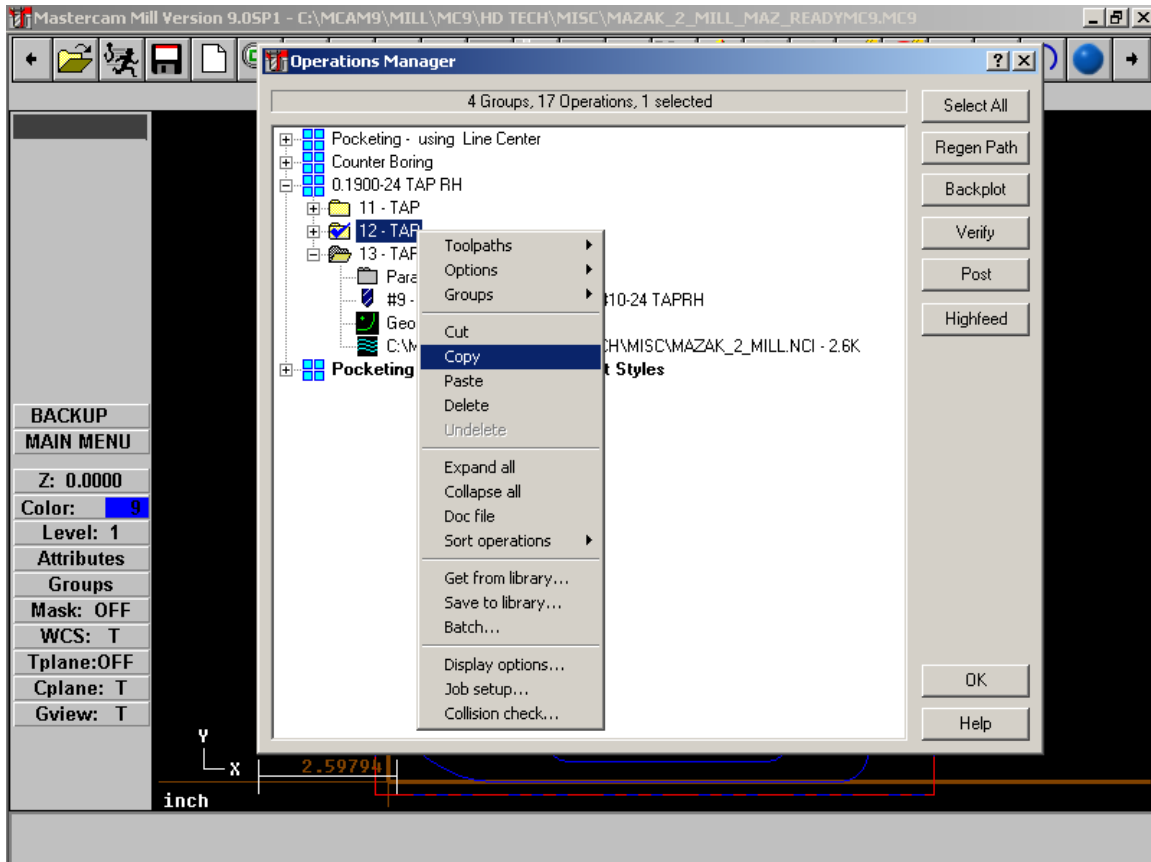
1. Using **EDIT COMMON PARAMETERS** highlight the Tapping Group in the **Operations Manager** set the Program # as follows:



2. Set all Drill Cycles to TAP as below:

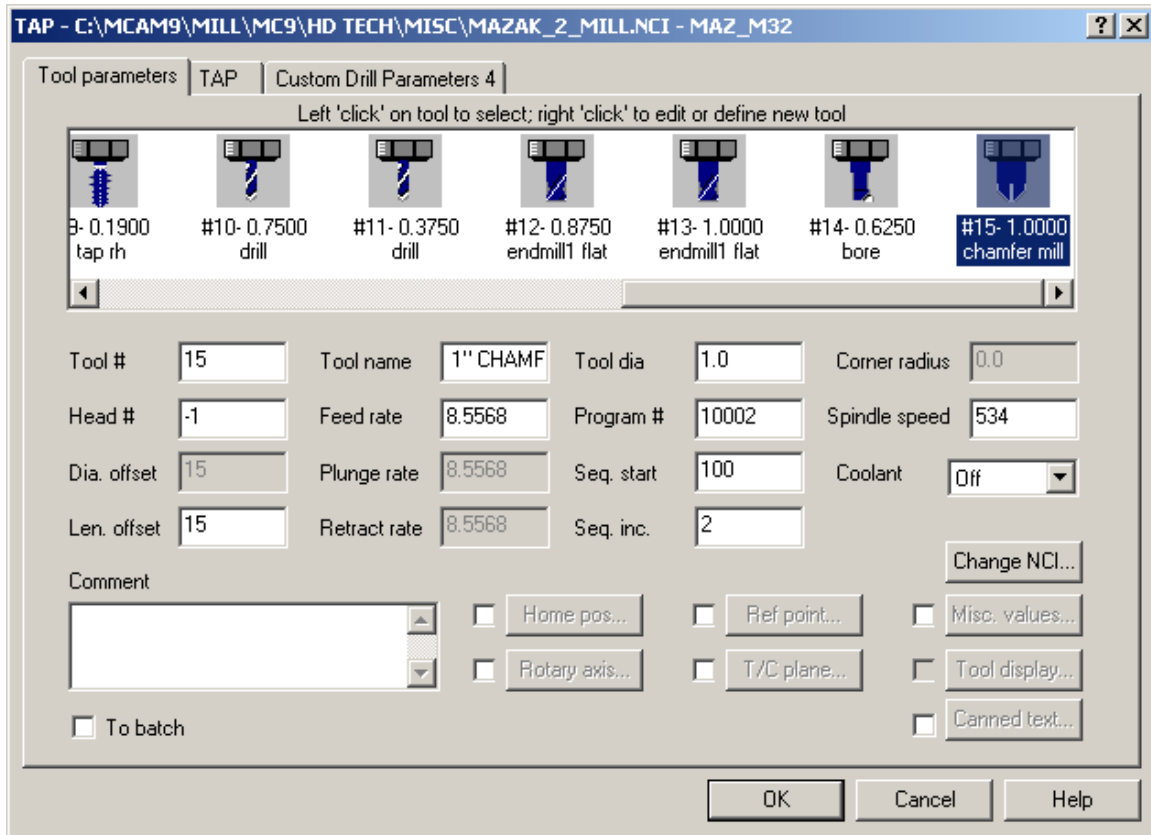


3. Now to add chamfering toolpath copy and paste the second operation within the group as shown



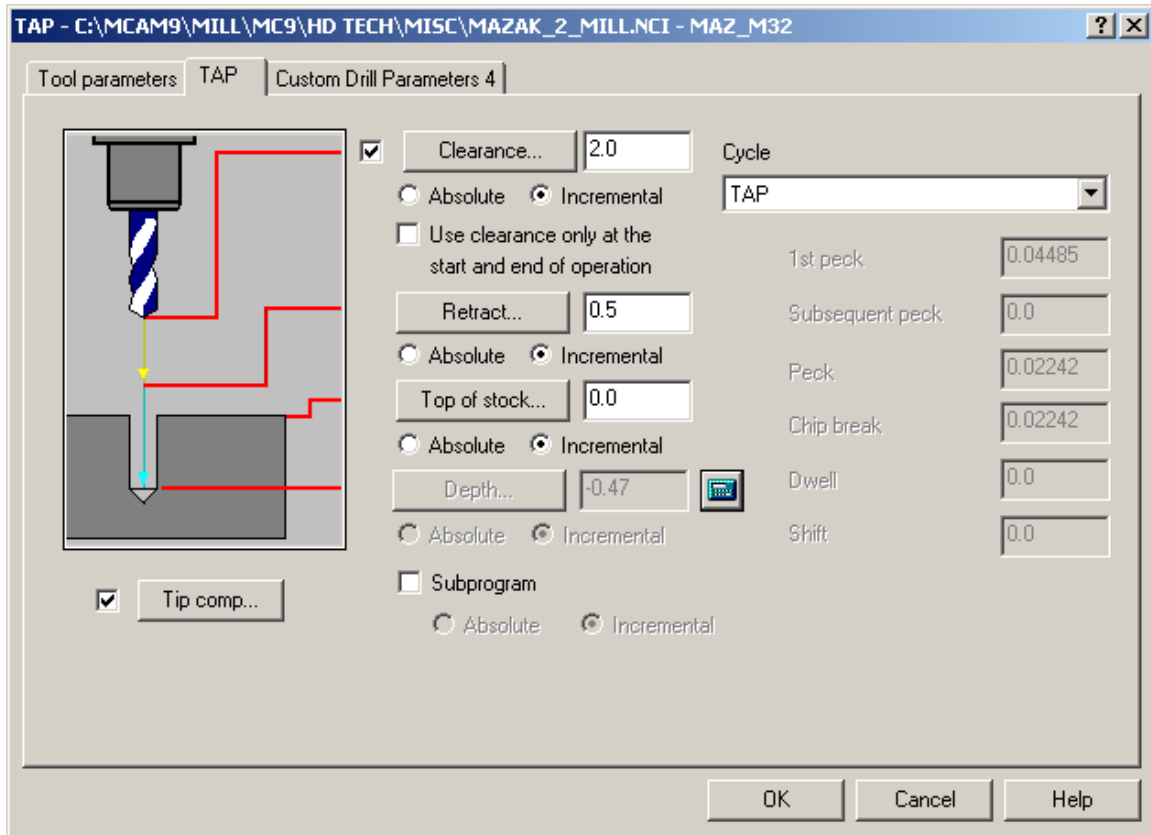
4. Then paste this operation so that it precedes the final tap operation.

5. Select Tool as shown.



As you have copied the operation within the group the Program # is still correct as shown.

6. Set **TAP** page as follows:



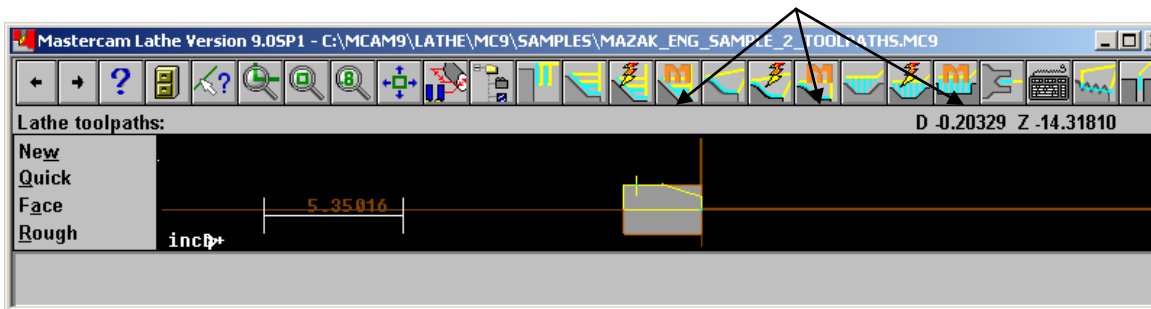
7. Select OK and REGEN path.

8. Save file and post to create Mazatrol Code.

Section 2 :Lathe

1. Programming a Basic Part.

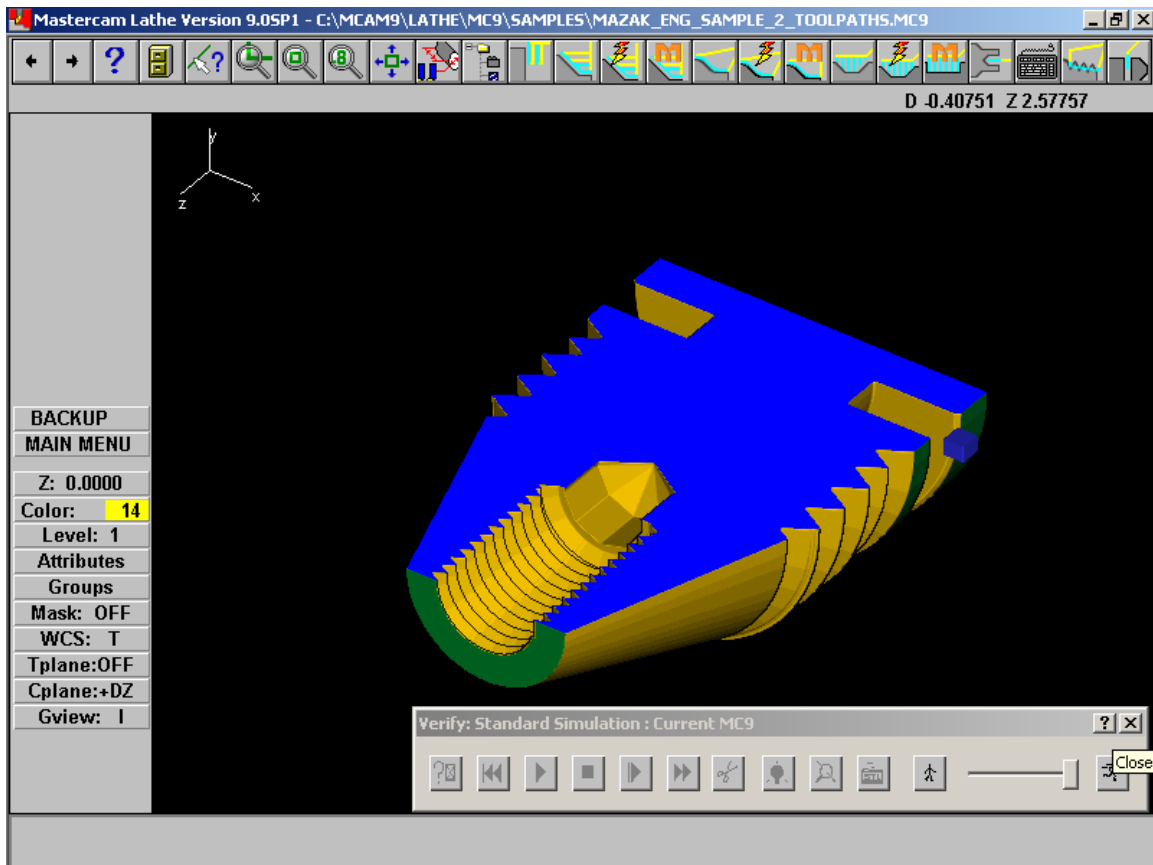
Mazatrol is designed to minimize the amount of information required to create a toolpath
 - therefore with this Post Processor interface we provide toolbar icons as seen below
 - these canned cycles are designated with the hopefully familiar ‘M’ for Mazak.



Obviously one can access these to create a toolpath but what is important to mention is the following: When programming for Mazatrol output use

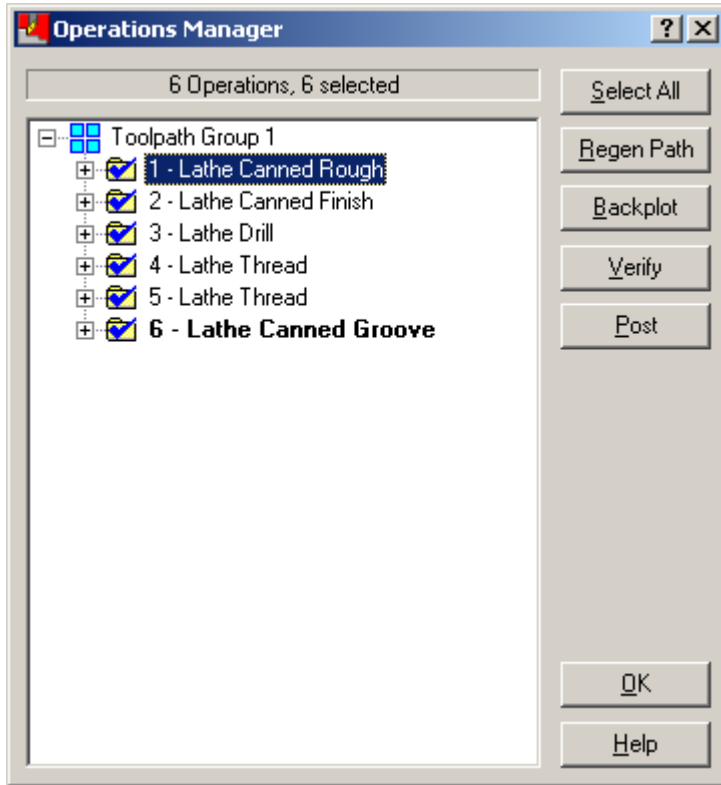
Mastercam	Mazatrol Output
Face	- EDG; FCE
Canned Rough	- BAR; IN, OUT, FCE, BAK Also some GRV
Canned Finish	- BAR; IN, OUT, FCE, BAK
Canned Groove	- GRV; IN, OUT, FCE, BAK
Thread	- THR
Drill	- DRL
Cutoff	- GRV

We will Rough and Finish outside profile, Drill and Thread ID then thread OD and final step will be grooving OD.



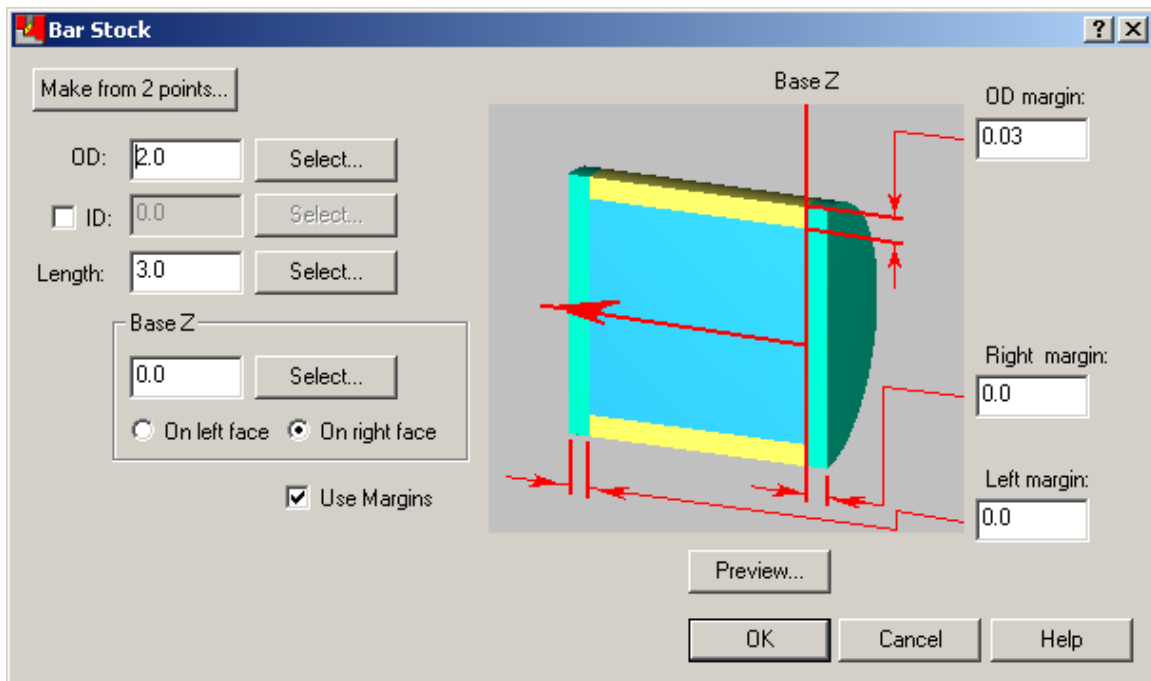
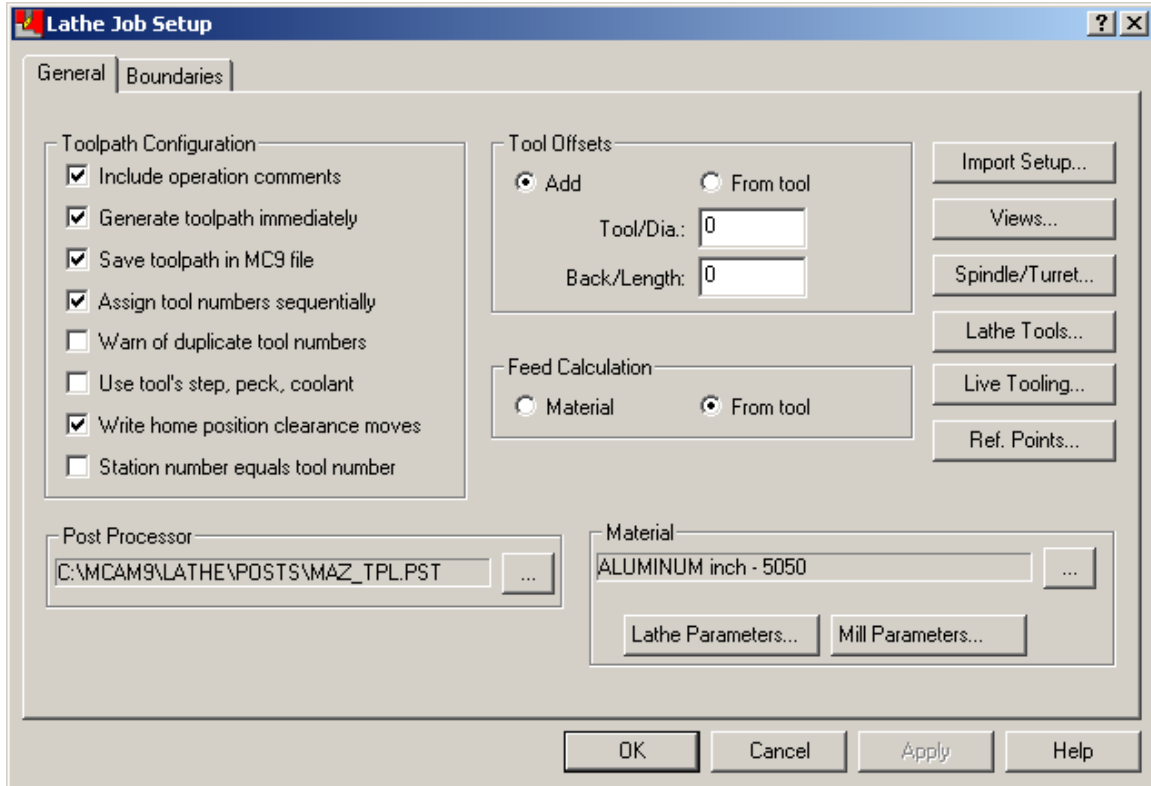
Finished part is shown verified

Operations List for Finished Part are listed below:

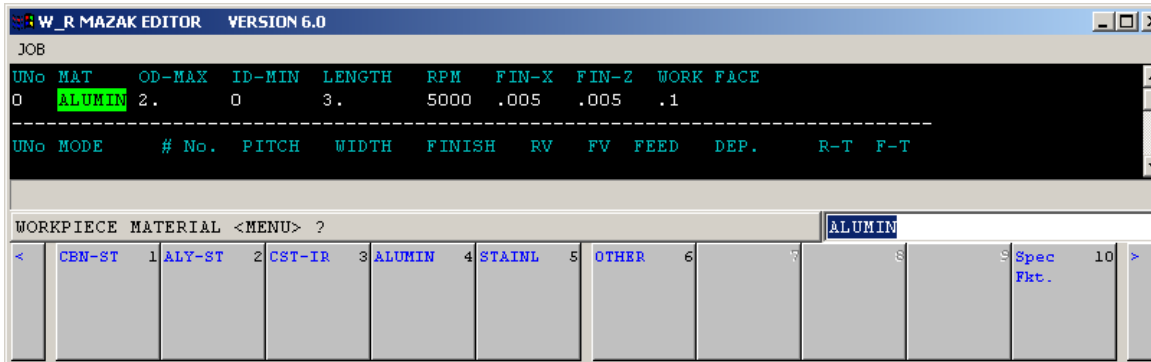


Exercise 1 - Opening the part file and Job Setup

1. Choose **Main Menu, File, Get**
2. Navigate to the folder with the tutorial parts.
3. Select **Mazak_ENG_Sample_2.mc9**; then choose **Open**.
4. Choose, **Main Menu, Toolpaths, Job Setup**
5. Enter settings as shown.



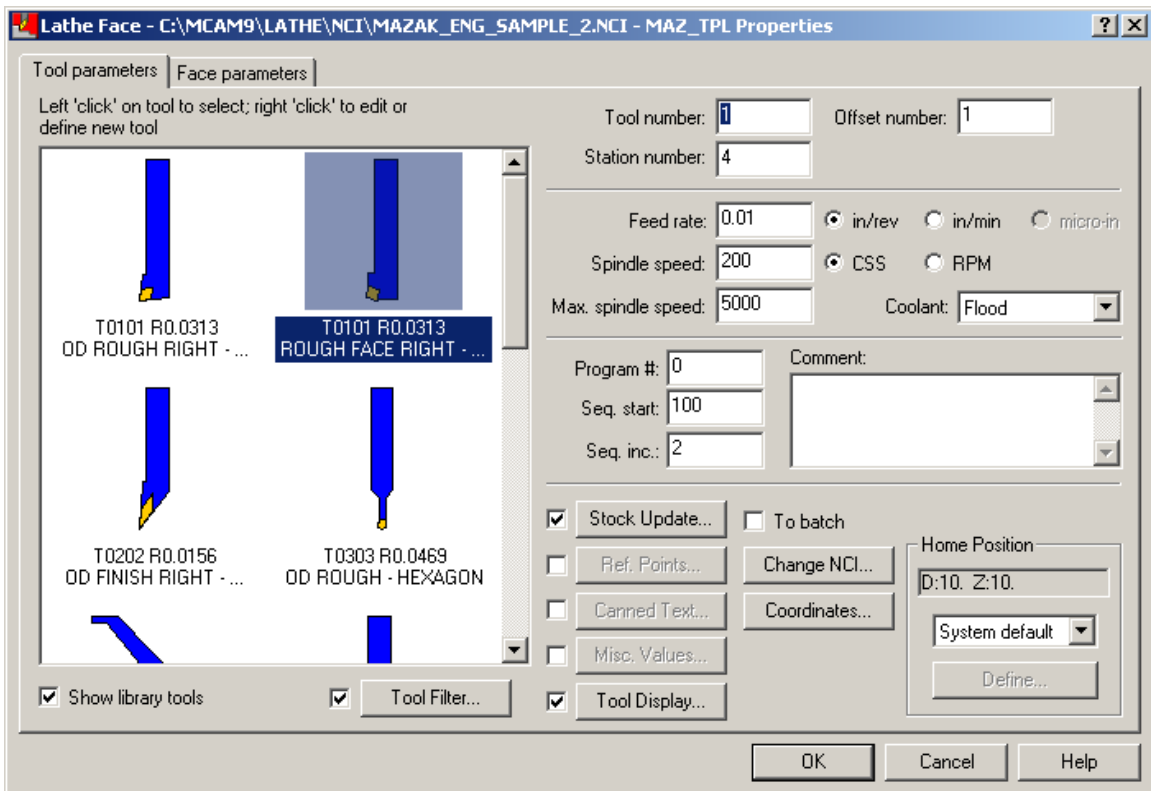
Note: Job Setup settings will affect the first line of the mazatrol UNO 0 and MAT data i.e. the material selected will be output and the OD will be OD-Max and Length will be Length will be based on the values entered in the job setup see below:

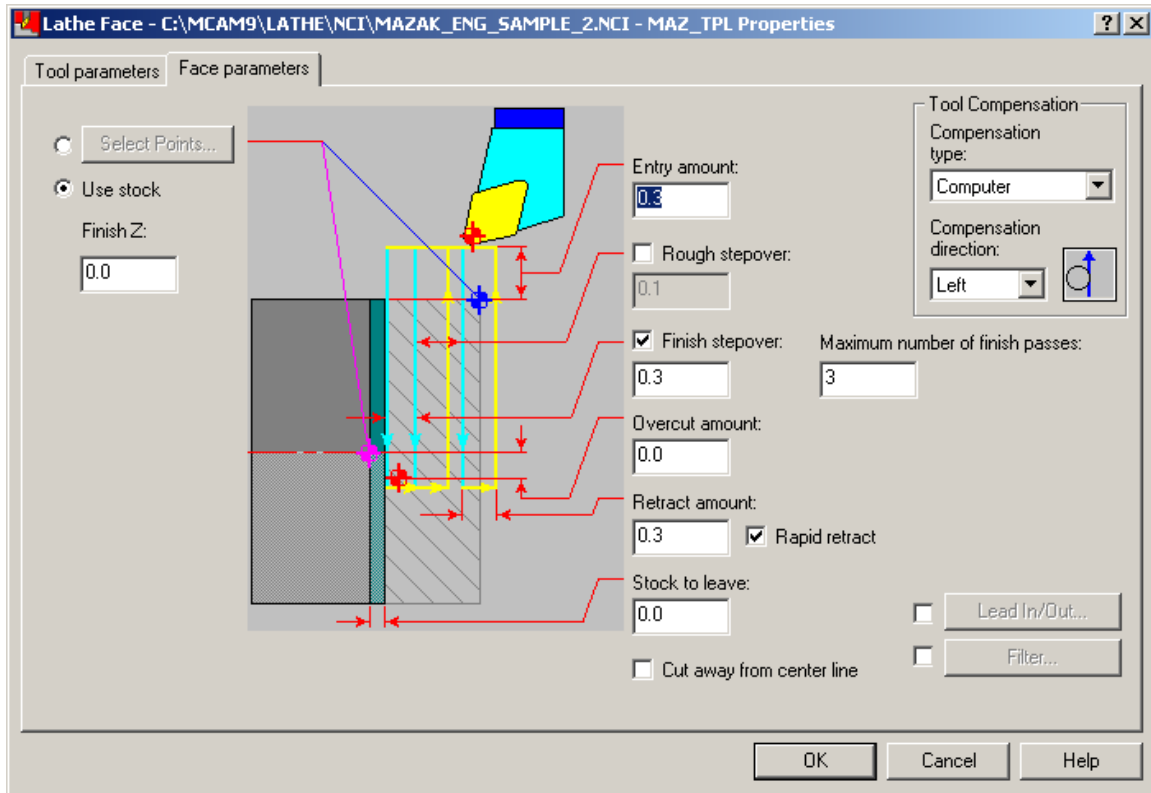


The other settings will have to be manually entered by the user if desired either using the editor (if available) or at the control.

Exercise 2 - Creating Facing Toolpath

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

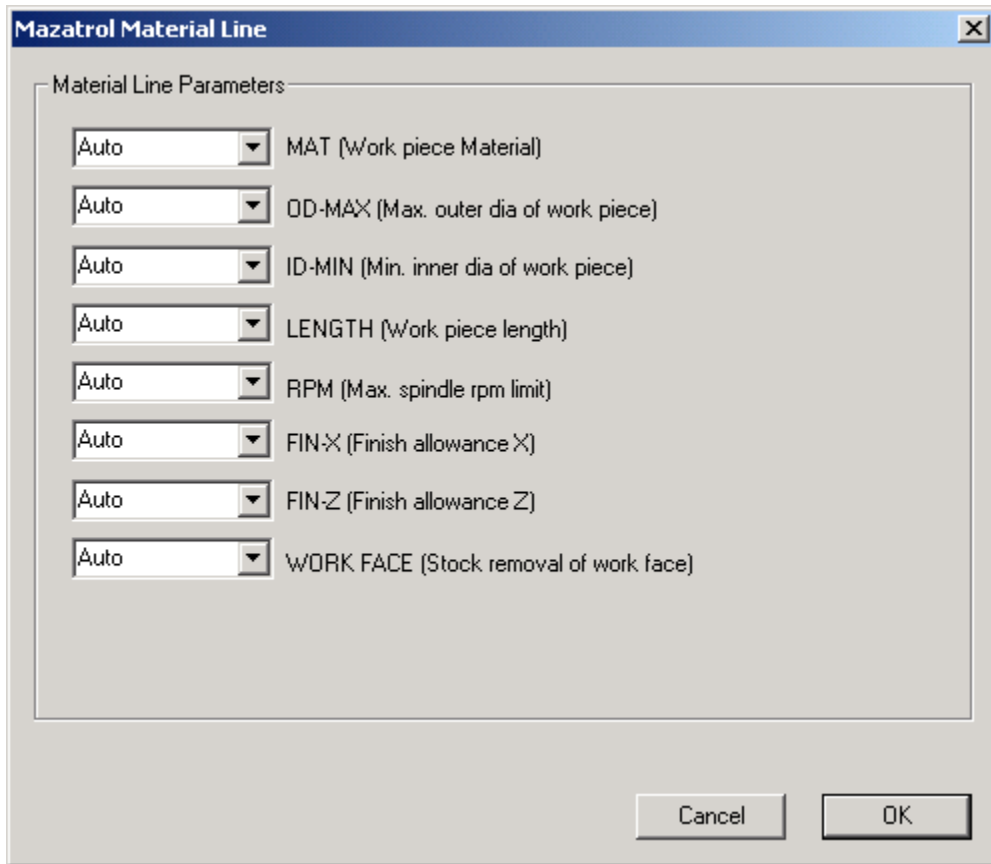




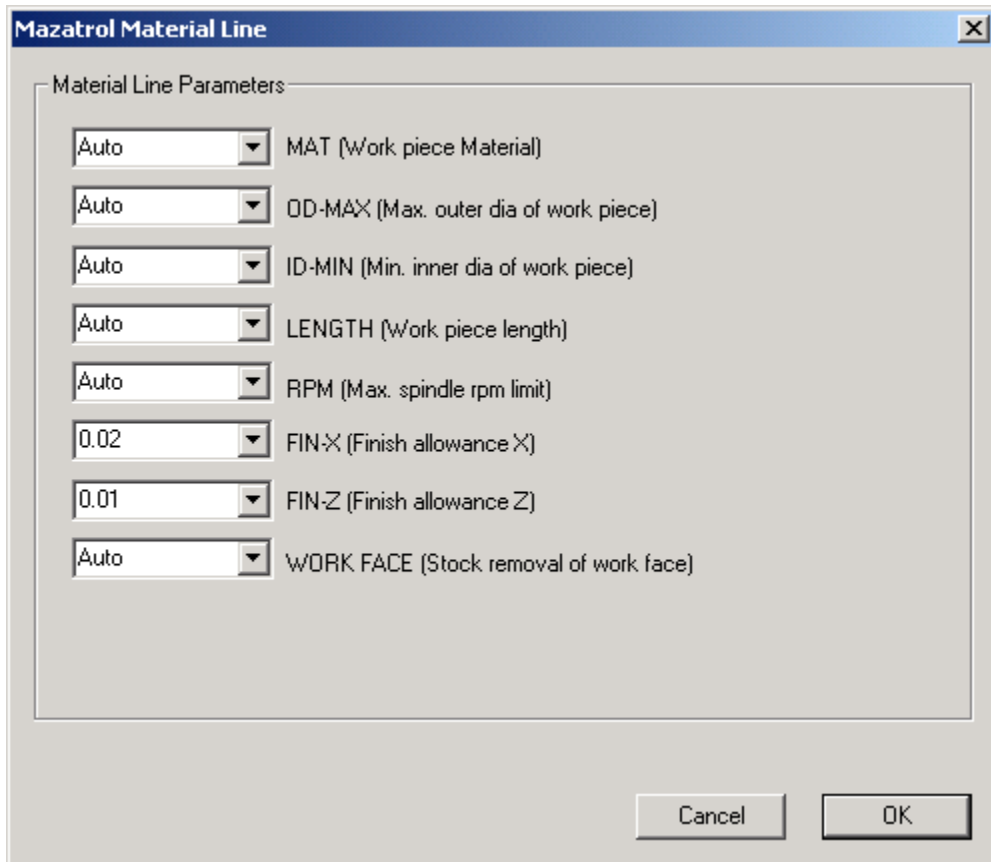
Important note: in Version 8.0.8 we introduced the access to the material line i.e. the first line of the Mazatrol Program where the size and material of the stock are defined. These values in some cases such as the length and diameter will overwrite previously defined values in the Job Setup. See below:

Go back to the first parameter page of the Facing Toolpath and click on **Misc. Values**.

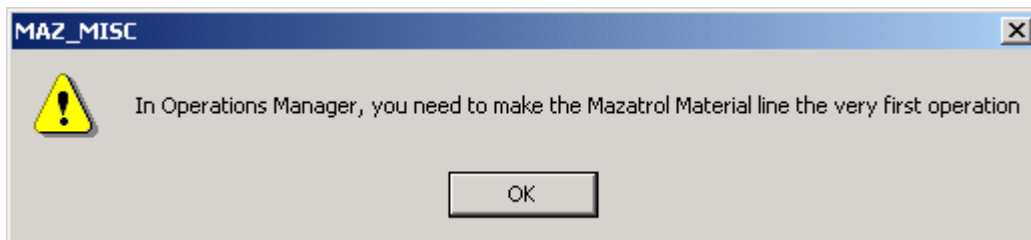
Then click on the **Material Line**. You should then see the following display:



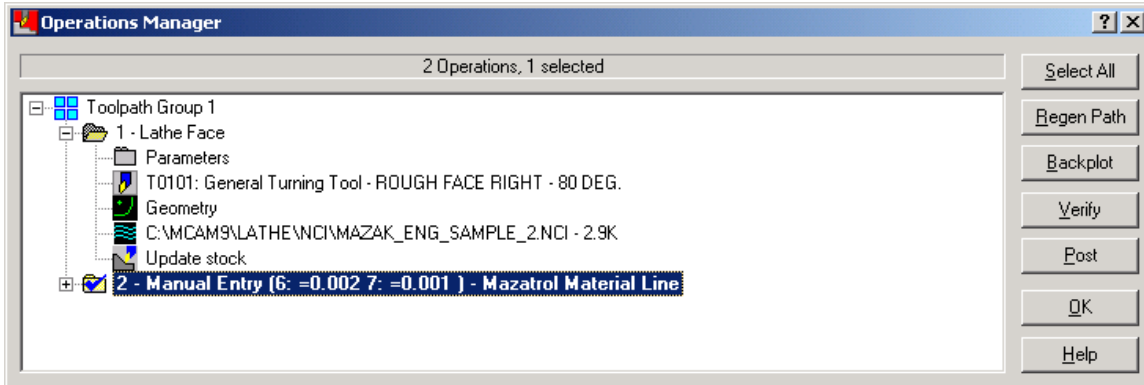
Let's change the finish allowance values to FIN-X = 0.02 and FIN-Z = 0.01 by entering in the dialog as shown below:



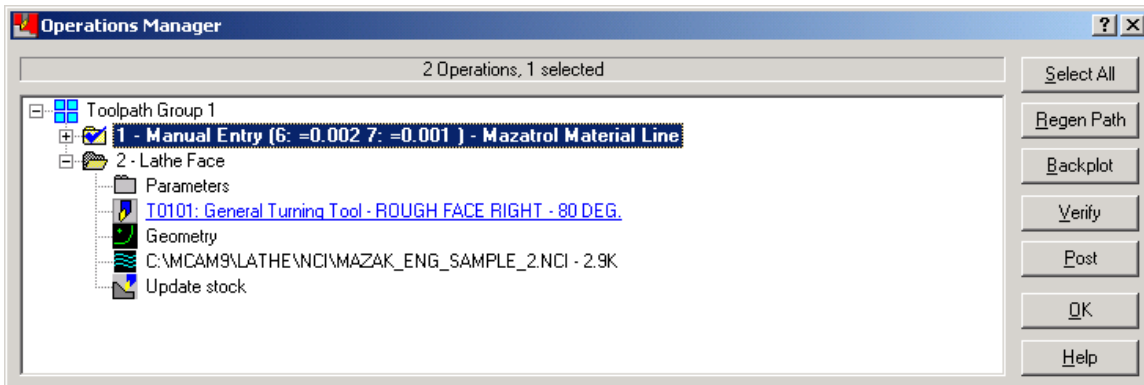
When you click OK you will get the following message



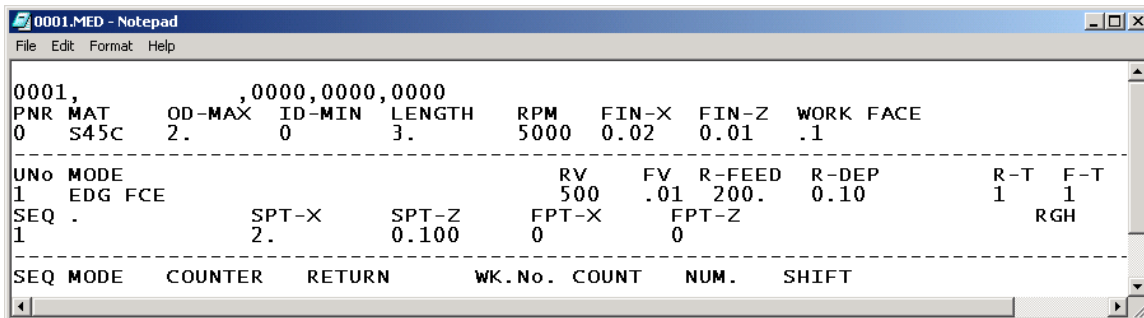
Click OK and in the Operations Manager Move this newly created **Manual Operation** to be the first operation.



Move to here



Sample output for material line and facing :

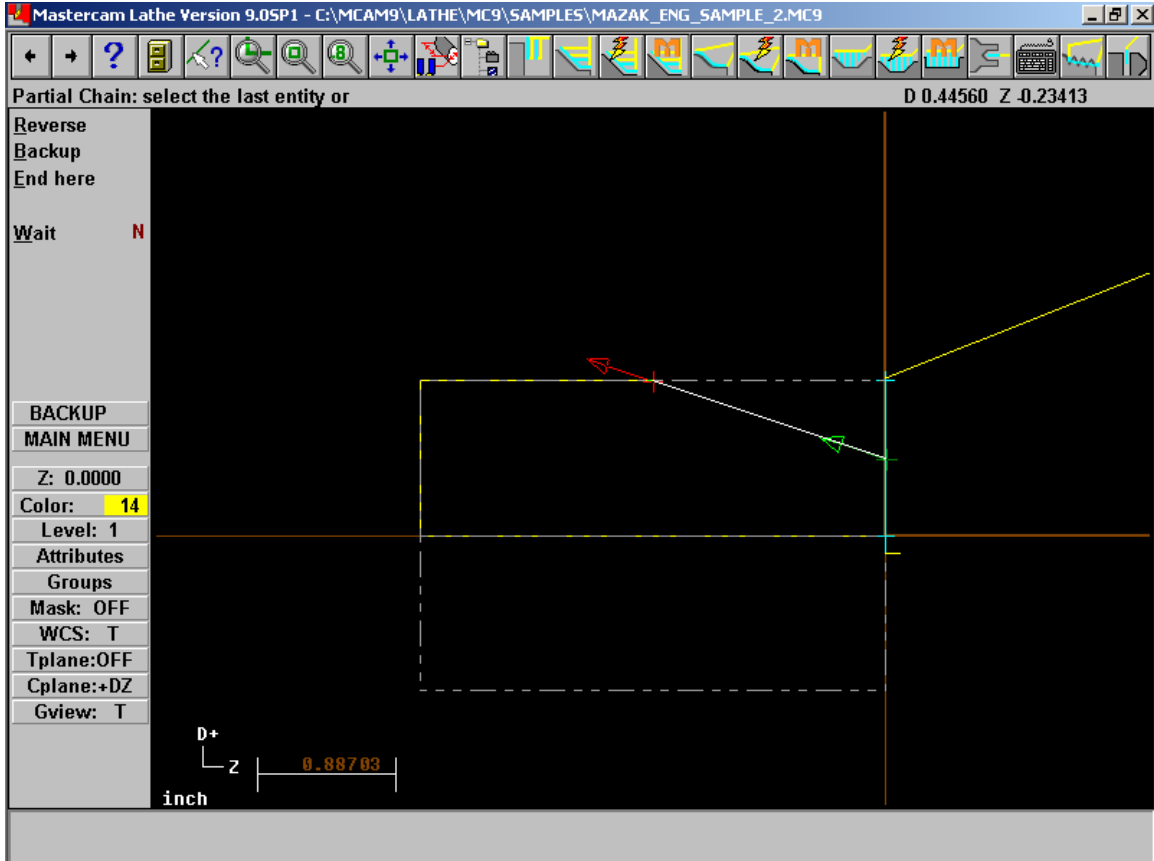


Exercise 3 - Creating Rough and Finish Toolpath

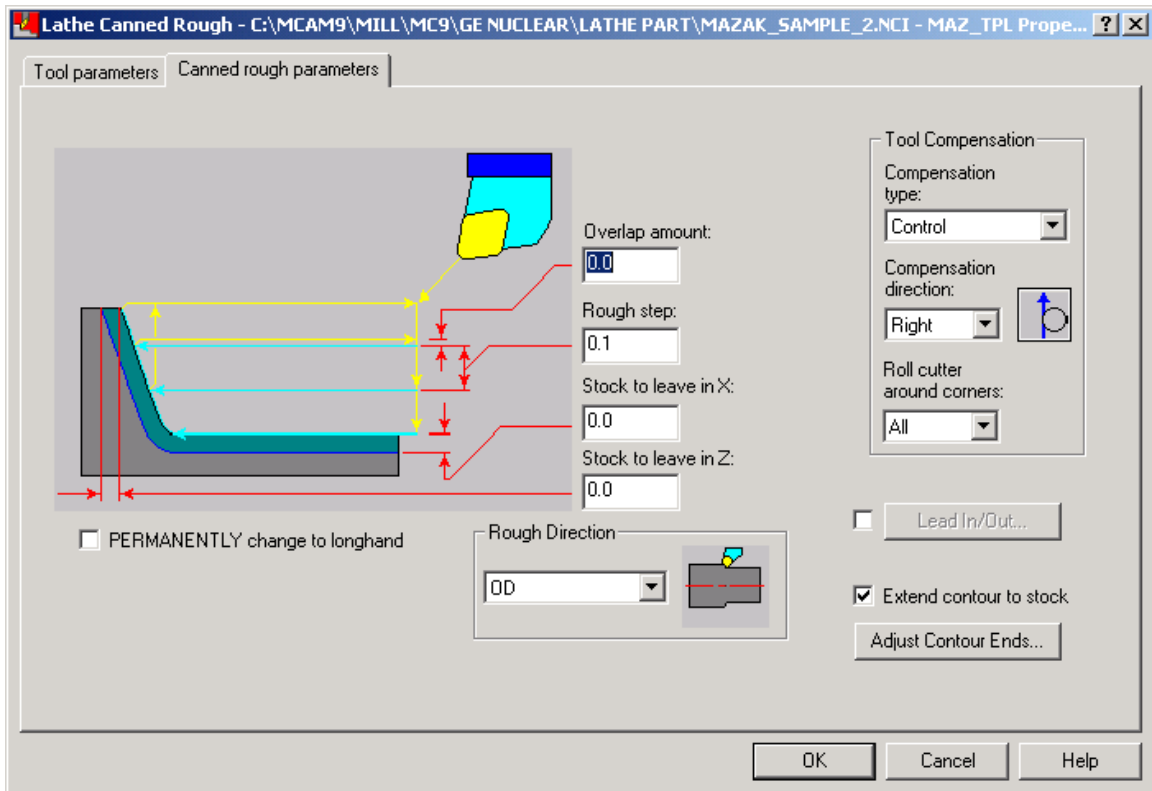
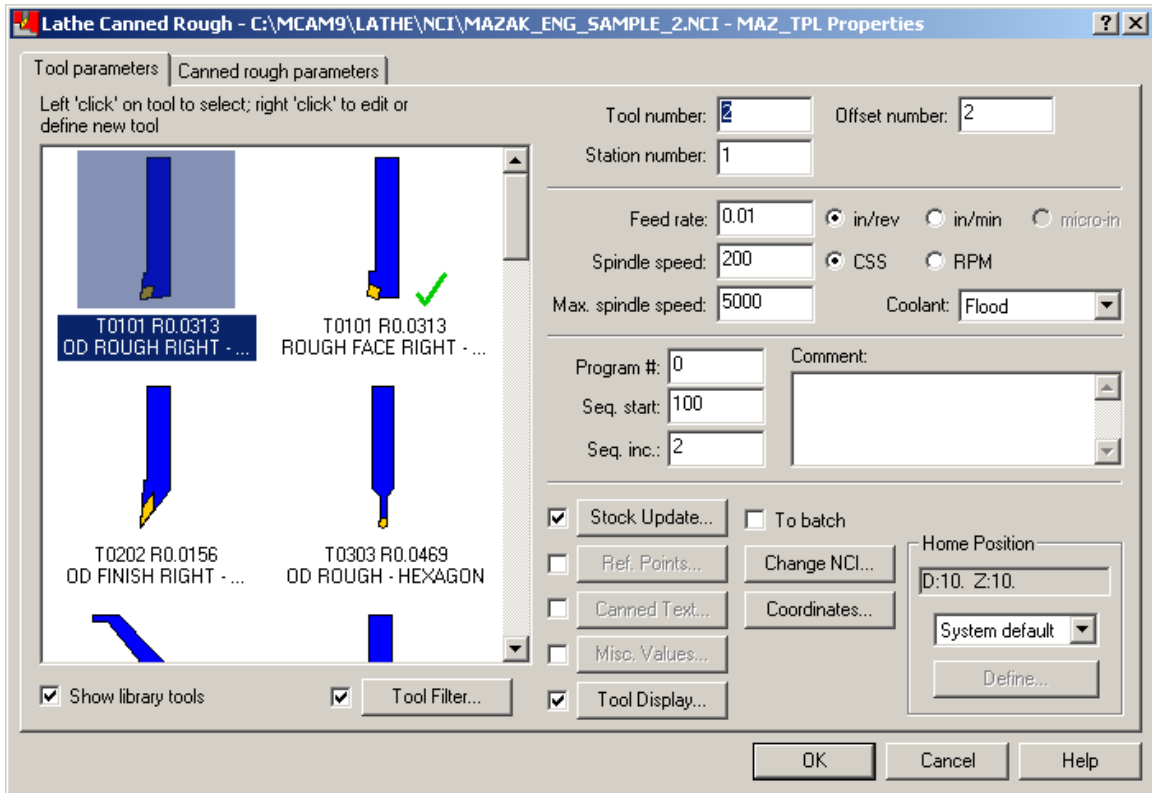
Select the following:

1. **Main Menu**
2. **Toolpaths**
3. **Canned Rough**

4. Select **Chain** as Shown Below



5. Set Parameters as in the following **Roughing Param. Pages**.



6. Click on **OK** when completed

If we were to post for output now, we would get window as shown below. We provide this to illustrate our progress. When we have completed the complete part program we will then document how to run the post and then send the program to the control.

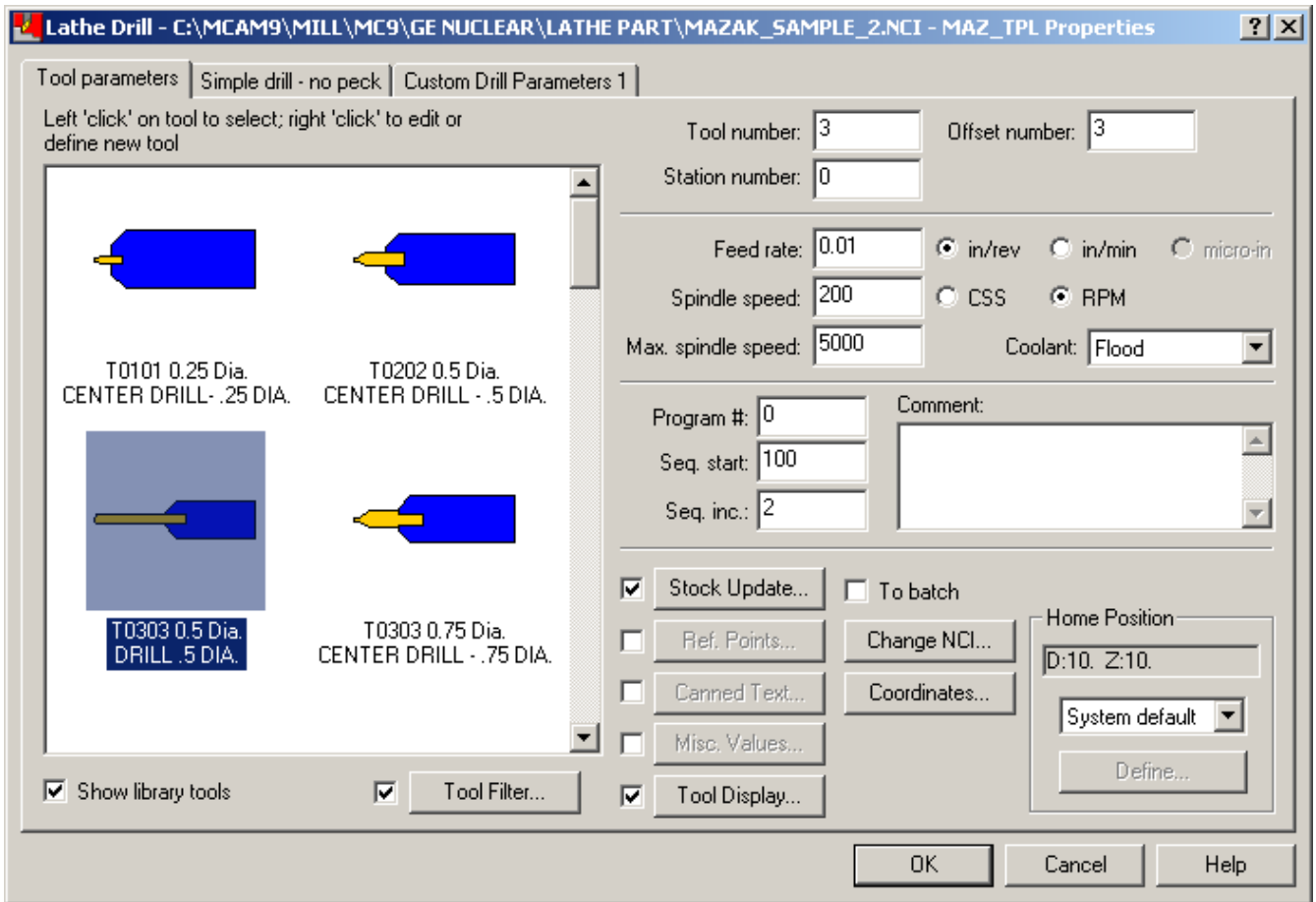
```

0001, ,0000,0000,0000
PNR MAT OD-MAX ID-MIN LENGTH RPM FIN-X FIN-Z WORK FACE
0 ALUM 2. 0 3. 5000 0.02 0.01 .1
-----
UNo MODE RV FV R-FEED R-DEP R-T F-T
1 EDG FCE 500 200. .01 0.10 1 1
SEQ . SPT-X SPT-Z FPT-X FPT-Z RGH
1 2. 0.100 0 0
-----
UNo MODE # CPT-X CPT-Z RV FV R-FEED R-DEP R-T F-T
2 BAR OUT 0 2. 0 150 200. .01 .1 1
SEQ SHP S-CNR SPT-X SPT-Z FPT-X FPT-Z F-CNR RADIUS RGH
1 LIN * * 1. 0 1. 0
2 TPR 1. 0 2. 1.5
-----
SEQ MODE COUNTER RETURN WK.No. COUNT NUM. SHIFT
3 END 1 1 1 1
    
```

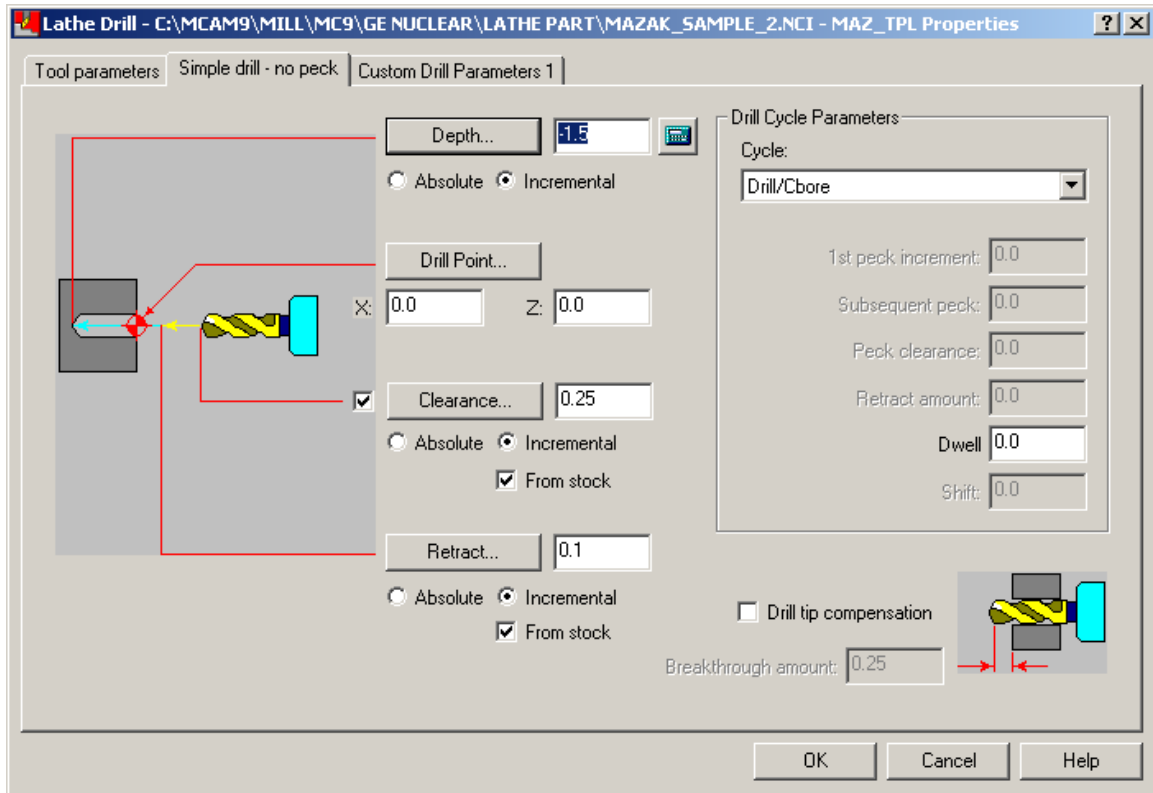
Exercise 4 - Creating Drill Toolpaths

To create Drill Toolpaths select the following:

1. **Main Menu**
2. **Toolpaths**
3. **Drill** - set Param. Pages as shown on the following pages



2. Select **Tool** as shown
3. Set next page as shown.

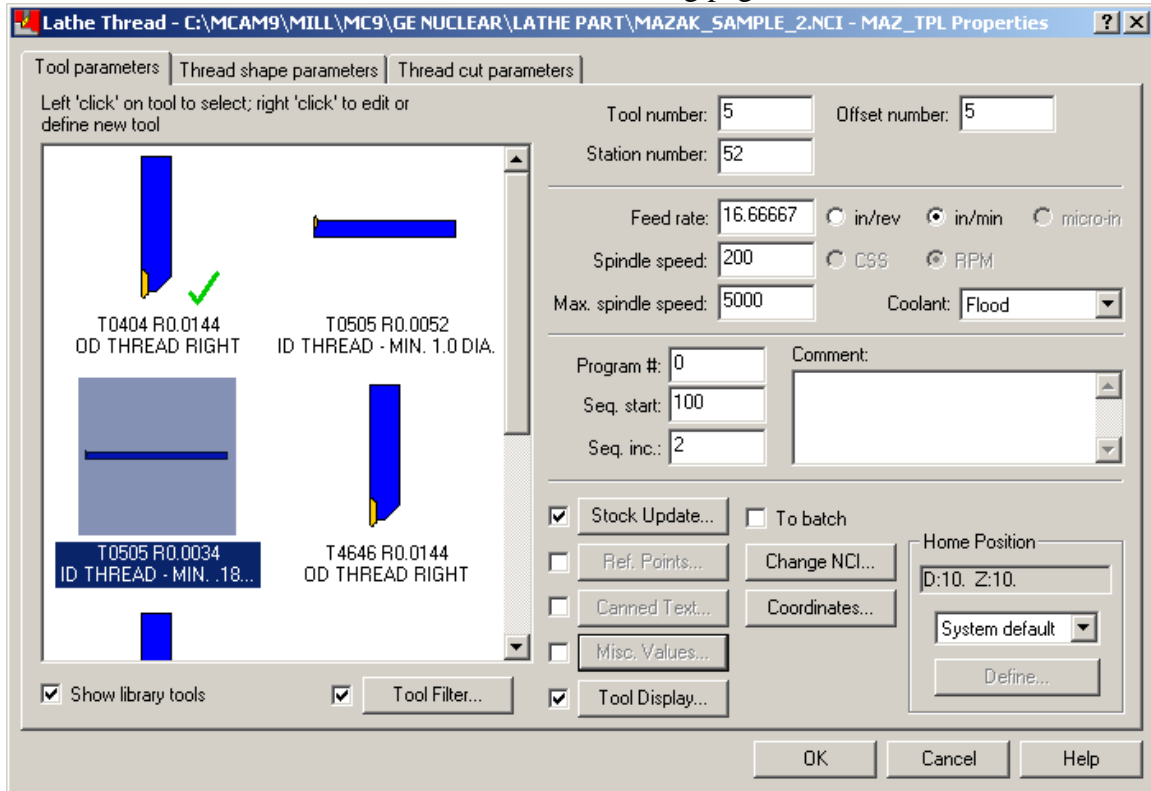


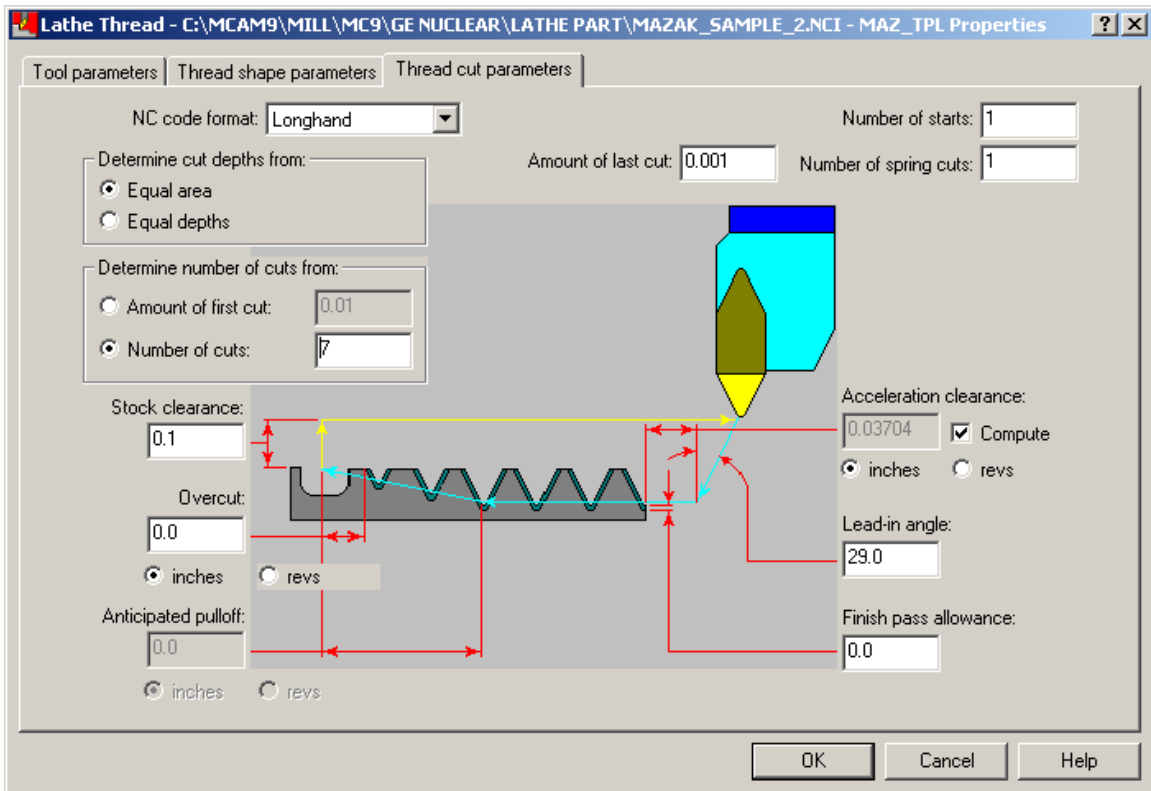
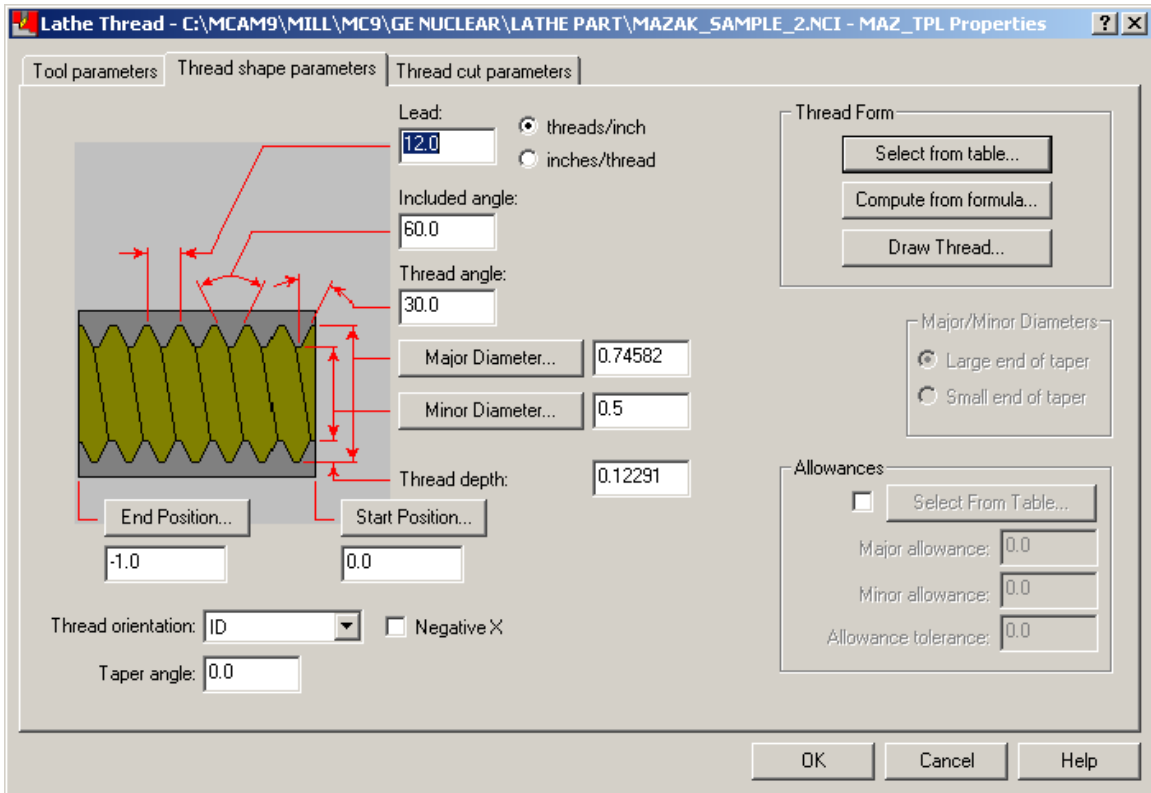
4. Click on **OK** when done
5. Set **Misc. Values** as shown

Exercise 5 - Threading Toolpaths

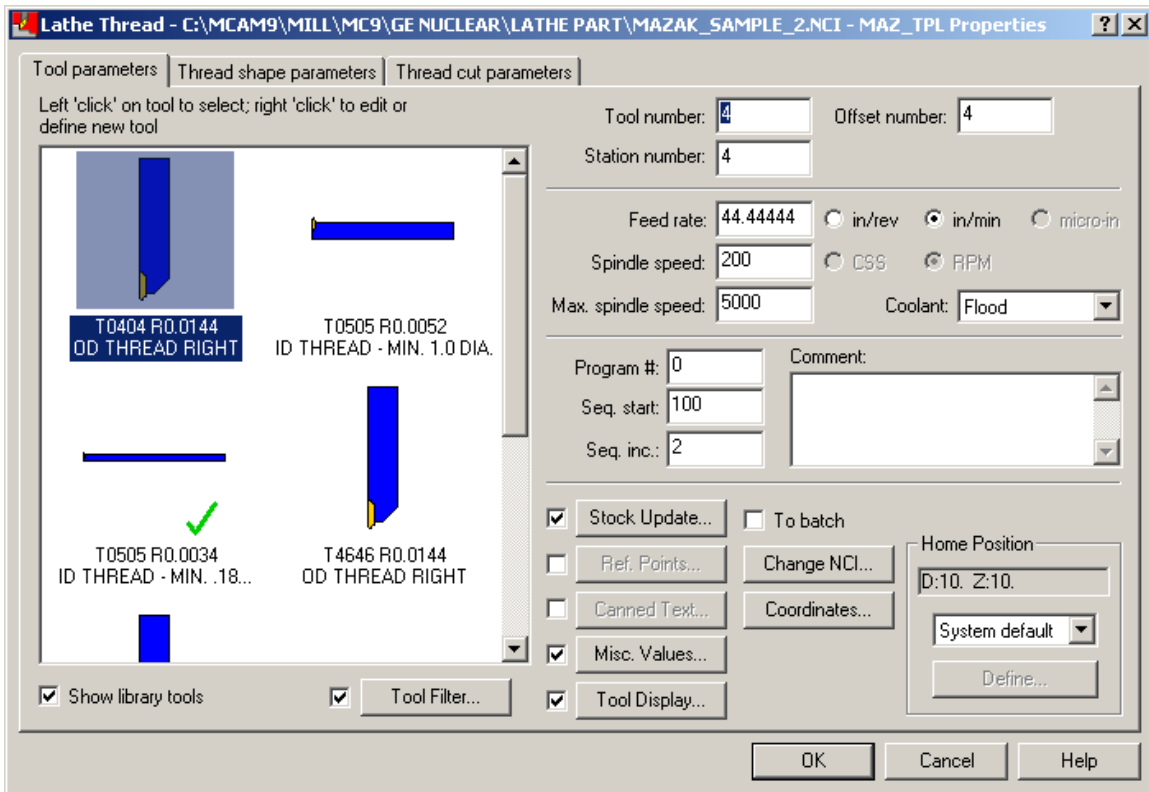
We will now Thread the ID

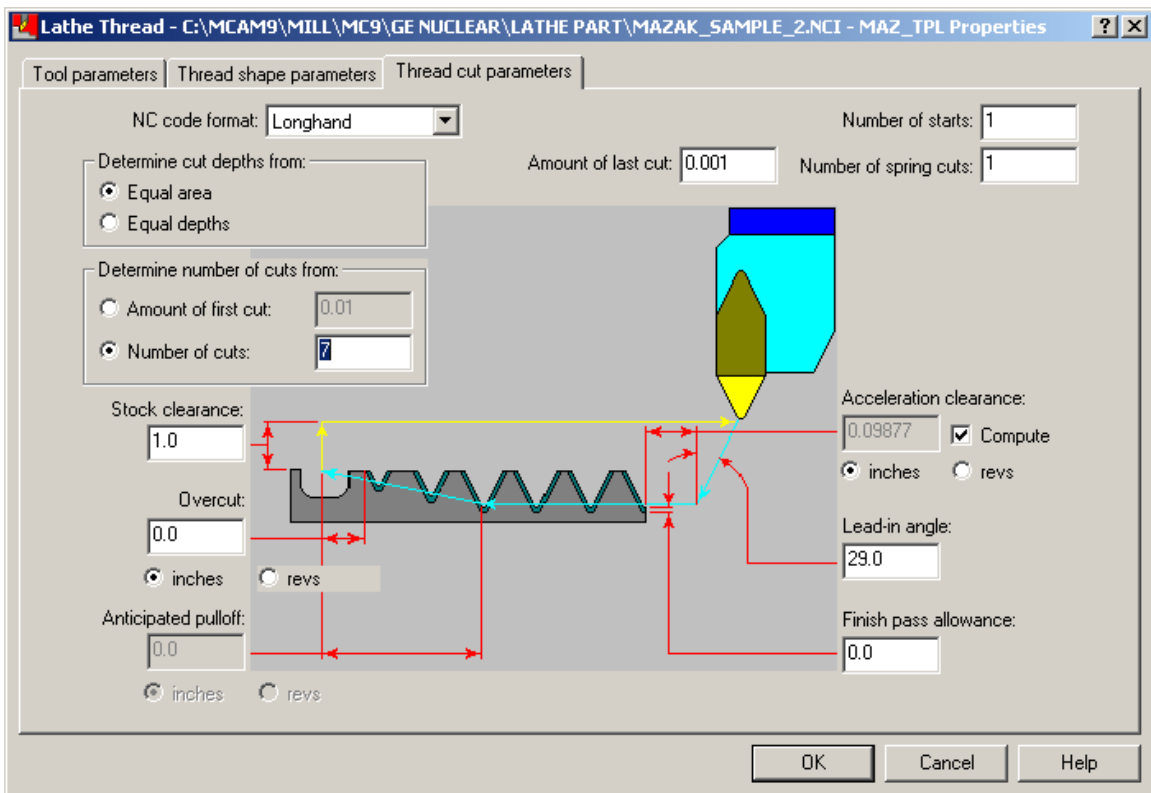
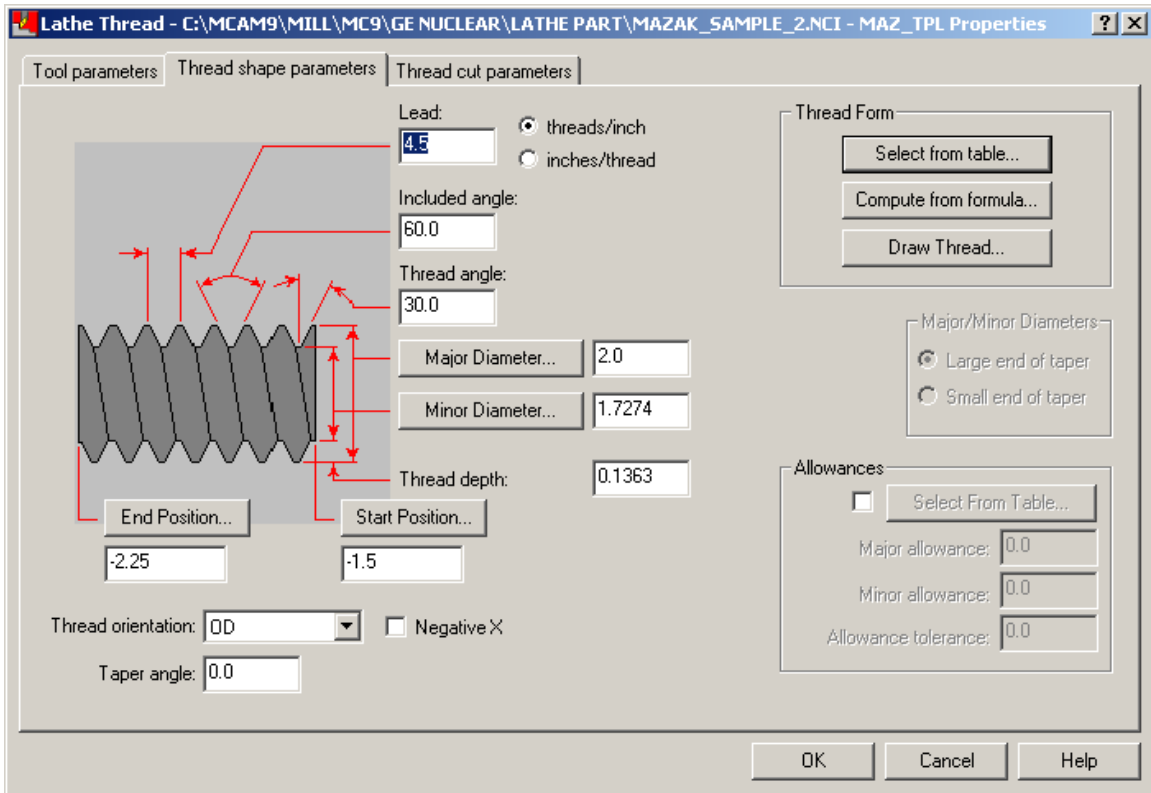
1. Set **Thread Parameters** as shown on the following pages.

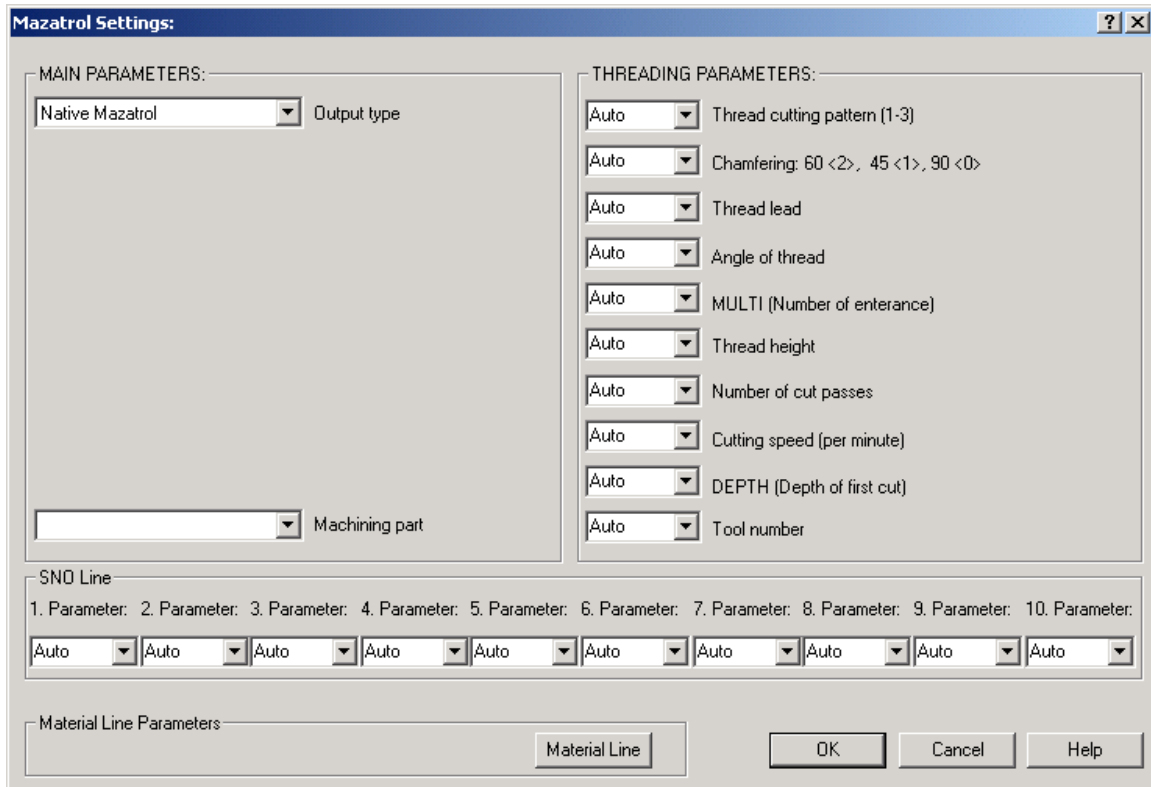




We will now Thread the OD, set the Parameters as shown below







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.

Select **OK** when done.

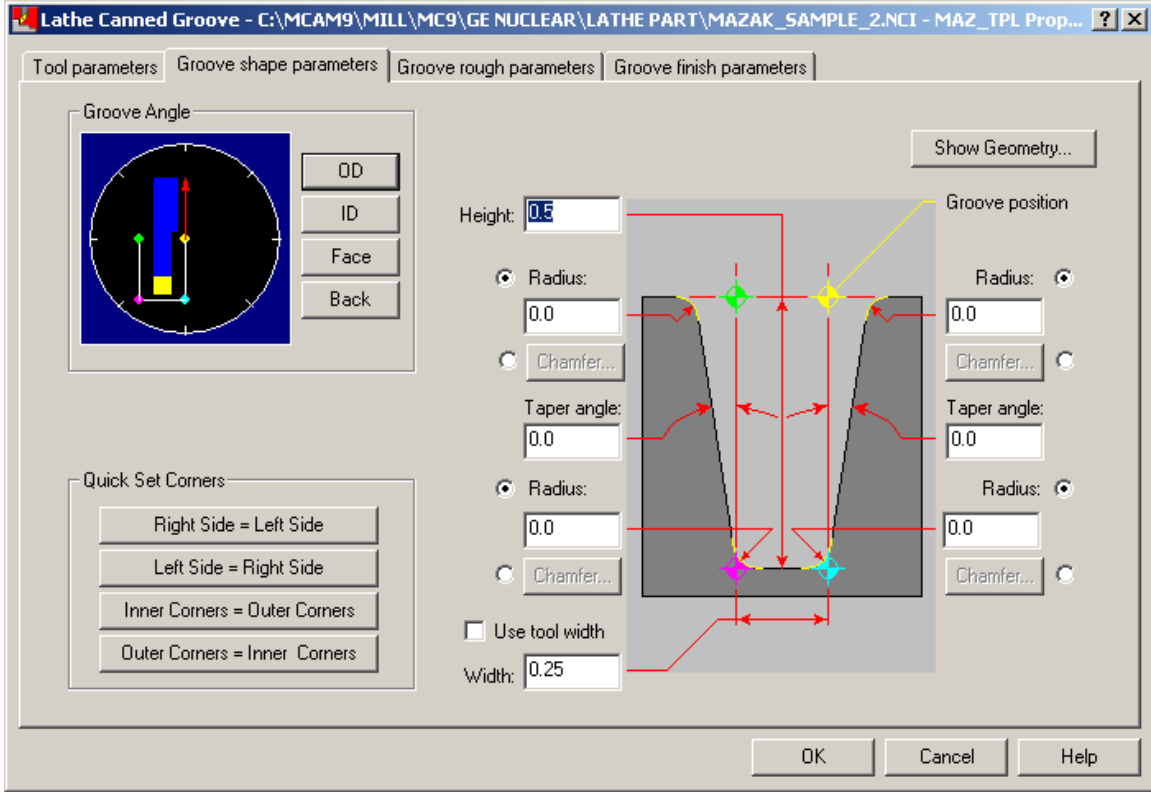
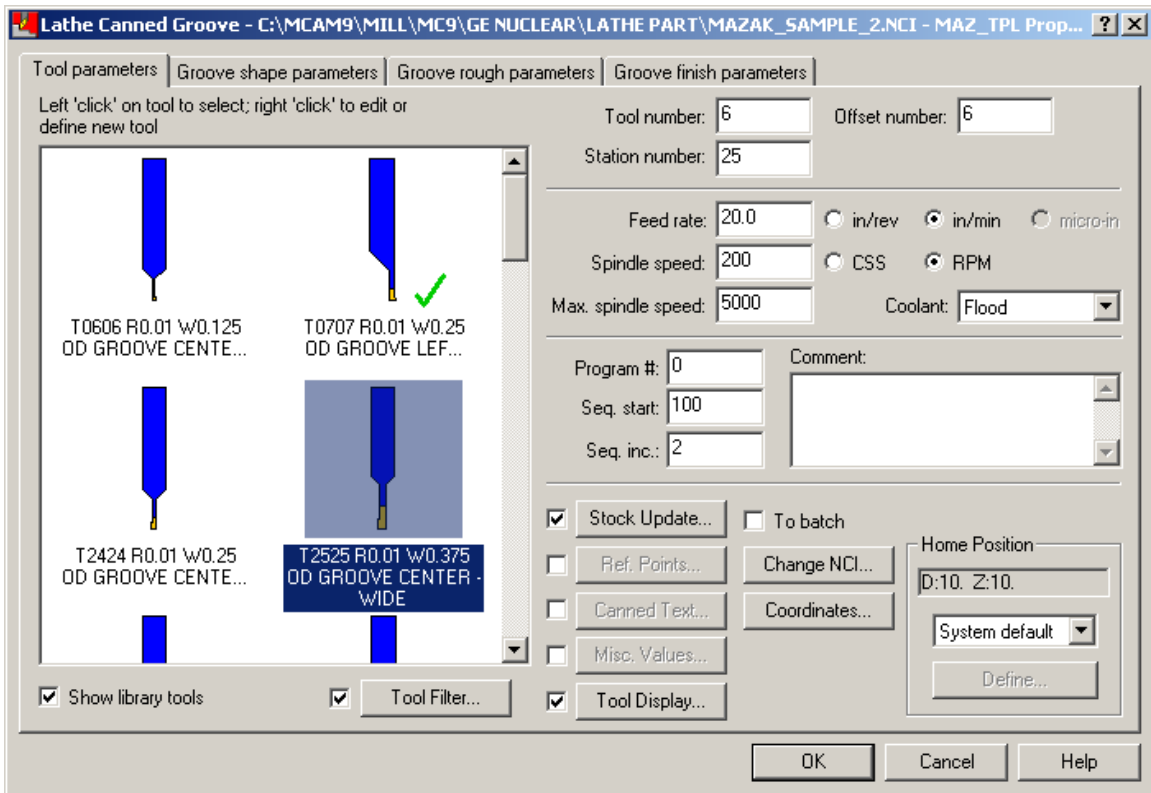
Exercise 5 - Creating Groove Toolpath

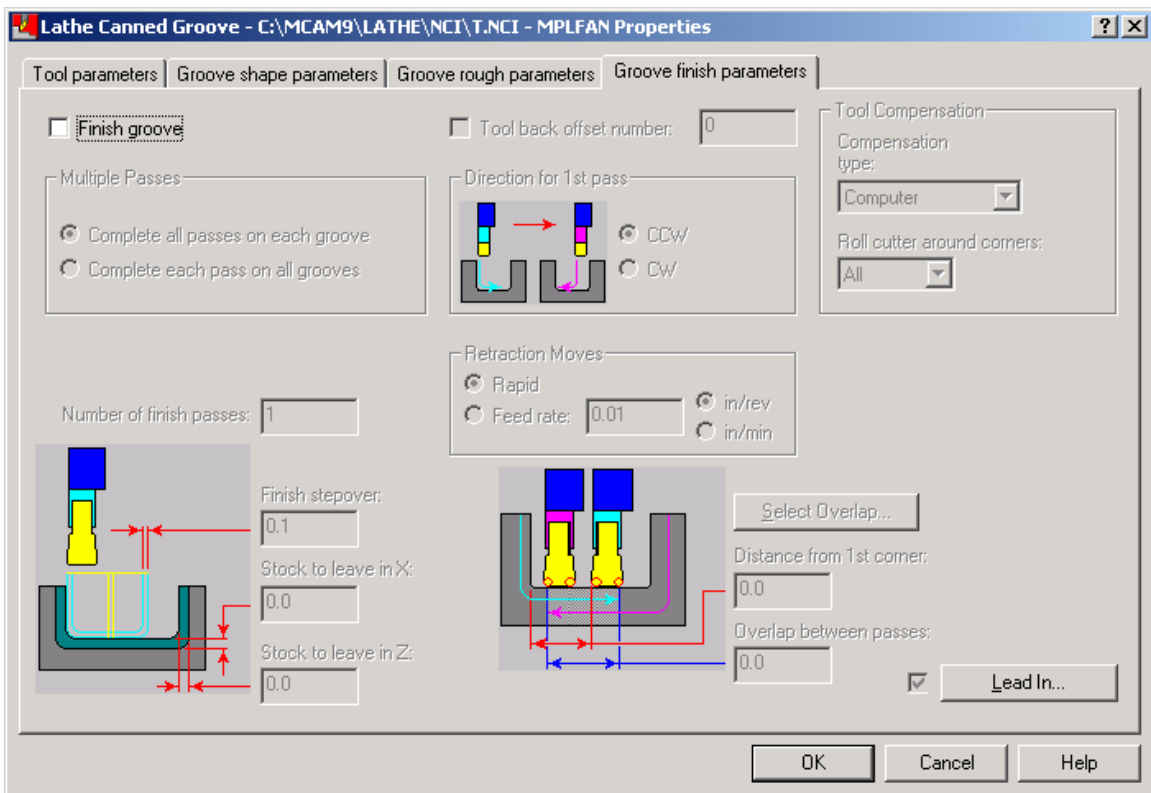
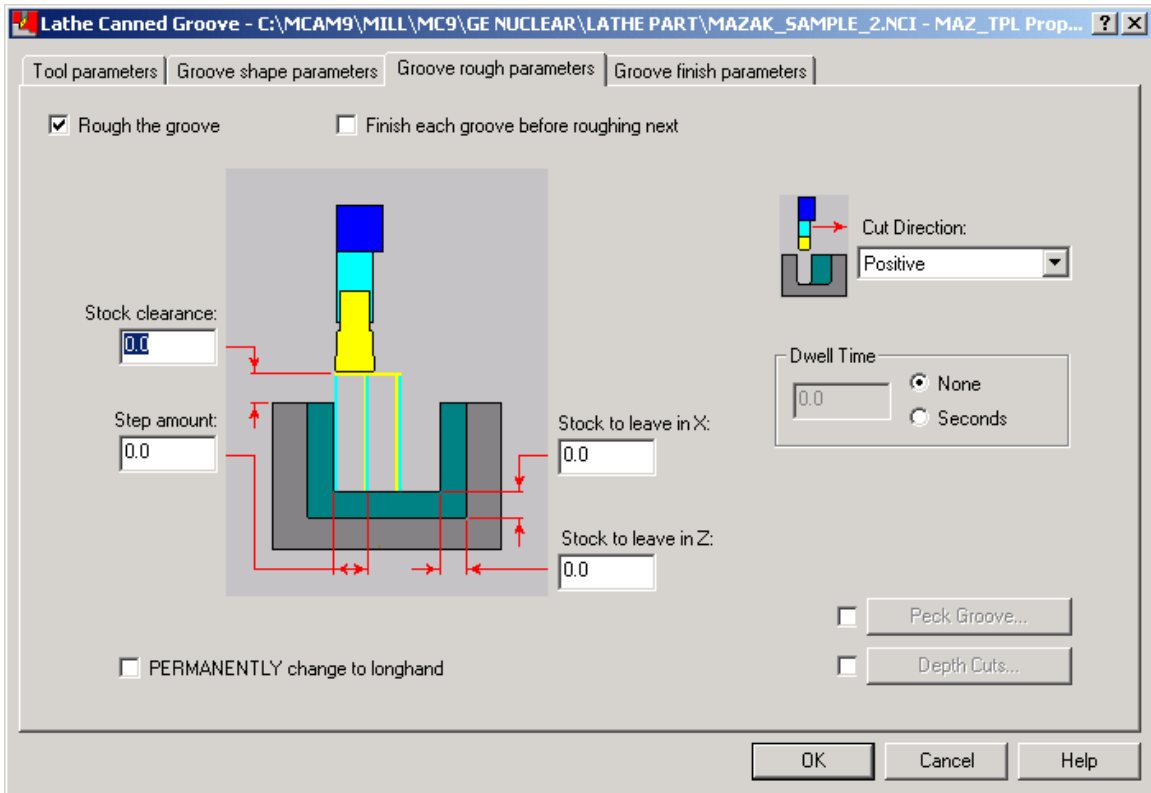
As with the Roughing and Finishing toolpaths it is generally unnecessary to have both a rough and a finish operation programmed in mastercam to get the correct output in Mazatrol

We will now create a 1 point groove on the OD.

Select the following:

1. **Main Menu**
2. **Toolpaths**
3. **Canned Groove**
4. **Select 1 pt**
5. Click pt shown on OD
6. Set Parameter Pages as shown



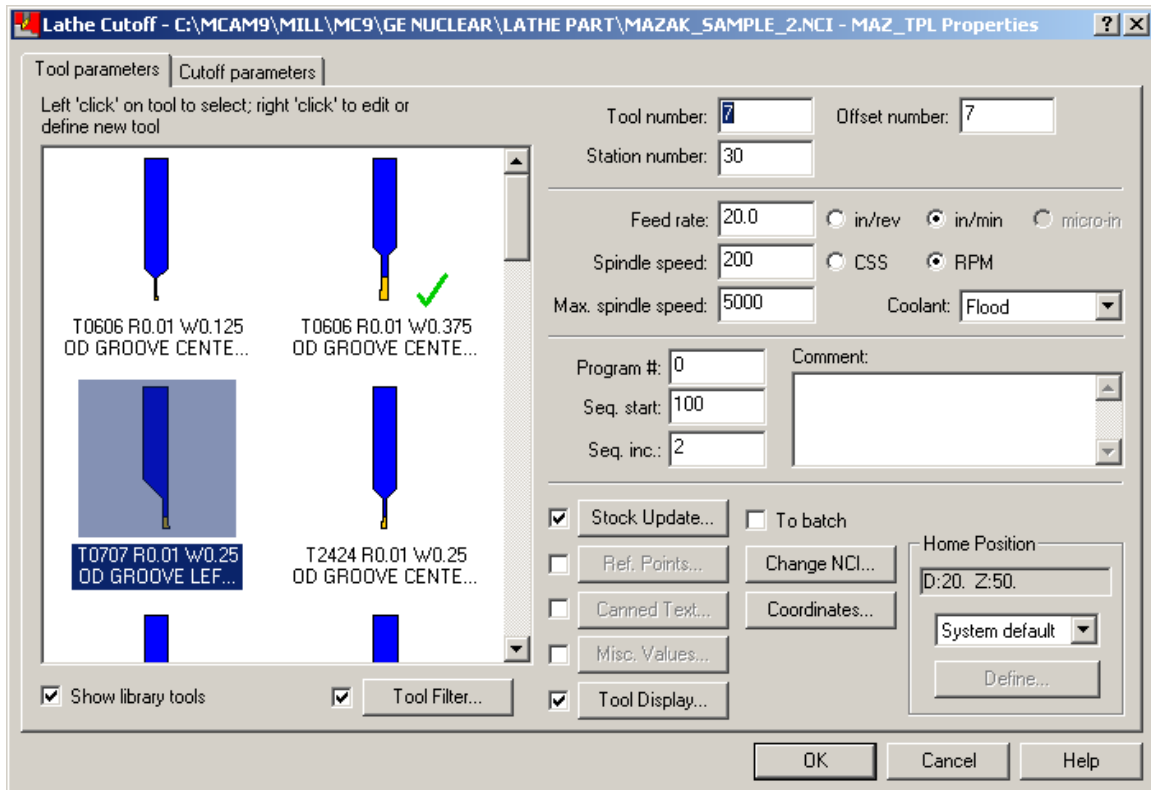


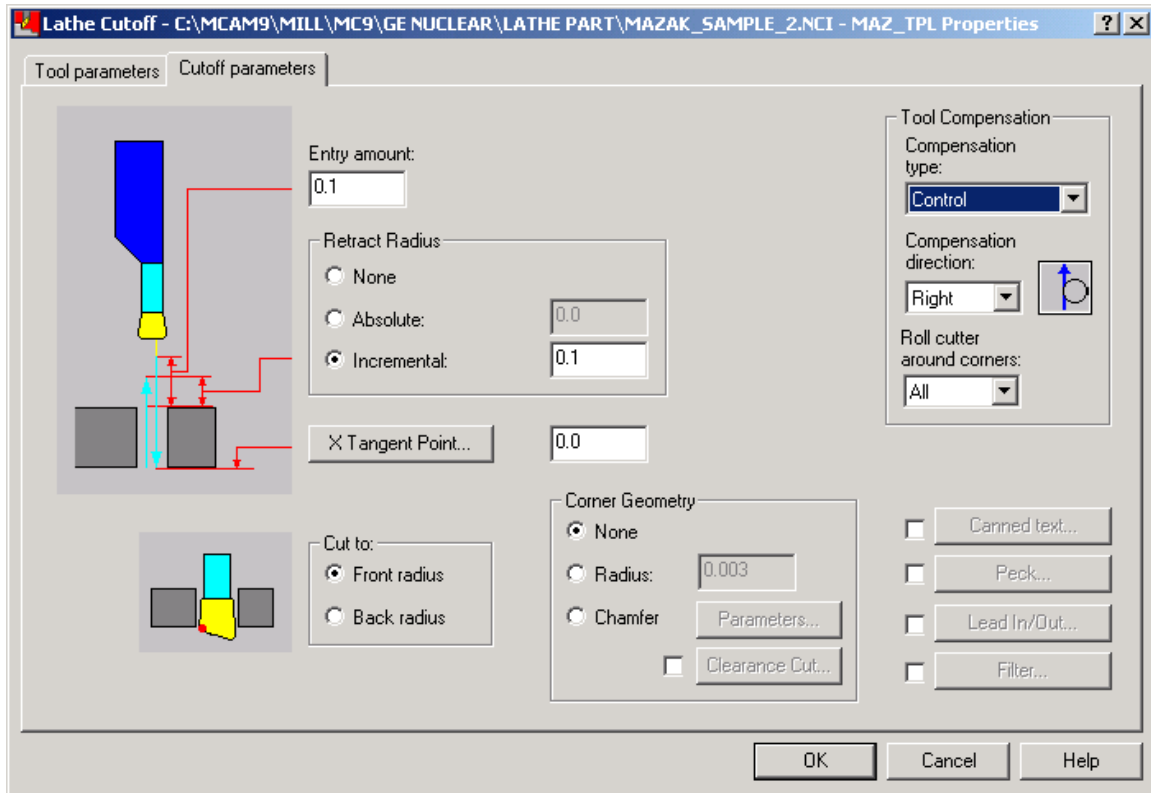
We do not need to select a finish for grooving. So set as shown above

Exercise 6 - Creating Cutoff Toolpath

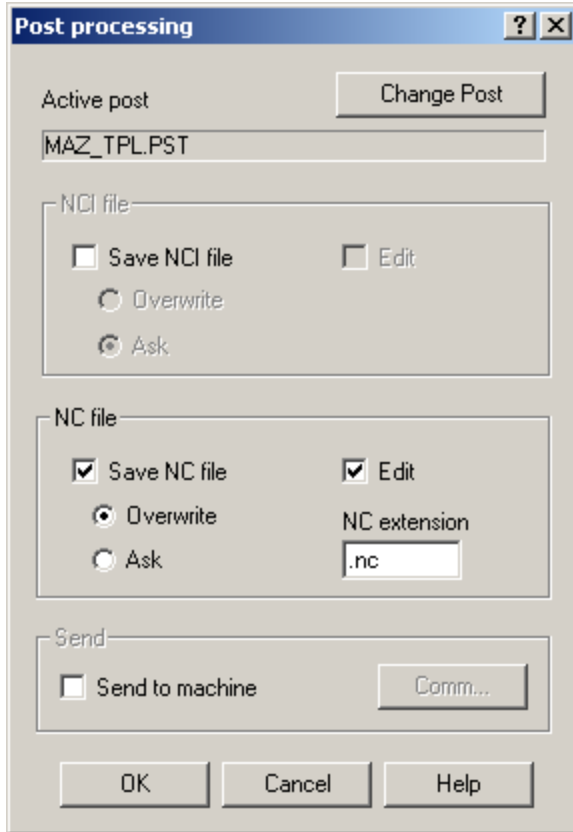
We will now finish the part by creating a cutoff operation.

Set Parameter Pages using Cutoff Toolpath





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 TPlus post-processor shown as MAZ_TPL.PST. Yours may vary but all the Mazatrol Post-Processors will have the format of MAZ_XXX.PST)

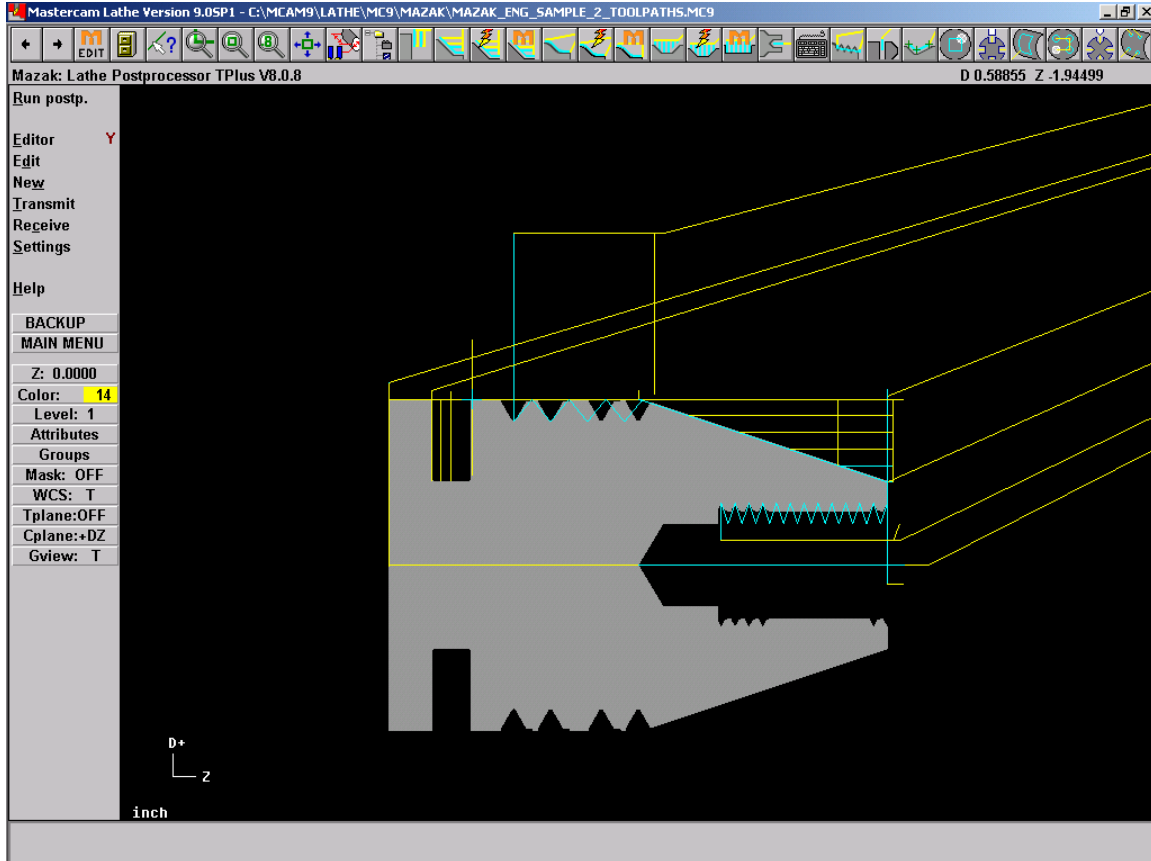


7. Select **OK**. The file name dialog should then appear as shown below:

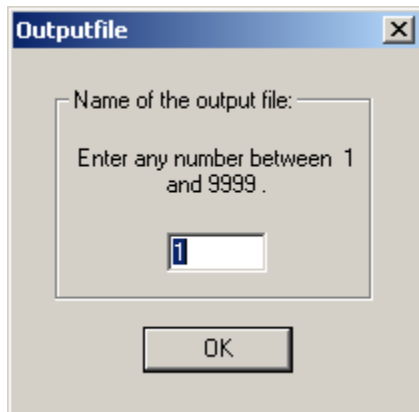
Note: We do not need to create an NC file but Mastercam needs to have this setting so that the post-processor can function

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 **Yes** in the Mazatrol Menu the output will open up in the Mazatrol Editor)

```

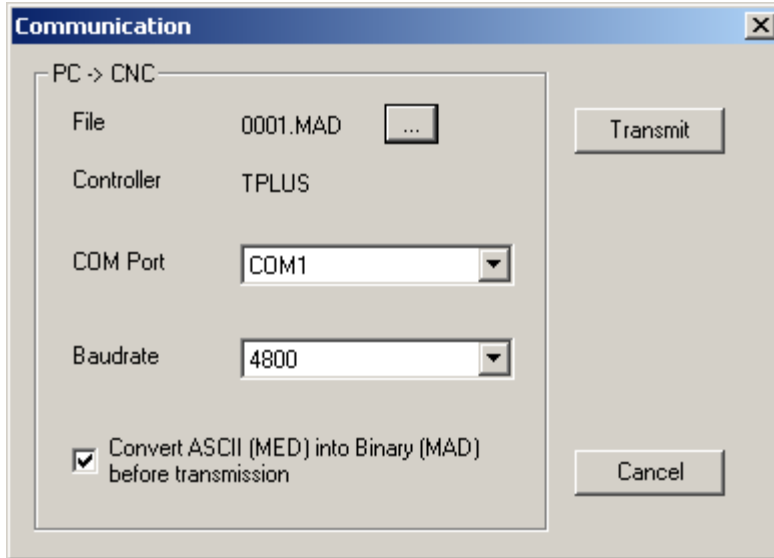
0001, ,0000,0000,0000
PNR MAT OD-MAX ID-MIN LENGTH RPM FIN-X FIN-Z WORK FACE
0 ALUM 2. 0 3. 5000 0.02 0.01 .1
-----
UNo MODE RV FV R-FEED R-DEP R-T F-T
1 EDG FCE 500 200. .01 0.10 1 1
SEQ . SPT-X SPT-Z FPT-X FPT-Z RGH
1 2. 0.100 0 0
-----
UNo MODE # CPT-X CPT-Z RV FV R-FEED R-DEP R-T F-T
2 BAR OUT 0 2. 0 150 200. .01 .1 2 2
SEQ SHP S-CNR SPT-X SPT-Z FPT-X FPT-Z F-CNR RADIUS RGH
1 LIN * * 1. 0 2.
2 TPR 1. 0 2. 1.5
-----
UNo MODE # DRL-DIA DEP-1 DEP-2 DEP-3 V FEED TOOL
3 DRL FCE 0 .5 0 0 0 200 .01 3
SEQ . SPT-Z FPT-Z
1 0. 1.5
-----
UNo MODE # CHAMF LEAD ANG MULTI HGT No. V DEPTH TOOL
4 THR IN 0 0 12. 60 1 .122 10 180 .01 5
SEQ . SPT-X SPT-Z FPT-X FPT-Z
1 .5 0 .5 1.
-----
UNo MODE # CHAMF LEAD ANG MULTI HGT No. V DEPTH TOOL
5 THR OUT 0 0 4.5 60 1 .136 10 180 .01 4
SEQ . SPT-X SPT-Z FPT-X FPT-Z
1 2. 1.5 2. 2.25
-----
UNo MODE # No. PITCH WIDTH FINI RV FV FEED DEP. R-T F-T
6 GRV OUT 0 1 1.09 .01 150 250 .1 .12 6 6
SEQ . S-CNR SPT-X SPT-Z FPT-X FPT-Z F-CNR ANGLE RGH
1 2.104 2.63 1.01 2.63
-----
UNo MODE # No. PITCH WIDTH FINI RV FV FEED DEP. R-T F-T
7 GRV OUT 0 1 2.22 .01 150 250 .1 .12 7 7
SEQ . S-CNR SPT-X SPT-Z FPT-X FPT-Z F-CNR ANGLE RGH

```

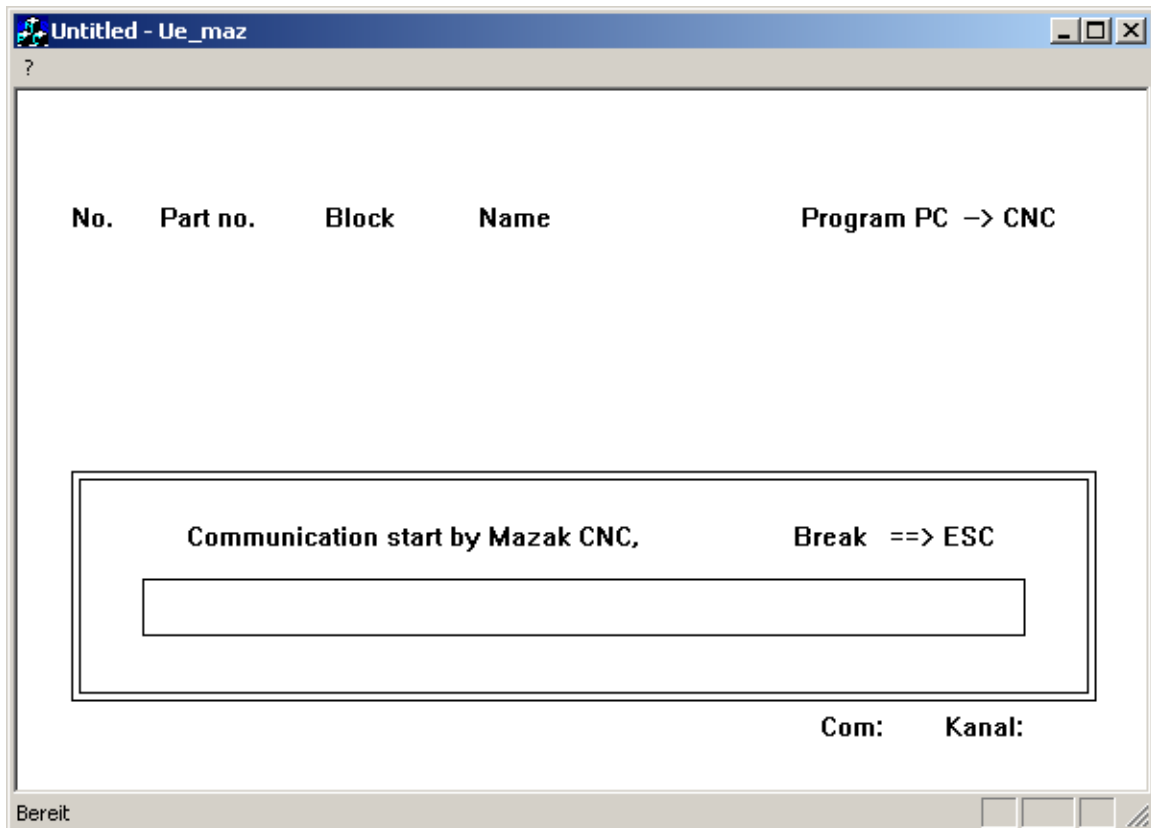
12. Close this window.

We will then send this program to the controller

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



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



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

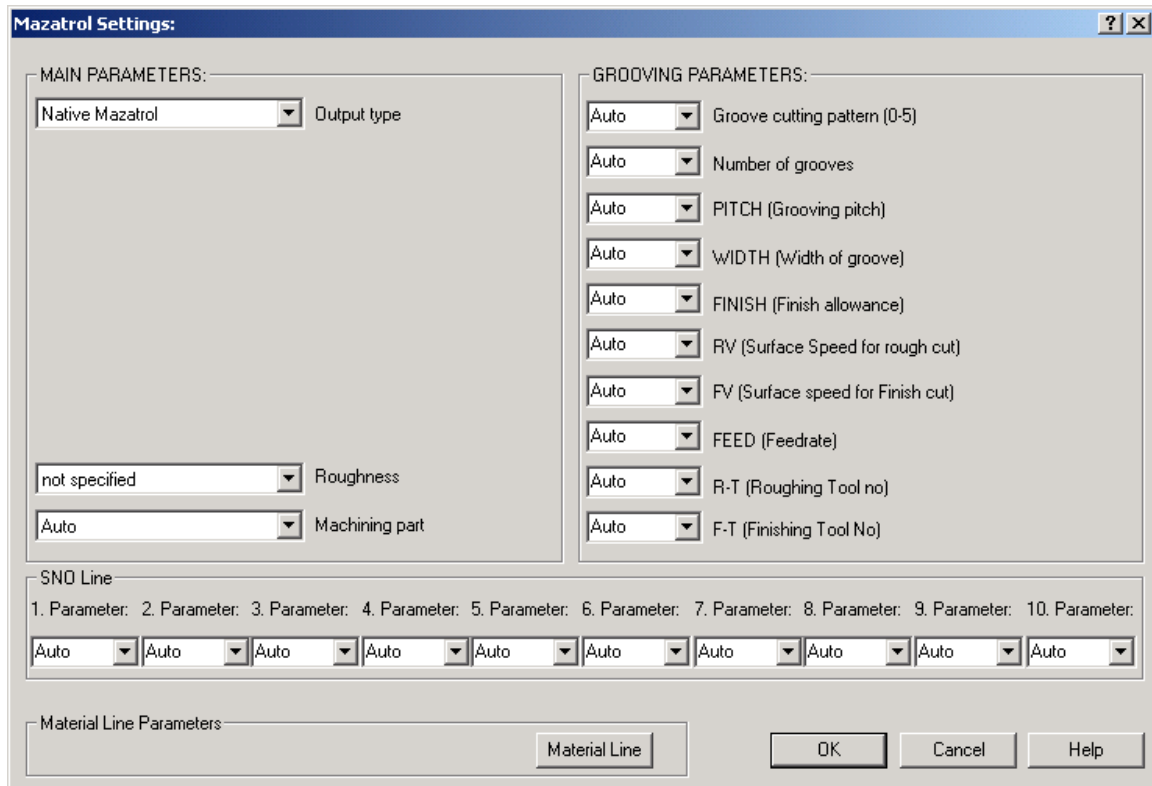
Appendix

Working with the Misc. Values Dialog to modify /override automatically generated output.

There will be times when you will wish to adjust the output at the mastercam programming stage or when a part has been programmed for a non-mazatrol control. As has been discussed earlier any value of the SNO and UNO lines can be overridden through the Misc. Values Page.

In the following example we will take the automatically generated groove of the previously programmed part and enter values which will then appear in the mazatrol code.

Below is Current Misc. values Dialog with current Auto Settings and then outputted code.



```

0001, ,0000,0000,0000
PNR MAT OD-MAX ID-MIN LENGTH RPM FIN-X FIN-Z WORK FACE
0 ALUMIN 2. 0 3. 5000 .005 .005 .1
-----
UNo MODE # No. PITCH WIDTH FINISH RV FV FEED DEP. R
1 GRV OUT 0 1 .2 .01 150 250 200 .12
SEQ . S-CNR SPT-X SPT-Z FPT-X FPT-Z F-CNR ANGLE
1 2.104 2.63 1.01 2.63
-----
SEQ MODE COUNTER RETURN WK.No. COUNT NUM. SHIFT
2 END 1 1 1
*
    
```

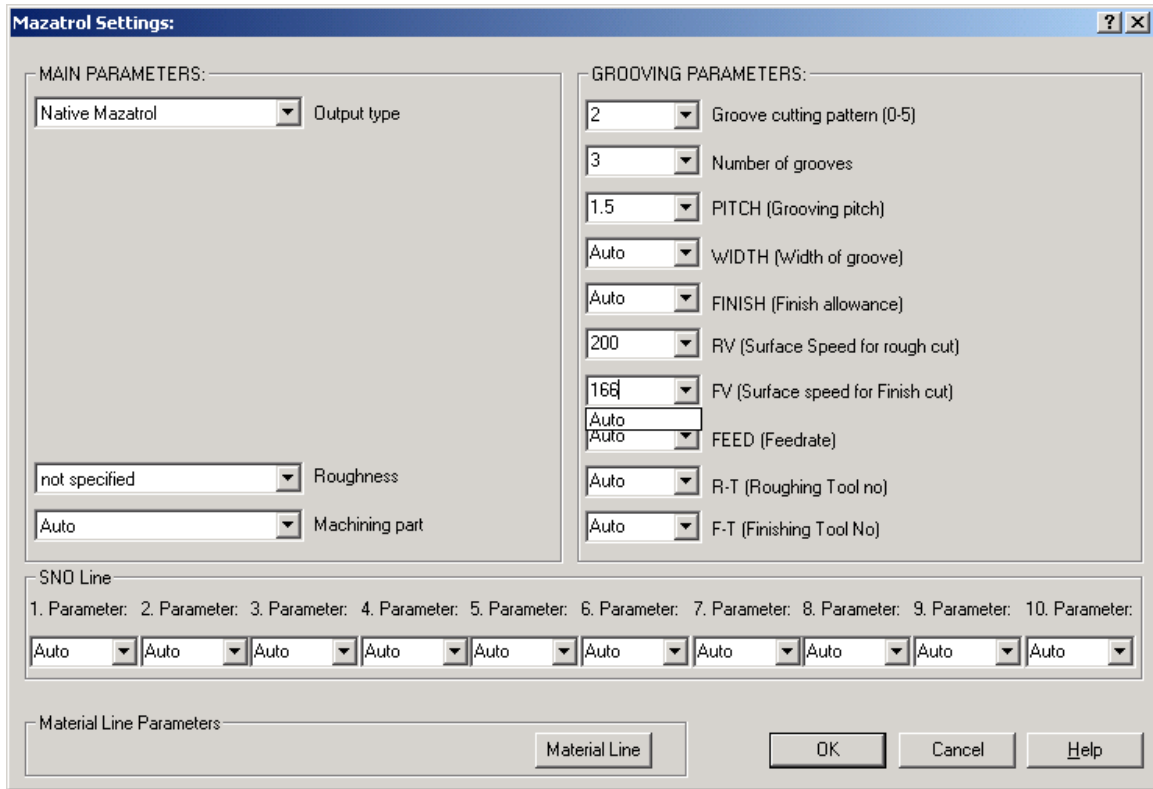
We will adjust the following:

We want a different grooving pattern say **#2 Right-tapered grooves**

Maybe multiple grooves based off of original **No.of 3 with a Pitch of 1.5**

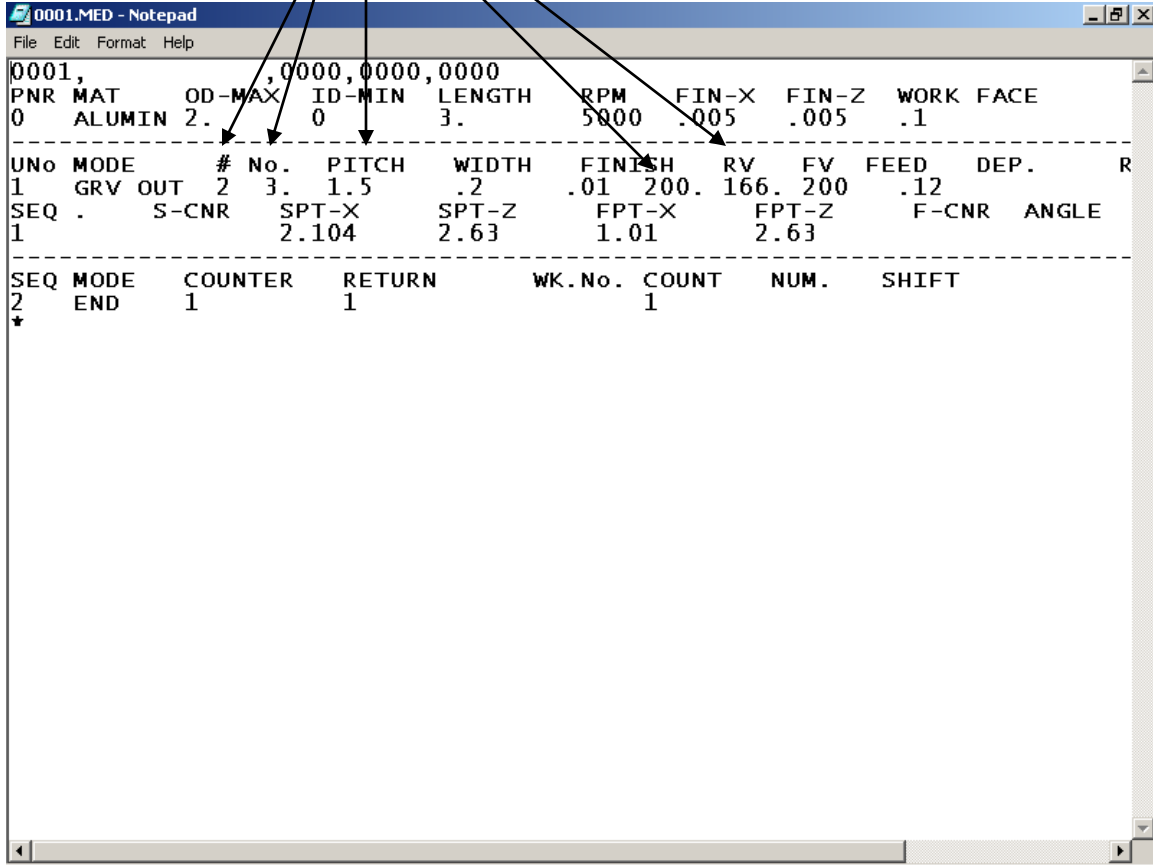
Maybe different values for feeds **200 for RV and 166 for FV**

We would modify the Misc. Values as shown



You can then see in the output below that those setting are now in transferred over.

Mastercam to Mazatrol Post-Processor Tutorial 11/1/2018



This can be done with every toolpath and operation and allows complete control to the programmer.

FOR ADDITIONAL INFORMATION ON THE USE OF THIS PRODUCT CONTACT:

Camaix USA
1515 South Mint St Suite C
Charlotte, NC 28203
704- 342-9292
INFOUSA@CAMAIX.COM