<u>Mastercam®to</u> <u>Mazatrol®Post-Processor Tutorial</u>

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

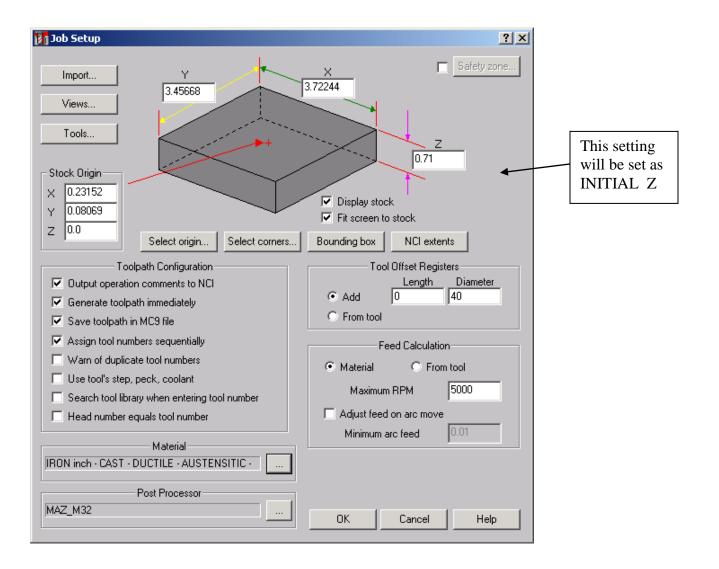
Note: This text was compiled using Version 8.0.8 of the Mazatrol Product – some dialogs presented may have changed or you may be using either an earlier or later version of the software.

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.



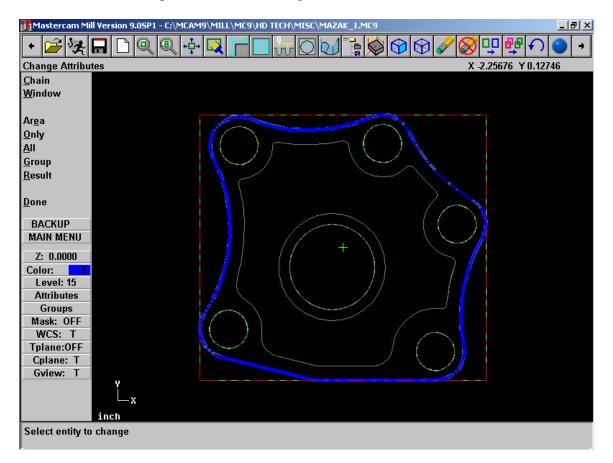
Note: Job Setup settings will affect the first line of the mazatrol PNR and MAT i.e. the material selected will be output and the Z depth of the material will be output as INITIAL-Z see below:

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.

Facing - C:\MCAM9\MILL	MC9\HD TECH\MISC\MAZ/	AK_1_MILL.N	EI - MAZ_M3	2	? ×
Tool parameters Facing p	Tool parameters Facing parameters				
	Left 'click' on tool to select; r	ight 'click' to ed	it or define nev	v tool	
#1- 1.5000 face mill					
Tool # 1	Tool name	Tool dia	1.5	Corner radius 0.0	
Head # -1	Feed rate 0.6	Program #	0	Spindle speed 50	
Dia. offset 41	Plunge rate 0.3	Seq. start	100	Coolant Off 💌	
Len. offset 1	Retract rate 0.3	Seq. inc.	2	Charges NICL	
Comment				Change NCI	
FACE		ime pos	🔲 Ref po	pint 🔲 Misc. values	
	E Ro	tary axis	T/C pl	ane 🔽 Tool display	
To batch				Canned text	
				OK Cancel H	lelp

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

Mazatrol Settings:	<u>? ×</u>
MAIN PARAMETERS:	FACE MILL PARAMETERS:
Native Mazatrol Output type	Auto Depth (Dist. from WPC Z=0 to fin. surface)
Face Machining unit	Auto SRV-Z (Z Axis stock removal)
Cutting Type	Auto BTM (Bottom roughness 1-9)
Rough and Finish	Auto FIN-Z (Finish allowance Z)
Bottom Roughness	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 •
Material Line Parameters	OK Cancel Help

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

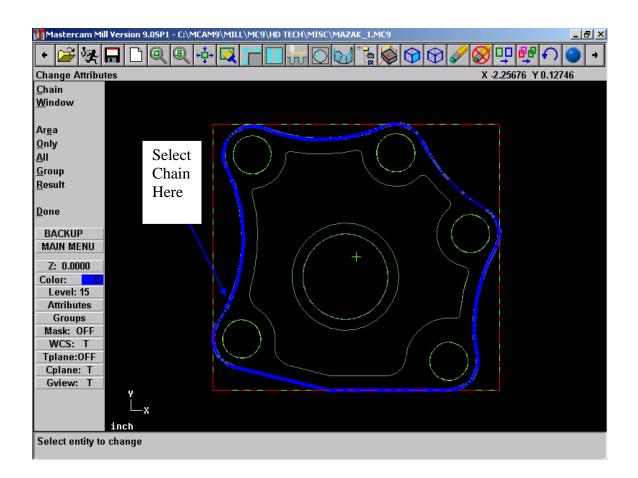
Facing - FACE - C:\MCAM9\MILL\M	1C9\HD TECH\MISC\MAZAK_1	MILL.NCI - MAZ_M32	<u>? ×</u>
Tool parameters Facing parameters			
	Clearance 2.0	Tip comp:	Tip
	Absolute C Incremental	Roll cutter around corners:	Sharp 💌
	Use clearance only at the start and end of operation	Cutting method:	jzag 💌
│ │ └╤ ┛ │┌─│□	Retract 0.25	Stepover: 75.	0 % 1.125
//	Absolute O Incremental	Auto angle	
	Feed plane 0.1 Absolute O Incremental	Roughing angle:	0.0
	Rapid retract	Move between cuts: Hid	ah speed loops
	Top of stock 0.0	Feed rate between cu	
	Absolute O Incremental		10.0
	Depth	Across overlap: 50.	0 % 0.75
	C Absolute 💿 Incremental	Along overlap: 110	0.0 % 1.65
Lineariz	zation tolerance: 0.0	Approach distance: 50.	0 % 0.75
Z stock	k to leave: 0.0	Exit distance: 50.	0 % 0.75
	Depth cuts		
		OK	Cancel Help

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.

Contour (2D) - (CONTOUR - C:	\MCAM9\MIL	.L\MC9\HD T	ech/misc//	MAZAK_1_M	ILL.NCI - MAZ_	M32	? ×
Tool parameters	Contour para	ameters						
	<u> </u>	eft 'click' on too	ol to select; rigł	nt 'click' to edit	or define new	tool		
-		P						
#1-1.50	, 2000 # 2-0.9	5000						
face mi		I1 flat						
Tool #	2	Tool name	1/2 FLAT I	Tool dia	0.5	Corner radius	0.0	
Head #	-1	Feed rate	0.456	Program #	1	Spindle speed	152	
Dia. offset	42	Plunge rate	6.4176	Seq. start	100	Coolant 🛛	Flood 💌	
Len, offset	2	Retract rate	6.4176	Seq. inc.	2			
Comment							Change NCI	
CONTOUR		<u> </u>	Hom	e pos	Ref poi	nt	vlisc. values	
		-	E Rota	ry axis	T/C plan	ne 🔽	Tool display	
🔲 To bato	h						Canned text	
					OK	Cancel	Help	

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

	Mazatrol Settings:	?×
		LINE MACHINING PARAMETERS:
	Native Mazatrol Output type	Auto Depth (Dist. from WPC Z=0 to fin. surface)
	Line machining	Auto SRV-Z (Z Axis stock removal)
	Auto. Cent/Left/Right Line machining	Auto SRV-R (X/Y Axis stock removal)
	Auto (CW/CCW/*)	4 RGH (Roughness 1-9)
Change		Auto CHMF (Chamfer width)
values	Use 1 Tool Rough and Finish	Auto FIN-Z (Finish allowance Z)
	► 4 Roughness	Auto FIN-R (Finish allowance R)
	SNO Line (END_MILL_SPHERICAL)	
	1. Parameter: 2. Parameter: 3. Parameter: 4. Parameter: 5. Parameter	r: 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

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.

Contour (2D) - CONTOUR - C:\MCAM9\MILI	L\MC9\HD TECH\MISC\MAZAK	_1_MILL.NCI - MAZ_M32
Tool parameters Contour parameters		
	Clearance 2.0	Compensation Computer
	Absolute C Incremental Use clearance only at the	Compensation Left 🔽 🚺
	start and end of operation	🗖 Optimize
	Retract 0.375	Tip comp Tip 🔽
	Absolute O Incremental Feed plane	Roll cutter around corners All
	Feed plane O Absolute O Incremental	Infinite look ahead
	Rapid retract	Linearization 0.0005
	Top of stock 0.0	Max. depth 0.001
	Absolute C Incremental	XY stock
	Depth0.71	T stock
	C Absolute 💿 Incremental	to leave
Contour type: 2D	Mult	i passes 🔽 Lead in/out
Chamfer Ramp Remachi	ning	oth cuts 🔲 Filter
		OK Cancel Help

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.

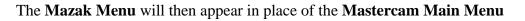
6. Select **Done**. This should return you to the operations manager. Select **Post** Modify settings as shown below. (In this example we are using the M32 post-processor shown as MAZ_32.PST. Yours may vary but all the Mazatrol Post-Processors will have the format of MAZ_XXX.PST)

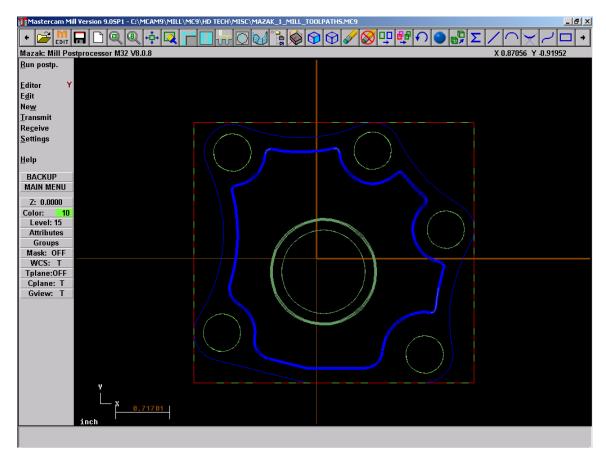
Post processing	? ×
Active post	Change Post
MAZ_M32.PST	
NCI file	
Save NCI file	🗖 Edit
C Overwrite	
🖲 Ask	
NC file	
🔽 Save NC file	🗖 Edit
Overwrite	NC extension
C Ask	.nc
- Send	
Send to machine	Comm
OK Cancel	Help

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

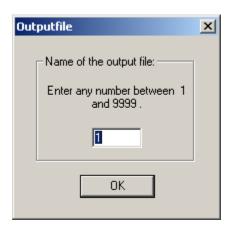
Note: We do not need to create an NC file but Mastercam needs to have this setting so that the post-processor can function

8. Click Save.





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)

	1.MED - Notepad						
	dit F <u>o</u> rmat <u>H</u> elp						
0001	L,	,0000	,0000,0000				
PNR	MAT	INITIAL-2	Z ATC MODE	MULTI	MODE MULTI	FLG PITCH-X	PITCH-Y
0	IRON	0.7100	1	OFF			
·							
UNO	UNO	DEPTH	SRV-Z	SRV-R			-R
1	FACE-MIL	0.0000	.001	Ŧ	1 *	0 0	
SNO		DM. NO.	APRCH-X	APRCH-Y	TYPE ZFD DEP	-Z WID-R C-SP	
1	F-MILL 1		? Y	?	XB1 G01 0.0		0.600 3 8
FIG	PTN	X	Y	r/0	I	ј Р	CNR
1	LINE	-3.0925	-1.6475				
12	LINE	2.6553	-1.6475				
13	LINE	2.6677	-1.6475		D 6677	4 9455	
4	CCW	2,6677	-0.7834	0.4321	2.6677	-1.2155	
1 2 3 4 5 6 7 8 9	LINE	-2.5141	-0.7834		5 5144	0 3514	
6	CW	-2.5141	0.0807	0.4321	-2.5141	-0.3514	
16	LINE	2.9560	0.0807		2 0560	0 5130	
ğ	CCW	2,9560	0.9448	0.4320	2.9560	0.5128	
	LINE	-2.3055	0.9448				
10	LINE	-2.3279	0.9448	A 433A	2 2220	1 1760	
11	CW	-2.3279	1.8089	0.4320	-2.3279	1.3709	
12	LINE	2.7487	1.8089				
	UNO	DEPTH	SRV-Z	SD//_D	RGH CHMF	FIN-Z	FIN-R
2	LINE-CTR		0.7100				0
			APRCH-X			•	
1).50 E	?	7		100 152	0.456 3 8
F TG	PTN	X	· Y	R/O	I] P	CNR
1	LINE	-2.1030	-0.0352		-	- 1	- m
12	LINE	-1.8035	-0.0524				
2 3	CCW	-1.4868	0.2299	0.3000	-1.7863	0.2471	
4	ČČŴ	-1.6745	1.1243	1.7501	-3.2340	0.3301	
4 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.

Cor	mmunication		×		
Г	PC -> CNC				
	File	0001.mad	Transmit		
	Controller	М32			
	COM Port	COM1 💌			
	Baudrate	4800			
Convert ASCII (MED) into Binary (MAD) before transmission					

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

🍂 U	ntitled	- Ue_maz				_ 0	×
?							
	No.	Part no.	Block	Name	Program I	PC -> CNC	
		Commu	nication star	t by Mazak CNC,	Break ==	> ESC	
					Com:	Kanal:	
ı Bereit	:						//.

This is the progress bar.

To complete the download complete the following steps at The <u>Mazak Controller</u>.

- 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.

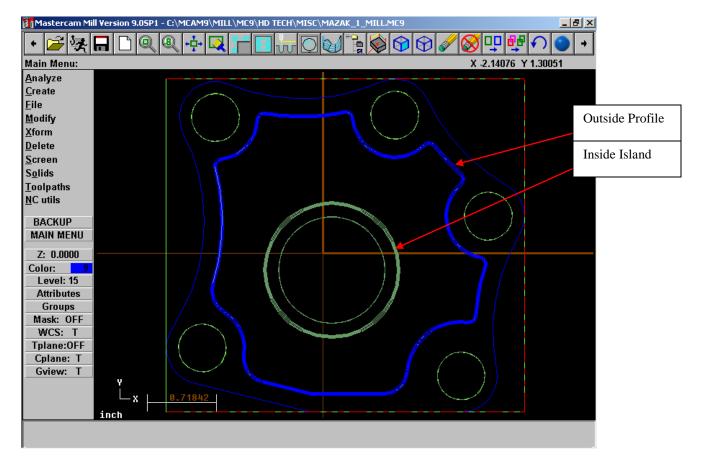
Save File as Mazak_1_Mill_1.mc9

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

Pocket (Standa	ard) - POCKE	r - C:\Mcam9\	MILL\MC9\	hd tech/mi	SC\MAZAK_1	I_MILL.NCI - MAZ_M32	? ×
Tool parameter	S Pocketing	parameters Ro	ughing/Finishi	ing parameters	1		
		Left 'click' on to	ol to select; rig	jht 'click' to edi	t or define new	tool	_
#1-1.50			0.3750				
face m			nill1 flat				
Tool #	3	Tool name	3/8 FLAT I	Tool dia	0.375	Corner radius 0.0	
Head #	-1	Feed rate	0.45066	Program #	1	Spindle speed 203	
Dia. offset	43	Plunge rate	6.33203	Seq. start	100	Coolant Flood 💌] [
Len, offset	3	Retract rate	6.33203	Seq. inc.	2		.
Comment						Change NCI.	<u> </u>
POCKET		<u> </u>	Hor	ne pos	Ref poi	int 🔽 Misc. values.	
		v	E Rota	ary axis	T/C pla	ne 🔽 Tool display.	
🔲 To bate	ch					Canned text.	
					ОК	Cancel	Help

7. Set **Tool Parameters** as shown

Mazatrol Settings:	? ×
MAIN PARAMETERS:	POCKET WITH ISLAND PARAMETERS:
Native Mazatrol Output type	Auto Depth (Dist. from WPC Z=0 to fin. surface)
PCKT MT Machining unit	Auto SRV-Z (Z Axis stock removal)
_	Auto SRV-R (X/Y Axis stock removal)
	Auto BTM (Bottom roughness 1-9)
CW (clockwise) Cutting Type	Auto VAL (Wall roughness 1-9)
Use 2 Tools Rough and Finish	0.0000 FIN-Z (Finish allowance Z)
3 Bottom Roughness	0.0200 TIN-R (Finish allowance R)
4 Wall Roughness	
SNO Line (END_MILL_SPHERICAL)	
1. Parameter: 2. Parameter: 3. Parameter: 4. Parameter: 5. Parameter	: 6. Parameter: 7. Parameter: 8. Parameter: 9. Parameter: 10. Parameter:
Auto 💌 Auto 💌 Auto 💌 Auto	r Auto 🔻 Auto 💌 Auto 💌 Auto 💌
Material Line Parameters	
Init Z 0.7 ATC Mode 1	OK Cancel Help

- 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 Machining direction-Clearance... П 2.0 Climb Conventional Absolute C Incremental Use clearance only at the Tip comp Tip start and end of operation Roll cutter 0.25 Sharp • П around corners C Incremental O Absolute Linearization 0.001 tolerance Feed plane... 0.1 0.0 XY stock to leave C Absolute Incremental Rapid retract 0.0 Z stock to leave 0.0 Top of stock... Г Absolute O Incremental Create additional finish operation Depth... -0.15625 Absolute C Incremental Depth cuts.. Filter.. Pocket type: Standard • Advanced.. Remachining.. Open pockets .. Facing.. 0K Cancel Help

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

Pocket (Standard) - POCKET - C:\MCA	M9\MILL\MC9\HD TECH\MISC\MAZAK_1_MILL.NCI - MAZ_M32
Tool parameters Pocketing parameters	Roughing/Finishing parameters
Rough	Cutting method: Parallel Spiral
Zigzag Constant Overlap Spiral	Parallel Spiral Parallel Spiral, Morph Spiral High Speed One Way
Stepover percentage 75.0 Stepover distance 0.28125	Minimize tool burial Entry - ramp High Speed
Roughing angle 0.0	
Image: Finish No. of passes	Finish pass spacing 0.01
Finish outer boundary	Cutter compensation control
Start finish pass at closest entity	
Keep tool down	Machine finish passes only at final depth Machine finish passes after roughing all pockets Lead in/out
	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	DEP	TH SRV-	Z SRV	/- R	BTM	WAL FIN	-Z FIN-	R	
1 PCKT.M	IT 0.09	12 0.0912	*	11	0	0			
SNO SNO	NOM.	NO. APRC	H-X A	PRCH	I-Y TY	PE ZFD D	DEP-Z WI	D-R C-SP F	R M M
1 E-MILL	0.38 E	?		?	CW	G0 1	1 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 $(5) \ge 0.5$ dia circles
- 5. Done
- 6. Done

DRILLING-no peck - C:\M	CAM9\MILL\MC9\HD TECH	\misc\mazai	K_1_MILL.NCI	- MAZ_M32	<u>?</u> ×					
Tool parameters DRILLING-no peck Custom Drill Parameters 1										
	Left 'click' on tool to select;	right 'click' to edi	it or define new	tool	-					
	7	V	7							
	2- 0.5000 #3- 0.3750 dmill1 flat endmill1 flat	#4- 0.5000 center drill	#5- 0.5000 drill							
Tool # 4	Tool name 1/2 CENT	Tool dia	0.5	Corner radius 0.0]	Set				
Head # -1	Feed rate 0.1935	Program #	10001 ┥	Spindle speed 129		Program				
Dia. offset 44	Plunge rate 0.1935	Seq. start	100	Coolant Off]	# to 10001				
Len. offset	Retract rate 0.1935	Seq. inc.	2		,					
Comment		,		Change NCI						
		ome pos	🔲 Ref poir	nt 🔽 Misc. values	.					
		otary axis	T/C plan	ne 🗖 Tool display						
To batch				Canned text.						
			OK	Cancel H	elp					

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)

Mazatrol Settings:	<u>? ×</u>
MAIN PARAMETERS:	DRILLING-NO PECK PARAMETERS:
	: 6. Parameter: 7. Parameter: 8. Parameter: 9. Parameter: 10. Parameter: Auto Auto Auto Auto Auto Auto Auto OK OK Cancel Help

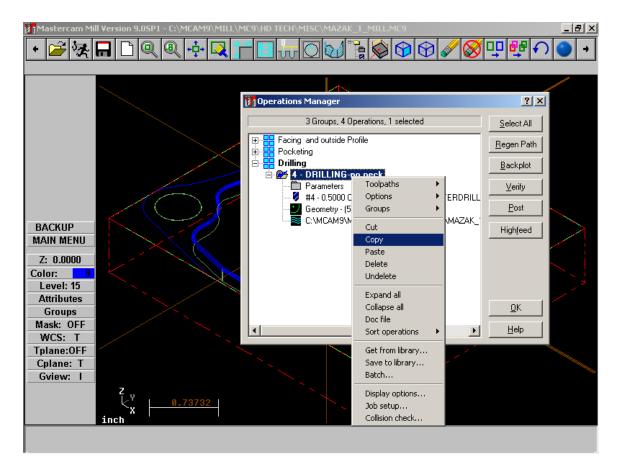
7. Misc. Values Leave settings on Auto as shown

	IILL\MC9\HD TECH\MISC\MAZAK_ * Custom Drill Parameters 1	1_MILL.NCI - MAZ_M32	<u>?</u> ×
Tip comp	 Clearance Absolute C Incremental Use clearance only at the start and end of operation Retract Absolute C Incremental Top of stock 0.0 Absolute C Incremental Depth -0.25 Absolute C Incremental Subprogram Absolute C Incremental 	Cycle DRILLING-DRI DRILLING-PK1 DRILLING-PK2 DRILLING-PK1 TAP BORE-S1 BORE-S1 RGH.CBOR REAMING not used RGH.BCB BK.CBORE CIR.MIL CBOR.TAP BORE-T1 BORE-T2 Custom cycle 16 Custom cycle 17	
		OK Cancel	Help

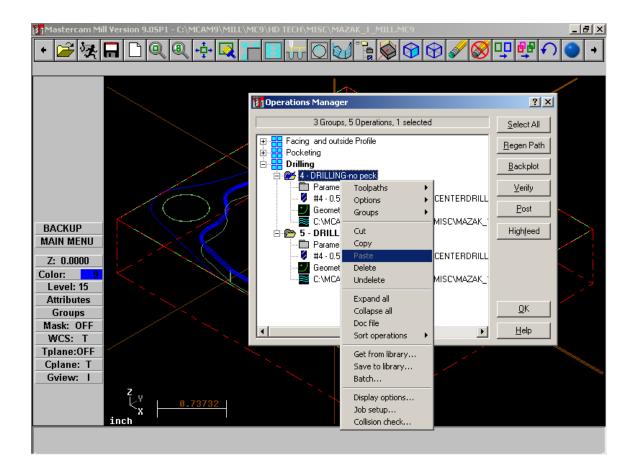
8. Go to **Drilling** and set **DRILLING – DRI** as shown

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

DRILLING-no peck - C:\MCAM9\MILL\MC9\HD TECH\MISC\MAZAK_1_MILL.NCI - MAZ_M32 ? × Tool parameters DRILLING-no peck Custom Drill Parameters 1 Left 'click' on tool to select; right 'click' to edit or define new tool C #5-0.5000 #1-1.5000 #2-0.5000 #3-0.3750 #4-0.5000 endmill1 flat endmill1 flat face mill center drill drill 1/2 DRILL 0.5 Corner radius 0.0 5 Tool dia Tool # Tool name 1 0.2322 10001 Spindle speed 129 Head # Feed rate Program # 100 45 0.2322 Dia. offset Plunge rate Seq. start Coolant Off • 2 Len. offset 5 Retract rate 0.2322 Seq. inc. Change NCI... Comment Home pos... E Ref point.. Misc. values... ۸ E Rotary axis.. T/C plane. Tool display.. -Canned text. П 🔲 To batch ΟK Cancel Help

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

12. Select 0.5" tool as shown

DRILLING-no peck - C:\MCAM9\MIL	L\MC9\HD TECH\MISC\MAZAK	1_MILL.NCI - MAZ_M32	? ×
Tool parameters DRILLING-no peck	Custom Drill Parameters 1		
Tip comp	Clearance 2.0 Absolute Incremental Use clearance only at the start and end of operation Retract 0.1 Absolute Incremental Top of stock 0.0 Absolute Incremental Depth 71 Absolute Incremental Subprogram Absolute	Cycle DRILLING-DRI 1st peck Subsequent peck Peck Chip break Shift	 0.1 0.1 0.1 0.1 0.1 0.0 0.0
		OK Cancel	Help

13. Set **Depth** as shown.

14. **OK**.

After posting the output will appear as shown below.

2	CW		-0	.7	904		-0.	17	45	Ο.	882	29	(0.09	25	-	0.17	45										
	CCW LINE		-0 -0							Ο.	224	19	-	-1.0	154		0.17	745										
													IF															
	DRILI SNO												. т а	שתת	DET		и Б1	זידיםי		.	210		15					
	C.DR																		129									
	DRILI																											
	PTN :															2111			100				P					
	PTN -											7	7		*		1	7		*	*	*	o	ο	Ο			
	PTN -	-0.0	000	ο.	746	4	1.	43	40	*		7	ŧ		*		-	ŧ		*	*	*	Ο	Ο	Ο			
	PTN -	-0.0	000	1.	717'	7	Ο,	38	48	*		7	Ŧ		*		7	t		*	*	*	0	Ο	0			
	PTN -	-0.0	000	1.	436'	7	-1	.2	726	*		7	t		*		7	t		*	*	*	0	0	0			
	PTN -	-0.0	000	-1	.25	17	-0).9	826	*		7	t		*		1	t		*	*	*	0	0	0			
0	UNO				co	NTI			NUM	BER		ATO	:		x													
	END PTNLI				0				0			0			0			0			0							
	FINLI																											
IC:	HING	UNIT	C <me< td=""><td>ENU</td><td>></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td>ENI</td><td>D</td><td></td><td></td><td></td><td></td><td>_</td></me<>	ENU	>																	ENI	D					_
Τ	M-COD	E	1 SUB			2 111	IS			RUM			ALE:	-		ROCE	ss	6 IN	DEX	7	OF	s		8	H−≻J	AUSW	Spec	LO
1			PRO	GRA	M				C	HANG	E	C	HAN(GE	E	ND											Fkt.	

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

perations Manager	<u>? ×</u>
3 Groups, 13 Operations, 1 selected	Select All
Pocketing Counter Boring	Regen Path
	Backplot
	Verify
	Post
	Highfeed
	ОК
	Help

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

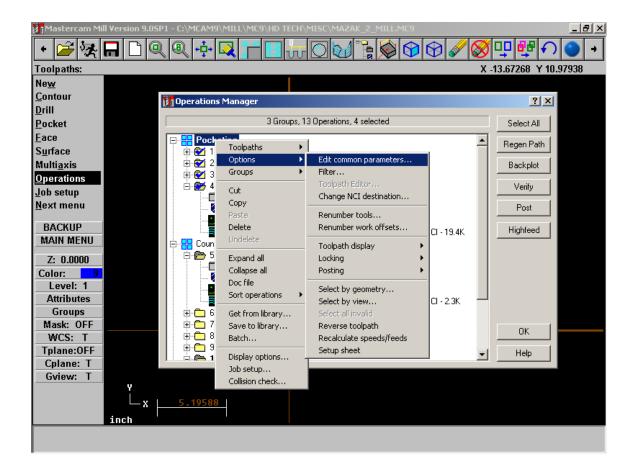
We have two options in this case. We could program all the pockets using line-center and modifying settings as shown below – this would take advantage of mastercam's many different type of pocketing strategies available when setting the Roughing component or we could program separate areas of the part using either Mazatrol's Pocket or Pocket MT. In this section we will program output as Line center.

d: Constant Overlap Spiral Parallel Spiral, Morph Spiral Clean Corners Minimize tool burial Spiral inside to outside Big Spiral Spiral Spiral Spiral inside to outside
Clean Corners Minimize tool burial Entry - ramp
Clean Corners Minimize tool burial Entry - ramp
Minimize tool burial
Spiral inside to outside High Speed
spacing 0.01
ensation control 💌
cutter comp in control
e finish passes only at final depth
e finish passes after roughing all pockets 🛛 🗌 Lead in/out

1. Fill in settings as shown below:

Mazatrol Settings:	?×
MAIN PARAMETERS:	
Native Mazatrol Output type	Auto Depth (Dist. from WPC Z=0 to fin. surface)
Line machining Machining unit	Auto SRV-Z (Z Axis stock removal)
Auto. Cent/Left/Right Line machining	Auto SRV-R (X/Y Axis stock removal)
Auto (CW/CCW/*)	Auto RGH (Roughness 1-9)
	Auto CHMF (Chamfer width)
Use 1 Tool Rough and Finish	□ FIN-Z (Finish allowance Z)
3 Roughness	0.02000 TIN-R (Finish allowance R)
SNO Line (END_MILL_SPHERICAL)	
	6. Parameter: 7. Parameter: 8. Parameter: 9. Parameter: 10. Parameter:
Auto 💌 Auto 💌 Auto 💌 Auto	Auto 💌 Auto 💌 Auto 💌 Auto 💌
Material Line Parameters	OK Cancel Help

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 Y R/0 I FIG PTN Х Р J CNR 1 LINE 8.7198 9.2517 8.7600 9.4741 0.6350 8.1250 9.4741 2 CCW 3 LINE 8.7600 10.0514

Mazatrol Settings:	? ×
MAIN PARAMETERS:	LINE MACHINING PARAMETERS:
Native Mazatrol Output type	Auto Depth (Dist. from WPC Z=0 to fin. surface)
Line machining Machining unit	Auto SRV-Z (Z Axis stock removal)
Auto. Cent/Left/Right Ine machining	Auto SRV-R (X/Y Axis stock removal)
Auto (CW/CCW/*) Line cutting direction	Auto RGH (Roughness 1-9)
	Auto CHMF (Chamfer width)
Use 1 Tool Rough and Finish	Auto FIN-Z (Finish allowance Z)
3 Roughness	Auto FIN-R (Finish allowance R)
SNO Line (END_MILL_SPHERICAL)	
	: 6. Parameter: 7. Parameter: 8. Parameter: 9. Parameter: 10. Parameter:
Auto 🗸 Auto 🗸 Auto 🗸 Auto	V Auto V Auto V Auto V
L	
Material Line Parameters	OK Cancel Help

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:

Mazatrol Settings:	? ×
MAIN PARAMETERS:	
Native Mazatrol Output type Auto Depth (Dist. from WPC Z=0 to fin. surface)	
Line machining Machining unit 0.25500 SRV-Z (Z Axis stock removal	Change
Auto. Cent/Left/Right Line machining 0.12500 SRV-R (X/Y Axis stock removal)	values
Auto (CW/CCW/*) Line cutting direction Auto RGH (Roughness 1-9)	
Auto CHMF (Chamfer width)	
Use 1 Tool Rough and Finish Auto FIN-Z (Finish allowance Z)	
3 Roughness Auto FIN-R (Finish allowance R)	
SNO Line (END_MILL_SPHERICAL)	
1. Parameter: 2. Parameter: 3. Parameter: 4. Parameter: 5. Parameter: 6. Parameter: 7. Parameter: 8. Parameter: 9. Parameter: 10. Pa	arameter:
Auto VAuto VAuto VAuto VAuto VAuto VAuto VAuto VAuto	
Init Z 0.7 ATC Mode 1 V OK Cancel I	Help

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

        I E-MILL 0.50
        ?
        *
        G01 0.2550 *
        1069 6.417 3
        9

        FIG PTN
        X
        Y
        R/0
        J
        P
        CNR

        1
        LINE
        8.7198
        9.2517
        J
        P
        CNR

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
           9.0401 11.1456
5 LINE
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 <u>Roughing</u> 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:

Pocket (Standard) - Main Pocket with Islands - C:\MCAM9\MILL\MC9\HD TECH\MISC\MAZAK_2_MILL.N ? 🗙		
Tool parameters Pocketing parameters Roughing/Finishing parameters		
Rough	Cutting method: Constant Overlap Spiral	
Zigzag Constant F Overlap Spiral	Parallel Spiral Parallel Spiral, Morph Spiral High Speed One Way Clean Corners	
Stepover percentage 75.0	Minimize tool burial	
Stepover distance 0.375	Spiral inside to outside	
Roughing angle 0.0		
🔽 Finish		
No. of passes 1	Finish pass spacing 0.01	
Finish outer boundary	Cutter compensation control	
🔲 Start finish pass at closest entity	Optimize cutter comp in control	
🔲 Keep tool down	Machine finish passes only at final depth	
	Machine finish passes after roughing all pockets	
	OK Cancel Help	

Set **Misc. Values** as below:

Mazatrol Settings:	<u>? ×</u>	
MAIN PARAMETERS:	POCKET MILL PARAMETERS:	
Native Mazatrol Output type	Auto Depth (Dist. from WPC Z=0 to fin. surface)	
PCKT Machining unit	Auto SRV-Z (Z Axis stock removal)	
	Auto SRV-R (X/Y Axis stock removal)	
	Auto BTM (Bottom roughness 1-9)	
CW (clockwise) Cutting Type	Auto 💌 WAL (Wall roughness 1-9)	
Use 1 Tool Rough and Finish	Auto FIN-Z (Finish allowance Z)	
3 Bottom Roughness	Auto FIN-R (Finish allowance R)	
4 Wall Roughness	Auto ??	
	Auto CHMF (Chamfer width)	
SNO Line (END_MILL_SPHERICAL)		
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 S OK Cancel Help		

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:

3 Groups, 13 Operations, 1 selected	
	Select All
Pocketing - using Line Center	Regen Path
E	
⊕-	Backplot
	Verify
🗄 💼 8 - DRILLING-PECK 2	
9 - DRILLING-no peck	Post
🖻 😭 10 - DRILLING-no peck	Highfeed
i ⊡ 📴 0.1900-24 TAP RH i ⊡ 🚰 11 - DRILLING-no peck	
🔰 🔰 #7 - 0.5000 SPOT DRILL - 1/2 SPOTDRILL	
Geometry - (4) point(s)	
C:\MCAM9\MILL\MC9\HD TECH\MISC\MAZAK_2_MILL.NCI - 2.6K	
🖻 👚 12 - DRILLING-PECK 2	
Parameters #8 - 0.1495 DRILL - #25 DRILL	
Geometry - (4) point(s)	
C:\MCAM9\MILL\MC9\HD TECH\MISC\MAZAK_2_MILL.NCI - 2.6K	
📕 🗄 🗁 13 - TAP 📃	ок (
■ ■ #9 - 0.1900 × 24.00 TAP RH - #10-24 TAPRH	Help

Common parameters						<u>? ×</u>
Tool parameters Com	mon Operation Parameters					
	Left 'click' on tool to selec	t; right 'click' to ed	it or define new	tool		
		7	7	7	Ţ	
#1- 0.5000 endmill1 flat	#2- 0.2500 #3- 0.1250 endmill1 flat endmill1 flat	#4-0.1250 center drill	#5- 0.0156 drill	#6- 0.1875 drill	#7- 0.5 spot c	
	originar net		Gilli	Gim	•	
		_				
Tool # 3	Tool name 1/8 FLA	T I Tool dia	0.125	Corner radius	0.0	
Head # 1	Feed rate 6.1609	Program #	10001	Spindle speed	4278	
Dia. offset 3	Plunge rate 6.1609	Seq. start	100	Coolant 0	ff 💌	
Len. offset 3	Retract rate 6.1609	 Seq. inc.	2			
, Comment				Cł	hange NCI	
		Home pos	🗖 🛛 Ref poi	nt 🔽 Mi	isc. values	
		Rotary axis	T/C pla	ne 🔽 T	ool display	
To batch					anned text	
			OK	Cancel	Hel	P

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

DRILLING-no peck - C:\MCAM9\MILL\MC9\HD TECH\MISC\MAZAK_2_MILL.NCI - MAZ_M32 ? × Tool parameters DRILLING-no peck Custom Drill Parameters 1 $\mathbf{\nabla}$ Clearance... 2.0 Cycle C Absolute 💿 Incremental DRILLING-DRI • DRILLING-DRI Use clearance only at the ٠ DRILLING-PK2 DRILLING-PK1 start and end of operation TAP 0.5 Retract... BORE-S1 BORE-S1 RGH.CBOR C Absolute 🔘 Incremental REAMING Top of stock... 0.0 not used C Absolute 💿 Incremental RGH.BCB **BK.CBORE** -0.0625 Depth... CIR.MIL CBOR.TAP C Absolute 📀 Incremental BORE-T1 BORE-T2 Subprogram Custom cycle 16 Tip comp.. Custom cycle 17 C Absolute C Incrementa ΟK Cancel Help

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

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

👔 Operations Manager	<u>?</u> ×
4 Groups, 18 Operations, 6 selected	Select All
Pocketing - using Line Center □	Regen Path
🖻 😥 5 - RGH.CBOR	Backplot
	Verify
🕀 😥 8 - RGH.CBOR	
	Post
⊕ <mark></mark>	Highfeed
Pocketing -Using Mazatrol Pocket Styles	
	ОК
	Help

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:

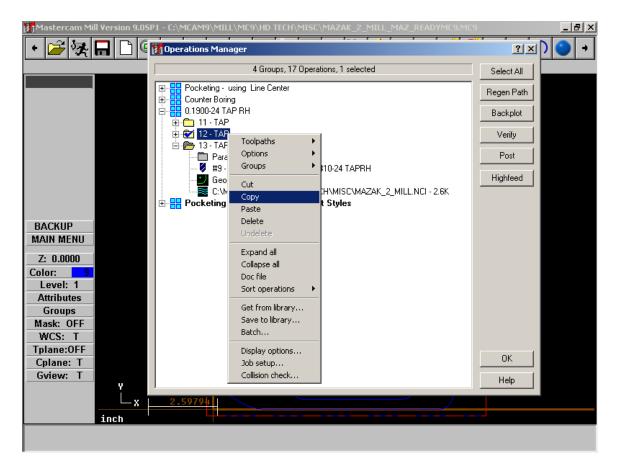
Common parameters	5						<u>? ×</u>
Tool parameters Cor	mmon Operation Paramet	ers					
	Left 'click' on too	I to select; rig	jht 'click' to edi	t or define new	tool		
			Ţ	7	7	Ţ	
#1-0.5000 endmill1 flat).1250 nill1 flat	#4- 0.1250 center drill	#5- 0.0156 drill	#6- 0.1875 drill	#7- 0.5 spot c	
•						►	
Tool # 3	Tool name	1/8 FLAT I	Tool dia	0.125	Corner radius	0.0	
Head # -1	Feed rate	6.1609	Program #	10002	Spindle speed	4278	
Dia. offset 3	Plunge rate	6.1609	Seq. start	100	Coolant Off	•	
Len. offset 3	Retract rate	6.1609	Seq. inc.	2			
Comment					Ch	ange NCI	
	<u> </u>	Hor	ne pos	🔲 Ref poi	nt 🔽 Mis	c. values	
	-	E Rot	ary axis	T/C plan	ne 🔽 To	ol display	
🔲 To batch						nned text	
				OK	Cancel	Hel	P

2. Set all Drill Cycles to TAP as below:

DRILLING-no peck - C:\MCAM9\MILL\MC9\HD TECH\MISC\MAZAK_2_MILL.NCI - MAZ_M32 ? × Tool parameters DRILLING-no peck Custom Drill Parameters 1 $\mathbf{\nabla}$ Clearance... 2.0 Cycle C Absolute 💿 Incremental DRILLING-DRI • DRILLING-DRI Use clearance only at the ٠ DRILLING-PK2 DRILLING-PK1 start and end of operation TAP 0.5 Retract... BORE-S1 BORE-S1 RGH.CBOR C Absolute 🔘 Incremental REAMING Top of stock... 0.0 not used RGH.BCB BK.CBORE C Absolute 💿 Incremental -0.095 CIR.MIL CBOR.TAP Depth... C Absolute 💿 Incremental BORE-T1 BORE-T2 Subprogram Custom cycle 16 Tip comp.. Custom cycle 17 C Absolute Incremental ΟK Cancel Help

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

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.

TAP - C:\MCAM9\MILL\MC9	hd tech/misc/maz	ak_2_mill.nci	- MAZ_M32		? ×
Tool parameters TAP Cu	stom Drill Parameters 4				
	Left 'click' on tool to seled	st; right 'click' to ed	it or define new tool		_
	7				
3- 0.1900 #10- 0.7500 tap rh drill				0.6250 #15- 1.000 ore chamfer m	-
•					D
Tool # 15	Tool name 1" CHA	MF Tool dia	1.0 C	orner radius 0.0	
Head # -1	Feed rate 8.5568	Program #	10002 Sp	indle speed 534	
Dia. offset 15	Plunge rate 8.5568	Seq. start	100 C	oolant Off	-
Len. offset 15	Retract rate 8.5568	Seq. inc.	2		_
Comment				Change NC	<u></u>
		Home pos	E Ref point	Misc. value:	s
		Rotary axis	T/C plane	Tool display	
To batch				Canned tex	i
			OK	Cancel	Help

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

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

TAP - C:\MCAM9\MILL\MC9\HD TEC Tool parameters TAP Custom Drill	H\MISC\MAZAK_2_MILL.NCI - MAZ_ Parameters 4	M32	<u>?</u> ×
	Clearance 2.0 Cyc Absolute Incremental TA Use clearance only at the start and end of operation 5 Retract 0.5 Absolute Incremental Top of stock 0.0 Absolute Incremental Depth -0.47 Absolute Incremental		 ▼ 0.04485 0.0 0.02242 0.02242 0.02242 0.0 0.0
Tip comp	Subprogram C Absolute © Incremental	OK Cancel	, Help

- 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

 $\mathbf{\Lambda}$

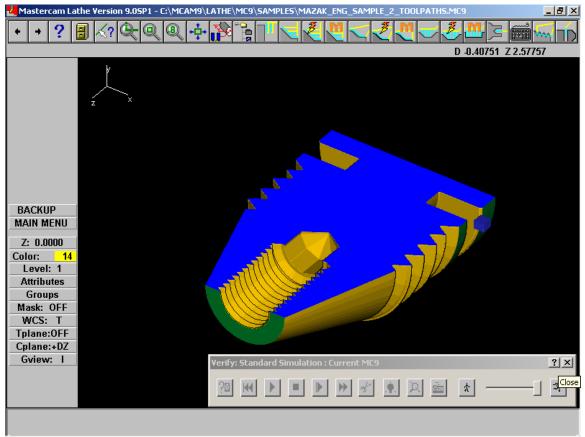
- these canned cycles are designated with the hopefully familiar 'M' for Mazak.

🛃 Mastercam Lathe Ve	rsion 9.05P1 - C:\MCAM9\LATHE\MC	9\SAMPLES\MAZAK_ENG	_SAMPLE_2_TOOLNATHS.MC9	_ 🗆 >
+ + ? 🔒 🗸	🗣 🔍 🍭 🕂 🏠 🎙</td <td>╞╨╲╣╝</td> <td></td> <td>1. Norm</td>	╞╨╲╣╝		1. Norm
Lathe toolpaths:			D -0).20329 Z -14.31810
Ne <u>w</u>				
Quick				
F <u>a</u> ce	5.35016			
Rough incl	▶+			

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

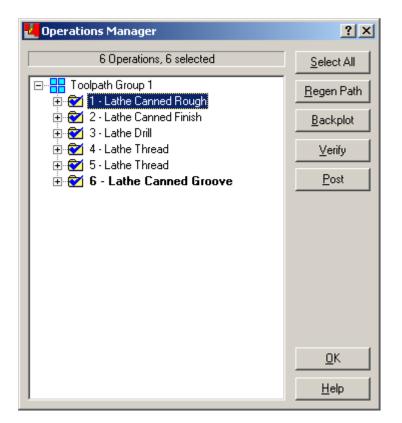
Mastercam	Mazatrol Output
Face	- EDG; FCE
Canned Rough	- BAR; IN, OUT, FCE, BAK Also some GRV
Canned Finish	- BAR; IN, OUT, FCE, BAK
Canned Groove	- GRV; IN, OUT, FCE, BAK
Thread	- THR
Drill	- DRL
Cutoff	- GRV

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

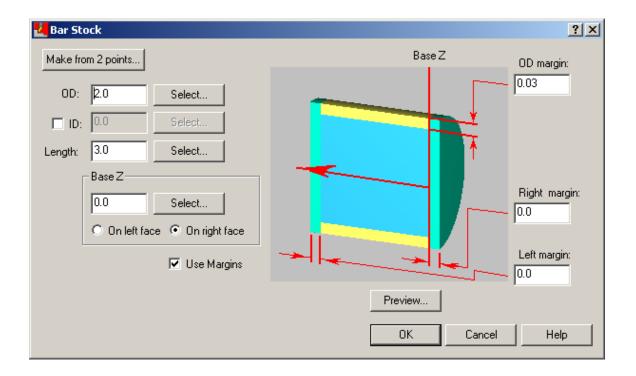
Operations List for Finished Part are listed below:



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.

General Boundaries Toolpath Configuration Tool Offsets Import Setup	tup
Import Setup	taries
✓ Include operation comments ✓ Add ○ From tool ✓ Import occup ✓ Generate toolpath immediately ✓ Tool/Dia.: ○ ✓ Views ✓ Assign tool numbers sequentially ✓ Back/Length: ○ ✓ Spindle/Turret ✓ Warn of duplicate tool numbers ✓ Seed Calculation ✓ Live Tooling ✓ Write home position clearance moves ○ Material ○ From tool ✓ Station number equals tool number ✓ Ref. Points	inport Setup Import Setup inport Setup Import Setup inport Setup Views inport Setup Spindle/Turret inport Setup Spindle/Turret inport Setup Lathe Tools inport Setup Live Tooling inport Setup Ref. Points
Post Processor Material C:\MCAM9\LATHE\POSTS\MAZ_TPL.PST Lathe Parameters Mill Parameters OK Cancel	ATHE\POSTS\MAZ_TPL.PST Material Material Lathe Parameters Mill Parameters



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:

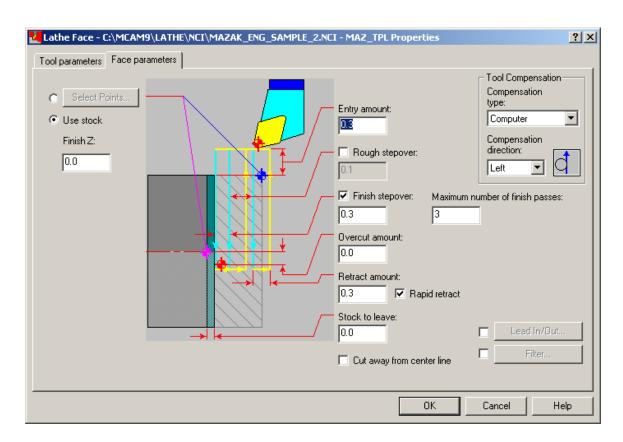
₩ 8 ₩_R M	AZAK EDIT	OR VERS	ION 6.0							_ _ ×
JOB										
UNO MAT			MIN LENG			FIN-Z WOR	K FACE			4
U ALU.	MIN 2.	U 	з. 	5000 	.005 	.005 .1			_	_
UNO MOD	E #	No. PI	TCH WID	TH FINIS	H RV	FV FEED	DEP.	R-T F-T		
										_
WORKPIE	CE MATE	RIAL <me< td=""><td>NU> ?</td><td></td><td></td><td>_</td><td></td><td>ALUMIN</td><td></td><td></td></me<>	NU> ?			_		ALUMIN		
WORKPIE				ALUMIN 4 S	TAINL 5	OTHER 6	7	ALUMIN	9 Spec	10 >
				ALUMIN 4 S	TAINL 5	OTHER 6	7	ALUMIN	Spec Fkt.	10 >
				ALUMIN 4 S	TAINL 5	OTHER 6	7	ALUMIN		10 >

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

🛃 Lathe Face - C:\MCAM9\LATHE\NCI\MAZAK_ENG_SAM	MPLE_2.NCI - MAZ_TPL Properties	٢
Tool parameters Face parameters		
Left 'click' on tool to select; right 'click' to edit or define new tool	Tool number: 1	
T0101 R0.0313 OD ROUGH RIGHT	Station number: 4 Feed rate: 0.01 • in/rev C in/min • micro-in Spindle speed: 200 • CSS • RPM Max. spindle speed: 5000 Coolant: Flood • Program #: 0 Comment: • • Seq. start: 100 • • •	
T0202 R0.0156 OD FINISH RIGHT OD ROUGH - HEXAGON	Stock Update To batch Ref. Points Change NCI Canned Text Coordinates Misc. Values System default <	
	OK Cancel Help	



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:

Mazatrol Material Line		×
– Material Line Paramete	NS	
Auto	MAT (Work piece Material)	
Auto	OD-MAX (Max. outer dia of work piece)	
Auto	ID-MIN (Min. inner dia of work piece)	
Auto	LENGTH (Work piece length)	
Auto	RPM (Max. spindle rpm limit)	
Auto	FIN-X (Finish allowance X)	
Auto	FIN-Z (Finish allowance Z)	
Auto	WORK FACE (Stock removal of work face)	
L		
	Cancel OK]

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:

Mazatrol Material Line		×
⊢ Material Line Paramete	18	1
Auto	MAT (Work piece Material)	
Auto	OD-MAX (Max. outer dia of work piece)	
Auto	ID-MIN (Min. inner dia of work piece)	
Auto	LENGTH (Work piece length)	
Auto	RPM (Max. spindle rpm limit)	
0.02 💌	FIN-X (Finish allowance X)	
0.01 💌	FIN-Z (Finish allowance Z)	
Auto	WORK FACE (Stock removal of work face)	
	Cancel OK	

When you click OK you will get the following message

MAZ_MIS	SC X
⚠	In Operations Manager, you need to make the Mazatrol Material line the very first operation
	ОК

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

Operations Manager	?)
2 Operations, 1 selected	<u>S</u> elect All
Toolpath Group 1 Toolpath Group 1 Toolpath Group 1 Toolpath Group 1 Tooloan Turning Tool - ROUGH FACE RIGHT - 80 DEG. Geometry C:\MCAM9\LATHE\NCI\MAZAK_ENG_SAMPLE_2.NCI - 2.9K Update stock C:\MCaM9\LATHE\NCI\MAZAK_ENG_SAMPLE_2.NCI - 2.9K C:\MCAM9\LATHE\NCI\NAZAK_ENG_SAMPLE_2.NCI - 2.9K C:\MCAM9\LATHE\NCI\NAZAK_ENG_SAMPLE_SAMPLE_2.NCI - 2.9K C:\MCAM9\LATHE\NCI\NAZAK_ENG_SAMPLE_SAMPLE_SAMPLE_SAMPLE_SAMPLE_SAMPLE_SAMPLE_SAMPLE_SAMPLE_SAMPLE_SAMPLE_SAMPLE_SAMPLE_SAMPLE_SAMPLE_SAMPLE_SAMPLE_SAMPLE_SAMPLE_SAMPLE_SAMPLE_SAMPLE_SAMPLE_SAMPLE_SAMPLE_SAMPLE_SAMPLE_SAMPLE_SAMPLE_SAMPLE_SAMPLE_SAMPLE_SAMPLE_SAMPLE_SAMPLE_SAMPLE_SAMPLE_SAMPLE_SAMPLE_SAMPLE_SAMPLE_SAMPLE_SAMPLE_	<u>B</u> egen Path <u>B</u> ackplot ⊻erify <u>P</u> ost <u>O</u> K <u>H</u> elp

Move to here

🛃 Operations Manager	? ×
2 Operations, 1 selected	<u>S</u> elect All
□	<u>R</u> egen Path
2 - Lathe Face	<u>B</u> ackplot
T0101: General Turning Tool - ROUGH FACE RIGHT - 80 DEG. Geometry	⊻erify
C:\MCAM9\LATHE\NCI\MAZAK_ENG_SAMPLE_2.NCI - 2.9K	<u>P</u> ost
	<u>0</u> K
	<u>H</u> elp

Sample output for material line and facing :

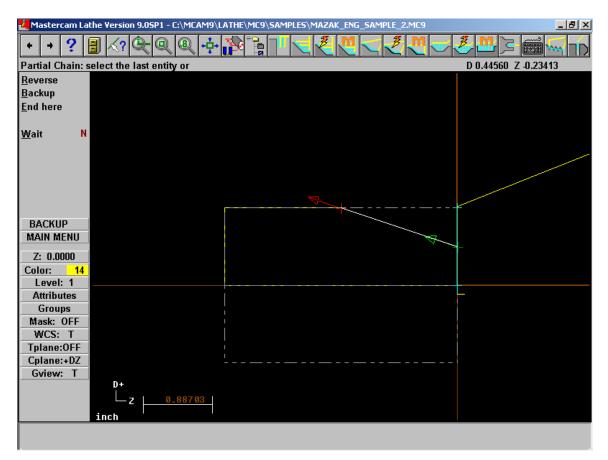
<i>ब्यू</i> ०००	1.MED - Notep	pad								
File E	dit Format H	łelp								
		,0 od-max 2.	000,0000 ID-MIN 0		RP M 5000		FIN-Z 0.01	WORK FACE		
	MODE EDG FCI		 РТ-Х •	SPT-Z 0.100	RV 50 FPT- 0	0.01	200.	R-DEP 0.10	R-T F- ⁻ 1 1 RGH	т
SEQ 1	MODE	COUNTER	RETUR	N W	κ. Ν ο. C	OUNT	NUM.	SHIFT		

Exercise 3 - Creating Rough and Finish Toolpath

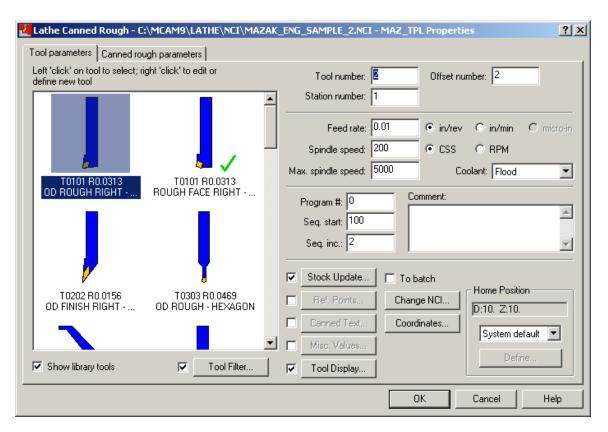
Select the following:

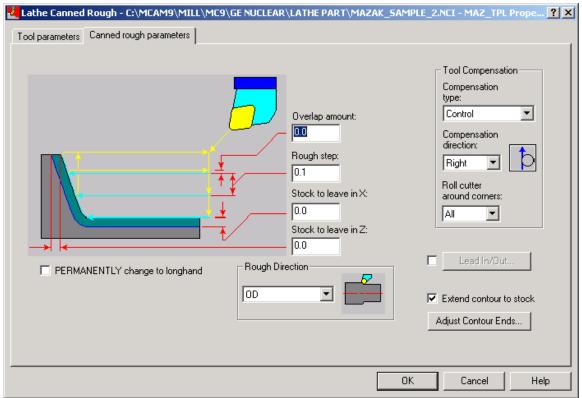
- 1. Main Menu
- 2. Toolpaths
- 3 Canned Rough

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.

<i>ब्यू</i> ०००	1.MED - Note	pad								_ 🗆 🗵
	dit Format H			0000						
PNR		,0(OD-MAX 2.					FIN-Z 0.01	WORK FA	CE	
UNO 1 SEQ 1	MODE EDG FC •	—	РТ-Х •	SPT-Z 0.100			R-FEED 01 PT-Z		R- ⁻ 1	Г F-T 1 RGH
2	BAR OU	# CPT T 0 2. -CNR SI * 1	Ō	_	RV 150 FPT-> 1. 2.) 200 (F 0	R-FEED 01 PT-Z .5		RADIUS	r F−T RGH
SEQ 3	MODE END	COUNTER 1	RETUR 1	N	WK.No. CC 1	UNT	NUM.	SHIFT		▼ //

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

Lathe Drill - C:\MCAM9\MILL\MC9\GE NUCLEAR\LATI Tool parameters Simple drill - no peck Custom Drill Parameter	
Left 'click' on tool to select; right 'click' to edit or define new tool	Tool number: 3 Offset number: 3 Station number: 0
T0101 0.25 Dia. CENTER DRILL25 DIA. CENTER DRILL5 DIA.	Feed rate: 0.01 Introv Introv Introvenia Spindle speed: 200 CSS Introvenia Max. spindle speed: 5000 Coolant: Flood Program #: 0 Comment: Introvenia Seq. start: 100 Introvenia Introvenia Seq. inc.: 2 Introvenia Introvenia
T0303 0.5 Dia. DRILL .5 DIA. CENTER DRILL75 DIA.	✓ Stock Update □ To batch □ Ref. Points Change NCI □ □ Canned Text Coordinates □ □ Misc. Values □ System default □ ✓ Tool Display □ □ □ □
	OK Cancel Help

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

Lathe Drill - C:\MCAM9\MILL\MC9\G	E NUCLEAR\LATHE PART\MAZAK	_SAMPLE_2.NCI - MAZ_TPL Properties
Tool parameters Simple drill - no peck Cu	ustom Drill Parameters 1	
	Depth	Drill Cycle Parameters Cycle: Drill/Cbore
×	Drill Point 0.0 Z: 0.0	1st peck increment: 0.0 Subsequent peck: 0.0 Peck clearance: 0.0
₩	Clearance 0.25 O Absolute O Incremental	Retract amount: 0.0 Dwell 0.0
	From stock	Shift: 0.0
	Retract 0.1 C Absolute Incremental	Drill tip compensation
	✓ From stock	Breakthrough amount: 0.25
		OK Cancel Help

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

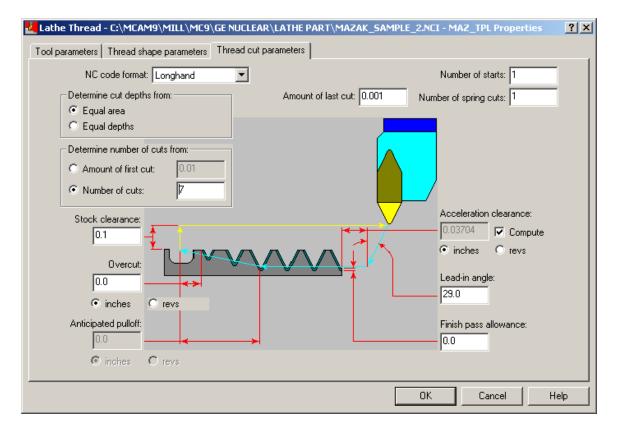
Exercise 5 - Threading Toolpaths

We will now Thread the ID

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

Lathe Thread - C:\MCAM9\MILL\MC9\GE NUCLEAR\LA	ATHE PART\MAZAK_SAMPLE_2.NCI - MAZ_TPL Properties
Left 'click' on tool to select; right 'click' to edit or define new tool	Tool number: 5 Offset number: 5
▲	Station number: 52
	Feed rate: 16.66667 O in/rev O in/min O micro-in
	Spindle speed: 200 CSS © RPM
T0404 R0.0144 T0505 R0.0052 OD THREAD RIGHT ID THREAD - MIN. 1.0 DIA.	
	Program #: 0 Comment:
	Seq. inc.: 2
	Stock Update
T0505 R0.0034 ID THREAD - MIN18 OD THREAD RIGHT	Bef Points. Change NCL. Home Position
	Canned Text
	Misc, Values
Show library tools	Tool Display
	OK Cancel Help

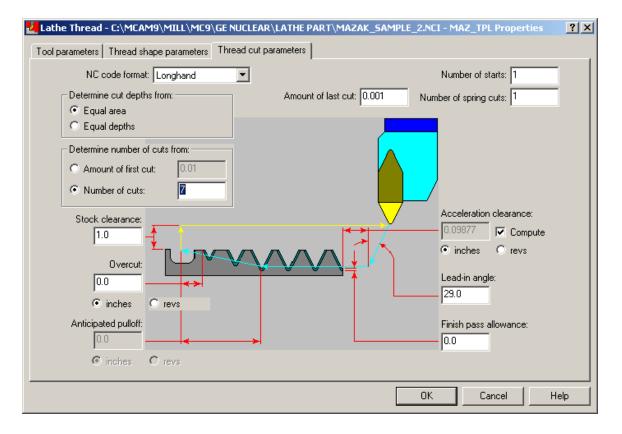
Lathe Thread - C:\MCAM9\MILL\MC9 Tool parameters Thread shape parameters	\GE NUCLEAR\LATHE PART\MAZAK_SAMI	PLE_2.NCI - MAZ_TPL Properties
End Position 1.0 Thread orientation: ID Taper angle: 0.0	Lead: threads/inch inches/thread Included angle: 60.0 Thread angle: 30.0 Major Diameter 0.74582 Minor Diameter 0.5 Thread depth: t Position	Thread Form Select from table Compute from formula Draw Thread Major/Minor Diameters Large end of taper Carge end of taper Small end of taper Small end of taper
		OK Cancel Help



Lathe Thread - C:\MCAM9\MILL\MC9\GE NUCLEAR\LA Tool parameters Thread shape parameters Thread cut param	ATHE PART\MAZAK_SAMPLE_2.NCI - MAZ_TPL Properties
Left 'click' on tool to select; right 'click' to edit or define new tool	Tool number: Offset number: 4
T0404 R0.0144 OD THREAD RIGHT T0505 R0.0052 ID THREAD - MIN. 1.0 DIA.	Station number: 4 Feed rate: 44.44444 in/rev in/min in/min in/mioro-in Spindle speed: 200 Max. spindle speed: 5000 Coolant: Flood Program #: 0 Seq. start: 100 Seq. inc.: 2
T0505 R0.0034 T4646 R0.0144 ID THREAD - MIN18 OD THREAD RIGHT	✓ Stock Update To batch ■ Ref. Points Change NCl ■ Canned Text Coordinates ✓ Misc. Values System default ✓ Tool Display Define
	OK Cancel Help

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

Lathe Thread - C:\MCAM9\MILL\MC9 Tool parameters Thread shape parameters	GE NUCLEAR\LATHE PART\MAZAK_SAMP	PLE_2.NCI - MAZ_TPL Properties ? X
	Lead: threads/inch inches/thread Included angle: 60.0 Thread angle: 30.0 Major Diameter Minor Diameter 1.7274 Thread depth: 0.1363 tPosition	Thread Form Select from table Compute from formula Draw Thread Major/Minor Diameters Large end of taper C Small end of taper Select From Table Major allowances 0.0
-2.25 -1.5 Thread orientation: OD Taper angle: 0.0	Negative X	Minor allowance: 0.0 Allowance tolerance: 0.0 OK Cancel Help



Mazatrol Settings: ? X MAIN PARAMETERS: THREADING PARAMETERS: Native Mazatrol Output type Thread cutting pattern (1-3) Auto Auto Chamfering: 60 <2>, 45 <1>, 90 <0> Thread lead Auto Auto Angle of thread Auto MULTI (Number of enterance) Auto Thread height Auto Number of cut passes Auto Cutting speed (per minute) Auto DEPTH (Depth of first cut) Machining part Tool number Auto SNO Line 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 Material Line ΟK Cancel Help

Mastercam to Mazatrol Post-Processor Tutorial11/1/2018

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.

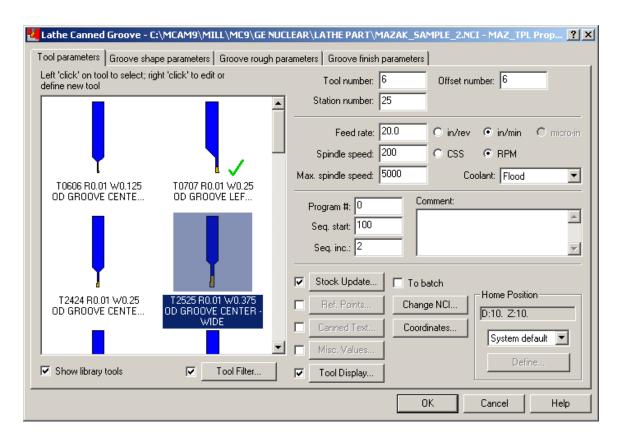
Select **OK** when done.

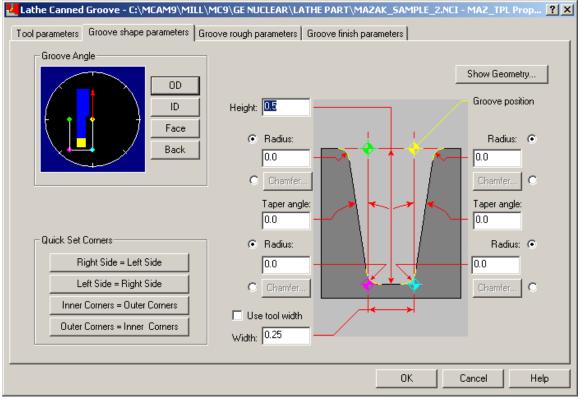
Exercise 5 - Creating Groove Toolpath

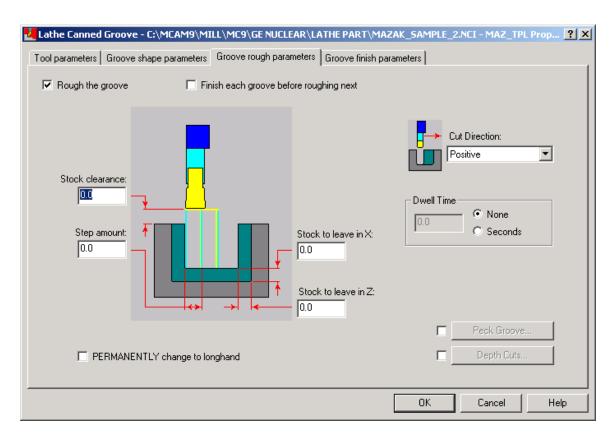
As with the Roughing and Finishing toolpaths it is generally unnecessary to have both a rough and a finish operation programmed in mastercam to get the correct output in Mazatrol

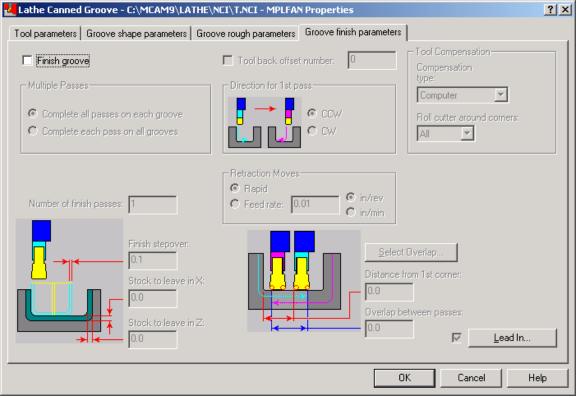
We will now create a 1 point groove on the OD. Select the following:

- 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

Lathe Cutoff - C:\MCAM9\MILL\MC9\GE NUCLEAR\LAT	THE PART\MAZAK_SAMPLE_2.NCI - MAZ_TPL Properties
Tool parameters Cutoff parameters	
Left 'click' on tool to select; right 'click' to edit or define new tool	Tool number: 7
	Station number: 30
	Feed rate: 20.0 O in/rev O in/min O micro-in
	Spindle speed: 200 C CSS © RPM
T0606 R0.01 W0.125 T0606 R0.01 W0.375	Max. spindle speed: 5000 Coolant: Flood
OD GROOVE CENTE OD GROOVE CENTE	Program #: 0 Comment:
	Seq. start: 100
	Seq. inc.: 2
	Stock Update
T0707 R0.01 W0.25 DD GROOVE LEF DD GROOVE CENTE	Bef Points
	Misc. Values
Show library tools	Tool Display
	OK Cancel Help

		ATHE PART\MAZAK_SAMPLE_2.NCI - M.	AZ_TPL Properties
Tool parameters Cutoff paramet	ers		
	Entry amount: 0.1 Retract Radius None Absolute: Incremental: X Tangent Point	0.0	Tool Compensation Compensation type: Control
	Cut to: Front radius Back radius	Corner Geometry None Radius: Chamfer Parameters Clearance Cut	Canned text Peck Lead In/Dut Filter
		OK	Cancel Help

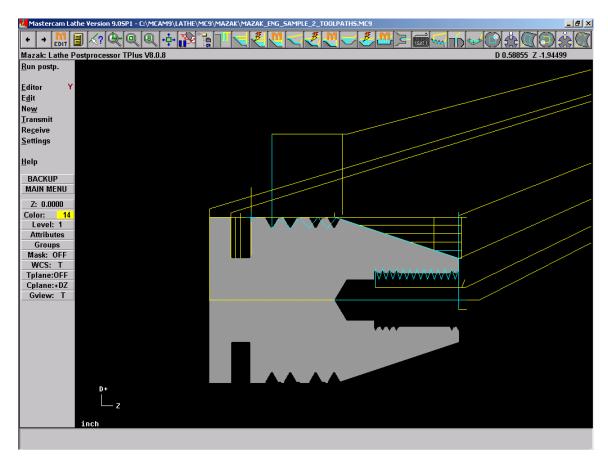
6. Select **Done**. This should return you to the operations manager. Select **Post** Modify settings as shown below. (In this example we are using the TPlus post-processor shown as MAZ_TPL.PST. Yours may vary but all the Mazatrol Post-Processors will have the format of MAZ_XXX.PST)

Post processing	<u>? ×</u>
Active post	Change Post
MAZ_TPL.PST	
NCI file	
Save NCI file	🗖 Edit
C Overwrite	
🖲 Ask	
NC file	
🔽 Save NC file	🔽 Edit
Overwrite	NC extension
C Ask	.nc
– Send	
Send to machine	Comm
OK Cancel	Help

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)

	1.MED - Notep dit Format He							
0001 PNR	L, MAT	OD - N	,0000,000 IAX ID-MIN 0	00,0000 N LENGTH 3.	RPM F 5000 0	IN-X FIN-2 .02 0.01	Z WORK FACE .1	<u> </u>
UNO 1	MODE EDG FCE				RV	EV R-FE	ED R-DEP 0.10	R-T F-T 1 1 RGH
UN0 2 SEQ 1 2	MODE BAR OUT SHP S- LIN TPR	# -CNR	CPT-X CF 2. 0 SPT-X + 1.	PT-Z SPT-Z ★ 0	RV 150 FPT-X 1. 2.	FV R-FEF 20001 FPT-Z 0 1.5	ED R-DEP .1 F-CNR RADIU	R-T F-T 2 2 IS RGH
UNO 3 SEQ 1	MODE DRL FCE	# E 0	DRL-DI4 .5	A DEP-1 0 SPT-Z 0.	DEP-2 0	DEP-3 0 FPT-Z 1.5	V FEED 200 .01	тооL 3
UNO 4 SEQ 1	MODE THR IN	# 0	CHAMF L 0 J SPT-X .5	EAD ANG L2. 60 SPT-Z 0	MULTI 1 FPT-X .5	HGT No. .122 10 FPT-Z 1.	V DEPTH 180 .01	TOOL 5
UNO	MODE	#		EAD ANG	MULTI	HGT No.		TOOL 4
UNO 6 SEQ 1	MODE GRV OUT . S-	# 0-CNR	No. PITCH 1 SPT-X 2.104	H WIDTH 1.09 SPT-Z 2.63	FINI RV .01 150 FPT-X 1.01	FV FEED 250 .1 FPT-Z 2.63	DEP. .12 F-CNR ANGLE	R-T F-T 6 6 RGH
UNO 7	MODE GRV OUT	r 10	No. PITCH	H WIDTH 2.22	FINI RV .01 150	FV FEED 250 .1	DEP. .12 F-CNR ANGLE	R-T F-T 7 7

12. Close this window.

We will then send this program to the controller

13. From the Mazatrol Menu select Transmit.

Communi	ication				×
_ PC → C					
File		0001.MAD		Transmit	
Cont	roller	TPLUS			
СОМ	l Port	COM1	•		
Baud	drate	4800	•		
	Convert ASCI before transm	l (MED) into Bi ission	nary (MAD)	Cancel	

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

<mark>de</mark> l	Untitled	- Ue_maz				[<u> </u>
?							
	No.	Part no.	Block	Name	Program F	PC -> CNC	
		Commu	nication star	t by Mazak CNC,	Break ==>	> ESC	
					Com:	Kanal:	
Bere	it						

This is the progress bar.

To complete the download complete the following steps at The <u>Mazak Controller</u>.

- 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.

Mazatrol Settings:	? ×
MAIN PARAMETERS:	GROOVING PARAMETERS:
Native Mazatrol Output type	Auto Groove cutting pattern (0-5)
	Auto Vumber of grooves
	Auto PITCH (Grooving pitch)
	Auto VIDTH (Width of groove)
	Auto 🗾 FINISH (Finish allowance)
	Auto RV (Surface Speed for rough cut)
	Auto V (Surface speed for Finish cut)
	Auto FEED (Feedrate)
not specified Roughness	Auto R-T (Roughing Tool no)
Auto Machining part	Auto F-T (Finishing Tool No)
r SNO Line	
1. Parameter: 2. Parameter: 3. Parameter: 4. Parameter: 5. Parameter:	6. Parameter: 7. Parameter: 8. Parameter: 9. Parameter: 10. Parameter:
Auto V Auto V Auto V Auto V	Auto 💌 Auto 💌 Auto 💌 Auto 💌
Material Line Parameters	terial Line OK Cancel Help

<i>2</i> 1000	1.MED - Notep	ad								_ & ×
	dit Format He			_	_					-01~
PNR	L, MAT ALUMIN	,00 OD-MAX 2.	000,0000 ID-MIN 0	,0000 LENGTH 3.	RPM 5000	FIN-X .005			FACE	*
1			РТ-Х	.2 SPT-Z	FINIS .01 1 FPT-2 1.01	50 250 × FF	200 PT-Z	.12	DEP. R ANGL	R R
SEQ 2 *	MODE END	COUNTER 1	RETUR 1	N 1	WK.No. Ci	OUNT N	NUM	SHIFT		
•										× •

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

Mazatrol Settings:	<u>? ×</u>
MAIN PARAMETERS:	GROOVING PARAMETERS:
Native Mazatrol Output type	2 Groove cutting pattern (0-5)
	3 Number of grooves
	1.5 PITCH (Grooving pitch)
	Auto VIDTH (Width of groove)
	Auto FINISH (Finish allowance)
	200 RV (Surface Speed for rough cut)
	166 FV (Surface speed for Finish cut)
	Auto Auto In FEED (Feedrate)
not specified Roughness	Auto 🗾 R-T (Roughing Tool no)
Auto Machining part	Auto F-T (Finishing Tool No)
SNO Line	
1. Parameter: 2. Parameter: 3. Parameter: 4. Parameter: 5. Parameter	: 6. Parameter: 7. Parameter: 8. Parameter: 9. Parameter: 10. Parameter:
Auto Auto Auto Auto	r Auto V Auto V Auto V Auto V
└ Material Line Parameters	
	aterial Line OK Cancel <u>H</u> elp

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

Mastercam to Mazatrol Post-Processor Tutorial11/1/2018	
Image: State	_B×
0001, ,0000,0000,0000 PNR MAT OD-MAX ID-MIN LENGTH RPM FIN-X FIN-Z WO 0 ALUMIN 2. 0 3. 5000 005 .005 .1	
UNo MODE # No. PITCH WIDTH FINITSH RV FV FEED 1 GRV OUT 2 3. 1.5 .2 .01 200. 166. 200 .1 SEQ S-CNR SPT-X SPT-Z FPT-X FPT-Z F 1 2.104 2.63 1.01 2.63	
SEQ MODE COUNTER RETURN WK.No.COUNT NUM. SHI 2 END 1 1 1 *	FT
•	▼ ▶

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