

POST PROCESSOR MANUAL FOR



NT(X) Series



This manual was prepared with the assumption that the intended reader does have working knowledge of ESPRIT and machine programming experience so that he fully understands the information it contains.

Table of Contents

1.	Intro	oduction	. 1
2.	Set	ting up the Esprit Environment	. 1
2.	1.	Required Add-ins	. 1
	2.1.	1. How to Load an Add-in in ESPRIT	. 1
2.	2.	Mori Seiki ESPRIT for Mapps Addin	. 4
2.	3.	Stock Automation	. 5
2.	4.	AutoSubStock (optional)	.7
2.	5.	Turning Work Coordinates Addin	. 8
	2.5.	1. Setting Up the Turning WC addin	. 8
2.	6.	AutoPark Addin (required on NT1000 machines)	12
3.	Mad	chine Setup	14
3.	1.	Introduction	14
3.	2.	Set the Program Name, Program Number	14
3.	3.	Define the Turning Stock	15
3.	4.	Tool Path Optimization with NURBS	17
3.	5.	Selecting Post Processors and Outputting NC Code	18
3.	6.	Machine Parameters	20
	3.6.	1. Introduction	20
	3.6.2	2. Output of Program End (M02), Sub-Program End (M99) or Program End and	
	Rew	vind (M30)	20
	3.6.	3. TCP Type	21
	3.6.4	4. Set position of optional stop code (M01) in the NC code	21
	3.6.	5. Station on Turret 2 for Transfer	21
	3.6.0	6. Tail Stock Type	22
	3.6.	7. Block Skip Method	22
	3.6.8	8. C-Axis Rollover	22
4.	Wo	rk Coordinates	24
5.	Тоо	Is (T-function)	26
5.	1.	T-function Specifications	26
	5.1.	1. Upper Head (B-Axis Head):	28
	5.1.2	2. Lower Turret:	29
5.	2.	Tool Life Management	30

5.3.	Tool Nose Direction (G43 H_ T_)	32
5.4.	Second home position	33
5.5.	G361/362 Tool Change Command Setting	
5.6.	Tool Change position	38
6. Sp	indle Direction, Speed, and Feedrate	39
6.1.	Turning Spindle Direction and Milling Tool Spindle Direction	39
6.2.	Spindle Speed output for CSS Unit	41
6.3.	Feedrate for 4-axis Milling Operations	42
6.4.	Feedrate option for rapid moves in 5 axis operations	43
6.5.	Turning on AI Contour Control	43
7. 5-a	axis Simultaneous Milling Options	45
7.1.	Introduction	45
7.2.	Definitions	45
7.2	2.1. Rotate With Part	
7.2	2.2. Fixed	
7.3.	Operation Types	46
7.3	a.1. RTCP Type 1 WKP=0 (G43.4)	
7.3	2.2. RTCP Type 1 WKP=1 (G43.4)	
7.3	3.3. RTCP Type 2 WKP=0 (G43.5)	47
7.4.	Additional Information	48
8. Sir	multaneous Turning Operations: Turning on the same spindle with two	turrets49
8.1.	Balance Roughing Cycle	
8.2.	Other Turning Cycles	51
9. Pa	rk Operation	53
9.1.	Introduction	53
9.2.	How to park a tool	53
9.3.	Stop Code Options	56
9.4.	Setting the B axis angle	57
9.5.	Park head/ turret to its minimum travel position (G53)	58
9.6.	Park lower turret prior to a transfer (Z configurations only)	60
9.7.	Setting up the park operation with the AutoPark addin	61
10. I	How to Program a Part Transfer	62
10.1.	Introduction	62
10.2.	SP1-CutOff-BarFeed-First	64

10.3. SP	1-CutOff-BarFeed-Last	66
10.4. SP	1 To SP2- BarFeed-Cutoff	68
10.5. SP	1 To SP2- BarPull-Cutoff	70
10.6. SP	1 To SP2-Cutoff-BarFeed	72
10.7. SP	1 To SP2-Release	74
10.8. SP	2 To SP1-Release	76
10.9. Op	eration Tabs in the Workpiece Transfer Utility	78
10.9.1.	Park	
10.9.2.	Part Eject	79
10.9.3.	Bar Feed	
10.9.4.	Pick Up	81
10.9.5.	Bar Pull	
10.9.6.	Cutoff	83
10.10. C	Custom Settings for Part Transfer Cycles	
10.10.1.	Workpiece Pushing Check (G38)	85
10.10.2.	C axis angle for pick operation	86
10.10.3.	Bar Feed positions relative to G53	87
10.10.4.	Workpiece Ejector (NT-S and NT-SZ Only)	
10.10.5.	Work Coordinate for Pickup Operation	89
10.10.6.	Smart Loader Shutter (M37) (NT1000 only)	
11. Workir	ng with Long Parts: Pickup\Barpull then Simultaneous Machining	91
12. Stock	Repositioning	
13. How to	Output Coolant Codes	95
13.1. Intr	oduction	95
13.2. Set	tting First Coolant Code	95
13.3. Se	tting the second coolant code (optional)	97
13.4. Set	tting the High Pressure Coolant Pressure options	97
13.4.1.	Upper Head Option	
13.4.2.	Lower Turret Options	
14. Multipl	e Repetitive (Canned) Cycles	100
14.1. Intr	oduction	100
14.2. Ro	ughing Cycle	102
14.2.1.	Introduction	102
14.2.2.	O.D./I.D. Roughing	105

14.2.3	3. Face Roughing	106
14.3.	Grooving Cycle	107
14.3.	1. Introduction	107
14.3.2	2. Face Grooving	110
14.3.	3. O.D./I.D. Grooving	111
14.4.	Threading Cycle	112
14.4.	1. Introduction	112
14.4.2	2. Threading with Canned Cycle Set to Single Path	114
14.4.3	3. Threading with Canned Cycle Set to OFF	116
14.4.4	4. Threading with Canned Cycle Set to Multiple Path	118
15. Ba	axis Turing Cycle	120
16. Ho	le Machining Canned Cycles	122
16.1.	Introduction	122
16.2.	Deep Hole Drilling Cycle (G74)	126
16.3.	Face and Side High-Speed Deep Hole Drilling Cycle (G83.5/G87.5)	127
16.4.	Face and Side Deep Hole Drilling Cycle (G83.6/G87.6)	129
16.5.	Face and Side Spot Drilling Cycle (G83/G87)	131
16.6.	Face and Side Tapping Cycle (G84/G88)	132
16.7.	Tapping at Center of Spindle (G32)	133
16.8.	Synchronized Tapping (M329)	134
16.8.	1. Limitations of Synchronized Tapping	134
16.8.2	2. Face and Side Synchronized Tapping (M329 G84/G88)	134
16.8.	 Face and Side (High-Speed) Deep Hole Synchronized Tapping (M329 C 136 	384/G88)
16.8.4	4. Spindle Synchronized Tapping (M329 G84) (Option)	137
16.8.	5. Face and Side Boring Cycle (G85/G89)	139
16.8.0	6. Boring in Turning Mode	140
16.9.	Spindle/Rotary Tool Spindle Simultaneous Operation Mode	141
17. 3D	Coordinate Conversion (G68.1)	144
18. 4-/	Axis Wrap Milling Cycles with Interpolation	144
18.1.	Introduction	144
18.2.	Cylindrical Interpolation	144
18.3.	Polar Interpolation	146
19. Ta	ilstock	148
19.1.	Live Center on Lower Turret	148

19.1.	1. Defining Tailstock in Machine Setup	148
19.1.2	2. Creating Tailstock Operation	149
19.2.	Second Live Center on Lower Turret	. 150
19.3.	Independent Tail Stock	. 152
19.3.	1. Defining Tailstock in Machine Setup	152
19.3.2	2. Creating Tailstock Operation	153
19.4.	Live Center on Sub Spindle	. 154
20. Ste	eady Rest	. 154
20.1.	Lower Turret Steady Rest	. 154
20.1.	1. Defining Steady Rest in Machine Setup	154
20.1.2	2. Defining Steady Rest in Machine Setup Utility (Mori Seiki Add-in)	156
20.1.3	3. Creating Steady Rest Operation	157
20.2.	Independent Steady Rest	. 159
20.2.	1. Defining Steady Rest in Machine Setup	159
20.2.2	2. Defining Steady Rest in Machine Setup Utility (Mori Seiki Add-in)	160
20.2.3	3. Creating Steady Rest Operation	161
20.2.4	4. G478 Steady Rest/Sub Spindle Synchronize Command (NT-S 6000 only)	163
21. Sp	ecial considerations when Programming an NT1000	. 164
21.1.	Introduction	. 164
21.2.	Mori Seki Macro Executer update required to work with ESPRIT 2012 and	d
newer		. 164
21.3.	Setting the Sub Spindle work offset value in the machine control	. 166
21.4.	Notion Axis setup on ESPRIT for sub spindle operations on upper nead.	. 166
21.5.	Setting the work Support/Sub Spindle angle (Bvv Axis)	. 167
21.6.	Conditions need to allow the Moving of the BW axis	. 169
21.7.	Conditions need to allow the Moving of the Baxis	. 169
21.8.	Setting A-axis Position with the AutoPark Addin	. 170
21.9.	Parking the Lower Turret	. 173
21.10.	Parking the A and Bw axis Prior to Tool change	. 175
21.11.	Parking the B axis head Prior to Bw Angle Change	. 177
21.12.	Parking the BW axis Prior to part transfer	.178
21.13. head:	iviaximize clearance when on the face of the main spindle with the B ax 179	IS
21.14.	Part Release from Sub Spindle	. 180
21.15.	Location of part release within the ESPRIT file:	. 182

2 ⁻ N	1.16. T1000-	Turret synchronization requirements for programming N	T1000-WZ and 184
	21.16.1 operati	Parking the lower turret before a BW axis angle change mac n: 184	de by an upper turret
	21.16.2 spindle	Machining on the Main Spindle with the B axis head while with lower turret:	machining on sub 184
	21.16.3 spindle	Machining on the sub spindle with the B axis head while ma with the lower turret:	achining on the main 185
	21.16.4 turret o	Changing BW axis angle with B axis head operation while the main spindle	machining with lower
	21.16.5	Changing A axis position between two lower turret operation	ons in a row: 186
	21.16.6	Machining with one head\ turret while other head\turret is w	vaiting: 186
	21.16.7	Part release on sub spindle	
	21.16.8	Part Transfer	
	21.16.9 operati	When in Long Part Machining mode and changing from Tu n on the same head\turret	rning to MillTurn 188
22.	Cus	om Settings Index	
2	2.1. (perations Page	
2	2.2.	ools Page	

1. Introduction

This is for ESPRIT 2012 or newer post processors and requires ESPRIT 2012 R3 or newer (Build B19.0.3.1325 or above.

Please refer to NT programming manual for detailed information on the related G and M-codes.

2. Setting up the Esprit Environment

2.1. Required Add-ins

2.1.1. How to Load an Add-in in ESPRIT

To load an add-in in ESPRIT, from the Tools menu, select Add-In...:



You will then see the add-in window:

Available Add-Ins	Load Beha [,]	vior	OK
3D Connexion Motion Controller AutoSubStock Citizen Cut20 Technology Add-In EDM Features Group Manager Monster MoriSeiki AddIn Smart Toolbar Soft Syncing Add-In SpinningTurningTool Addin Turning Work Coordinates	Startup / Lo Startup / Lo Startup / Lo	oaded oaded oaded	Cancel Help
escription		- Load Behavio	r
Adds support for the 3D Connexion Motion Contro space ball)	oller 🔺	Loaded/L	Jnloaded Startup I Line

To load an add-in, highlight it in the Available Add-Ins list and check in Load Behavior the box Loaded/Unloaded.

Add-In Manager		X
Available Add-Ins 3D Connexion Motion Controller AutoSubStock Citizen Cut20 Technology Add-In EDM Features Group Manager Monster MoriSeiki AddIn Smart Toolbar Soft Syncing Add-In SpinningTurningTool Addin Turning Work Coordinates	Load Behavior Startup / Loaded Startup / Loaded Startup / Loaded Startup / Loaded	OK Cancel Help
Description Create Work Coordinates for Turning F	ile	or /Unloaded Startup nd Line

Check box Load on Startup so the addin will automatically load each time you start ESPRIT

For the Mori Seiki NT series, you will need to load the following add-ins:

• Turning Work Coordinates

Optional Addins that can also be loaded if needed:

- Spinning Turning Tool Addin
- AutoSubStock

2.2. Mori Seiki ESPRIT for Mapps Addin

Installation of the Mori Seiki ESPRIT for MAPPS Addin is required. The Mori Seiki addin has a number of components that are required for the programming of the Mori Seiki NT machines.

The Mori Seiki ESPRIT for MAPPS Addin is installed with the Mori Seiki Global installer.

The Mori Seki AutoPark addin is integrated with the ESPRIT Utilities.

ESPRIT for MAPPS addin version I60011C04 or newer is required

lame	Version
ESPRIT	19.0.4.1343
MORISEIKI AddIn	I60011C04
	Class

2.3. Stock Automation

In ESPRIT, from the Tools menu, select Options...



On the Machining tab, if you check Enable Stock Automation, The AutoSubStock add-in should not be used since the Stock Automation will compute the state and shape of your stock present in both the spindle 1 and spindle 2:

ptions	
Attributes Input Advanced Machining 1 space File Locations	
Tech Page Defaults	٦
🔽 Custom Page	
Stock	
Enable Stock Automation	
Stock Automation Tolerance 0.1	
Stock Transparency	
OK 3 Cancel Default Help	_
	_

Please refer to the ESPRIT help file for additional information on stock automation.

2.4. AutoSubStock (optional)

Important note: Note: If the Stock Automation function (above) is used, do not use the AutoSubStock Addin.

The AutoSubStock add-in needs to be turned on prior to opening a file. This add-in will allow you to correctly simulate the sub spindle work. It will also allow the simulation of production work, when cutting on the main and the sub spindles at the same time.

Once you are done programming a part, simply play the entire simulation and once complete click on Auto Sub Stock. This will save the sub stock in the sub spindle and stop the simulation.



If you restart the simulation, you will be able to see the cut on the main and sub spindles side all at the same time.

2.5. Turning Work Coordinates Addin

The Turning Work Coordinate add-in will sort your operations in the appropriate work coordinates and also offset the NC code for the sub spindle operations. It is necessary to run this add-in in order to generate correct NC code.

When the Addin is run the operations that have the Main Spindle selected will go under the Work Coordinate (WC) created for the Main Spindle and the operations that have the Sub Spindle. If an operation(s) are machined with the B axis at an angle other than -90, 0, +90 will be placed in a "Baxis" version of the WC for the spindle the "tilted" operation is working on.

2.5.1. Setting Up the Turning WC addin

When first staring a new file in Esprit from the Create menu, select Turning Work Coordinates.



You will then see the Turning Work Coordinates dialog:

E Turning Work	Coordinates		×	
_ MainSpindle		_ SubSpindle		
WC Name	G54	WC Name	G55	
WC Numbers	54 0	WC Numbers	55 0	
Z Offset	0	Z Offset	-36	
Spindle Orientation	Z+	Spindle Orientation	Z-	
Options				
Keep Z axis par	allel with Tool axis		/ith Tilted Orientation	
Reverse Z axis of WC if spindle Z axis is reversed				
Autorun Mode		BI	EFORE POSTING	
			OK Cancel	

The Turning WC addin will look at your Machine Setup file and determine how the dialog box needs to be setup for your machine. The Spindle names and orientation information in the dialog box are taken directly coming from the Machine Setup.

• WC Name: Corresponds to the Work Coordinate name. Please note that the name is just informative and will not affect the NC code. This name will appear in the feature list inside of Esprit.

• WC Numbers: Both fields are used by the post. The Machine control has two sets of offsets that can be used. The standard work coordinates (G54-G59) and the extended work coordinates (G54.1 Pxx)

Standard WC Field		Additiona WC Field	al l
Turning Work Coordinates			×
MainSpindle	SubSpindle		
WC Name G54	WC Name	G55	•
WC Numbers 54 0	WC Numbers	55	0
Z Offset 50	Z Offset	-85.725	
Spindle Orientation	Spindle Orientation	Z-	
Options			
Keep Z axis parallel with Tool axis		With Tilted Oriental	tion 💌
Reverse Z axis of WC if spindle Z axis is revers	ed	NO	-
Autorun Mode		BEFORE POSTING	& SIMUL/ 👻
		ОК	Cancel

• The Standard WC number is used to output G54-G59. Enter 54 to have G54 in NC code. 55 to have G55 in NC code, etc. If a machine is equipped with 2 spindles a value must be set for both spindles. When using the machines Standard Work Coordinates (G54-G59) a zero (0) must be entered into the Additional offset field.

Note: The NT1000 post has G59 is reserved for use on the Sub spindle when it is tilted. If programming an NT1000 do not enter 59 into a WC in Esprit).

• The Additional WC number is used to output extended work coordinate in the NC code (G54.1 Pxx). For example if a 1 is entered into the Additional WC filed a G54.1 P1 will be output in the NC code. When outputting the extended work coordinates (G54.1 Pxx) a value of zero (0) must be entered into the Standard WC field.

• Z Offset: Used to setup the location of the work coordinates in Esprit. This is most commonly only used for the sub spindle.

- Main Spindle: Typically set to zero. This means the program zero for main spindle is P0 (World Zero) in Esprit.
- Sub Spindle: The sub spindle value is typically set to the negative value of the finish part length. The value is pre-loaded from the Part Stock Length field of the Machine Setup (on the General tab). For example if the finish part length is 100mm you would enter -100 in the Sub Spindle Z offset Field. The work coordinate will be offset by minus this amount from P0.

• The Z offsets MUST have the correct values in them to have correct Z axis values in your NC code.

Machine Name NT4200/1500SZ Stock: Type Bar Start Position Z 0 Bar Diameter 100 Start Position X 0 Incide Diameter 5 Start Position Y 0 Casting Feature 110	Maximum Diameter Total Bar Length Part Stock Length	200 200 36	_
Start Position Z 0 Bar Diameter 100 Start Position X 0 Inside Diameter 6 Start Position Y 0 Casting Feature 110	Total Bar Length Part Stock Length	200 36	=
Start Position X 0 Inside Diameter 6 Start Position Y 0 Casting Feature 1 MilTum Rotary Retract Movement Maceilaneous 1	Part Stock Length	36	
Start Position Y 0 Casting Feature			- 1
MilTum Rotary Retract Movement Macellaneous			
a card the second secon			
Retract Position Tool Change • NC Offset Simulation Along Lead-in/out •	Dogleg Rapid	Off	•
Potary Descance 2 Optimize Tool Path Of Park Position Optimize Tolerance 0.0003937007874015	Rotary Index Time	0.28	
X.Y.Z 0 0 0 RTCP Coordinate Mode Fixed	-		

• Spindle Orientation: This information is coming from the Machine Setup and is just informative.

E Turning Work Coordinates	X
MainSpindle WC Name G54 WC Numbers 54 0 Z Offset 50 Spindle Orientation	SubSpindle WC Name G55 WC Numbers 55 0 Z Offset -85.725 Spindle Z- Orientation
 Options Keep Z axis parallel with Tool axis Reverse Z axis of WC if spindle Z axis is rever Autorun Mode 	with Tilted Orientation
	OK Cancel

Options Section:

- Keep Z axis parallel with Tool axis: This MUST be set to "With Tilted Orientation"
- Reverse Z axis of WC if spindle Z axis is reversed: MUST be set to NO
- Auto Run Mode: it is best to have the "Before Posting and simulation" Option selected. This will cause the addin to run every time you simulate and post NC code and ensure that the operations fall under the correct WC, this ensuring your NC code is correct.

2.6. AutoPark Addin (required on NT1000 machines)

The ESPRIT AutoPark addin was designed to help users park the turrets to the extreme travel limits without having known the exact location of the travel limits in the machine coordinate system. The NT machines are setup to use the Auto Park addin to positions the Head\Turret to the left most position in the travel envelop of the NT machines.

In addition the AutoPark addin is used to position the A axis (Sub Spindle) in the NT1000 machines to ensure correct NT code and simulation.

The AutoPark Addin is only for use with ESPRIT 2012 or newer and the ESPRIT 2012 or newer Mori Seki addin.

In addition the AutoPark addin will only function correctly if you have the latest Machine Setups created for the ESPRIT 2012 or newer packages. ESPRIT files created prior to ESPRIT 2012 will need the Machine Setup pages updated to use the AutoPark addin.

For information as to how to use the addin please refer to the <u>Setting up the park operation with</u> <u>the AutoPark addin</u> and <u>Setting A-axis Position with the AutoPark Addin</u> sections of this manual.

3. Machine Setup

3.1. Introduction

Some important settings regarding the NC output are set in the Machine Setup.

To open the Machine Setup, click on Common Machining and then on Setup:



3.2. Set the Program Name, Program Number

To set the program name and number output at the beginning of your NC code on each channel, you will need to go to the NC Output tab of the Machine Setup. Under General Properties of the Turret Program Output, enter the name of your program, its number. You can also specify here the unit of your NC code and the coordinate mode.

Ge	neral Assembly NC Out	put C 3					
	NC Output Order	Turret	Number of Hea	ads 1	+ Head IC	Head-1	¥
ſ	Sync String Codes			Sync Codes	10		
	Sync ID and Name	· · ·		Start and End N	umber 100	900	
L		N	Turnet Dee	arran Outau t			
	Program Number	1000 ships	Unit for NC Coo	de Metric			
	R-Avis Head	Shared •	Shared •	Shared -	Shared *	Shared -	
	Bottom Turret	Shared •	Shared 💌	Shared v	Shared 👻	Shared -	
		Shared 👻	Shared 👻	Shared 💌	Shared 👻	Shared -	
	l l	Shared 💌	Shared 💌	Shared 💌	Shared 💌	Shared 💌	
		Shared 🔄	Shared 💌	Shared 💌	Shared 💌	Shared 💌	
			N 2-				

3.3. Define the Turning Stock

On the General tab of the Machine Setup, you can define your turning stock. For turning operations, if you are using the Stock Type Automation, your NC output will be linked to the defined turning stock.

Use Start Position Z to position the stock along the Z axis. It will be used, for example, to define the front face facing amount.

Use Stock Type to define the shape of your stock: Bar, Tube (Inside Diameter will then be available) or Casting (Casting Feature will then be available for selection).

Use Bar Diameter to specify the diameter of your stock. Use Total Bar Length to define the total length of your stock.

Finally use Part Stock Length to define the length of your finish part. This will be used by the Turning Work Coordinates add-in.

eral Assembly N	C Output Custom					
Machine Definition -		Stock Configuration	-	_		
Machine Name	NTX1000-SZM	Stock Type	Bar	Maximum Diameter	60	
Start Position Z	1	Bar Diameter	50.00000	Total Bar Length	87.725	
Start Position X	0	Inside Diameter	152.4	Part Stock Length	88 725	1

Note: ESPRIT supports a number of different stock configurations. Please consult the ESPRIT Help file for detailed information.

In order to use the stock configuration from the machine setup dialog box you must have the Simulate Turning Stock field (Found in the Simulation Parameter dialog box) checked (as shown below):

Status Display	Step Control				
Coolant Status	Multi Step Mode	Deerstion	Bi	ocks	10
Diameter Comp. Feed Rate	Image: Strength of the strengt of the strength of the strength of the strength of the				
Operation Comment	- Miscellaneous				
Operation Number	Collision Detection	None	▼ To	lerance	0.2
Operation Type Power Spindle Direction Spindle Speed	Simulation Type	Full	*		
	Simulate Threads	Segment	•		
	Plot Geometry	1	Simulate	Turnina	Stock
Tool Comment	Trace Tool Path	1	Automat	ic Slug F	lemoval
Tool Coordinate	- Colors -			Spee	d
Tool Id	Rapid Move			1:	Fast
Tool Number	Rest Material		1	+	
Wire Angles	Tool			1	
Wire Coordinates	Tool Shank			1	
Work Coordinate	Inactive Tool		•	1	
	Transport			1.00	Slow

3.4. Tool Path Optimization with NURBS

With the Mori Seiki NT series machines, it is possible to translate non-uniform rational B-splines (NURBS) used in modeling sculptured or curved surfaces directly in NC units. This is especially helpful when programming complex three dimensional molds and tool dies.

To turn on NURBS Optimization in ESPRIT, navigate to the Machine Setup page and the General tab. Under "Miscellaneous" there is a pull down labeled "Optimize Tool Path" open this and scroll down to "NURBS Curves" to select NURBS optimization. Click OK to complete this function.

Machine Setup General Tab

	DUT LODG LLODGET	Stock Configuration			Inco
Machine Name	NT4250/10005Z	Stock Type	Bar	Maximum Diameter	250
Start Position Z	0	Bar Diameter	101.000000	Total Bar Length	150
Start Position X	0	Inside Diameter	152.4	Part Stock Length	75
Start Position Y	0	Casting Feature			
MillTum Rotary Retrac	t Movement	Miscellaneous			
Retract Position	Tool Change 💌	NC Offset Simulation	Along Lead-in/out 👻	Dogleg Rapid	Off 💌
Flotary Clearance	50.8	Optimize Tool Path	Off 👻	Rotary Index Time	0.28
- Park Position		Optimize Tolerance	Off		
×, Y, Z 0	0 0	5-Axis	Arcs G17		
		RTCP Coordinate M	Arcs G17-G19	-	
		Calculate Links	Setting		

3.5. Selecting Post Processors and Outputting NC Code

Starting in ESPRIT 2011, there is now a new tool to output NC code from any ESPRIT file. Now, the user simply needs to select NC Code from the Common Machining toolbar. The first time that NC code is posted, ESPRIT will prompt the user to select the file location of the post processor(s) to be used. The user will see the following window pop up upon first selecting NC Code:



By selecting the "Add NC Output" button, the user is able to select the appropriate file location for their post processor as well as determine the file location of the resultant NC code as well as the name of the NC code file and file extension type:

Min NC Output File	? X
Add Post Processor files and configure NC Output. Selecting several post processors will combine the output to a sing	le NC file
Post Processors	
	\mathbf{x}
NC File Folder	
C:\Users\Default\Documents	
NC File Name	
<documentname></documentname>	
Extension	
Preview: C:\Users\Default\Documents\NT1000SZ_Sample-08.nc	
ОК	Cancel

For multi-turret machines such as the NT-SZ, NT1000WZ and NT1000SZ, the user should select each turrets post processor separately and give each file a unique NC File Name. Once the user has selected one post processor, they are able to select Add NC Output again and create a new NC Output file for the other post processor, again with a unique NC File Name.

Once the user selects all necessary post processors and selects OK from the NC Output Settings window, NC code will be generated and appear in the users selected NC code editor.

The next time the user chooses NC Code from the Common Machining toolbar, the NC code will automatically be generated, without having to interface with the NC Output Settings window again.

Note: If the user receives new post processors or would like to make any changes to the file location or file name, they are able to do this by navigating to the Machine Setup in the Common Machining toolbar. On the "NC Output" tab, there is a section on the right hand side called NC Output Settings that will bring the user back into the NC Output Settings window and allow them to make the necessary changes.

neral Assembly NC C	Output Custom					
NC Output Order	Tunet	Number of Head	<u>1</u>	Head ID	Head-1 •	
Sync String Codes Number of Syncs Sync ID and Name	4 <u>↓</u>		Sync Codes Code Increment Start and End Number	10 +	900	
General Properties		Turret Progr	am Output			
Program Name		Coordinate Mode	Absolute 👻	NC Code Output	Settings	
Program Number	0	Unit for NC Code	Metric			

3.6. Machine Parameters

3.6.1. Introduction

Machine parameters can either be set on the Machine Parameters tab of the Machine Setup Utility (part of the MoriSeik AddIn).

Program Eng Code	M30 -	Custom String 1	
TCP Type	TCP Type 1	Custom String 2	
Tool Number Output	Add 1000 to T 🗸	Custom String 3	
Position of M01 Code	At Syncs 🗸	Custom String 4	
Station on Turret 2 for Transfer	0	Custom String 5	
Custom Setting 6		Custom String 6	
Tailstock Type	0	Custom String 7	
Block Skip Method	Comment Out 🗸	Custom String 8	
C Axis Roll Over	Roll Over Off 🔹	Custom String 9	
Custom Setting 10		Position of Sequence Numbers (N)	At Operations
Custom Setting a		Custom Setting k	
Custom Setting b		Custom Setting I	
Custom Setting c		Custom Setting m	
Custom Setting d		Custom Setting n	
		Custom Setting o	
Custom Setting e		Custom Sotting p	
Custom Setting e Custom Setting f		custom setting p	
Custom Setting e Custom Setting f Custom Setting g		Custom Setting q	
Custom Setting e Custom Setting f Custom Setting g Custom Setting h		Custom Setting q Custom Setting r	
Custom Setting e Custom Setting f Custom Setting g Custom Setting h Custom Setting i		Custom Setting q Custom Setting r Custom Setting s	

3.6.2. Output of Program End (M02), Sub-Program End (M99) or Program End and Rewind (M30)

On the POST Output Configuration, set **Program End Code** to **M02** to output M02, set it to **M30** to output M30 or set it to **M99** to output M99 at the end of the NC code of each turret.

Please note that M30 is the default.

3.6.3. TCP Type

This setting is to choose between G43.4 and G43.5 modes when machining in 5 axis simultaneous mode. **Type 1** is G43.4 and **Type 2** is G43.5. Please see section <u>5-axis</u> <u>Simultaneous Milling Options</u> for more information on these options.

3.6.4. Set position of optional stop code (M01) in the NC code.

Upper head only machines (turrets (NT-S, NT1000-S, NT1000-W) have 2 options for outputting the optional stop code (M01).

- At Operations (at the end of every operation)
- At Tool Cancel (at the end of a tool or at the end of an operation where the B axis angle, spindle changes, or anything else that causes the post to output tool cancel section of NC code)

By default, optional stop codes are output at the end of every operation.

Machines equipped with a lower turret (NT-SZ, NT1000-SZ, NT1000-WZ) have an a 3 options for outputting the optional stop code:

- At Syncs (at every sync in the ESPRIT file's operation list)
- At Operations (at the end of every operation as well as at every sync)
- At Tool Cancel (at the end of a tool or at the end of an operation where the B axis angle, spindle changes, or anything else that causes the post to output tool cancel section of NC code, as well as every sync)

By default, optional stop codes will be output on each head after sync codes in the NC code on a lower turret machine this way, optional stop can be turned on while running the NC code on both heads at the same time (production mode).

The "At Operations" and "At Tool Cancel" can only run with the optional stop mode on when running the machine at one head at a time (NC code proofing mode).

3.6.5. Station on Turret 2 for Transfer

If this value is not zero, this station 1 will be used as empty station during part transfer and part release on release operations in the lower turret (NT-S and NT-SZ). If the tool station for is left zero the turret will be left at the current station when the turret is parked by the transfer section in the post.

By default this value is 0

Note: The lower turret can be parked to the lower turret park location with the park operation before the transfer is programed. If this is done, the post will not move the turret or change the station of the turret even if this setting is not zero.

For additional information see the <u>Park Cycles</u> section of this manual.

3.6.6. Tail Stock Type

This option is used to determine an "independent" tail stock is in the sub spindle of the machine used as a live center or a standard tails stock mounted on the machine.

If a value of 0 is entered in the Tail Stock Type setting, the post will assume the machine is using an independent tail stock.

If a value of **1** is entered the post will assume the center is in the sub spindle of the machine.

By default this setting is 0

For additional information see the <u>Tailstock</u> section of this manual.

3.6.7. Block Skip Method

When using spindle priority on turning cycles on lower turret type machines (see part <u>Other</u> <u>Turning Cycles</u> section for additional information), you can use choose which symbol will be output to skip the turning spindle control (spindle start and spindle stop as well as the spindle speeds) on the slave turret.

Comment Out will put the spindle commands inside of a comment as follows "(G97 S2139 M04)" in the slave turrets program

Block Skip will output a block skip "/" before the spindle commands in the slave turrets program

Block Skip 2 will output a block skip "/2" before the spindle commands in the slave turrets program

By default Comment out is used.

3.6.8. C-Axis Rollover

For the Mori Seiki NT-SZ machine, it is sometimes necessary to limit the C-axis rotation angle to between -360 and 360 degrees.

Roll Over Off will turn off roll over in the post. **Roll Over On** will turn on C-axis roll over in the post.

By default Roll Over Off is used.

The Roll over function will need to match the mode of the machines control:.

On machine control, if <Roll over for C-axis> is invalid (NC Parameter 1008 bit 0 = 0):

- min. C-axis value is -99999.999°
- max. C-axis value is +99999.999°

In ESPRIT use Roll Over Off

On machine control if <Roll over for C-axis> is valid (NC Parameter 1008 bit 0 = 1):

- min. C-axis value is -359.999°
- max. C-axis value is 359.999°

In ESPRIT use Roll Over On

When C-axis roll over is on, C-axis will be indexed between -360° and 360°:

- C-axis will be indexed with C values.
- SolidMillTurn 4-axis wrap and rotary operations with interpolation off cannot be programmed.

4. Work Coordinates

The most trouble free method of setting up Work Coordinates in Esprit is by using the Turing Work Coordinate Addin as described in <u>Turning Work Coordinates Addin</u> section. However the NT post will allow you manually setup the work coordinates in the Esprit file.

The only reason to setup work coordinates manually would be to have more than one offset per spindle.

To setup Work coordinates manually care must be taken to ensure the work coordinates are properly created\setup as well as every operation is under the correct work coordinate.

Work Coordinate	x
Define Work Coordinate Work Coordinate Name Standard, Additional Work Coordinate Number 54 0	
X, Y, Z Coordinate 0.000000 0.000000 0.000000	4
Offset Angle 0.000000 Rotary Clearance 0.000000	
Work Plane YZX 💌 Rotate With Part None	•
OK Cancel He	*lp

• The Standard WC number is used to output G54-G59. Enter 54 to have G54 in NC code. 55 to have G55 in NC code, etc. If a machine is equipped with 2 spindles a value must be set for both spindles. When using the machines Standard Work Coordinates (G54-G59) a zero (0) must be entered into the Additional offset field.

Note: The NT1000 post has G59 is reserved for use on the Sub spindle when it is tilted. If programming an NT1000 do not enter 59 into a WC in Esprit).

• The Additional WC number is used to output extended work coordinate in the NC code (G54.1 Pxx). For example if a 1 is entered into the Additional WC filed a G54.1 P1 will be output in the NC code. When outputting the extended work coordinates (G54.1 Pxx) a value of zero (0) must be entered into the Standard WC field.

• XYZ coordinates: X and Y values are typically left at 0, Z can be any value desired to shift the offset from the selected work plane.

- Offset Angle: Must be set to 0
- Rotary Clearance: is not currently used by the NT posts
- Work Plane: it is Critical that the Work plane selected is either YZX or a plane that is parallel to the YZX plane.
- Rotate with Part MUST be set as follows:
 - For turning operations any MillTurn operations at B-90, B0, and B90 set this to "None"
 - For any MillTurn operation that is NOT at B-90, B0, and B90, set this to "Point only"

For additional information on the use of work coordinates please refer to the help file within ESPRIT.

5. Tools (T-function)

5.1. T-function Specifications



	Turret and S	pindle to Use	T Code to Specify
1	Turret 1	Spindle 1	T1001 – T1099
2	Turret 2	Spindle 1	T0101-T1212
3	Turret 2	Spindle 2	T0131 – T1242
4	Turret 1	Spindle 2	T1001 – T1099

Turning Tool General Page:

Tool Number	12	Directation Angle	0.000000
Turret Name	Bottom Turret	Compensation	Comer •
Station Name	Station:12	Length Register	30
Spindle Direction	CCW	-	
Coolant Simulation Cut Color	On	Touch Off Angle	0.00000
Tool Change			
Movement X.Z.Y	Home	• Home •	Home
Position VSZ/Y	10.000000	0.000000	10.000000
Tool Shift			
Tool Shift X, Z, Y	0.000000	0.000000	0.000000
NMG432 ROUGH FA	CE AND TURN		

Milling Tool General Page:

Milling To	ools - Drill		X
Milling To	Cools - Drill General Tool ID Tool Number Length Comp Register Coolant Spindle Direction Initial Clearance Unit Simulation Cut Color Feeds and Speeds Tool Material	DRILL 3 3 On CW 80.000000 Metric V	
Comme	nt		OK Cancel Help

5.1.1. Upper Head (B-Axis Head):

For the NT machine series when machining on Spindle 1 (Main Spindle) or Spindle 2 (Sub Spindle) with the tool mounted in the Tool Spindle (Head1/B-Axis), if the "Tool Number Output" option in the Post Output Configuration "Tool Number Output" will determine if the post processor will add 1000 to the number entered or not. (See <u>Machine Parameters</u>)

If the "Tool Number Output" is set to "Add 1000 to T" and the tool number entered into the tool page is less than 1000, the post will add 1000 to the T code. If not, the tool number will be output as entered on the tool's dialog box

NOTE: All turning tools that are defined on the B-Axis head for the NT series machines should be set on the "General" tab of the tool page in either a "3V" or "4V" orientation (whichever is the position of the tool in the D1 orientation of the milling spindle). ESPRIT will automatically adjust the B-Axis head angle to the correct orientation based on the tool orientation selected in a given machining operation.

Tool ID	WNMG LH main	Orientation	3V	
Tool Number	1	Orientation Angle	0.00000	0
Turret Name	B-Axis Head	Compensation	Comer	•
Station Name	Station:1	tion:1 Length Register		
Spindle Direction	CCW	*		
Coolant	On	Touch Off Angle	0.00000	0
Simulation Cut Color				
Tool Change				
Movement X.Z.Y	Home	✓ Home	• Home	
Position XZ/Y	10.000000	6.000000	10.0000	00
Tool Shift				
Tool Shift X, Z, Y	0.000000	0.000000	0.00000	0
CNMG432 ROUGH FA	CE AND TURN			_

5.1.2. Lower Turret:

For the NT machine series when machining on spindle 1 (Main Spindle) with the tool mounted on the Lower Turret (Head 2/Turret 2), the tool number will be output directly as entered in the Tool Number Dialogue Box. If 1010 is entered in the Tool Number Dialogue Box, T1010 will be output in the NC code. Also, if the user has entered three numbers in the Tool Number Dialogue Box, the post processor will add a leading 0 to the tool number. If 101 is entered in the Tool Number Dialogue Box, T0101 will be output in the NC code.

In the event that the user has entered two or less numbers in the Tool Number Dialogue Box, the post processor will combine the numbers in the Tool Number Dialogue Box and the Length Register (Length Comp Register for milling tools) Dialogue Box. If 10 was entered in the Tool Number Dialogue Box and 27 was entered in the Length Register Dialogue Box, 1027 will be output in the NC code.

For the NT machine series when machining on spindle 2 (Sub Spindle), the post processor will add <u>30</u> to the tool number entered in the Tool Number Dialogue Box if that tool is mounted on the Lower Turret (Head 2/Turret 2). If 1010 is entered in the Tool Number Dialogue Box, T1040 will be output in the NC code. Also, if the user has entered three numbers in the Tool Number Dialogue Box, the post processor will add a leading 0 to the tool number. If 101 is entered in the Tool Number Dialogue Box, T0131 will be output in the NC code.

In the event that the user has entered two or less numbers in the Tool Number Dialogue Box, the post processor will combine the numbers in the Tool Number Dialogue Box and the Length Register (Length Comp Register for milling tools) Dialogue Box and add <u>30</u> to the combination. If 10 was entered in the Tool Number Dialogue Box and 27 was entered in the Length Register Dialogue Box, 1057 will be output in the NC code.
5.2. Tool Life Management

For the Mori Seiki NT Series machines, it is possible to use Tool Life Management to record data about a particular tools usage. To turn on Tool Life Management, navigate to the Custom tab of the Tool Page and set the "Tool Life Management" setting to 100 (Tool Custom Setting 8). This will output an M300 at the end of each operation with that tool and also modify the tool call depending on if you are using a milling or turning tool.

Turning tool page:

Turning Tools - Turning Ins	sert			
Insert Holder General	Custom			
_ Values		1		1
Second Coolant	0.000000			
Custom Setting 2	0.000000			
G30 or G28	0.000000			
Custom Setting 4	0.000000			
Custom Setting 5	0.000000			
Custom Setting 6	0.000000			
Custom Setting 7	0.000000			
Tool Life Management	300.000000			
Custom Setting 9	0.000000			
Custom Setting 10	0.000000			
		ОК	Cancel	Help

Milling tool page:

Second Coolant 0.000000 Custom Setting 2 0.000000 G30 or G28 0.000000 Custom Setting 4 0.000000 Custom Setting 5 0.000000 Custom Setting 6 0.000000 Custom Setting 7 0.000000 Custom Setting 7 0.000000 Custom Setting 9 0.000000 Custom Setting 9 0.000000 Custom Setting 9 0.000000 Custom Setting 9 0.000000 Custom Setting 10 0.000000	Values		
Custom Setting 2 0.000000 G30 or G28 0.000000 Custom Setting 4 0.000000 Custom Setting 5 0.000000 Custom Setting 6 0.000000 Custom Setting 7 0.000000 Custom Setting 9 0.000000 Custom Setting 9 0.000000 Custom Setting 10 0.000000	Second Coolant	0.000000	
G30 or G28 0.000000 Custom Setting 4 0.000000 Custom Setting 5 0.000000 Custom Setting 6 0.000000 Custom Setting 7 0.000000 Custom Setting 9 0.000000 Custom Setting 9 0.000000 Custom Setting 10 0.000000	Custom Setting 2	0.000000	
Custom Setting 4 0.000000 Custom Setting 5 0.000000 Custom Setting 6 0.000000 Custom Setting 7 0.000000 Tool Life Management 300.000000 0.000000 Custom Setting 9 0.000000 Custom Setting 10 0.000000	G30 or G28	0.000000	
Custom Setting 5 0.000000 Custom Setting 6 0.000000 Custom Setting 7 0.000000 Tool Life Management 300.000000 Custom Setting 9 0.000000 Custom Setting 10 0.000000	Custom Setting 4	0.000000	
Custom Setting 6 0.000000 Custom Setting 7 0.000000 Tool Life Management 300.000000 Custom Setting 9 0.000000 Custom Setting 10 0.000000	Custom Setting 5	0.000000	
Custom Setting 7 0.000000 Tool Life Management 300.000000 Custom Setting 9 0.000000 Custom Setting 10 0.000000	Custom Setting 6	0.000000	
Tool Life Management 300.000000 Custom Setting 9 0.000000 Custom Setting 10 0.000000	Custom Setting 7	0.000000	
Custom Setting 9 0.000000 Custom Setting 10 0.000000	Tool Life Management	300.000000	
Custom Setting 10 0.000000	Custom Setting 9	0.000000	
	Custom Setting 10	0.000000	

5.3. Tool Nose Direction (G43 H_ T_)

When using a turning tool in the B-axis head of the Mori Seiki NT, it is possible to specify the direction of the tool nose to be offset according to the following diagram:



The corresponding G code is:

G43 H_. T_.

To program this in ESPRIT, go to the Custom tab of any operation using a turning tool and set the Tool Nose Direction (1-0) (Custom Setting 7) field to any value between 1 and 9. Values >9 or <1 will omit the T_{-} output.

₩ 0	K 🗙 Cancel 🖗 🗸	🖉 Help
leral	Settings	
Ger	Custom Setting 1	0.000000
2	Percentage of Spinning Tool RPM	0.000000
trate	Custom Setting 3	0.000000
S	Custom Setting 4	0.000000
Itour	G332 Setting Value	0.000000
Co	Custom Setting 6	0.000000
E	Tool Nose Direction (1-9)	2.000000
Custo	G361/G362	0.000000
	Custom Setting 9	0.000000
	G30 or G28	0.000000
	Features Tools EB Operatio	ons <u>SolidTurn - Contouring</u>

5.4. Second home position

On the Custom tab of the tool page, set "G30 or G28" setting to 30 (Tool Custom Setting 3) to use the second zero return of the machine (G30 P4) for this tool.

Turning tool page:

Turr	ning Tools - Turning In	sert				×
In	sert Holder General	Custom				
	-Values		٦			
	Second Coolant	0.000000				
	CustomSetting2	0.000000				
	G30 or G28	30.000000				
	CustomSetting4	0.000000				
	CustomSetting5	0.000000				
	CustomSetting6	0.000000				
	CustomSetting7	0.000000				
	Tool Life Management	0.000000				
	CustomSetting9	0.000000				
	CustomSetting10	0.000000				
l						
_			Γ	ОК	Cancel	Help
						· ·]

Milling tool page:



Milling To	ols - Drill				×
	Values Second Coolant CustomSetting2 G30 or G28 CustomSetting4 CustomSetting5 CustomSetting7 Tool Life Managemen1 CustomSetting9 CustomSetting10	0.000000 0.000000 30.000000 0.000000 0.000000 0.000000 0.000000 0.000000 0.000000 0.000000			
	u				
			ОК	Cancel	Help

This will trigger the output of G30 P4 (instead of G28) whenever a turret needs to be sent home.

If you omit (or enter a wrong value) to enter the value in this field, turrets will be sent to the machine zero point with G28.

This can also be set within a specific operation page as well as shown below with the "G30 or G28" (Operation Custom setting 10)

🧼 0	K 🗙 Cancel 🛛 🚰 👻			🕜 Help
tegy	Values			
Stra	Rapid Feedrate (G01 F)	0.000000		
e	Custom Setting 2	0.000000		
Pock	Custom Setting 3	0.000000		
_	Custom Setting 4	0.000000		
lguos	G332 Setting Value	0.000000		
"	Custom Setting 6	0.000000		
inks	Custom Setting 7	0.000000		
	G361/G362	0.000000		
stor	Custom Setting 9	0.000000		
õ	G30 or G28	30.000000		
	Footures 📆 Tools 🖽	Desertises	Calid Mill Turn Deckating	

5.5. G361/362 Tool Change Command Setting

For the Upper Turret of the NT-SZ machine, it is possible to output the tool change command with either G361 or G362 (via 4th zero point).

For any operation where the part is being held by only one spindle (where G361 is standard), navigate to the Custom Settings tab, and to output G362 instead of G361, set the "G361/G32" field to 362 (Operation Custom Setting 8). If any value other than 362 is input, G361 will be output in the NC Code.

√ 0	K 🗙 Cancel 🛛 🚰 🕶	🕡 Help
ieral	Settings	
Ger	Custom Setting 1	74.000000
8	Percentage of Spinning Tool RPM	0.000000
trate	Custom Setting 3	0.000000
S	Custom Setting 4	0.000000
Itour	G332 Setting Value	0.000000
Cor	Custom Setting 6	0.000000
E	Tool Nose Direction (1-9)	0.000000
Clusto	G361/G362	362.000000
<u> </u>	Custom Setting 9	0.000000
	G30 or G28	0.000000
	Features 178 Tools 199 Oceantia	SolidTum - Contouring

For operations where both spindles are holding the part (simultaneous machining), the G362 tool change call is standard. If the user would like to instead output G361, set the "G361/G32" to 361. If any value other than 361 is input, G362 will be output in the NC code.

₩ 0	🛩 OK 🔭 Cancel 🛛 🚰 🗝		
teral	Settings		
Ger	Custom Setting 1	74.000000	
2	Percentage of Spinning Tool RPM	0.000000	
trate	Custom Setting 3	0.000000	
S	Custom Setting 4	0.000000	
tour	G332 Setting Value	0.000000	
Ğ	Custom Setting 6	0.000000	
E	Tool Nose Direction (1-9)	0.000000	
Clusto	G361/G362	361.000000	
	Custom Setting 9	0.000000	
	G30 or G28	0.000000	
	Features 🔀 Tools 🖪 Operatio	ons 🗳 SolidTurn - Contouring	

5.6. Tool Change position

NT-S 6600 machines have the ability to change the tool change position. This is done with an R value on G361/G362 line. This can be programed by setting the desired position number into the Tool Change Position (Custom Setting 2) field.

Taal Change Basking (NITCCOD)	0.000000	1	
	0.000000		
G30 or G28	0.000000	_	
Custom Setting 4	0.000000		
Coolant Pressure	0.000000		
Custom Setting 6	0.000000		
Custom Setting 7	0.000000		
Tool Life M(300)	0.000000		
Custom Setting 9	0.000000		
Custom Setting 10	0.000000		

6. Spindle Direction, Speed, and Feedrate

6.1. Turning Spindle Direction and Milling Tool Spindle Direction

For turning operations using a cutting insert (Roughing, Balanced Roughing, Contouring, Grooving and Threading), the spindle direction will automatically be computed by the post processors and will output the correct M-code (M03, M04, M203 or M204) based on the hand of the tool on the holder and the orientation of the tool. The field Spindle Direction on the General tab has not effect on the output.

urning Tools - Turning losert	x	Turning Tools - Turning	Insert	-	
Invent Holder General	Angles	Innert Holder Genera	1		
Holder ID	Lead Argle (LA) 300000	Tool ID	Turing meet GD Top Left	Overstation	3/
Holder Type [Turning]	Note Angle (NA)	Tool Number	3	Treasure works	Carriero
	Heter Angle (ISA) (S2.000000	Tunet Name	Upper Left Turrel 💌	Compensation	Corner 💌
Holder Pagerfeit State 1.31 and	Generally	Station Name	Station.3	Length flegister	þ
Shark full		Spixelle Direction	CW 👱		
Clarge Stela States	TH	Coolani	On 💌	Touch Olf Angle	000000
Costance Ande N 0"	5.0.5	Simulation Cut Color			
Hand of Tool Plate -	57	Tool Overge		6 1243	91 40
Intel Shape	E NA	MovementXZY	Home 🔳	Hume 📃	Home 💌
And a second		100000K21	5.00000	asoum.	Januara
	F TO RA	Tool Shift	la soona	In anoma	E anno
	A 19.050 C 101.60 E 31.750	Loci ShiftX, Z, T	lanoone	Income	No occor
	B 19.050 0 00000 \$ 31.750				
	OK Cancel Help		1	OK Ca	noei Helo

Custom Turning tools do not have the option to select if the tool is a right or left hand tool. In this case we use the Holder Thickness field. If this value is a positive value the post will know this is a left hand tool, if it is a negative value it is a right hand tool.

Turning Tools - Custom Insert	X
Insert General Custom	
OK Cancel He	elp

For the turning Drilling and milling operations, the turning spindle direction or milling tool spindle direction is specified by the Spindle Direction pull down on the first tab of the tool page.

Milling To	ools - Drill		×	۲
	General Tool ID Tool Number Length Comp Register Coolant Spindle Direction Initial Clearance Unit Simulation Cut Color Feeds and Speeds Tool Material	SPOT 04 Lower 12 12 0n • Cw • C		
	ni.			
			OK Cancel Help	

Proje	ct Manager		×
₩ 0	K 🗡 Cancel 🖺 🗸		🕜 Help
General	Operation Name		
6 G	S Tool Selection		
Strat	Tool		
yđno	😻 Work Setup		
č	Spindle Name	MainSpindle	•
inish	HeadId	Head-1	
	Feeds and Speeds		
ustom	Speed RPM, CSS	0	0
Ō	Feedrate PM, PR	0.000000	0.000000 🗘
	Reference Diameter	60.000000	
	Speed Unit	CSS	•
	Feed Unit	Per Revolut	tion 🔽
	Maximum RPM	5000	
	Spindle Range	Off	_
	Spindle Priority	Off	•
	Use Feed and Speed KB	No	
	S Comment		
	Features 🚮 Tools	🖪 Operations	편 SolidTurn - Grooving

6.2. Spindle Speed output for CSS Unit

On a turning operation page, if you choose to output the turning spindle speed in the unit CSS (Constant Surface Speed), a warm up speed will first be output with the G97 code before the first positioning move. The speed value is computed at the first diameter from where the CSS command will be turned on. Next, after positioning the tool, the CSS tuned on by the output of G96 S. The cut is done and finally, at the last diameter, the CSS is cancelled by the output of G97 S with S computed at the current last diameter.

6.3. Feedrate for 4-axis Milling Operations

For Wrap Pocketing and Wrap Contouring operations with Cylindrical Interpolation set to No or Rotary Face Pocketing and Rotary Face Contouring operations with Polar Interpolation set to No, a rotary feedrate will be computed by the post processor, based on the linear and rotary (Caxis) moves of the cut.



This computation is necessary since two types of feedrate are involved for 4-axis wrap cuts: linear feedrates (linear moves along XYZ) in inch/min (or mm/min) and rotary feedrates (angular moves around the C-axis) in deg/min.

In ESPRIT, when programming such operations, simply enter the desired XY and Z PM (per minute) feedrate values. The post will then, based on these values; compute the correct 4-axis feedrate

Project Manager				
🔶 O	ж 🗙 Cancel 🛛 🛍 🗸		🕜 Help	
eral	S Operation Name			
Gen				
윤	Tool Selection			
1×	ToolD			
-B]		
Strat	😵 Work Setup			
_	Spindle Name	MainSpindle	•	
Iguor	HeadId	Head-1	_]	
	▼ Feeds and Speeds			
Link	Cut Speed RPM, SPM	0		
ε	XY Feedrate PM PT	0.000000	0.000000	
Custo	Z Feedrate <mark>PM</mark> . PT	0.000000	0.00000 🔨	
	Feed Unit	Per Revolution		
	Const. Removal Rate	No No		
	Corner Slow Down			
	Use Feed and Speed KB	No	•	
	S Comment			
	Features	Operations 🕺 🖉 Calida	dillTurn - Wran Pooksting	
			minum - wrap Fockeung	

6.4. Feedrate option for rapid moves in 5 axis operations

At times it is better to program a high feedrate move rather than rapid moves (G00). This is particularly true in 5 axis mode.

In order to output a G01 and a high feedrate instead of a G00 rapid type move in a 5axis operation set the "Rapid Feedrate" filed to the desired feedrate for the rapid moves within the 5x operation.

If this value is left 0 G00 will be output for the rapid moves.

This setting is only active in the operation programed. If you want to use the G1 F option for multiple operations you will need to set this in each operation.

	Reald Feederate (CO1 E)	E000 000000
		0.000000
Ц	Custom Setting 2	0.00000
	Custom Setting 3	0.000000
	Custom Setting 4	0.000000
	G332 Setting Value	2.000000
H	Custom Setting 6	0.000000
H	Custom Setting 7	0.000000
H	G361/G362	0.000000
H	Custom Setting 9	0.000000
	G30 or G28	0.000000
ì		

6.5. Turning on Al Contour Control

This function is provided for high-speed, high-precision machining. It enables suppression of the delays in machine movement during acceleration/deceleration that become larger as the feedrate increases, and machining profile errors are reduced as a result.

To use the AI Contour Control, the user must first select which cutting mode they would like to use for an operation. There are four cutting modes possible with AI contour control. They are:

- R1: Time Priority Mode
- R2: Middle Mode
- R3: Accuracy Priority Mode
- R4: Custom Mode

For further details about each of these cutting modes, please see the NT Programming Manual.

Once the user has selected which cutting mode is appropriate for a given operation, navigate to the "Custom" tab of the operation and in the "G332 Setting Value" field (Operation Custom Setting 5), enter a one (1), two (2), three (3), or four (4) to select R's 1-4 respectively. If the G332 Setting Value is zero (0), AI Contour Control will not be activated.

Once this setting has been changed, click OK to exit the operations page.

NOTE: The user must set Operation G332 Setting Value for EVERY operation that they would like to use AI Contour Control with. ESPRIT will automatically output the corresponding code to turn off this function at the end of each operation.

<i>•</i> 0)K 🏋 Cancel 🛛 🚰 👻		🕜 Help
leral	Settings		
Ger	Rapid Feedrate (G01 F)	5000.000000	
÷	Custom Setting 2	0.000000	
ol Pa	Custom Setting 3	0.000000	
Ê	Custom Setting 4	0.000000	
Б	G332 Setting Value	2.000000	
entat	Custom Setting 6	0.000000	
ō	Custom Setting 7	0.000000	
S	G361/G362	0.000000	
	Custom Setting 9	0.000000	
E	G30 or G28	0.000000	
Custo			
1	CAD Feature	📴 Tools 🔛 Operations 🖏	Composite

7. 5-axis Simultaneous Milling Options

7.1. Introduction

When programming a simultaneous 5-axis operation in ESPRIT for the NT machine, there are three options for the format of the output code:

- Fixed Type1 Mode (G43.4, XYZ BC WKP=1)
- Rotate with Part Type1 Mode (G43.4, XYZ BC WKP=0)
- Rotate with Part Type2 Mode (G43.5, XYZ IJK WKP=0)

When cutting in a 5-axis simultaneous mode on an on the sub spindle of an NT1000 machine, the BW axis MUST be at BW0.

The post supports 5-axis operations in the upper head, even if the operation is only used for wrap type machining.

7.2. Definitions

7.2.1. Rotate With Part



- Coordinates are output relative to the table/spindle. The coordinate system is attached to the work piece and rotates with the part.
- After the RTCP command is specified, the workpiece coordinate system that is fixed on the table at that point in time becomes the programming coordinate system. Thereafter, the programming coordinate system rotates as the table turns around.
- The coordinate system does not rotate with the tool head.
- Type1 outputs code with a G43.4 and coordinates expressed with XYZ for linear axes and ABC for rotary axes (See sample code below).
- Type2 outputs code with a G43.5 and coordinate expressed with XYZ for linear axes and IJK for rotary axes (See sample code below).



- Coordinates are output relative to the current work piece origin (local work coordinate in ESPRIT). In this case, the programming coordinate system does NOT rotate as the table turns around but remains fixed on the workpiece coordinate system.
- This mode changes the coordinates output by the postprocessor when there is at least one table rotary axis.
- For a tool/tool rotary axis configuration, this mode has no effect. With the Rotate With Part option, the XYZ coordinates in the NC code are similar to coordinates output for a machine with two rotary axes on the milling head (tool/tool configuration).
- Type1 outputs code with a G43.4 and coordinates expressed with XYZ for linear axes and ABC for rotary axes.

7.3. Operation Types

7.3.1.RTCP Type 1 WKP=0 (G43.4)

ESPRIT Machine Setup 'RTCP Coordinate Mode' must be set to Rotate With Part. TCP Type must be set to "TCP Type 1" in the Post Output Configuration Utility.

5-Axis		
RTCP Coordinate Mode	Rotate With Part	•
Calculate Links	Settings	

7.3.2. RTCP Type 1 WKP=1 (G43.4)

ESPRIT Machine Setup 'RTCP Coordinate Mode' must be set to Fixed. TCP Type must be set to "TCP Type 1" in the Post Output Configuration Utility.

- 5-Axis		
RTCP Coordinate Mode	Fixed	-
Calculate Links	Settings	

7.3.3.RTCP Type 2 WKP=0 (G43.5)

•

TCP Type must be set to "TCP Type 2" in the Post Output Configuration Utility as well as 'RTCP Coordinate Mode' must be set to Rotate With Part on the Machine Setup\General Tab

5-Axis		
RTCP Coordinate Mode	Rotate With Part	•
Calculate Links	Settings	

7.4. Additional Information

Machine Parameter

The following parameters need to be set on the Mori Seiki NT-SZ machine before performing 5-axis machining:

1. G43.4 XYZBC WKP=1 -> 19696-bit5=1 2. G43.4 XYZBC WKP=0 -> 19696-bit5=0 3. G43.5 XYZIJK WKP=0 -> 19696-bit5=0 4. 19741 =Max. B-pos. 5. 19742 =Min. B-pos.

NOTE that you may only choose ONE RTCP mode for an entire ESPRIT file.

To turn on RTCP Mode navigate to the General tab and to the section called "RTCP" and set this to "Yes". You must perform this for each 5-axis operation that you create.



8. Simultaneous Turning Operations: Turning on the same spindle with two turrets

8.1. Balance Roughing Cycle

When programming a Balanced Roughing cycle, you have the choice between two different balanced modes:

- Trailing Tool: there is a user defined trailing distance between the two inserts.
- Simultaneous: the cut is mirrored between the turrets, the feedrates can be doubled.

Proje	ct Manager	×
₩ 0	K 🟋 Cancel 🛛 📆 👻	🕡 Help
teral	S Cutting Strategy	
Ger	Type of Work	OD 🔽
trategy		
o J	Balanced Mode	Trailing Tool
Rough		
tom	Trailing Distance	0.000000
O	Rough Pass	Yes
	Finish Pass	No

Proj	ect Manager		<u>×</u>
1	OK 🗙 Cancel 🛛 🕅 👻		🕜 Help
heral	😵 Cutting Strategy		
Ger	Type of Work	OD	
itrategy			
0,	Balanced Mode	Simultaneous	
Rough			
tom	Rough Pass	Yes	
Cus	Finish Pass	No	•

For the Trailing Tool balanced mode, a wait code (soft sync) will be output in between each pass. For the Simultaneous balanced mode, the Balanced Cut Mode (G68) will be turned on on

both involved heads and will be canceled at the end of the cut by the Balanced Cut Mode Cancel code (G69).

The spindle speed will only be output in Head 1 (B-axis), not in Head 2.

8.2. Other Turning Cycles

When two heads are cutting at the same time on the same spindle, the spindle command of a head can override the spindle command of the other head.

The head with the longest operation must have the control of the spindle speed.

The head controlling the spindle and the head "slave" are set on the operation page of turning operations, using the Spindle Priority setting:

Proje	ect Manage	ar 👘	U		×	
~	DK 🗙 Cano	el 🛛 🔁 👻			🕜 Help	
eral	© Operation Name					
Gen						
tegu	S Tool Se	election				
Stra	Tool		??????	??????		
ntour	🖲 Work S	ietup				
Õ	Spindle Na	ame	MainSpindle			
tom	HeadId		Head-1		•	
Cns	区 Feeds a	and Speeds				
	Speed RP	M, CSS	0	0		
	Feedrate PM, PR Reference Diameter		0.000000	0.000000	⊥	
			60.000000	60.000000		
	Speed Un	iit	CSS	CSS Per Revolution 100.000000 5000 Off		
	Feed Unit		Per Revolution			
	Transvers	e Feedrate %	100.000000			
	Maximum	RPM	5000			
	Spindle Ra	ange	Off			
	Spindle Pr	iority	Off			
	Use Feed	and Speed K	.B Off On			
	😵 Comme	ent	Other Operatio	n		
	Ċ					
	Features	🚺 Tools	📰 Operations	<u>≰</u> SolidTurn - Co	ntouring	

Set Spindle Priority to:

- On to output the spindle speed (this head is controlling the spindle)
- Other Operation: the other head is controlling the spindle and the spindle speeds are output as shown in <u>Machine Parameters</u> section "Under Block Skip Methods"

Note: If using one of the block skip options make sure to activate the appropriate block delete function on the control.

9. Park Operation

9.1. Introduction

The park operation can be used to move either head to any location within the machine travels.

The park operation is most often used in the lower turret to avoid collisions between the B axis head on the lower turret when working on the face of the part with the upper head at B+/-90 degrees, when transferring parts from spindle to spindle, doing bar pull operations with the sub spindle or part release operations.

With the Park cycle of ESPRIT, you will have different possibilities to park a tool located on head 1 (or 2 for lower turret configurations). It is a great way to park a tool above a spindle to a clearance position, so you can for example freely machine on the same spindle with another turret for lower turret configuration machines. You can also send a tool home along the X, Y (Y-axis specifications only) and/or Z axis. You finally can park the turret to its far most left position (left or right) or anywhere in-between using the machine work coordinate system (G53).

The NT1000 machines also use the park operation to position the A axis (Sub spindle) and BW axis allow for more clearance when working on the main spindle with the B axis head.

When using the Park operation with the upper head, the Position type park modes are not supported as the simulation will not match the machine movements. The Home and Machine modes are fully supported.

In addition the B axis may also be parked with the park operation. Parking the C axis is not supported by the post at this time..

When parking the lower turret, all modes (Position, Home, and Machine) are supported.

9.2. How to park a tool

On the Park tab of the Park operation page, using the Park Position X, Y & pull-downs and fields, you can set how and where you want to park your tool.

Project Manager						
🛩 OK 🗙 Cancel 🛍 🗝 🔞 Help						
neral	Park Position					
- B	Park Position X	None	0.000000			
ž	Park Position Y	None	0.000000	R		
Ъ	Park Position Z	Machine	0.000000	L3		
stom	Stop Code	Position		•		
ð	😵 Rotary Axes					
	Park Axis 1	No		•		
	Park Axis 2	No		•		
	Features Too	ols _ 🔃 Operations	🛃 Solid Tur	m - Park		

Set Park Position X, Y or Z to:

• None: The tool will not move along the selected axis. If you set for example Park Position X to None, the tool will not move along the X-axis when going to the park position.

• **Home:** The tool will be sent to the machine zero point with G28 along the selected axis. Note that second home position G30 can be output instead of G28, please see part <u>Second</u> home position section for additional information.

• Machine: The tool will move along the selected axis in the machine work coordinate (G53) to the specified position in Position X, Y, Z fields.

S Park Position						
Park Position X	Machine	•	10.000000	ß		
Park Position Y	None	•	0.000000	B		
Park Position Z	None	•	0.000000	B		
Stop Code	Optional Stop			•		

• **Position:** The tool will be parked at the specified position in Position X, Y, Z fields at rapid traverse rate (G00). Position mode is only supported by the lower turret.

Park Position				
Park Position X	Position	•	10.000000	R
Park Position Y	None	•	0.000000	B
Park Position Z	None	•	0.000000	B
Stop Code	Optional Stop			•

Note: The X; Y & Z positions entered in ESPRIT are absolute in YZX. So the post processor will compute and output the correct values from this absolute position based on the turret, tool and the spindle (work coordinate) in use.

Once your tool movements correctly defined, you will be able to set the Return Mode. It controls how the axes move to the park position. This setting is available only when at least 2 axes are allowed to move during the park.

🔶 C)K 🗙 Cancel 🛛 🔮	•		🕜 Help
ieral	Park Position			
Ger	Park Position X	Home	0.000000	\$
÷	Park Position Y	Home	0.000000	5
Pa	Park Position Z	Home -	0.000000	(A)
E	Return Mode	None		-
Cust	Stop Code	None X First		
	😵 Rotary Axes	Y First 7 First		
	Park Axis 1	X Then Y		
	Park Axis 2	Y Then X		
_		Z Then X		1

The different existing Return Modes are:

- None: All axes move directly to the park position. All axes allowed to move will move together.
- **X First:** The X axis is moved first to Park Position X. Then all other axes move in a second step.
- Y First: The Y axis moves first. Then all other axes move together.
- **Z** First: The Z axis moves first. Then all other axes move together.
- **X Then Y:** X moves first, then Y, then Z.
- X Then Z: X moves first, then Z, then Y.
- **Y Then X:** Y moves first, then X, then Z.
- **Y Then Z:** Y moves first, then Z, then X.
- **Z** Then X: Z moves first, then X, then Y.
- **Z** Then Y: Z moves first, then Y, then X.

The options available for Return Mode depend on the axes selected for movement and their positions type (None, Home, Machine or Position). See the examples below:

• Example 1:

If Park Position X is set to Home, Park Position Y set to None and Park Position Z set to Home: the choices for Return Mode become None, X First or Z First.

• Example 2:

If Park Position X is set to Home, Park Position Y set to Home and Park Position Z set to Machine: the choices for Return Mode become Z First, X Then Y, X Then Z, Y Then X, Y Then Z, Z Then X or Z Then Y.

Since movements in machine work coordinate system (G53) cannot be output in the NC code on the same line as movements home (G28), ESPRIT is filtering the forbidden combinations. These combinations will always have Z movement output on its own line in the NC code.

• Example 3:

If Park Position X, Y & Z are all set to Home: all Return Modes become available.

Note that If your machine does not have a Y-axis on the lower turret, any Y-axis movement programmed with the Park cycle will be ignored.

9.3. Stop Code Options

On the Lathe Park tab of the operation, if you set Stop Code to Stop, a Program Stop Code (M00) will be output. If you set Stop Code to Optional Stop an Optional Stop Code (M01) will be output. If ether Optional Stop or Stop Code are used, the post will repeat the tool change information after the park operation to ensure all the necessary commands are active for the next operation.



9.4. Setting the B axis angle

If parking the upper head, the angle of the B axis can be controlled by the park operation if desired. To do this set the Park Axis 2 to Yes and enter the desired angle into the Angle of Axis 2 field.

If the park operation makes a tool change, the B axis will always move to the position of the B axis in the Rotary angles properties (normally B0) even Park Axis 2 field is set to No. Therefore if the park operation is making a tool change it is best to set the Park Axis 2 field is set to Yes and set the desired B axis angle.

If the park operation does not make a tool change the B axis will only be moved Park Axis 2 field is set to Yes, otherwise the B axis will be left at its current angle.

If you park the B axis and you select the G53 Option the machine will move the Z axis home (G29/G30 W0) prior to moving the B axis, then move to the G53 position.

😵 Rotary Axes		
Park Axis 1	No	•
Park Axis 2	Yes	•
Angle of Axis 2	-90.000000	

Note: Parking the C axis (Park Axis 1) is not implemented in the NT posts.

In Addition, since the B axis can only be moved if the XYZ axis are at the home position, the XYZ axis must all be set to HOME if the B axis is going to be moved by the park operation.

Park Position				
Park Position X	Home	-	0.000000	[Lz]
Park Position Y	Home	-	0.000000	[k]
Park Position Z	Home	-	0.000000	La
Return Mode	X First			•

If you would like the XYZ axis to be at a different location after the B axis is positioned, you will need to do this in a second park operation.

9.5. Park head/ turret to its minimum travel position (G53)

Moving the head/turret to the far left position by setting the appropriate locations in the park operation or you can enter in a value in to the park op shown below in the park operation. If this is done the simulation will match the NC code.

Prior to ESPRIT 2012 we did not have the ability to match the simulation with the NC code so we provided a method of parking the head/turret by putting a value of "-53" in Position with G53 field (Custom Setting 1) of the Park operation.

If you set Position with G53 to -53, the active turret will be parked to its left maximal travel position regardless how the park operation is setup but simulation may not match the NC code.

This is the location in which the lower turret needs to be parked prior to a part transfer or part release. If the Esprit file does not park the turret the post will park the turret before the transfer or the part release.

Proje	ct Manager	×
₩ 0	K 🗙 Cancel 🛃 🗸	🕐 Help
heral	Settings	
Ger	Position with G53	-53.000000
×	Output M09	0.000000
Ъ.	CustomSetting3	0.000000
Ę	CustomSetting4	0.000000
Custo	CustomSetting5	0.000000
	CustomSetting6	0.000000

Minimum travel position values (in the machine coordinate system G53) for NT machine configurations:

Machine Model	Bed Length	Head/Turret	Left position(mm)	Left position(inch)
1000		Head 1	G53 Z-230.	G53 Z-9.05
		Lower turret	G53 Z-305.	G53 Z-12.008
2000	1500	Head 1	G53 Z-780.	G53 Z-30.7
		Lower turret	G53 Z-751.	G53 Z-29.56
3100	500	Head 1	G53 Z-470.	G53 Z-18.5
3150		Lower turret	G53 Z-470.	G53 Z-18.5
3200				
	1000	Head 1	G53 Z-540.	G53 Z-21.25
		Lower turret	G53 Z-485.	G53 Z-19.09
4200	700	Head 1	G53 Z-560.	G53 Z-22.04
4250		Lower turret	G53 Z-505.	G53 Z-19.88
4300				
	1000	Head 1	G53 Z-560.	G53 Z-22.04
		Lower turret	G53 Z-505.	G53 Z-19.88
	1500	Head 1	G53 Z-817.5	G53 Z-32.18
		Lower turret	G53 Z-762.5	G53 Z-30.01
5400	1200	Head 1	G53 Z-670.	G53 Z-26.37
		Lower turret	G53 Z-615.	G53 Z-24.21
	1800	Head 1	G53 Z-970.	G53 Z-38.18
		Lower turret	G53 Z-915.	G53 Z-36.02
6600	3000	Head 1	G53 Z-1575.	G53 Z-62.0
	4000	Head 1	G53 Z-2075.	G53 Z-81.69
	6000	Head 1	G53 Z-3075.	G53 Z-121.06

Notes

That if the tool selected in the park operation is not the same as the previous tool the turret will be sent to the tool change location, change the tool, then move to the park location.

The -53 option is now obsolete with the new AutoPark Addin, see the <u>Setting up the park</u> operation with the AutoPark addin section below.

9.6. Park lower turret prior to a transfer (Z configurations only)

Once correctly setup, you can park the turret using the Park cycle (with G53) prior to a transfer.

If you want to select a blank station on your turret for the park operation you will first need to create a dummy tool on the turret, in the right station. Set the Tool Number to whatever you have decided to use as the empty station and the Length Comp Register to 0.



Then, simply program a Park operation using this previously defined tool prior to the transfer sequence. Note that since the tool call will be made without tool wear offset number, you will

need to set Position with G53 to -53 (see section <u>Park head/ turret to its minimum travel position</u> (G53)) on the Park operation Custom tab. The turret will be parked using G53.

DUMMY TOOL		
🛃 Park turret	٩	Main Spindle
┣遢 Pickup	٩	Sub Spindle
Cutoff Insert		
= ^B / _{2≫} Cutoff	٩	Main Spindle

Note: Prior to a park operation the spindle is always stopped. After a park operation the post will output a complete tool change section even if no tool change takes place after a park operation. This is done to ensure all the necessary functions are re-started after a park operation.

9.7. Setting up the park operation with the AutoPark addin

The AutoPark addin is an alternative method of setting up the park operation to move the Head\Turret to the minimum travel limits. The advantages of the addin is that it automatically setups up your park locations of the head/turret for both correct NC code and simulation.

If the AutoPark addin has been run and has the turret parked at the AutoPark location, the -53 option in the "Position with G53" field (Custom Setting 1) will be ignored by the post.

If after you park the turret with the AutoPark addin you change a setting in the park operations locations, the AutoPark addin will then reset its Park Turret Code field to "None" and the Park operations positions or the -53 option will be used by the post.

To use the AutoPark addin perform the following steps:

- 1. Create a park operation, select the tool/or Turret, Spindle as you would normally do
- 2. Do not set any values in the park tab for the park location of the XYZ axis
- 3. Press OK to create operation
- 4. Group park operation
- 5. Look in the properties dialog box and find the "Mori Seki AutoPark" heading
- 6. Press "+" to expend the fields under the heading
- 7. Select the Park Turret Code option in the pull down list.

Кеу	Value
Froup Count	2
🗄 General	
Technology	
Total Cycle Time:	00:00:00
Name	
Mori Seiki AutoPark	
Park Turret Code	G53 Z-505 🔹

10. How to Program a Part Transfer

10.1. Introduction

The part transfers can be programed using the Mori Seiki Workpiece Transfer utility. This utility streamlines the creation of the multiple operations used a typical transfers on the NT machines.

In order for the post to create the proper output for the part transfer there must be at least one operation in the lower turret's operation list, even if it is just a park operation.

			<u>×</u>		
Кеу		Value		<u> </u>	
Group Count		1	A		
🕀 General				9	
			=	8,6	
				610	
Work Coordinate		G54		08	
				8	
Total Cycle Time:		00:00:03	-	At	
]∢	1	11	•		
				The last	
< <sync>></sync>	100%	- 🙂 🕶 🐨			Workpiece
B-Axis Head		Bottom Turret	A		Transfer Utility
Name	S S	Name	S S		Icon
🖉 WNMG LH sub		🕢 drill Z+ L			
🕰 face sub	🗳 S.	wrap drilling main face L	🕹 M	44	
		(A) C face main	🕹 M		
Sync				- 🚌 📗	
		C/WNMG LH sub L	2 4	0	
Supc			S	•	
		U U U U			

The Workpiece Transfer Utility can be found on the Mori Seiki addin as shown below:

Selecting this Icon brings up the Workpiece Transfer dialog box. In the Workpiece Transfer dialog box you will noticed that your machine type is selected automatically and the list of the most common transfers are shown.

	📕 Workpiece Transfer		×
List of	Machine NT-SZ	Machine Type	
List of transfer types available in the utility for selected machine	Machine M1-52 Templates Sp1-CutOff-BarFeed-First Sp1ToSp2-BarFeed-CutOff Sp1ToSp2-BarFeed-CutOff Sp1ToSp2-BarFeed-CutOff Sp1ToSp2-BarFeed-CutOff Sp1ToSp2-BarFeed Sp1ToSp2-BarFeed Sp1ToSp2-BarFeed Sp1ToSp2-BarFeed Sp1ToSp2-BarFeed Sp1ToSp2-BarFeed Sp1ToSp2-BarFeed Sp1ToSp2-Release Sp2ToSp1-Release Sp2ToSp1-Release Bar Stock Feed Length 78.175	Operations Operations Park /Head_2 /Spindle_1 Part Eject /Head_2 /Spindle_2 Part Eject /Head_1 /Spindle_1 Part Eject /Head_1 /Spindle_1 Part Eject /Head_1 /Spindle_1 Part Eject /Head_1 /Spindle_1 Part Eject /Head_1 /Spindle_1	List of operations created by selected transfer
	Tabs for each op necessary setting	eration with the js	
		0	Cancer

At this point select the transfer you would like to use and fill in the appropriate values and the transfer will be created for you in the ESPRIT file.

Notes:

- Below are a summary of each of the transfer types supported by the utility.
- In the <u>Operation Tabs in the Workpiece Transfer Utility</u> section is a detailed expiation of the settings in operation tabs.
- The Workpiece Transfer Utility does not cover every situation yet. You can always modify what has been created to suite your specific needs. If you find errors or missing transfers please contact support and let us know so we can update and improve the utility.
- If your machine type is not selected you are not using the correct Machine Setup file. You will need to down load the latest machine setup file and update your ESPRIT file.
- In addition if you do not see the list of transfers you have an installation problem and you should contact support for help in resolving this matter.

10.2. SP1-CutOff-BarFeed-First

This transfer type really is not a transfer. This type of transfer would be used when only machining on the main spindle.

The following operations will be created:

- Bar feed by stopper on upper head at the top of the operation list
- Cutoff Operation. On the upper head.

chine NT-SZ	•	
emplates		Operations
p1-CutOff-BarFeed-First p1-CutOff-BarFeed-Last p1ToSp2-BarFeed-CutOff p1ToSp2-BarPull-CutOff p1ToSp2-CutOff-BarFeed p1ToSp2-CutOff-BarFeed p1ToSp2-Release p2ToSp1-Release		Image: Deparations At Beginning : Bar Feed /Head_1 /Spindle_1 Image: Operations At Last : Image: Deparation Bt Las
- Bar Feed = S Cut Off Tool Selection		
Tool	drill 10	•
Stopper Position		
G53 Z Position	0	↑ X
G53 X Position	0	
		_
Bar Stock		
Bar Stock Feed Length	78.175	
Bar Stock Feed Length	78.175	

Refer to the <u>Operation Tabs in the Workpiece Transfer Utility</u> section for a detailed expiation of the settings in operation tabs.

You may or may not need to park the lower turret before the bar feed operation to avoid collision with the upper head. This can be done after the transfer is created.



Below is an example of a file after the transfer was applied to an NT-SZ file:

Note: This same option exists on all of the NT machines, non-lower turret machines will not have the sync added, otherwise they are the same.
10.3. SP1-CutOff-BarFeed-Last

SP1-CutOff-BarFeed-last is very similar to the <u>SP1-CutOff-BarFeed-First</u> shown above.

The following operations will be created:

- Cutoff Operation. On the upper head.
- Bar feed by stopper on upper head

Achine NT-SZ	•		
emplates		Operations	
p1-CutOff-BarFeed-First p1-CutOff-BarFeed-Last p1To Sp2-BarFeed-CutOff p1To Sp2-BarPull-CutOff p1To Sp2-CutOff-BarFeed p1To Sp2-CutOff-BarFeed p1To Sp2-Release p2To Sp1-Release			vindle_1 Spindle_1
면 tool Selection]		
Tool	Grooving Cutoff	•	
Feeds and Speeds			
Feedrate PR	0.1	Speed RPM	500
Rough Passes			
Stock Allowance	0	Peck Increment	2
Stock Clearance	2	Peck Clearance	0.1
Miscellaneous			
Use Part Catcher	Yes	Cutoff Z	-75

Below is an example of a file after the transfer was applied to an NT-SZ file:



Note: This same option exists on all of the NT machines, non-lower turret machines will not have the sync added, otherwise they are the same.

10.4. SP1 To SP2- BarFeed-Cutoff

This is a transfer from the main spindle to the sub spindle.

The following operations will be created:

- Park operation on the lower turret (not used on single turret machines)
- Part Eject Cycle (on lower turret on NT-SZ, on upper turret on all other machines)
- Bar feed by stopper on upper head
- Pickup Operation by sub spindle
- Cutoff Operation. On the upper head.

achine NT-SZ	▼	
Templates		Operations
Sp1-CutOff-BarFeed-First Sp1-CutOff-BarFeed-Last Sp1ToSp2-BarFeed-CutOff Sp1ToSp2-BarPull-CutOff Sp1ToSp2-CutOff-BarFeed Sp1ToSp2-Release Sp2ToSp1-Release		Operations At Last : Park /Head_2 /Spindle_1 Part Eject /Head_2 /Spindle_2 Bar Feed /Head_1 /Spindle_1 F3 Pickup /Head_1 /Spindle_2 A Cut Off /Head_1 /Spindle_1
♥ Park │ /월 Part Eject Tool Selection Tool	🕌 Bar Feed 뵭 🖥 Pick Up 🖷 Cu	Cut Off
Park Position		
Position X	None	• 0
Position Y	None	• 0
Position Z	None	• 0

L face sub L 🕹 S Drill 10 Z+ Main Spindl dummy tool Park Lower Turret Part Eject S. 7barfeed stopper + Bar Feed By Stopper 🕹 М 🗳 S. Ξ Pickup Grooving Cutoff Cutoff 🗳 М Tools Operations 📇 Features Added Syncs and Operations Notes:

Below is an example of a file after the transfer was applied to an NT-SZ file:

This same option exists on all of the NT machines, non-lower turret machines will not have the sync or park operation added and the part eject cycle will be on the upper turret, otherwise they are the same.

NT000-SZ and NT000-WZ machines will also have the part eject cycle on the upper head

10.5. SP1 To SP2- BarPull-Cutoff

This is a transfer from the main spindle to the sub spindle.

The following operations will be created:

- Park operation on the lower turret (not used on single turret machines)
- Part Eject Cycle (on lower turret on NT-SZ, on upper turret on all other machines)
- Pickup Operation by sub spindle
- Bar Pull with sub spindle
- Cutoff Operation. On the upper head.

	•		
mplates		Operations	
p1-CutOff-BarFeed-First p1-CutOff-BarFeed-Last p1ToSp2-BarFeed-CutOff p1ToSp2-BarFeed-CutOff p1ToSp2-CutOff-BarFeed p1ToSp2-CutOff-BarFeed p1ToSp2-Release p2ToSp1-Release		Operations At Last : Park /Head_2 /Spindle_1 Part Eject /Head_2 /Spindle_2 Pickup /Head_1 /Spindle_2 Bar Pull /Head_1 /Spindle_1 Cut Off /Head_1 /Spindle_1	
V Park) 월 Part Eject 불 Tool Selection	ਤੋਂ Pick Up ਸ਼ੋ+ Bar Pull =ਾਨੂ (Cut Off	
1001			
Park Position			
Position X	None	▼ 0	
Position Y	None	• 0	
Position Z	None	▼ 0	

Below is an example of a file after the transfer was applied to an NT-SZ file:



Notes:

This same option exists on all of the NT machines, non-lower turret machines will not have the syncs or park operation added and the part eject cycle will be on the upper turret, otherwise they are the same.

NT000-SZ and NT000-WZ machines will also have the part eject cycle on the upper head

10.6. SP1 To SP2-Cutoff-BarFeed

This is a transfer from the main spindle to the sub spindle.

The following operations will be created:

- Park operation on the lower turret (not used on single turret machines)
- Part Eject Cycle (on lower turret on NT-SZ, on upper turret on all other machines)
- Pickup Operation by sub spindle
- Cutoff Operation. On the upper head.
- Bar feed by stopper on upper head

chine NT-SZ	•		
emplates p1-CutOff-BarFeed-First p1-CutOff-BarFeed-Last p1ToSp2-BarFeed-CutOff p1ToSp2-BarPull-CutOff p1ToSp2-Release p2ToSp1-Release p2ToSp1-Release		Operations	
♥ Park 월 Part Eject Chute Position Position Z	물 Pick Up 특 Cut Off) 971	⊨• Bar Feed	
Position Y Use Part Catcher	Yes	▼ Eject Type Off	•



Below is an example of a file after the transfer was applied to an NT-SZ file:

This same option exists on all of the NT machines, non-lower turret machines will not have the syncs or park operation added and the part eject cycle will be on the upper turret, otherwise they are the same.

NT000-SZ and NT000-WZ machines will also have the part eject cycle on the upper head

10.7. SP1 To SP2-Release

This is a transfer from the sub spindle to the main spindle.

The following operations will be created:

- Park operation on the lower turret (not used on single turret machines)
- Part Eject Cycle (on lower turret on NT-SZ, on upper turret on all other machines)
- Pickup Operation by main spindle
- Release Operation by the sub spindle.

workpiece in	ransfer		-
achine	NT-SZ	•	
Templates			Operations
Sp1-CutOff-Bar Sp1-CutOff-Bar Sp1ToSp2-Bar Sp1ToSp2-Bar Sp1ToSp2-Cut Sp1ToSp2-Rel Sp2ToSp1-Rel	rFeed-First rFeed-Last rFeed-CutOff rPull-CutOff tOff-BarFeed lease lease		Operations At Last : Park /Head_2 /Spindle_1 Part Eject /Head_2 /Spindle_1 Part Eject /Head_1 /Spindle_1 Part Eject /Head_1 /Spindle_1 Part Eject /Head_1 /Spindle_1
Park 🕌	물 Part Eject 틀콩 sition	Pick Up	
Position Z		971	
Position Y			
Position Y Use Part C	Catcher	Yes	▼ Eject Type Off ▼
Position Y Use Part C	Catcher	Yes	▼ Eject Type Off ▼
Position Y Use Part C	Catcher	Yes	▼ Eject Type Off ▼
Position Y Use Part C	Catcher	Yes	▼ Eject Type Off ▼
Position Y Use Part C	Catcher	Yes	▼ Eject Type Off ▼
Position Y Use Part C	Catcher	Yes	▼ Eject Type Off ▼

	face sub			4	S	🖑 wrap drilling main face L	4	М	
				-		C face main	ě	Μ	
	Sync					Sync			
						WNMG LH sub L			
						Lace sub L	4	S.	
	Sync					Sync			
	况 Drill 10 Z+								
	A Main Spindle	- Top Right `	Turret	4	М				
	Sync					🕒 Sync			
						dummy tool			
						Park Lower Turret	4	Μ	
	🔴 Sync					🔴 Sync			
						/∄ Part Eject	4	S.	_
	😑 Sync					Sync			=
	┣권 Pickup			4	S.				
	/🛃 Release			4	М				
									Ŧ
	Features	Tools		eratio	ons	; \			
						,			
						Added Syncs and			
						Operations			
Notes:						Operations			

Below is an example of a file after the transfer was applied to an NT-SZ file:

This same option exists on all of the NT machines, non-lower turret machines will not have the syncs or park operation added and the part eject cycle will be on the upper turret, otherwise they are the same.

NT000-SZ and NT000-WZ machines will also have the part eject cycle on the upper head

10.8. SP2 To SP1-Release

This is a transfer from the main spindle to the sub spindle. The following operations will be created:

- Park operation on the lower turret (not used on single turret machines)
- Part Eject Cycle (on lower turret on NT-SZ, on upper turret on all other machines)
- Pickup Operation by sub spindle
- Release Operation by the main spindle.

🕅 Workpiece Transfer	×
Machine NT-SZ	
Templates	Operations
Sp1-CutOff-BarFeed-First Sp1-CutOff-BarFeed-Last Sp1ToSp2-BarFeed-CutOff Sp1ToSp2-BarPull-CutOff Sp1ToSp2-CutOff-BarFeed	Operations At Last : Park /Head_2 /Spindle_1 Part Eject /Head_2 /Spindle_2 Pickup /Head_1 /Spindle_2 Pickup /Head_1 /Spindle_1
Sp2ToSp1-Release	
🛃 Park / 🗟 Part Ejec	╞ 꿀 Pick Up
Chute Position	
Position Z	971
Position Y	
Use Part Catcher	Yes Eject Type
	OK Cancel

	Lace sub		S	wrap drilling main face L	4	M	
	Sync			Sync			
				Lace sub L	4	S	
	Drill 10 Z+			U Sync			
<u>ا</u>	Sync	- 4	M	Sync			
				✓ dummy tool Park Lower Turret		M	
	Sync			Sync			
	Sync			/≓ Part Eject ● Sync	4	S.	Ξ
	╞퀄 Pickup /달 Release	4 4 4	S. M				
							Ŧ
	Features 🚺 Tools 🔜 C	peratio	ons	<u>ا</u> ا			
Notos				Added Syncs and Operations			

Below is an example of a file after the transfer was applied to an NT-SZ file:

This same option exists on all of the NT machines, non-lower turret machines will not have the syncs or park operation added and the part eject cycle will be on the upper turret, otherwise they are the same.

NT000-SZ and NT000-WZ machines will also have the part eject cycle on the upper head

10.9. Operation Tabs in the Workpiece Transfer Utility

In this section the specific settings used by the different tabs (Operations) created by the part transfer utility and how to modify them once the operation(s) are created.

10.9.1. Park

The park tab is used for lower turret machines. It is used to park the lower turret under the main spindle during prior to a transfer. The settings on this page will be used in the creation of a park operation.

Park Position			
Position X	Home	• 0	
Position Y	Home	• 0	
Position Z	Machine	-505	

The Park Tab has the following settings:

- Tool: The addin does not allow us to not select a tool as the new park operation does, but you can just select the same tool that is used by the previous operation.
- Park Position X: Recommend setting is "Home"
- Park Position Y: Recommend setting is "Home"
- Park Position Z: Recommend setting is "Machine" and the Far left position of your machines travel (see chart in <u>Park head/ turret to its minimum travel position (G53)</u> section for this location on your machine model)

10.9.2. Part Eject

The part Eject Tab is used to fill in specific settings used in the Part Release operation to release the part from the sub spindle. The settings on this page will be used in the creation of a release operation.

locition 7	071	
VOSICION Z	971	
Position Y		
Jse Part Catcher	Yes	Off 🔹

The Eject Tab has the following settings:

- Chute Position Z: This is only used for simulation; this does not affect the NC code. This is the location on the screen relative to Esprit World Zero that the part will be released at.
- Use Part Catcher (Yes/No): This set the part chute filed in the part release operation.
 - If set to NO the post will output M0's in the NC code to stop the machine to allow the operator to manually remove the part from the machine. (Lower Turret machines will have the M00 in both turrets NC code)
 - If set to yes the part will be removed automatically via the part chute.
- Eject Type (Off/M47) If selected an M47 will be output in the NC code in the part release.
 - Then M47 is not supported on the NT1000 machines this needs to be set to OFF for an NT1000 machine.
 - Then NT-S also uses an M360\M361. The addin does not currently support the M360\M361 but you can set this manually after the operation is created. See <u>Workpiece Ejector (NT-S and NT-SZ Only)</u> section below.
- See <u>Smart Loader Shutter (M37) (NT1000 only)</u> section below for information on activating the Smart Loader Shutter on and NT1000 machine. This can be set after the release operation is created since this feature is not supported by the workpiece transfer utility at this time.

Note: At the end of the release cycle on an NT1000 machine A axis will be at G330 P4 and the BW axis will be at BW0

10.9.3. Bar Feed

The bar feed tab is used to setup a bar feed by stopper type of operation. The settings on this page will be used in the creation of a bar feed operation.

1001	bar feed stopper 👻	
Stopper Position		
G53 Z Position		†×
G53 X Position	0	
Bar Stock		
Feed Length	78.175	

The Bar Feed Tab has the following settings:

- Tool: This is the tool used for the bar stopper
- Stopper Position G53 Z position: Position in the NC code for the bar stopper location
- Stopper Position G53 X position: Position in the NC code for the bar stopper location
 - Note if the Stopper position is left zero the NC code will move the bar stopper to the Z position equal to the value in the bar feed distance.
 - Refer to <u>Bar Feed positions relative to G53</u> section below for additional information on these values and how to change them after the operation is created.
- Bar Stock Feed length: feed length of the stock. This value is critical as it is used to shift the Z axis for the cutoff location.
 - This value must be the exact as every operation following the bar feed operation will be shifted in Z by this distance.
 - Set the Feed Length (2) equal to Length of the part + Cut-off tool width + Length of stock for facing the front side and the backside of the part.

Notes:

- The bar feed operation is only setup for the upper head and takes place at B-90 Degrees.
- The simulation of the B axis head will not match the NC code unless you modify the XZ positions in the bar feed operation.

10.9.4. Pick Up

The Pickup Tab is used to set the values needed for the pick up portion of the transfer. The settings on this page will be used in the creation of a pickup operation.

Feedrate PM	50 Spee	ed RPM	0
Sync Spindles			
Sync Type	Phase Sync (M34) C Ind	dex Angle	0
Pickup			
Position Z	0 Clea	rance	2
Pushing Check (G38)			
Pushing Check	Off	action Stroke (K)	0
Feedrate (F)	0 Posit	tion Tolerance (Q)	0

The Pick Up Tab has the following settings:

- Feeds and Speeds
 - Feedrate: Feedrate used in the pickup operation
 - Speed RPM: Spindle RPM used in the pickup operation (used in M34 or M35 modes only)
- Sync Type:
 - Phase Sync (M34): Option used for cutoff type transfers
 - This option will set the Sync Spindle field in the pickup operation to "Oriented"
 - Speed Synced (M35): Option used for cutoff type transfers
 - This option will set the Sync Spindle field in the pickup operation to "Speed and Direction Only"
 - Spindle Orient (M19): Option used for non-cutoff type transfers
 - C-Axis Connect (M45): Option used for non-cutoff type transfers
 - C Index Angle: Used when C-Axis Connect Sync type is used. This is the C axis angle of the spindles during the transfer. Typically set to 0. Refer to <u>C axis angle for pick operation</u> section below for additional information on these values and how to change this value after the operation is created
- Pickup
 - Position Z: Z position relative to World zero in ESPRIT for pickup point (note this will always be A0 in the NC code)
 - Clearance: Distance added to the pickup point to transition from rapid to feed (in addition to the clearance distance the post adds the sub spindle jaw length to the initial rapid position)

- Pushing Check (G38)
 - Pushing Check (On\Off): turns on or off the G38 Option
 - Feedrate: Feedrate for G38 command
 - Retraction Stroke (K): K value for G38 command
 - Position Tolerance (Q): Q value for G38 command
 - Refer to <u>Workpiece Pushing Check (G38)</u> section below for additional information on these values and how to change them after the operation is created
- For information about setting up a different work coordinate for the pickup operation see the <u>Work Coordinate for Pickup Operation</u> section below. This can be set after the pickup operation is created since this feature is not supported by the workpiece transfer utility at this time.

10.9.5. Bar Pull

When programming a transfer type with a barpull with the sub spindle, the barpull tab only has one setting. The settings on this page will be used in the creation of a pickup operation.

🛃 Park 🛛 🖓 Part Ejec	t ฿∄ Pick Up 📴 Bar Pull 📲 Cut Off	
Feed Length	78.175	
		OK Cancel

Set the Feed Length (2) equal to Length of the part + Cut-off tool width + Length of stock for facing the front side and the backside of the part.

10.9.6. Cutoff

The Cutoff Tab is used to set the values needed for the cutoff portion of the transfer. The settings on this page will be used in the creation of a cutoff operation.

Tool	Grooving Cutoff	•	
Feeds and Speeds			
Feedrate PR	0.1	Speed RPM	500
Rough Passes			
Stock Allowance	0	Peck Increment	2
Stock Clearance	2	Peck Clearance	0.1
Miscellaneous			
Use Part Catcher	Yes	✓ Cutoff Z	-75

The Cutoff Tab has the following settings:

- Tool: Sets the tool used for the cutoff operation
- Feeds and speeds section: Sets Feedrate and RPM for cutoff operation
- Rough passes section
 - Stock allowance: Sets Rough Stock allowance in the cutoff operation
 - Stock Clearance: Sets the Stock Clearance in the cutoff operation
 - Peck Increment: Sets Peck Increment in the cutoff operation
 - Peck Clearance: Sets Peck Clearance in the cutoff operation
- Miscellaneous
- Use Part Cather: Sets Use Parts Cather in cutoff operation
- Cutoff Z: Z location of the cutoff operation feature that is created by the addin to be used for the cutoff operation. This is typically the finish length of the part (assuming that the main spindle work coordinate is a world zero in ESPRIT). This value is always from World Zero in ESPRIT. This is loaded with the value in the Part Stock Length in the ESPRIT machine setup page.

10.10. Custom Settings for Part Transfer Cycles

While programming a part transfer cycle for the Mori Seiki NT Series machines, there are several options that the user has to tailor the cycle to their needs. Below is a short description of how to program these special functions in ESPRIT.

Some of these features are supported by the Workpiece Transfer utility and can be setup when the operations are created, others are not supported by the utility at this time but can be setup after the operation is created.

Once the operation is created these settings can be modified in the given operation(s) by navigating to the custom tab in the operation and editing the settings to the desired values.

10.10.1. Workpiece Pushing Check (G38)

In the workpiece transfer operation, when the position error of the A-axis servomotor of headstock 2 reaches the parameter set value, the workpiece is judged to have been brought into contact with the reference face of chuck 2 or chuck 1. Then, chuck 2 or chuck 1 clamps. In ESPRIT, it is possible to set the feedrate (F_), tolerance (Q_), and A-axis retraction stroke (K_) for a G38 workpiece pushing check cycle. The format of the operation is as follows:

G38 A_K_F_Q_

The Push Check settings are entered into the Pickup Operation in Esprit. The Mori Seiki Transfer will enter these values into the page, but if you need to modify them the values are in the following locations:

- Push Check (G38) (Custom Setting 1) needs to be set to a value of 38 If this is zero the settings below will have no effect on the NC code.
- G38 K value (Custom Settings 2) is the a-axis retraction stroke value K_.
- G38 Feedrate (Custom Setting 3) is the federate F_.
- G38 Q value (Custom Setting 4) is the tolerance Q_.

Enter the appropriate values for these variables in their respective custom setting box to output them in the code.

Push Check (G38)	0.000000	
G38 K Value	0.000000	
G38 Feedrate	0.000000	
G38 Q Value	0.000000	
Custom Setting 5	0.000000	
Spindle Orientation	0.000000	
C Index for M45	0.000000	
Work Coordinate	0.000000	
Custom Setting 9	0.000000	
Custom Setting 10	0.000000	

10.10.2. C axis angle for pick operation

During a non-cutoff type of transfer, the turning spindles are stopped.

Using the Spindle Orientation field (Custom Setting 6) on the Custom tab of the Pickup operation, the user can specify how to orient the spindles. Set Spindle Orientation field to 19 to use spindle orientation (M19 / M219) or set Spindle Orientation field to 45 to use the milling C-axis mode (M45 / M245).

When using milling C-axis mode, set the C axis Index for M45 field (Custom Setting 7) to specify the C angle value to position the C axis to during the transfer. Make sure to enter a correct value for C. For example, if C-axis roll-over is on, C has to be contained between -360 and 360.

Push Check (G38)	0.00000
G38 K Value	0.00000
G38 Feedrate	0.000000
G38 Q Value	0.00000
Custom Setting 5	0.000000
Spindle Orientation	0.000000
C Index for M45	0.000000
Work Coordinate	0.000000
Custom Setting 9	0.000000
Custom Setting 10	0.000000

NOTE: The milling C-axis mode is the default. So if the user omits a value in the Spindle Orientation field (leaves it set to zero) or enters a wrong value, the spindles will be indexed using this mode.

10.10.3. Bar Feed positions relative to G53

The Bar feed operation uses two custom settings to output the stopper position relative to machine zero (G53)

On the Custom tab, enter in the Stopper G53 X field (Custom Setting 4) the X position (in G53 work coordinate) of the stopper tool and in Stopper G53 X field (Custom Setting 5) the Z position (in G53 work coordinate) of the stopper tool.

If these values are left zero the post will position the bar stopper to the Z location in the operation relative to the active work coordinate for the spindle selected in the bar feed operation (typically this is the main spindle).

₩ 0)K 🗙 Cancel 🔆 🔸			🕜 Help
eral	Settings			
Ger	Custom Setting 1	0.000000		
g	Custom Setting 2	0.000000		
r Fe	Custom Setting 3	0.000000		
Ba	Stopper G53 X	0.000000		
ш	Stopper G53 Z	0.000000		
Cus	Custom Setting 6	0.000000		
	Custom Setting 7	0.000000		
	Custom Setting 8	0.000000		
	Custom Setting 9	0.000000		
	Custom Setting 10	0.000000		
			7	
e	Features 🔣 Tools	Operations	^b r [™] / ₂₄ + SolidTurn - Bar Feed	

10.10.4. Workpiece Ejector (NT-S and NT-SZ Only)

In the case that the workpiece is small or delicate, it is possible to use the Workpiece Ejector to ensure that the workpiece has been removed from the chuck. To initiate this function, navigate to the Custom tab of the Release operation used to eject the part from the sub spindle.

10.10.4.1. NT-S

The NT-SZ has the option of Set the Workpiece Ejector (Custom Setting 9) filed equal to 360 to output M360/M361 or 47 to output M47 in the part release section of the NC code.

10.10.4.2. NT-SZ

Set the Workpiece Ejector (Custom Setting 9) filed equal to 47 to output M47 in the part release section of the NC code.

🧼 O	🥔 OK 🗙 Cancel 🔆 🚽 🕜 Help						
ase	Settings						
Bele	Custom Setting 1	0.000000					
E	Custom Setting 2	0.000000					
Custo	Custom Setting 3	0.000000					
	Custom Setting 4	0.000000					
	Custom Setting 5	0.000000					
	Custom Setting 6	0.000000					
	Custom Setting 7	0.000000					
	Custom Setting 8	0.000000	_				
	Workpiece Ejector (M47)	47.000000					
	Custom Setting 10	0.000000					
	Features 🔣 Tools 📃	Operations Solid Turn - Release					

10.10.5. Work Coordinate for Pickup Operation

The turning work coordinate forces the part pickup operation into the Main Spindle Work coordinate. If the users would like to use a different work coordinate for a pickup operation the Turning Work Coordinate would not be able to be left in the auto run mode. This is a dangerous situation.

To avoid this we have setup the Work Coordinate (Custom Setting 8) field in the pickup operation for the work coordinate for the pickup operation. If this value is left zero or not 54, 55, 56, 57, 58, or 59, the post will output the work coordinate the pickup operation is tied to in the NC code.

The most common use of using alternative work coordinate for the pickup operation is when the using multiple pickup operations within the same file for doing long part machine and\or stock repositioning.

~ 0	K 🗡 Cancel 🌾 🝷	🕑 Help		
kup	Settings			
ä	Push Check (G38)	0.000000		
tom	G38 K Value	0.000000		
Clis	G38 Feedrate	0.000000		
	G38 Q Value	0.000000		
	Custom Setting 5	0.000000		
	Spindle Orientation	0.000000		
	C Index for M45	0.000000		
	Work Coordinate	58.000000		
	Custom Setting 9	0.000000		
	Custom Setting 10	0.000000		
				r
	Features 🚺 Tools	Operations	Solid Turn - Pickup	

A axis position for part release (NT1000 only)

The NT1000 needs to know where the sub spindle will be when the part is released from the sub spindle. This is set in Custom setting 9 in the part release cycle. This value needs to be relative to the machine coordinate system. See section <u>Special considerations when Programming an NT1000</u> sub section <u>Part Release from Sub Spindle</u> section for additional information.

10.10.6. Smart Loader Shutter (M37) (NT1000 only)

If the Part Chute option is used on the NT1000 Part eject you have the option to use or not use the Smart Loader Shutter. This is activated by putting a value of 37 in the Smart Loader Shutter (Custom Setting 10) in the part release operation.

See section <u>Special considerations when Programming an NT1000</u> sub section <u>Part Release from</u> <u>Sub Spindle</u> section for additional information.

<i>P</i> 0	0K 🗙 Cancel 🔆 🕶		🕢 Help
ease	Settings		
E E	Custom Setting 1	21.000000	
Ε	Custom Setting 2	0.000000	
Usto	Custom Setting 3	0.000000	
	Custom Setting 4	0.000000	
	Custom Setting 5	0.000000	
	Custom Setting 6	0.000000	
	Custom Setting 7	0.000000	
	Custom Setting 8	0.000000	
	A-axis Position	0.000000	
	Smart Loader Shutter (M37)	0.000000	
	Features 🛛 🗱 Tools 🛛 📰 O	perations 🛛 🗄 Solid Turn - Release	

11. Working with Long Parts: Pickup\Barpull then Simultaneous Machining

To support the stock with both spindles, you first need to pick it up (and, if needed, pull it) with the spindle 2. You can after program your operations and in the code will be output the necessary spindles synchronization and un-synchronization codes. Finally, once machining is completed, you can program one of the transfer scenarios described in the previous parts at the end of the operation list.

In order for the post to create the proper output to hold onto the part with both spindles transfer there must be at least one operation in the lower turret's operation list, even if it is just a park operation.

Note: On the General tab of the Machine Setup, set the Start Position Z: Start Position Z = Length of stock for facing the front side of the part or position of the stock in spindle 1 before pick up for synchronized machining

9	SolidTurn Machine Setup							
	General Assembly NC Output Custom							
]	-Mach	ine Definitio	n				
		Mach	ine Name		NZ2000-T2Y2			
	Start Positi		Position Z		1			
	Start		Position X		0			
		Start I	Position Y		0			

The typical steps to follow are:

- 1. Machining the parts on the spindle 1 and spindle 2 sides
- 2. Catching the finished part in the spindle 2
- 3. Picking the part in the spindle 1 with the spindle 2
- 4. Pulling the bar with the spindle 2 (if needed)
- 5. Machining the part on the spindle 1 side while the spindles are synchronized and are both supporting the part
- 6. Perform one of the previously described transfer scenarios to transfer the part to spindle 2 side
- 7. Looping for next parts

Notes:

It is possible to program multiple Pickup\Bar Pull operations in a row with no machining between them) to pull the part out of the spindle further then you have travel in the A axis.

Sync			Sync		
╞据 Solid Turn - Pickup	4	S.			
	4	М			
┣据 SolidTum - Pickup	4	S.			
➡+ SolidTum - Bar Feed	4	М			
A Supe			Sime		

It is possible to program several Pickup\Bar Pulls with machining between them to machine the part at different locations as you pull the part through the machines

If multiple pickup\bar pulls are programed the C axis orientation options will only be read from the first pickup operation.



Long part machining can be canceled by either performing a release operation or a part transfer.

12. Stock Repositioning

The post supports the ability to pull stock out of either the main or sub spindles with the opposite spindle with Pick\Barpull\Release operations. This is used to machine some operations with the

part close to the spindle, then pull the part out longer to machine operations with the sub spindle while supporting the part with a Steady Rest, Tailstock or unsupported.

Performing these operations is the same as doing a Pickup\Bar pull as described above in the <u>Working with Long Parts: Barpull and Simultaneous Machining</u> above, but with a release operation added after the Pickup\Bar pull

Mulitple Pickup\Barpulls can be performed in a row to allow you to pull the part out further than the machines A axis allows, below is an example showing this:



13. How to Output Coolant Codes

13.1. Introduction

The Mori Seiki NT Series can handle multiple different types of coolant codes:

M Code	Description
M08	Coolant ON
M09	Coolant OFF
M278	Through-spindle coolant ON <spindle 2=""></spindle>
M279	Through-spindle coolant OFF <spindle 2=""></spindle>
M382	Shower coolant ON
M383	Shower coolant OFF
M458	Tool tip air blow ON
M459	Tool tip air blow OFF
M478	Through-spindle coolant ON <spindle 1=""></spindle>
M479	Through-spindle coolant OFF <spindle 1=""></spindle>
M484	Through-spindle coolant ON (tool-spindle) /Through-spindle coolant ON for Knoll I/F
M485	Through-spindle coolant OFF (tool-spindle) /Through-spindle coolant OFF for Knoll I/F
M684	Through-spindle air blow ON (tool-spindle)
M685	Through-spindle air blow OFF (tool-spindle)

13.2. Setting First Coolant Code

The coolant code can be selected using the Coolant pull-down menu on the tool pages.

Coolant options supported by the NT Post:

• Off to output M09

.

- On or Flood to output M08 (M09)
- Mist to output M382 (M383)
- Flood2 to output M458 (M459)
- On Through Spindle to output M484 (M485)
- Mist Through Spindle to output M684 (M685)
- Flood Through Spindle to output M478 (M479)
- Flood 2 Through Spindle to output M278 (M279)

Turning tool page:

Turning Tools - Turning I	nsert			x
Insert Holder General	Custom			
Common Parameters-				
Tool ID	Tool 13	Orientation	3∨ ▼	
Tool Number	1	Orientation Angle	0.000000	
Turret Name	(Unmounted)	Compensation	Corner 💌	
Station Name	Station:1	Length Register	1	
Spindle Direction	CW			
Coolant	On 💌	Touch Off Angle	0.000000	
Simulation Cut Color	Off On			
Tool Change	Mist Flood			-1
Movement XZ,Y	Flood 2 On Through Spindle	Home 💌	Home 💌	
Position X,Z,Y	Mist Through Spindle	25.000000	0.000000	
- Tool Shift	Flood 2 Through Spindle			
Tool Shift X, Z, Y	0.000000	0.000000	0.000000	
				-1
		OK Ca	ncel Help	

Milling tool page:

Milling To	ools - Drill			×
	ools - Drill General Tool ID Tool Number Length Comp Register Coolant Spindle Direction Initial Clearance Unit Simulation Cut Color Feeds and Speeds Tool Material	Tool 13 13 13 13 0n Mist Flood 2 0n Through Spindle Therefore The state of the state o		×
Comme	nt			
			OK Cancel H	lelp

13.3. Setting the second coolant code (optional)

If you desire to output a second coolant code (which is optional), enter the coolant code value on the Second Coolant field (Custom Setting 1) field of the Custom tab of the tool page. For example M484 is desired, enter 484 In the Second Coolant Field (See captions below Coolant Pressure Options section.)

13.4. Setting the High Pressure Coolant Pressure options

13.4.1. Upper Head Option

Set Custom Setting 5 to 2041-2047 to output M2041-M2047. In order to use these you must have M484 coolant type selected as either the primary or secondary coolant options

13.4.2. Lower Turret Options

Set Custom Setting 5 to 440-447 to output M440-M447.

Turning Tool Page:

- Values	La 000000	-		
Second Coolant	0.00000	-		
Custom Setting 2	0.00000	_		
G30 or G28	0.000000	_		
Custom Setting 4	0.00000	-		
	2041.000000	_		
Custom Setting 6	0.000000	-		
Custom Setting /	0.00000	_		
Tool Life M(300)	0.000000			
Custom Setting 9	0.000000			
Custom Setting 10	0.000000			

Milling tool page:

Values Second Coolant	0 00000			
Second Coolant	0.00000		1	1
Custom Setting 2	0.000000			
G30 or G28	0.00000			
Custom Setting 4	0.000000	_		
Coolant Pressure	2041.000000			
Custom Setting 6	0.000000	-		
Custom Setting 7	0.000000			
Tool Life M(300)	0.000000			
Custom Setting 9	0.000000			
Custom Setting 10	0.000000			
t		3833		

14. Multiple Repetitive (Canned) Cycles

14.1. Introduction

The multiple repetitive cycles simplify the programs for rough and finish cutting processes on O.D./I.D. and face.

Roughing processes that require several blocks of commands can be specified by a single block of commands preceded by a G code calling a multiple repetitive cycle, and blocks that define the finished shape. The tool paths for rough cutting cycles are automatically determined.

For example, by defining only the workpiece finish shape, the tool paths for executing rough cutting operation are automatically generated.

G71	O.D./I.D. rough cutting cycle/pocket cutting (Cutting along the Z-axis, infeed along the X-axis) In O.D./I.D. cutting operation, when the finish shape is defined in the program, the workpiece is machined to the shape which includes a finishing allowance on the defined shape. The tool paths used for rough cutting are automatically determined.
G72	Rough facing cycle/pocket cutting (Cutting along the X-axis, infeed along the Z-axis) In facing operation, when the finish shape is defined in the program, the workpiece is machined to the shape which includes a finishing allowance on the defined shape. The tool paths used for rough cutting are automatically determined.
G70	Finishing cycle After completing a rough cutting cycle for O.D., I.D., or end face, called by the G71, G72, or G73 command, the shape is finished by the G70 cycle.

O.D./I.D. and face rough and finish cycles:

O.D./I.D.	and face groove	, cutoff and	threading cycles:	
• • • • • • • •		,		

G74	Intermittent Feed	Face cut-off cycle, face grooving cycle, deep hole drilling cycle Infeed is made along the Z-axis intermittently. If a command for the depth of cut along the X-axis is omitted, the deep hole drilling cycle is called.
G75	Intermittent Feed	O.D./I.D. grooving cycle, cut-off cycle Infeed is made along the X-axis intermittently. If a command for the depth of cut along the Z-axis direction is omitted, the cut-off cycle is called.
G76		Multiple thread cutting cycle The thread cutting pattern is repeated by gradually infeeding the cutting tool.
14.2. Roughing Cycle

14.2.1. Introduction

In ESPRIT, on the Strategy tab of the SolidTurn Roughing operation page, set Canned Cycle to Yes to trigger the multiple repetitive cycle output.



Based on the selected work strategy (Type of Work), the output G-code will be different. G71 will be output if your profile is on the O.D. or the I.D. of the part. G72 will be output if you cut on the face of the part.

roject	Manager			🕜 Help
2 6	Cutting Strategy			®
Gene	Tupe of Work	[nn		
Strategy	Finish Pass		5	
lough	Feature Extension			
	Start Extension	0.000000		
Custom	End Extension	0.000000		
Ĩ	8 Rapid Approach/Exit			
	Entry Mode	None		•
	Exit Mode	None		
C	Collision Detection			
	Undercutting Mode	Yes	•	
	Collision Detection	Yes		•
	Front Clearance Angle	0.000000		
	Back Clearance Angle	0.000000		
	Tool Blend	No		
G	Machine Functions			
	Cutter Comp NC	No		•
	Length Register	0		
	Canned Cycle	Yes		
F	eatures [🐻 Tools 🛛 🖽	Operations	💐 SolidT	urn - Roughing

A finish pass will only be output with G70 if you set Finish Pass to Yes.

Proje	ct Manager		×
🔶 OI	K 🗡 Cancel 🛛 📆 👻		🕢 Help
Teral	😢 Cutting Strategy		
trategy Ger	Type of Work	OD	
S	Finish Pass	Yes	•
lough	Feature Extension		
	Start Extension	0.000000	
Finisł	End Extension	0.000000	
ε	😵 Rapid Approach/Exit		
Custo	Entry Mode	None	•
	Exit Mode	None	<u> </u>
	S Collision Detection		
	Undercutting Mode	Yes	
	Collision Detection	Yes	•
	Front Clearance Angle	0.000000	
	Back Clearance Angle	0.000000	
	Tool Blend	No	<u> </u>
	S Machine Functions		
	Cutter Comp NC	No	•
	Length Register	0	
	Canned Cycle	Yes	
	Features 🚺 Tools	💶 Operations	🚅 SolidTurn - Roughing 🗌

Information about some of the parameters to enter for the following cycles:

- The relief amount R will be output as entered in ESPRIT.
- The feedrate F (and the cut speed S) can be specified on the General tab of the operation.

14.2.2. O.D./I.D. Roughing

G71 U(1) R_; G71 P_Q_U(2) W_F_;



U(1) and W will be output as entered in ESPRIT. U(2)/2 will be multiplied by 2 to have the correct value in the NC code U(2). R will be computed based on the entered Maximum Depth of Cut and Retract % of Depth value: R = Maximum Depth of Cut * Retract % of Depth / 100. For example, if you set Maximum Depth of Cut to 5 and Retract % of Depth to 75, R output in the NC code will be R3.75 (5*75/100).

14.2.3. Face Roughing

G72 W(1) R_; G72 P_Q_U_W(2) F_;

Proje	ct Manager	×
√ 0	K 🗙 Cancel 🛛 🕄 👻	🕡 Help
eral	Stock	
Gen	Туре	Length
trategu		
5 L	Stock Length	0.000000
Boug	Stock Allowance	W(2) U/2
ε	Rough Stock Z, X	0.000000 0.000000
Custo	😵 Passes	
	Rough Pattern	Parallel to Axis
	Depth Variation	Constant 📃
	Maximum Depth of Cut	5.000000 W(1)
	Clearance Along Cut	2,10000
	Depth Clearance	2.000 R
	S Retract	
	Retract % of Depth	100.000000
	Pullout Mode	Along Feature
	🗵 Lead-In/Out	
	Lead-In Type	Normal
	Normal Distance	0.000000
	Lead-Out Type	Normal
	Normal Distance	0.000000
	Features 🚺 Tools	🖪 Operations 🛛 齾 SolidTurn - Roughing

W(1) and W(2) will be output as entered in ESPRIT. U/2 will be multiplied by 2 to have the correct value in the NC code U. R will be computed based on the entered Maximum Depth of Cut and Retract % of Depth value: R = Maximum Depth of Cut * Retract % of Depth / 100.

14.3. Grooving Cycle

14.3.1. Introduction

In ESPRIT, on the Strategy tab of the SolidTurn Grooving operation page, set Canned Cycle to Yes to trigger the multiple repetitive cycle output.

Proje	ct Manager	×
~ 0	K 🗙 Cancel 🛛 📆 👻	🕜 Help
neral	Strategy	
Bel	Type of Work	OD 🗾
ð		
Strate		
5	Rough Pass	Yes
Boug	Finish Pass	Yes
	Rapid Approach/Exit	
Finish	Entry Mode	None
stom	Exit Mode	None
Ő	Entry && Exit Clearance	2.000000
	😸 Sharp Edges	
	Tool Blend	No
	Machine Functions	
	Cutter Comp NC	No
	Length Register	0
	Edge Shift Register	0
	Canned Cycle	Yes
	Features 8 Tools	Uperations Solid Furn - Grooving

Based on the selected work strategy (Type of Work), the output G-code will be different. G74 will be output if you cut on the face of the part. G75 will be output if your profile is on the O.D. or the I.D. of the part.

Proje	ct Manager	×
₩ 0	K 🗙 Cancel 🛛 🛍 👻	🕜 Help
lera	S Cutting Strategy	
Ger	Type of Work	
Strategy	Rough Pass	
Hough	Finish Pass	Yes I
	🗵 Rapid Approach/Exit	
Finish	Entry Mode	None
stom	Exit Mode	None
Ő	Entry && Exit Clearance	2.000000
	😻 Sharp Edges	
	Tool Blend	No
	S Machine Functions	
	Cutter Comp NC	No
	Length Register	0
	Edge Shift Register	0
	Canned Cycle	Yes
	Features 🐻 Tools 🖽	Operations 📲 SolidTurn - Grooving

Information about some of the parameters to enter for the following cycles:

- The return amount R(1) will be output as entered in ESPRIT.
- P is an unsigned value, in radius. Q is an unsigned value. Specify P and Q in mm, it will then be output in units of 0.001 mm without decimal point. For example, if you enter 2 for P in ESPRIT, the output will be in the NC code P2000.
- The feedrate F (and the cut speed S) can be specified on the General tab of the operation.

Note: For Face Grooving, the value P designates the Step Over and the value Q designates Peck Increment. For O.D./I.D. Grooving, these values are REVERSED, i.e. P is Peck Increment and Q is Step Over.

NOTE: Grooving canned cycles can only be used on grooves with straight wall geometry. Grooves with a side taper angle must be programmed with a regular groove operation.

NOTE: Grooving canned cycles can only be used when groove type is set to single plunge or multiple plunges. If you set groove type to zigzag or zigzag diagonal, you will get incorrect NC Code.



14.3.2. Face Grooving

 $\begin{array}{l} G74\ R(1);\\ G74\ X_\ Z_\ P_\ Q_\ F_\ ; \end{array}$

Stock	
Туре	Length
	· • • • • • • • • • • • • • • • • • • •
Stock Length	0.000000
Stock Allowance	
Rough Stock Z, X	0.000000 0.000000
Passes	
Groove Type	Multiple Plunge
Step Over, % of Tool Width	0.000000 P
Step Over Mode	Smart
Pre-finish	No
Dwell Time	0.000000
Plunges	
Plunge Pattern	Consecutive
Plunge Direction	Forward
Retract Feed Move	No
🖲 Pecking	_
Peck Increment	0.500000 Q
Peck Order	Down Then Across
Clearance	0.000000 R(1)
Full Retract IDepth	0.000000
Sub Groove Order	Region

14.3.3. O.D./I.D. Grooving

G75 R(1); G75 X_Z_P_Q_F_;



14.4. Threading Cycle

14.4.1. Introduction

In ESPRIT, on the Strategy tab of the SolidTurn Threading operation page, set Canned Cycle to Single Path to trigger the G76 multiple repetitive cycle output. If you set this parameter to Off, the threading cycle will be output with G32 or G34 if Lead Variation is different from 0. When Canned Cycle is set to Multiple Path, the threading cycle will be output with G92.

Project Manager	×
🛩 OK 🗙 Cancel 🔛 👻	🕜 Help
👸 区 Cutting Strategy	
Type of Work	
Strategy	
😨 😵 Rapid Approach/Exit	t
Entry Mode	None
Exit Mode	None
ී 🗵 Lead-In/Out	
Clearance	2.000000
Lead-In Type	Rapid
Lead-Out Type	Rapid
S Machine Functions	
Length Register	0
Canned Cycle	Off
	Off Single Path Multiple Path
Features Tools	🖽 Operations 🛛 🥁 SolidTurn - Threading

Information about some of the parameters to enter for the following cycles:

- The lead of the thread F will be output as entered in ESPRIT.
- Specify the angle of shift of the thread cutting start angle Q in $^{\circ}$, it will then be output in the units of 0.001° without a decimal point. For example, if you enter 2 for Q in ESPRIT, the output will be in the NC code Q2000.

• The cut speed S can be specified on the General tab of the operation.

On the Strategy tab, if you set the Lead-Out Type to Chamfer and the Chamfer Length greater than 0, you will have M23 (chamfer ON) output prior to the G76 or G92 commands. With a different lead-out set, the M-code output will be M24 (chamfer OFF).

Proje	ct Manager	×
₩ 0	K 🗙 Cancel 🛛 🔛 👻	🕜 Help
eral	S Cutting Strategy	
Gen	Type of Work	
	.,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,	
rateg		
ũ	Bapid Approach/Evit	
ead		
Thu	Entry Mode	None
ε	Exit Mode	None
Custo		
	Clearance	2.000000
	Lead-In Type	Rapid
	Lead-Out Type	Chamfer
	Chamfer Length	1.000000
	Machine Functions	
	Length Begister	0
	Congarinoglator	0
	Cannied Lycle	
	Features 🐻 Tools 🗉	Operations 🛛 🚟 SolidTurn - Threading

Proje	ct Manager		×
🔶 O	K 🗙 Cancel 🛛 🔛 👻		🕜 Help
eral	S Geometry		
Gen	Thread Definition	From Profile	
Ba	Thread Lead	2.000000	F
trate;	Thread Depth	1.000000	P(2)
S	Major Diameter	32.873700	
read	Minor Diameter	30.873700	
É	Lead Variation	0.000000	
ų m	Thread Angle	60.000000	Thead angle
Cust	Start Angle	0.000000	
	Threads / Lead	1	
	S Feature Extensions		
	Start Length, Leads	4.000000	2.000000
	End Length, Leads	4.000000	2.000000
	Passes		
	Depth Variation	Even Chip (Cross Section
	First Depth of Cut	0.400000	Q(2)
	Minimum Depth Of Cut	0.000000	Q(1)
	Number Of Rough Passes	7	
	Stock for Finish	0.000000	R(1)
	Number Of Finish Passes	0	P
	Thread Cutting Mode	Off	Number of finish
	Pecking Distance	0.000000	cuts to be repeated
	Features 🛛 🗱 Tools 🛛 🖭	Operations	于 SolidTurn - Threading

Proje	ct Manager	×
₩ 0	K 🗙 Cancel 🛛 🟦 👻	🕡 Help
eral	🗵 Cutting Strategy	
Gen	Type of Work	
а		
itrate		
57	S Rapid Approach/Exit	
hread	Entry Mode	None
F		Numero and
Istom		None
8	S Lead-In/Out	
	Clearance	2.000000
	Lead-In Type	Rapid
	Lead-Out Type	Chamfer
	Chamfer Length	5.000000
	S Machine Fur	P
	Length Register Thread	chamfering size
	Canned Cycle	Off
	Fashing Task	Occurring V. Collations Theory Con

R(1) will be output as entered in ESPRIT. Q(1), P(2) and Q(2) are unsigned values, in radius. Specify them in mm, they will then be output in units of 0.001 mm without decimal point. For example, if you enter 2 for Q(1) in ESPRIT, the output will be in the NC code Q2000.

14.4.3. Threading with Canned Cycle Set to OFF

 $\begin{array}{ccc} 14.4.3.1. & \mbox{Lead Variation Set to Zero (0)} \\ \mbox{G32 } X_Z_F_Q_; \end{array}$

Proje	ct Manager	<u>×</u>
🔶 C	K 🗙 Cancel 🛛 📆 👻	🕜 Help
eral	S Geometry	
Gen	Thread Definition	From Profile
а	Thread Lead	2.000000 F
Itrate	Thread Depth	1.000000
^o	Major Diameter	40.100000
rread	Minor Diameter	38.100000
Ê	Lead Variation	0.000000 G32
tom	Thread Angle	60.000000
Cus	Start Angle	0.000000 Q
	Threads / Lead	1
	Feature Extensions	
	Start Length, Leads	4.000000 2.000000
	End Length, Leads	4.000000 2.000000
	Sector Passes	
	Depth Variation	Even Chip Cross Section 🗾
	First Depth of Cut	0.400000
	Minimum Depth Of Cut	0.000000
	Number Of Rough Passes	7
	Stock for Finish	0.000000
	Number Of Finish Passes	0
	Thread Cutting Mode	Off 📃
	Pecking Distance	0.000000
	Features 📑 Tools 📰	Operations KolidTurn - Threading

Proje	ct Manager	<u>×</u>
₩ 0	K 🗙 Cancel 🛛 🟦 👻	🕢 Help
eral	S Geometry	
Gen	Thread Definition	From Profile
-Bi	Thread Lead	2.000000 F
Strate	Thread Depth	1.000000
-	Major Diameter	40.100000
hread	Minor Diameter	38.100000
F	Lead Variation	0.500000 G34; K
stom	Thread Angle	60.000000
Ő	Start Angle	0.000000 Q
	Threads / Lead	1
	Feature Extensions	
	Start Length, Leads	4.000000 2.000000
	End Length, Leads	4.000000 2.000000
	I Passes	
	Depth Variation	Even Chip Cross Section
	First Depth of Cut	0.400000
	Minimum Depth Of Cut	0.000000
	Number Of Rough Passes	7
	Stock for Finish	0.000000
	Number Of Finish Passes	0
	Thread Cutting Mode	Off
	Pecking Distance	0.000000
	Features	Operations K SolidTurn - Threading

K will be output as entered in ESPRIT.

14.4.4. Threading with Canned Cycle Set to Multiple Path

The post processor will automatically adjust the output if you are cutting a straight or a tapered thread.

Project Manager			
🛩 OI	K 🗙 Cancel 🛛 🔛 👻	🕡 Help	
eral	S Geometry		
Gen	Thread Definition	From Profile	
а	Thread Lead	2.000000 F	
Strate	Thread Depth	1.000000	
0,	Major Diameter	40.100000	
rread	Minor Diameter	38.100000	
È	Lead Variation	0.000000	
E E	Thread Angle	60.000000	
Cus	Start Angle	0.000000 Q	
	Threads / Lead	1	
	Feature Extensions		
	Start Length, Leads	2.000000	
	End Length, Leads	4.000000 2.000000	
	🛛 Passes		
	Depth Variation	Even Chip Cross Section	
	First Depth of Cut	0.400000	
	Minimum Depth Of Cut	0.000000	
	Number Of Rough Passes	7	
	Stock for Finish	0.000000	
	Number Of Finish Passes	0	
	Thread Cutting Mode	Off	
	Pecking Distance	0.000000	
	🔚 Features 🛛 🚯 Tools 🛛 🖽 Operations 🛛 🕌 SolidTurn - Threading		



15. B axis Turing Cycle

When programing a B axis turning cycle there are a few special consideration.

The tool Compensation filed MUST be set to "Center" for ESPRIT to correctly calculate the tool path for a B axis turning cycle.

Furning Tools - Turning Insert				
Insert Holder General Custom				
Common Parameters				
Tool ID	Boring Bar	Orientation	3V –	
Tool Number	2	Orientation Angle	0.000000	
Turret Name	B-Axis Head	Compensation	Center	
Station Name	Station:1	Length Register	2	
Spindle Direction	CW]		
Coolant	On 💌	Touch Off Angle	0.000000	
Simulation Cut Color]		
Tool Change				
Movement X,Z,Y	Home	Home 💌	Home 💌	
Position X.Z.Y	125.000000	125.000000	0.000000	
Tool Shift				
Tool Shift X, Z, Y	0.000000	0.000000	0.000000	
		OK Ca	Incel Help	

At the machine it is possible to still set the offset value at the corner of the insert by setting the Nose Radius Offset field (Custom Setting 1) on the B axis contouring page to the tools nose radius value.

If this value is set (not zero) the post will adjust the tool path by the value in the Nose Radius Offset field..

When programing a B axis turning cycle in ESPRIT the tool nose offset must be set to the center of the tool

₩ 0	🛩 OK 🗙 Cancel 🔆 🔹 🔞 Help			
eral	Settings			ן
Gen	Nose Radius Offset	0.400000		
8	Custom Setting 2	0.000000		
trate	Custom Setting 3	0.000000		
S	Custom Setting 4	0.000000		
Axis	G332 Setting Value	0.000000		
ur B,	Custom Setting 6	0.000000		
ontoi	Tool Nose Direction (1-9)	0.000000		
	G361/G362	0.000000		
stom	A-axis Position	0.000000		
Ğ	G30 or G28	0.000000		
	🖀 Features 🛛 🚺 Tools 🛛 🖽 Operations 🛛 🙋 SolidMill Turn - B-Axis Contouring			

16. Hole Machining Canned Cycles

16.1. Introduction

Hole machining programs are so complicated, specified in several blocks. A hole machining canned cycle, however, allows the hole machining to be programmed in one block with the appropriate G codes below:

The Face Hole Machining Canned Cycle:

Hole machining is executed by the Z-axis with positioning executed by the combination of the X-, C- and Y-axes.

The Side Hole Machining Canned Cycle:

Hole machining is executed by the X-axis with positioning executed by the combination of the Z-, C- and Y-axes.

In ESPRIT, you can choose between the different drilling canned cycles using the Cycle Type pull down on the Drill tab for the SolidTurn Drilling cycle:

Project Manager 🛛 💆				
🖌 🗸	🛩 OK 🗙 Cancel 🔛 👻			
ieral	S Drill Cycle			
Ger	Cycle Type	Drill	•	
Dril	Depths	Drill Peck		
E	Total Depth	Bore		
ustor	Starting Depth	Drill 2 Reak 2		
O I	Tip Already Included	Tap 2	_	
	😒 Clearances	H Bole 2		
	Return Plane	Clearance	•	
	Clearance	2.000000		
	S Machine Functions			
	Length Comp Register	0		
	Canned Cycle	Yes		
	Dwell Time	0.000000		
	Features		Turn - Drilling	
_ <u>B</u>				

eral	S Drill Cycle		
Gen	Cycle Type	Drill	
-	Reverse	Drill	
ā	S Depths	Тар	
ш	Total Depth	Bore Drill 2	
Cust	Starting Depth	Peck 2	
	Use Chamfer Diameter	Bore 2	
	Tip Already Included	No	
ľ	S Clearances		
	Full Clearance	10.000000	
	Clearance	2.000000	
	Return Plane	Clearance	
	Last Pt Return Plane	Clearance	
ĺ	S Machine Functions		
	Canned Cycle	Yes	
	Dwell	0.000000	

On the Drill tab for the SolidMillTurn Drilling cycle:

Project Manager × 🖉 OK 🛛 🗙 Cancel 🛛 🔛 👻 🕜 Help General 😵 Type Of Work • Diameter Wrap Drill 😵 Drill Cycle Drill Ŧ Cycle Type Drill ٠ Reverse Custom Peck Тар 😵 Depths Bore Drill 2 Total Depth Peck 2 Use Chamfer Diameter Tap 2 Bore 2 Tip Already Included 😵 Clearances 2 10.000000 Full Clearance 2.000000 Clearance **Return Plane** Clearance ▼ Clearance • Last Pt Return Plane 😵 Machine Functions Yes Canned Cycle • 0.000000 Dwell [🖁 Tools 🐉 SolidMillTurn - Wrap Drilling 📇 Features E Operations

On the Wrap Drill tab for the SolidMillTurn Wrap Drilling cycle:

Also make sure to set Canned Cycle to Yes:

Proje	Project Manager 🛛 🕺			
~ 0	K 🗙 Cancel 🛛 🔛 👻		🕜 Help	
ieral	😵 Drill Cycle			
Ger	Cycle Type	Drill		
E	S Depths			
	Total Depth	0.000000	R	
Istom	Starting Depth	0.000000	L3	
G	Tip Already Included	No	_	
	S Clearances			
	Return Plane	Clearance	•	
	Clearance	2.000000		
	S Machine Functions			
	Length Comp Register	0		
	Canned Cycle	Yes	•	
	Dwell Time	0.000000		
	Features [🖁 Tools	📰 Operations	🚰 SolidTurn - Drilling	

Note that the correct G-code for the drill canned cycle will be output based on the cycle type (SolidTurn Drilling or SolidMillTurn Drilling/Wrap Drilling) and the orientation of the tool in use: you will not need to choose between side or face hole machining canned cycle, the post processor will do it for you.

Information about some of the parameters to enter for the following cycles:

• The depth of cut per one infeed motion Q is an unsigned value, in radius for side drilling. Specify Q in mm, it will then be output in units of 0.001 mm without decimal point. For example, if you enter 2 for Q in ESPRIT, the output will be in the NC code Q2000.

• Enter the dwell P in s, it will then be output in units of 0.001s without decimal point. For example, if you enter 1.5 for P in ESPRIT, the output will be in the NC code P1500.

• The feedrate F (and the cut speed S) can be specified on the General tab of the operation.

16.2. Deep Hole Drilling Cycle (G74)

CAUTION !!! Use SolidTurn Drilling



G74 R(1) G74 Z_ Q_ F_;

On the SolidTurn Drilling cycle, set the Cycle Type to Drill or Peck 2 to output your drilling cycle with G74.

Proje	Project Manager 🛛 🛛 📉			
~	OK 🗙 Cancel 🔡 🗸 👘	🕜 Help		
10	🖾 Drill Cucle			
ene				
6	Cycle Type	Drill		
ā				
	Total Depth	Bore		
ston	Starting Depth	Drill 2		
3	Tip Already Included	Peck 2		
		Bore 2		
	😒 Clearances			
	Return Plane	Clearance		
	Clearance	2.000000		
	S Machine Functions			
	Length Comp Register	0		
	Canned Cycle	Yes		
	Dwell Time	0.000000		
	🖁 Features 🛛 🚺 Tools 🛛 🖥	🖪 Operations 🛛 🚰 SolidTurn - Drilling		

If you select Drill, the depth of cut per one infeed motion Q will be set to the total depth of cut plus 5 mm (or plus 0.2 in) so the hole will be drilled in one infeed motion. If you select Peck 2, you will be able to specify the value of Q that will be output in the NC code:

S Drill Cycle			
Cycle Type	Peck 2		
First Peck Increment	0.000000		
Peck Increment	0.250000 Q		
Full Retract IDepth	0.000000		

R(1) corresponds to return amount. It will be output as entered in ESPRIT when Peck 2 is selected. R0.0 will be output when Drill is selected.

Project Manager 🛛 🗡			
₩ 0	K 🗙 Cancel 🛛 🕄 👻	🕢 Help	
ieral	S Drill Cycle		
Gen	Cycle Type	Peck 2	
-	First Peck Increment	0.000000	
ā	Peck Increment	0.250000	
E C	Full Retract IDepth	0.000000	
Cust	S Depths		
	Total Depth	0.000000	
	Starting Depth	0.000000	
	Tip Already Included	No	
	S Clearances		
	Return Plane	Clearance	
	Clearance	2.000000 R(1)	
	S Machine Functions		
	Length Comp Register	0	
	Canned Cycle	Yes	
	Dwell Time	0.000000	
	Features 🔀 Tools 🗄	Dperations SolidTurn - Drilling	

16.3. Face and Side High-Speed Deep Hole Drilling Cycle (G83.5/G87.5)

CAUTION !!! Use SolidMillTurn Drilling or SolidMillTurn Wrap Drilling

G83.5 X_ (Y_) C_ Z_ R_ Q_ P_ F_ ; G87.5 Z_ (Y_) C_ X_ R_ Q_ P_ F_ ;

Project Manager 🛛 🛛 📉				
~ (OK 🗡 Cancel 🛛 🟦 🗸 👘		🕜 Help	
a a	S Drill Cycle			
Gene	Cucle Tupe	Peck 2 G83 5/G87 5	_	
	First Peck Increment	0.500000		
D.	Peck Increment	0.250000 Q		
ε	Full Retract IDepth	0.000000		
Custo	Reverse	No	•	
Ľ		,		
	Total Depth	1.000000	R	
	Starting Depth	0.000000	R	
	Use Chamfer Diameter	No	•	
	Tip Already Included	No		
	S Clearances			
	Full Clearance	10.000000	R	
	Clearance	2.000000		
	Return Plane	Clearance	•	
	Last Pt Return Plane	Clearance	•	
	S Machine Functions			
	Canned Cycle	Yes	•	
	Dwell	0.100000 P		
	🖞 Features 🛛 🥻 Tools 📔	🖪 Operations 🔤 🍪 SolidMillTurn -	Drilling	

16.4. Face and Side Deep Hole Drilling Cycle (G83.6/G87.6)

CAUTION !!! Use SolidTurn Drilling, SolidMillTurn Drilling or SolidMillTurn Wrap Drilling

当会会避雨品を形ぷ 予 やよど 品 ≪ 結晶 🕄 ≪ S & & &

G83.6 X_ (Y_) C_ Z_ R_ Q_ P_ F_ ; G87.6 Z_ (Y_) C_ X_ R_ Q_ P_ F_ ;

Project Manager 🛛 🛛 🛛 🛛			
🔶 C	K 🗡 Cancel 🛛 🖄 🗸	🕜 Help	
eral	S Drill Cycle		
Gen	Cycle Type	Peck G83.6/G87.6	
-	First Peck Increment	0.500000	
ā	Peck Increment	0.250000 Q	
stom	Reverse	No	
0°	🖲 Depths		
	Total Depth	1.000000	
	Starting Depth	0.000000	
	Use Chamfer Diameter	No	
	Tip Already Included	No	
	Clearances		
	Full Clearance	10.000000	
	Clearance	2.000000	
	Return Plane	Clearance	
	Last Pt Return Plane	Clearance	
	S Machine Functions		
	Canned Cycle	Yes	
	Dwell	0.100000 P	
	Features 8 Tools	🖸 Operations 🛛 📅 SolidMillTurn - Drilling	

Settings for SolidTurn Drilling:

Proje	Project Manager 🛛 💆			
🛩 0	K 🗙 Cancel 🛛 🕄 👻	🕜 Help		
eral	S Drill Cycle			
Gen	Cycle Type	Peck G83.6		
-	First Peck Increment	0.000000		
ā	Peck Increment	0.250000 Q		
stom	S Depths			
Ö	Total Depth	0.000000		
	Starting Depth	0.000000		
	Tip Already Included	No		
	Clearances			
	Return Plane	Clearance		
	Clearance	2.000000		
	Machine Functions			
	Length Comp Register	0		
	Canned Cycle	Yes		
	Dwell Time	0.100000 P		
	Features	🖪 Operations 🔤 🕼 SolidTurn - Drilling		

16.5. Face and Side Spot Drilling Cycle (G83/G87)

CAUTION !!! Use SolidMillTurn Drilling or SolidMillTurn Wrap Drilling

	品書器	4 6 4	1 4 4 4 7 6	5 II 🕹 🖥 🛃	予 故 出 世	\$ \$ \$
--	-----	-------	-------------	------------	---------	----------

 $\begin{array}{l} G83 \ X_{(Y_{})} \ C_{} \ Z_{} \ R_{} \ P_{} \ F_{} \ ; \\ G87 \ Z_{(Y_{})} \ C_{} \ X_{} \ R_{} \ P_{} \ F_{} \ ; \end{array}$

Proje	ct Manager		×			
≁ 0	K 🗙 Cancel 🔛 👻		🕜 Help			
eral	😒 Drill Cycle					
Gen	Cycle Type	Drill G83/G87	•			
-	Reverse	No				
ā	S Depths		\equiv			
stom	Total Depth	1.000000	B			
Ous	Starting Depth	0.000000				
	Use Chamfer Diameter	No				
	Tip Already Included	No	•			
	Clearances					
	Full Clearance	10.000000	R			
	Clearance	2.000000				
	Return Plane	Clearance	•			
	Last Pt Return Plane	Clearance				
	S Machine Functions					
	Canned Cycle	Yes	•			
	Dwell	0.100000 P				
	Features	🖸 Operations 🛛 🖓 SolidMillTurn -	Drilling			

16.6. Face and Side Tapping Cycle (G84/G88)

CAUTION !!! Use SolidMillTurn Drilling or SolidMillTurn Wrap Drilling



Project Manager 🛛 🛛 🖄							
~ (OK 🗙 Cancel 🛛 🛣 👻		🕜 Help				
eral	S Drill Cycle						
Gen	Cycle Type	Tap G84/G88	•				
12	Reverse	No					
	S Depths						
rstom	Total Depth	1.000000	L ₈				
۵ ا	Starting Depth	0.000000	R				
	Use Chamfer Diameter	No					
	Tip Already Included	No					
Clearances							
	Full Clearance	10.000000	R				
	Clearance	2.000000					
	Return Plane	Clearance					
	Last Pt Return Plane	Clearance					
	Machine Functions						
	Canned Cycle	Yes	•				
	Dwell	0.100000 P					
	Features	🖪 Operations 🛛 🖓 SolidMillTurn -	Drilling				

16.7. Tapping at Center of Spindle (G32)

CAUTION !!! Use SolidTurn Drilling



 $G32 Z_F_;$

If you set the Cycle Type to Tap on the SolidTurn Drilling cycle, the tapping canned cycle will be output with G32.

Project Manager 🛛 🛛 🖄						
~	OK 🗙 Cancel 🛛 🛐 🗸 👘		🕜 Help			
Teral	😒 Drill Cycle					
Ger	Cycle Type	Tap G32				
Deil	S Depths					
	Total Depth	0.000000	R			
Istom	Starting Depth	0.000000				
0	Tip Already Included	No				
	S Clearances					
	Return Plane	Clearance				
	Clearance	2.000000				
	S Machine Functions					
	Length Comp Register	0				
	Canned Cycle	Yes				
	Dwell Time	0.000000				
		-				
	Features	Uperations	ঞৰ SolidTurn - Drilling			

16.8. Synchronized Tapping (M329)

16.8.1. Limitations of Synchronized Tapping

CAUTION !!! During the synchronized tapping cycle, the maximum speeds for the rotary tool is 3000 min⁻¹.

If the maximum speed for the rotary tool holder is less than the above, specify a speed that is less than the maximum speed for the holder.

16.8.2. Face and Side Synchronized Tapping (M329 G84/G88)

CAUTION !!! Use SolidMillTurn Drilling or SolidMillTurn Wrap Drilling



M329 S_; G84 X_(Y_) C_ Z_ R M329 S_; C ⁸⁸ Z_(Y_) C_ Y_ R	_ P_	F_;			
$\operatorname{Goo} \operatorname{Z}_{(1)} \operatorname{C}_{A} \operatorname{K}_{(1)}$	_ P_	Γ_;			ed l
		K X Cancel 1 🗐 👻			
	Gener				
	Dril	Tool Selection	-??????-		
	Custom	Spindle Name	MainSpindle		
		Head Id	Head-1		
		Cut Speed RPM, SPM	0	M329 S	
		% of Calculated Feed	0		
		S Comment	No		
		Features 🚺 Tools	🖪 Operations	🔠 SolidMillTurn	- Drilling

Proje	ct Manager		×
₩ 0	K 🗙 Cancel 🛛 📆 👻		🕜 Help
teral	S Drill Cycle		
Ger	Cycle Type	Tap 2 G84/G88	•
E	Reverse	No	
	S Depths		
Istom	Total Depth	1.000000	R
O	Starting Depth	0.000000	R
	Use Chamfer Diameter	No	•
	Tip Already Included	No	•
	S Clearances		
	Full Clearance	10.000000	R
	Clearance	2.000000	
	Return Plane	Clearance	•
	Last Pt Return Plane	Clearance	
	S Machine Functions		
	Canned Cycle	Yes	•
	Dwell	0.100000 P	
	Features Tools	🖪 Operations 🛛 🖓 SolidMillTurn -	- Drilling

16.8.3. Face and Side (High-Speed) Deep Hole Synchronized Tapping (M329 G84/G88)

In order to program Deep Hole Synchronized Taping Set the Cycle Type as described above in the Synchronized Tapping (M329) section and set the depth of cut per pass (Q) in the G84/G88 Q Value (Custom Setting 3) field of the SolidMill Drilling (or Wrap Drilling) operation.

	DK 🗙 Cancel 🔆 🗸) Help
ieral	Values			
Ger	Custom Setting 1	0.000000		
=	Custom Setting 2	0.000000		
ā	G84/G88 Q Value	2.000000		
E	Custom Setting 4	0.000000		
Custo	G332 Setting Value	0.000000		
	Custom Setting 6	0.000000		
	Custom Setting 7	0.000000		
	G361/G362	0.000000		
	Custom Setting 9	0.000000		
	G30 or G28	0.000000		
	Features 🖪 Tools	🖽 Operations 🛛 🖓 Soli	dMillTurn - Drilling	

16.8.4.Spindle Synchronized Tapping (M329 G84) (Option)CAUTION !!!Use SolidTurn Drilling



 $\begin{array}{l} M329 \; S_{\,\,}; \\ G84 \; X_{\,\,} Z_{\,\,} R_{\,\,} P_{\,\,} F_{\,\,}; \end{array}$

The M329 G84 command is used to execute a tapping operation at the center of the spindle (rotational center of workpiece).

This tapping cycle allows the Z-axis feed to be always synchronized with spindle revolution.

Set the Cycle Type to Tap 2 on the SolidTurn Drilling cycle to have the synchronized tapping canned cycle output with M329 G84

Project Manager 🛛 🛛 🛛 🛛						
₩ 0	K 🗙 Cancel 🛛 📆 🗸			🕜 Help		
eral	S Operation Name					
Gen						
-				\equiv		
ā				1 -		
E		-mm-				
Custo	😻 Work Setup					
	Spindle Name	MainSpindle		•		
	Head Id	Head-1		•		
	Feeds and Speeds			\equiv		
	Speed Unit RPM					
	Speed RPM, CSS	0	M329 S			
	Feed Unit	Per Revolution	n	•		
	Feedrate PM, PR	0.000000	0.000000	_ ▲		
	Maximum RPM	5000				
	Spindle Range	Off		•		
	Spindle Priority	Off		•		
	Use Feed and Speed KB	No		•		
	S Comment			\equiv		
]					
	Features 🔡 Tools 🗄	Operations	ু ঞিশ SolidTurn -	Drilling		
Proje	ect Manager	×				
-------	----------------------	---------------------------------				
🛩 C)K 🗙 Cancel 🗐 🗸	🕜 Help				
eral	🗵 Drill Cycle					
Gen	Cycle Type	Tap 2 G84				
E	🗵 Depths					
	Total Depth	0.000000				
Istom	Starting Depth	0.000000				
L C	Tip Already Included	No				
	S Clearances					
	Return Plane	Clearance				
	Clearance	2.000000				
	S Machine Functions					
	Length Comp Register	0				
	Canned Cycle	Yes				
	Dwell Time	0.100000 P				
L						
	Features 🚺 Tools 📃	Operations SolidTurn - Drilling				

16.8.5. Face and Side Boring Cycle (G85/G89)

CAUTION !!! Use SolidMillTurn Drilling or SolidMillTurn Wrap Drilling



$\begin{array}{l} G85 \; X_{-}\left(Y_{-}\right) C_{-} \; Z_{-} \; R_{-} \; P_{-} \; F_{-} \; ; \\ G89 \; Z_{-}\left(Y_{-}\right) C_{-} \; X_{-} \; R_{-} \; P_{-} \; F_{-} \; ; \end{array}$

Project Manager 🛛 🛛 💌			
🔶 C)K 🗡 Cancel 🖺 🗸	🕢 He	۶lp
a l	😒 Drill Cycle		ר
Gene	Eucle Tupe	Rev. 685/689	
3	Unset	0.000000	
	Reverse	No	IJ
stom	逐 Depths		ונ
Ő	Total Depth	1.000000	
	Starting Depth	0.000000	
	Use Chamfer Diameter	No	
	Tip Already Included	No	
	Clearances		٦l
	Full Clearance	10.000000	1
	Clearance	2.000000	1
	Return Plane	Clearance	
	Last Pt Return Plane	Clearance	
	S Machine Functions		٦ I
	Canned Cycle	Yes	
	Dwell	0.100000 P	
	Features 🚺 Tools 📱	🗄 Operations 🔄 🏭 SolidMillTurn - Drilling	g

16.8.6. Boring in Turning Mode CAUTION !!! Use SolidTurn Drilling



If you want to perform a boring cycle with the SolidTurn Drilling operation, set the Cycle Type to Bore and Canned Cycle to No. No canned cycle is supported for the turn boring cycle. The code will be output with linear G01 moves.

Proje	ct Manager		×
₩ 0	K 🗙 Cancel 🛛 🟦 👻		🕜 Help
heral	😺 Drill Cycle		
Ger	Cycle Type	Bore	_
-=	Offset	0.000000	
	S Depths		
stom	Total Depth	0.000000	
0	Starting Depth	0.000000	R
	Tip Already Included	No	•
	S Clearances		
	Return Plane	Clearance	•
	Clearance	2.000000	
	S Machine Functions		
	Length Comp Register	0	
	Canned Cycle	No	-
	Dwell Time	0.000000	
	Foshiros		

16.9. Spindle/Rotary Tool Spindle Simultaneous Operation Mode CAUTION !!! Use SolidTurn Drilling



M90 S_; M91; M290 S_; M291;

The M90 and M290 commands are used to rotate the spindle and the rotary tool simultaneously to machine a hole at the center of the spindle (workpiece) at a proper cutting speed if necessary cutting speed cannot be obtained by rotating only the spindle or the rotary tool.

On the Custom tab of the SolidTurn Drilling operation, set the Simultaneous Operation (Custom Setting 6) field to 90 to trigger the output of the M90 (or M290 if machining on the spindle 2 side) command.

On the General tab, set in Speed RPM to the desired total speed, which mean speed of the turning spindle plus speed of the rotary tool.

You can then specify with the Percentage of Spinning Tool RPM (Custom Setting 7) field the percentage of the total speed you want to apply to the rotary tool.

Example:

If you set the Speed RPM to 6000 and Percentage of Spinning Tool RPM to 30, the spindle speed will be 6000 - 30% * 6000 = 4200 rpm and the rotary tool speed will be 6000 * 30% = 1800 rpm.

Notes:

• Set on the General tab, the Feed Unit to Per Minute: You can determine the feedrate of the tool assuming the spindle speed of 6000 min^{-1} , which is the sum of the turning spindle speed (4200 min⁻¹) and the rotary tool speed (1800 min⁻¹).

• On the General tab, make sure to change the Maximum RPM value according to the specified Speed RPM value. If Speed RPM is set to 6000 rpm but Maximum RPM to 5000 rpm, the output will be computed using 5000 rpm, as a safety.

• The direction of rotation has to be specified on the Tool page by the pulldown Spindle Direction.

Proje	ct Manager		
₩ 0	K 🗙 Cancel 🛛 📆 🗸		🕜 Help
heral	S Operation Name		
Ger			
E	S Tool Selection		
ε	Tool	??????	
Custo	😻 Work Setup		
	Spindle Name	MainSpindle	
	HeadId	Head-1	•
	😵 Feeds and Speeds		
	Speed Unit	RPM	•
	Speed RPM, CSS	6000	0
0	Feed Unit	Per Minute	G98 💽
	Feedrate PM, PR	0.000000	0.000000
	Maximum RPM	6000	
	Spindle Range To	tal speed	<u> </u>
	Spindle Priority	100	
	Use Feed and Speed KB	No	
	S Comment		
	U		
	Features 🐻 Tools	🖽 Operations	🚰 SolidTurn - Drilling

🥪 0	OK 🗙 Cancel 🔆 🗸 🕜 Help		
eral	Settings		
Gen	Custom Setting 1	0.000000	
=	Custom Setting 2	0.000000	
2	Custom Setting 3	0.000000	
E	Custom Setting 4	0.000000	
Custo	G332 Setting Value	0.000000	
	Simultaneous Operation	90.000000	
	Percentage of Spinning Tool RPM	30.000000	
	G361/G362	0.000000	
	Custom Setting 9	0.000000	
	G30 or G28	30.000000	
6	Features 🛛 🗱 Tools 🛛 💀 Operatio	ons BaidTum - Drilling	

17. 3D Coordinate Conversion (G68.1)

By using the G68.1 command, it is possible to rotate the axis of the program in such a manner that the tool axis is always the Z axis in the NC code.

The 3D coordinate conversion function is always output when performing a non 5-Axis MillTurn operations with the B-axis head tilted to an angle other than 0 or \pm -90.

By using the <u>Turning Work Coordinates Addin</u> as described in this manual the operations will always be put into the correct work coordinate in ESPRIT ensuring that the NC code is correct.

Note that the center of rotation will always be at the local zero point (for example G54 X0; Y0; Z0) and the direction of the center axis of rotation (I0; J1; K0): the coordinate system will be tilted around the Y-axis. The angle of rotation R_ will automatically be computed for you based on the angle of the B-axis head.

18. 4-Axis Wrap Milling Cycles with Interpolation

18.1. Introduction

The NC code can be very long for wrap milling operations using the C-axis. But it can be shorten by turning on the interpolation.

18.2. Cylindrical Interpolation

The cylindrical interpolation can be turned on the SolidMillTurn Wrap Pocketing and Wrap Contouring pages.

On the Wrap tab of the operation pages, set Cylindrical Interpolation to Yes to output 4-axis wrap milling operations with cylindrical interpolation.

Proje	Project Manager 🛛 🛛 🗡				
🖌 🔶 OI	K 🟋 Cancel 🛛 📆 👻	🕢 Help			
heral	😸 Rotary Machining				
Ger	Working Diameter	25.000000			
æ	Type of Work	OD 🔽			
Ň	Wall Type	Radial Tool Axis			
-	Cylindrical Interpolation	Yes			
i i i i i i i i i i i i i i i i i i i	Tolerance	0.100000			
•					
	Features 🚺 Tools 📰 (Operations 🛛 💑 SolidMillTurn - Wrap Pocketing			

It will be turned on in the NC code by G107 C_{_} and turned off by G107 C0.

C corresponds to the radius of the workpiece (groove bottom). It can be specified (as a diameter value) on the Wrap tab in the Working Diameter field of the operation pages:

Proje	ct Manager	×
🔶 O	K 🗙 Cancel 🛐 🗸	🕐 Help
heral	S Rotary Machining	
Ge	Working Diameter	25.000000 C*2
æ	Type of Work	OD 🗾
Ň	Cylindrical Interpolation	Yes
tegy	S Transverse Move	
Stra	Move Type	Radial Tool Axis
	Preserve Radial Wall	No
-		
	Features 🚺 Tools 🛄	Operations 🕹 SolidMillTurn - Wrap Contouring

Cylindrical interpolation is used when cutting on cylinder circumference. Programming is made assuming a plane by developing the cylinder circumference into the plane.



Using cylindrical interpolation can shorten the program length because ESPRIT outputs the circular motions in the cylindrical plane.

Without cylindrical interpolation, circular motions in the cylindrical plane are approximated by small linear moves (quality varies with the tolerance of the operation page).

18.3. Polar Interpolation

The polar coordinate interpolation can be turned on the SolidMillTurn Wrap Face Pocketing and Wrap Face Contouring pages.

On the Strategy tab of the operation pages, set Polar Interpolation to Yes to output rotary face milling operations with polar interpolation.

Proje	Project Manager			
~ (OK 🗙 Cancel 🛛 🛍 🗸	🕡 Help		
eral	😵 Rotary Machining			
Gen	Polar Interpolation	Yes		
(eg)	S Cutting Strategy			
Stral	# of Rough Passes	1		
pa	Finish Pass	No		
Vanci	Cutting Strategy	Climb		
ΡA	Process Order	Width		
inks	Spiral Move	No		
Stock Allowance				
Istom	Stock Allowance Walls	0.000000		
0	Stock Allowance Floors	0.000000		
	🗵 Depths			
	Total Depth	3.000000		
	Incremental Depth	0.000000		
	Starting Depth	0.000000		
	Tapered Wall Angle	0.000000		
	S Compensations			
	Offset Side Computer	Left		
	Offset Tool Radius	Yes		
	Cutter Comp NC	Left		
	Offset Register NC	1		
	Offset Register Value	0.000000		
	Features	Operations 🔊 SolidMillTurn - Wrap Face Contouring		

It will be turned on in the NC code by G112 and turned off by G113.

Tool positions are output in rectangular coordinate system (X; C) (C = Y distance), instead of XC (C in degrees). The work system rotates to the start C position of the operation. The start point is always at C (Y distance) = 0.



19. Tailstock

19.1. Live Center on Lower Turret

When machining long work pieces with the Mori Seiki NT machines equipped with the lower turret, it is sometimes beneficial to use the lower turret as a live center (i.e. tailstock) to support the workpiece while machining.

19.1.1. Defining Tailstock in Machine Setup

Open the Machine Setup page and navigate to the Assembly tab. Click on Tailstock to view the Tailstock Properties window. Under Type select Turret. Under Turret Name select Bottom Turret. In Station Name, it is possible to select the tool station which will hold the tailstock fixture. This value will be output during the tool change for tailstock operations. It is also possible to define specific geometric properties of the tailstock in this section.

B-Axis Head Bottom Turret	New Turret	New Spindle	New Steady Rest	Delete
Main Spindle	×	New Axis	New Solid	New Station
Sub Spindle Talistock	- Tailstock Properties		3	
- Axes	Туре	Turret 💌		
- Solids	Turret Name	Bottom Turret 💌		
	Station Name	Station:6		
	Home Position X	-495.000000	GD GA	BD
	Home Position Y	0.000000		<u> </u>
	Home Position Z	0.000000		
	Custom Tailstock			
	Tailstock Length (TL)	100.000000	Grip Diameter (GD)	5.000000
	Grip Angle (GA)	50.000000	Base Diameter (BD)	50.000000

Machine Setup Assembly Page

19.1.2. Creating Tailstock Operation

To create a Tailstock operation, navigate to Solid Turn (1) and then Tailstock (2) in the tool bar.



In the Tailstock tab, select Main Spindle for Spindle Name to have the tailstock engage with the workpiece in the main spindle. Select Bottom Turret for Turret Name to signify that the tailstock is attached to the bottom turret.

~ (OK 🟋 Cancel 🛛 🚰 🗣		🕜 Help
tock	S Operation Name		
Tails	SolidTum - Tailstock Engage		
Custom	Work Setup		
	Spindle Name	Main Spindle	-
	Turret Name	Bottom Turret	•
	Head Id	Head-1	-

Under Tailstock you can select whether the operation is an Engage or Disengage. This tells the machine whether you would like to tailstock to engage with the workpiece or if you would like it to retract from the workpiece after machining has been completed.

Position X, Y, Z	0.000000	0.000000	0.000000	B
Tailstock	Engage			•

After selecting which type of operation to perform, click OK and you will be prompted to select the Tailstock reference engage point. This is the point that the tailstock will engage the part at or the point to which it will return after disengaging.

NOTE: There should be a Sync directly before and after any tailstock operations to ensure that no other machine movement is occurring while the tailstock is engaging /disengaging.

19.2. Second Live Center on Lower Turret

At times it may be necessary to have two live centers in the lower turret (one for the main spindle, one for the sub spindle). In Esprit we are currently limited to creating one lower turret center in the machine setup page.

We have created a method by which you can use the manual turning operation to program a second center on the lower turret by setting the Tail Stock Position field (Custom Setting 1) to either a 1 (engage) or -1 (disengage).

The Manual Turning is found on the SolidTurn tool bar as shown below:



It will be necessary to create a tool that you can use as the tail sock that will resemble the center you will use. You can even use a custom tool if necessary, but often a drill works well The spindle speed settings will not be used but the feedrate will be used in the same manner the feedrate is used in the tail stock operation.

Once you create the operation you then will select points on the graphics area (that you have previously created) to define the manual operation. Setup the points in such a way as to imitate the motions the tail machine will make as the tail stock is engaged or disengaged so your simulation will match the machine movements.

The manual operation MUST have ONE and ONLY ONE rapid move in the operation. If this is not done correctly the NC code will not be correct.

√ 0	K 🗙 Cancel 🛛 🚰 🗸	🕢 Help
Ibun	Settings	
Ma	Tailstock Position	1.000000
EO	Custom Setting 2	0.000000
Cust	Custom Setting 3	0.000000
	Custom Setting 4	0.000000
	G332 Setting Value	0.000000
	Custom Setting 6	0.000000
	Tool Nose Direction (1-9)	0.000000
	G361/G362	0.000000
	Custom Setting 9	0.000000
	G30 or G28	0.000000
	Features 🛛 🗱 Tools 🛛 🖽	Operations SolidTurn - Manual Turning

NOTE: There MUST be a Sync directly before and after any tailstock operation to ensure that no other machine movement is occurring while the tailstock is engaging /disengaging.

B-Axis Head			Bottom Turret			
Name	S	S	Name	S.	. S	
😑 Sync			😑 Sync			
			Tailstock Engage	4	M	
Sync			🔴 Sync			
DNMG 432 ROUGH						
🚅 Solid Tum - Roughing	- 4	М				
Sync			Sync			
DNMG 432 UPPER MAIN						
SolidTum - Contouring	4	М				=
SolidTum - Contouring	4	М				
SolidTum - Contouring	4	М				
Sync 🗧			Sync			
🕰 Laydown Thread						-
Solid Turn - Threading	<u></u>	М				
Sync			😑 Sync			
0.159 GARR UPPER						
Solid Mill Turn - Drilling	4	М				
Sync			Sync			
			Ci Tailstock Disengage	4	M	
			BLANK STATION			
			Solid Turn - Park	4	M	
Sync			Sync			
Frank Solid Turn - Pickup	4	S.				
늞-Solid Turn - Bar Feed	4	М				-

19.3. Independent Tail Stock

On machines that are equipped with a tail stock rather than a sub spindle you will use the Independent tail stock in ESPRIT to create a tail stock in the machine setup page.

19.3.1. Defining Tailstock in Machine Setup

Open the **Machine Setup** page and navigate to the **Assembly** tab. Click on **Tailstock** to view the **Tailstock Properties** window. Under **Type** select **Independent** for the NT-S machine. It is also possible to define specific geometric properties of the tailstock in this section.

New Turret	New Spindle	New Steady Rest	Delete
× I	New Axis	New Solid	New Steppin
Tailstock Properties		1	
Туре	Independent •		п <i>—</i>
Turret Name			1
Home Position Y	0.00000	GD GA	BD
Home Position Y	0.000000		1
Home Position Z	1016.000000		
Custom Tailstock	,		
Tailstock Length (TL)	100.000000	Grip Diameter (GD)	0.000000
Grip Angle (GA)	50.000000	Base Diameter (BD)	50.000000
9			
	New Turret Tailstock Properties Type Turret Name Station Name Home Position X Home Position Y Home Position Z Custom Tailstock Tailstock Length (TL) Grip Angle (GA)	New Turret New Spindle X New Axis Tailstock Properties Type Independent Turret Name Image: Comparison of the system Station Name Image: Comparison of the system Home Position X 0.000000 Home Position Y 0.000000 Home Position Z 1016.000000 Custom Tailstock Tailstock Length (TL) Tailstock Length (TL) 100.000000	New Turret New Spindle New Steady Rest X New Axis New Solid Tailstock Properties Independent Image: Comparison of the system Turret Name Image: Comparison of the system Image: Comparison of the system Station Name Image: Comparison of the system Image: Comparison of the system Home Position X 0.000000 Image: Comparison of the system Home Position Z 1016.000000 Image: Comparison of the system Custom Tailstock Image: Comparison of the system Image: Comparison of the system Tailstock Length (TL) 100.000000 Image: Comparison of the system Grip Angle (GA) 50.000000 Base Diameter (GD)

Machine Setup Assembly Page

19.3.2. Creating Tailstock Operation

To create a Tailstock operation, navigate to **Solid Turn** (1) and then **Tailstock** (2) in the tool bar.



In the **Tailstock** tab under Work Setup, select **Main Spindle** for **Spindle Name** to have the steady rest engage with the workpiece in the main spindle. Select **B-Axis Head** for **Turret Name** to signify that the tailstock is controlled by the B-Axis head.

Under the Tailstock section you can select whether the operation is an **Engage** or **Disengage**. This tells the machine whether you would like to tailstock to engage with the workpiece or if you would like it to retract from the workpiece after machining has been completed.

	S Operation Name	
	SolidTurn · Tailstock	
-	S Work Setup	
1	Spindle Name	Main Spindle
	Turret Name	B-Axis Head
ľ	Head Id	Head-1
	Feeds and Speeds	
	Feedrate PM	250.000000
	Speed RPM	0.000000
	Use Feed and Speed KB	No
	S Tailstock	
	Position X, Y, Z	0.000000 0.000000 🕞
ſ	Tailstock	Engage 🗸 🗸
I	Dwell Time	Disengage
ľ	S Comment	Engage
	Features Tools F	
-	Treductes [10] roots E	B obergroups

After selecting which type of operation to perform, click OK and you will be prompted to select the **Tailstock reference engage point**. This is the point that the tailstock will engage the part at or the point to which it will return after disengaging.

NOTE: If the machine is equipped with a lower turret there should have a sync directly before and after any tailstock operations to ensure that no other machine movement is occurring while the tailstock is engaging /disengaging.

NT6600 machines equipped with an independent tail stock also use Tailstock Position (Custom setting 1) field for the V value in the G479 command for the independent tail stock.

19.4. Live Center on Sub Spindle

When machining long work pieces with the Mori Seiki NT machine it is sometimes beneficial to use the sub spindle as a live center (i.e. tailstock in the sub spindle) to support the workpiece while machining.

Creation and programing of a tailstock in the sub spindle is exactly the same as the <u>Independent</u> <u>Tail Stock</u> with the addition of setting the "Tail Stock Type" in Post Output Configuration page. See the <u>Machine Parameters</u> section for more information on this setting.

NOTE: The simulation will not show the sub spindle moving with the tail stock center.

20. Steady Rest

20.1. Lower Turret Steady Rest

On a machine equipped with a lower turret it is possible to use the lower turret as a steady rest for machining long parts.

20.1.1. Defining Steady Rest in Machine Setup

Open the Machine Setup page and navigate to the Assembly tab. Click on Steady Rest button to create a new Steady Rest. A new Steady Rest will be created on the left side of the page.



To view the Steady Rest Properties window, select the newly created steady rest. Under Type select Turret for the machine model. Under Turret Name select Bottom Turret. Under Station Name, select the station on the lower turret that the steady rest will be located on. It is also possible to define specific geometric properties of the steady rest in this section by clicking on the Geometry button.

B axis head	New Turret	New Spindle	New Steady Rest	Delete
Bottom Turret Main Spindle	×	New Axis	New Solid	New Station
Sub Spindle	Steady Rest Properties	1		
- Steady Rest	Name	Steady Rest	Clamping Time	0.000000
	Number	1	Rapid Rate	0.000000
	Туре	Turret 💌		
	Turret Name	Bottom Turret 💌		
	Station Name	Station:2 -		
	Orientation Angle	0.000000		
	Home Position X Y Z	0.000000 0.000000	0.000000	
	STL Ref. Point X Y Z	0.000000 0.000000	0.000000	Geometry

20.1.2. Defining Steady Rest in Machine Setup Utility (Mori Seiki Add-in)

On the Steady Rests tab of the Machine Setup Utility of the MoriSeiki AddIn, you will be able to create your steady rest.

💼 Machine Setup Utility					×
Tool Holders Steady Rests Machin	ne Parameters				
Steady rest 1	XML File Name General Selection	Steady Rest 1	Orientation Angle	0.000000	
	Position XYZ Axis Parameters Axis Label	-150.000000 Z4	0.000000	150.000000	
	Min Position	222.500000	Max Position	1661.500000	
	Solid Selection Nar Solid 2 Solid 3 STL	me _ File Name	Front Col C:\DOCUME~1\Julien[lor D\LOCALS~1\1 Browse	
	Jaw Geometry Roller Diameter (RD)	47.000000		RD RD	
* *	Center Distance (CD) Arm Length (AL) Arm Thickness	103.000000 139.000000 45.000000			
	Add		Update	Delete	
			ОК	Cancel	

Note: Please refer to your Machine Setup Utility User Manual to have detailed instructions on how to create the steady rest with this utility.

20.1.3. Creating Steady Rest Operation

First navigate to the Solid Turn (1) tool bar and then select Steady Rest (2)



In the Steady Rest tab, select the spindle you would like the steady rest to be located on by selecting either the Main or Sub Spindle in Spindle Name. Then select Bottom Turret for Turret Name.

C	DK 🗙 Cancel 🔂	•	😡 He
5	S Operation Name	8	
- Annon	SolidTum - Steady	Rest	
5	S Work Setup		
III ONO	Steady Rest	Steady Rest	•
3	Spindle Name	Main Spindle	•
	Turret Name	Bottom Turret	
	Head Id	Head-1	

Input the Reference Diameter based on the Diameter of your part at the steady rest location.

Speed RPM, CSS	0 0		
Feedrate PM, PR	0.000000	0.000000	
Reference Diameter	100.000000		
Speed Unit	RPM		
Feed Heit	Per Revolution		
reeu unit	FOR THE FORMULA		

When you click OK, a new window will pop up called "Lathe Steady Rest." This window will primarily be used to create an accurate simulation in ESPRIT. The important aspect of this with regard to the NC code is the order of operations. If you want the steady rest to open, move to a new location, and then close, you must program an Open, a linear move, and then a Close. If you emit any of these, they will not be output in the NC Code. The post processor automatically moves the Z axis first, followed by the X axis.

Lathe Steady Rest
C Rapid © Feed
Open 300
Close 200
Dwell
Stop Optional Sto 💌
🔘 Cycle Stop 🔊 Back
X, Y, Z 299.7% 706.1% 0
Apply

NOTE: The values and positions selected here will only be used for simulation in ESPRIT. You must MANUALLY enter the machine coordinate (G53) values for Z and X after posting out code. The post processor will output code "Z***" and "X***" for you to enter the correct values.

12000	-	1	
B axis hea	bd	Bottom Tum	et
Name	Spindle ID	Name	Spindle ID
Turning Insert			
Sync 🔍			
FaceTurn Top	Main Spin		
Sync		Svnc	2
		SolidTum - Steady	Main Spindle
Sync 🗧			
CODTurn Top Main	Main Spin		i i

20.2. Independent Steady Rest

On machines equipped with an independent steady rest rather than a lower turret it is possible to use an independent steady rest for machining long parts.

20.2.1. Defining Steady Rest in Machine Setup

Open the **Machine Setup** page and navigate to the **Assembly** tab. Click on **Steady Rest** button to create a new Steady Rest. A new Steady Rest will be created on the left side of the page.

Turn Machine Setup					
neral Assembly NC Output	.t Custom				
Machine Base Baxis head	New Tur	ret	New Spindle	New Steady Rest	Delete
Bottom Turret Main Spindle	X	•	New Axis	New Solid	New Station

To view the **Steady Rest Properties** window, select the newly created steady rest. Under **Type** select **Independent**. It is also possible to define specific geometric properties of the steady rest in this section by clicking on the Geometry button.

Machine Base B-Axis Head Main Spindle Sub Spindle Tailstock	New Turret	New Spindle New Axis	New Steady Rest New Solid	Delete New Station
Taistock Steady Rest	Steady Rest Properties Name Number	Steady Rest	Clamping Time Rapid Rate	0.000000
	Type Turret Name Station Name Orientation Angle	Independent		
	Home Position XYZ STL Ref. Point XYZ	0.000000 0.000000	696.5000	Geometry

20.2.2. Defining Steady Rest in Machine Setup Utility (Mori Seiki Add-in)

On the Steady Rests tab of the Machine Setup Utility of the MoriSeiki AddIn, you will be able to create your independent steady rest.

	•						
SLU_1_3000	XML File Name	C:\Documents ar	nd Settings\My Documents\DP T	echnolog Browse			
	General Selection						
	Name	SLU_1_3000	Orientation Angle	0.000000			
	Position XYZ	0.000000	0.000000 179.0	00000			
	Axis Parameters	Axis Parameters					
	Axis Label	Z4]				
	Min Position	39.000000	Max Position	629.000000			
	Solid Selection						
	Solid 1 Na	ime	Front Color				
	Solid 3 ST	L File Name	C:\Documents and Settings\M	ichael V Browse			
	Jaw Geometry						
	Roller Diameter (RD)	19.000000		RD			
	Center Distance (CD)	37.000000		*[CD]+			
	Arm Length (AL)	50.500000		No.			
	Arm Thickness	20.000000		<u>}</u>			
	C		Undets	Delete			
	Add		Update	Delete			

Note: Please refer to your Machine Setup Utility User Manual to have detailed instructions on how to create the steady rest with this utility.

20.2.3. Creating Steady Rest Operation

First navigate to the Solid Turn (1) tool bar and then select Steady Rest (2)



In the Steady Rest tab, select the spindle you would like the steady rest to be located on by selecting either the **Main** or **Sub Spindle** in **Spindle Name**. Then select **B-Axis Head** for **Turret Name**.

Then input the Reference Diameter based on the diameter of your part.

SolidTum - Steady Rest Close Steady Rest Steady Rest Spindle Name Main Spindle Tunet Name BrAxis Head Head Id Head-1 Feeds and Speeds Speed RPM, CSS 0 Feedrate PM, PR 0.000000 Reference Diameter 100.000000 Speed Unit Per Revolution Maximum RPM 4000	10	K 🟋 Cancel 🛛 🕍 👻		ī	🥑 Hel	
SolidTurn - Steady Rest Close Steady Rest Steady Rest Spindle Name Main Spindle Turret Name B-Axis Head Head Id Head Id Head-1 Speeds Speed RPM, CSS 0 Feedrate PM, PR 0.000000 Reference Diameter 100.000000 Speed Unit RPM Feed Unit Per Revolution	ŝ	S Operation Name				
Steady Rest Steady Rest Spindle Name Main Spindle Turret Name Braxis Head Head Id Head-1 Speed RPM, CSS 0 Feedrate PM, PR 0.000000 Reference Diameter 100.000000 Speed Unit Per Revolution Maximum RPM 4000	· Anno	SolidTurn - Steady Re	est Close			
Steady Rest Steady Rest Spindle Name Spindle Name Main Spindle Image: Comparison of the spindle Torret Name B-Axis Head Image: Comparison of the spindle Head Id Head-1 Image: Comparison of the spindle Speed RPM, CSS 0 0 Feedrate PM, PR 0.000000 0.000000 Reference Diameter 100.000000 Speed Unit RPM Image: Comparison of the spinole Maximum RPM 4000	5	S Work Setup			_	
Spindle Name Main Spindle Tunet Name B-Axis Head Head Id Head-1 Speed RPM, CSS 0 Feedrate PM, PR 0.000000 Reference Diameter 100.000000 Speed Unit RPM Feed Unit Per Revolution ✓ Maximum RPM		Steady Rest	Steady Rest		~	
Tone B-Axis Head Head Id Head-1 Speed RPM, CSS 0 0 Feedrate PM, PR 0.000000 0.000000 Reference Diameter 100.000000 Speed Unit RPM Feed Unit Per Revolution ✓ Source Comment Second Comment Second Comment	5	Spindle Name	Main Spindle		~	
Head Id Head-1 Speed RPM, CSS 0 Feedrate PM, PR 0.000000 Reference Diameter 100.000000 Speed Unit RPM Feed Unit Per Revolution Maximum RPM 4000		Turret Name	B-Axis Head		~	
Speeds and Speeds Speed RPM, CSS 0 Feedrate PM, PR 0.000000 Reference Diameter 100.000000 Speed Unit RPM Feed Unit Per Revolution ✓ Feed Unit Ø Comment		Head Id	Head-1		~	
Speed RPM, CSS 0 0 Feedrate PM, PR 0.000000 0.000000 Reference Diameter 100.000000 • Speed Unit RPM • Feed Unit Per Revolution • Maximum RPM 4000 •		S Feeds and Speeds				
Feedrate PM, PR 0.000000 0.000000 Reference Diameter 100.000000 Speed Unit RPM ✓ Feed Unit Per Revolution ✓ Maximum RPM 4000 ✓		Speed RPM, CSS	0	0		
Reference Diameter 100.000000 Speed Unit RPM Feed Unit Per Revolution Maximum RPM 4000		Feedrate PM, PR	0.000000	0.000000		
Speed Unit RPM Feed Unit Per Revolution Maximum RPM 4000 Somment		Reference Diameter	100.000000			
Feed Unit Per Revolution Maximum RPM 4000 Source Comment	ľ	Speed Unit	RPM		~	
Maximum RPM 4000		Feed Unit	Per Revolution		~	
Comment		Maximum RPM	4000			
		S Comment				
NOK NOK						

When you click OK, a new window will pop up called "Lathe Steady Rest." This window will primarily be used to create an accurate simulation in ESPRIT. The important aspect of this with regard to the NC code is the order of operations. If you want the steady rest to open, move to a new location, and then close, you must program an Open, a linear move, and then a Close. If you emit any of these, they will not be output in the NC Code. The post processor automatically moves the Z axis first, followed by the X axis.

Lathe Steady Rest				
C Rapid	Feed			
Open	300			
Close	200			
Dwell				
Stop	Optional Sto 💌			
🔘 Cycle S	itop හ Back			
X, Y, Z 299.	72 706.12 0			
4	Apply			

NOTE: For the NT-S series machines, it is necessary to manually insert the V value in the G479 Automatic Steady Rest Centering command that is output by ESPRIT. To manually enter the value in ESPRIT, go to the Custom tab of the Steady Rest operation and enter the value in G479 V Value field (Custom Setting 1):

Custom Setting G479 V_input:

-0	K 🗙 Cancel 📑	•	🤪 Help
Rest	Settings		
(ippe)	G479 V Value	25.000000]
Ste	CustomSetting2	0.000000	
Custom	CustomSetting3	0.000000	
	CustomSetting4	0.000000	
_	CustomSetting5	0.000000	

20.2.4. G478 Steady Rest/Sub Spindle Synchronize Command (NT-S 6000 only)

The NT-S 6600 machine has the ability to synchronize the steady rest movement with the sub spindle. This can be done by entering the value of 478 into the G478 Synchronize field (Custom Setting 8) in the steady rest operation.

Note: Simulation in ESPRIT will not show the steady rest moving when the sub spindle moves

🥪 O	K 🗙 Cancel 🔆 🗸	🔞 Help
lest	Settings	
ady F	G479 V Value	15.000000
Ste	Custom Setting 2	0.000000
E	Custom Setting 3	0.000000
Custo	Custom Setting 4	0.000000
	Custom Setting 5	0.000000
	Custom Setting 6	0.000000
	Custom Setting 7	0.000000
	G478 Synchronize	478.000000
	Custom Setting 9	0.000000
	Custom Setting 10	0.000000
	Features Tools	Operations SolidTurn - Steady Rest

.

21. Special considerations when Programming an NT1000

21.1. Introduction

In ESPRIT 2012 as well as the post have improvements to continue to ease the level of difficulty in programming operations on the work support/sub spindle of the NT1000 series machines. These improvements include:

- New Park operation that lets us have realistic control of the turrets as well as the A and BW axis.
- Simplifying the tool change areas in the post, reducing the number of syncs needed within the Esprit file on the SZ and WZ machines.

21.2. Mori Seki Macro Executer update required to work with ESPRIT 2012 and newer post

Mori Seki has changed how the G43 and G68 commands function on the sub spindle of the NT1000 machines late in 2011.

In order to use the posts for ESPRIT 2012 and newer your machine may need to be updates as well as parameter changes.

Please contact your Mori Seki distribution about this update if you are unsure if your machine has the required update.

Below is a copy of the Mori Seki document as to what is needed:

	設計通知No	報書		20.90	200	7.48
設計通知	MD165125	10H	00078	平認	×22	部開発部
多品一葉	役務申請No				2012/0 山中	05/15 2012/05/15 牽島/江洋司
题目/TOPIC	NTX1000/2000:初	標準マクロエ	ジゼキュータ改 職	i		
	STANDARD MAC	RO EXECU	TOR MODIFIE	D:NTX1000/	2000	
部署 電装N NTX	IT課 G計チーム	担当者	桐山 景一		E更前設計通知No	
探)	115月 11時期 2009年07月機	より	配付先	12		
			千葉/組立/			
幾種	機番、採用時期	al al	千葉/千品/			
NTX1000 NTX2000			十葉/エル/ その他//千葉CA	м課 上原MA		
1172000			その他//北海道	MAPPS TM		
			example diversion			
			初品チェッ	ック	依頼伝票	No
			CAD BURGERS COLUMN TO	146670		
					什事分類	
			□素材		上 仕事分類 設計変	更
中南/CONTENTS			□素材 □組立	□加1 □部5	L 仕事分類 設計家	更
内容/CONTENTS	つかた博進つハロエバギた	과 또 파니 ㅎ	□素材 □組立	□加コ □部品	L 仕事分類 設計多	E.
内容/CONTENTS NTX1000/2000 31ii 変更内容: NTX10	3対応標準マクロエグゼを DOIにおけるミーリング工具	改版致しま &3次元座	□素材 □組立 す。(198755A17 標変換中&サ	□加コ □部点 →18) ブ主輪側加コ	L 仕事分類 設計到 L &D2369.0=	を更 = 1のときの工具補正指
内容/CONTENTS NTX1000/2000 31ii 変更内容: NTX10 命を修正します。20	3対応標準マクロエグゼを 00におけるミーリング工具 12年5月25日出荷機。	改版致しま &3次元座 り対応です	□素材 □組立 す。(198755A17 標変換中&サ 。	□加コ □部品 →18) ブ主輪側加コ	L 世事分類 設計多 L & D2369.0=	5更 = 1のときの工具補正指
内容/CONTENTS ITX1000/2000 31ii 変更内容: NTX10 合を修正します。20 Standard macro exi	3対応標準マクロエグゼを DOIにおけるミーリング工具 12年5月25日出荷機。	改版致しま &3次元座 り対応です 00 31iB	□素材 □組立 す。(198755A17 標変換中&サ 。	□加コ □部日 →18) ブ主輪側加コ	L 仕事分類 設計到 L & D2369.0=	5更 =1のときの工具補正指
内容/CONTENTS ITX1000/2000 31 変更内容:NTX10 合を修正します。20 Standard macro exi Contents: Modificat 2369.0=1 for NTX	3対応標準マクロエグゼを DOにおけるミーリング工具 12年5月25日出荷機。 Docutor for NTX1000/200 ion of tool offset comm 1000.	改版致しま &3次元座 り対応です 00 31iB and for mil	□素材 □組立 す。(198755A17 標変換中&サ 。	□加コ □部占 →18) ブ主軸側加コ	L 世事分類 設計多 L & D2369.0=	E更 = 1のときの工具補正指 & sub-spindle side &
内容/CONTENTS ITX1000/2000 31ii 変更内容: NTX100 合を修正します。20 Standard macro ex- Contents: Modificat 12369.0=1 for NTX Naplied: From 25 M	3対応標準マクロエグゼを DOにおけるミーリング工具 12年5月25日出荷機。 ecutor for NTX1000/200 ion of tool offset comm 1000. ay. 2012 for NTX1000	改版致しま &3次元座 り対応です 00 31iB and for mil	□素材 □組立 す。(198755A17 標変換中&サ	□加コ □部品 →18) ブ主軸側加コ	L 世事分類 設計到 I & D2369. 0=	E更 = 1のときの工具補正指 & sub-spindle side &
内容/CONTENTS ITX1000/2000 31ii 変更内容: NTX10 合を修正します。20 Standard macro ex Contents: Modificat 2369.0=1 for NTX pplied: From 25 M 原因/CAUSE	3対応標準マクロエグゼを DOIにおけるミーリング工具 12年5月25日出荷機。 acutor for NTX1000/200 ion of tool offset comm 1000. ay. 2012 for NTX1000	改版致しま &3次元座 り対応です 00 31iB and for mil	□素材 □組立 す。(198755A17 標変換中&サ 。	□加コ □部店 →18) ブ主軸側加:	L L L L L L L L L L L L L L L L L L L	E更 = 1のときの工具補正指 & sub-spindle side &
内容/CONTENTS NTX1000/2000 31 変更内容: NTX100 令を修正します。20 Standard macro exe Contents: Modificat 2369.0=1 for NTX 2369.0=1 for NTX 29plied: From 25 M 原因/CAUSE NTX1000におけるミー す。	B対応標準マクロエグゼを DOにおけるミーリング工具 12年5月25日出荷機。 ecutor for NTX1000/200 ion of tool offset comm 1000. ay. 2012 for NTX1000	改版致しま &3次元座 り対応です 00 31iB and for mil	□素材 □組立 す。(198755A17 標変換中&サ 。 ling cutting pro	□加コ □部占 →18) ブ主軸側加コ ocess on 3-c	L 世事分類 設計す L & D2369.0= dimension mode a 0=1のときのエ	E更 = 1のときの工具補正指 & sub-spindle side & 具補正指令を修正しま
内容/CONTENTS (TX1000/2000 31) 変更内容: NTX10 合を修正します。20 Standard macro ex Contents: Modificat)2369.0=1 for NTX)2369.0=1 for NTX)2369.0=1 for NTX)pplied: From 25 M 原因/CAUSE (TX1000におけるミー た。 dodification of tool or NTX1000	B対応標準マクロエグゼを DOIにおけるミーリング工具 12年5月25日出荷機。 acoutor for NTX1000/200 ion of tool offset comm 1000. ay. 2012 for NTX1000 -リング工具 & 3次元座標 offset command for mil	改版致しま &3次元座 り対応です 00 31iB and for mil 変換中&1	□素材 □組立 す。(198755A17 標変換中&サ っ ling cutting pro サブ主軸側加工 g process on 3	□加コ □部品 →18) ブ主軸側加: ocess on 3-c : &D2369. -dimension r	L & D2369.0 I & D2369.0 dimension mode a 0=1のときのエ mode & sub-spin	E更 = 1のときの工具補正指 & sub-spindle side & 具補正指令を修正しま dle side & D2369.0=1.
内容/CONTENTS NTX1000/2000 31 変更内容: NTX100 令を修正します。20 Standard macro exc Contents: Modificat 2369.0=1 for NTX 2369.0=1 for NTX 2369.0=1 for NTX 2369.0=1 for NTX 2369.0=1 for NTX 2369.0=1 for NTX 2000におけるミー のはfication of tool or NTX1000. 作業指示/PROCES	B対応標準マクロエグゼを DOにおけるミーリング工具 12年5月25日出荷機。 ecutor for NTX1000/200 ion of tool offset comm 1000. ay. 2012 for NTX1000 -リング工具&3次元座標 offset command for mil S DIRECTIVE	改版致しま &3次元座 り対応です 00 31iB and for mil 変換中&4	□素材 □組立 す。(198755A17 標変換中&サ 。 ling cutting pro サブ主軸側加工 g process on 3	□加コ □部占 →18) ブ主軸側加コ ocess on 3-c 3-c 3-c 3-c 3-c 3-c 3-c 3-c 3-c 3-c	L AD2369.0= dimension mode 0=1のときのエ mode & sub-spin	E更 = 1のときの工具補正指 & sub-spindle side & 具補正指令を修正しま dle side & D2369.0=1.
内容/CONTENTS NTX1000/2000 31ii 変更内容: NTX10 合を修正します。20 Standard macro ex Contents: Modificat 02369.0=1 for NTX Applied: From 25 M 原因/CAUSE NTX1000におけるミー 。 Modification of tool or NTX1000. 作業指示/PROCES	B対応標準マクロエグゼを DOIにおけるミーリング工具 12年5月25日出荷機。 acoutor for NTX1000/200 ion of tool offset comm 1000. ay. 2012 for NTX1000 -リング工具 & 3次元座標 offset command for mil S DIRECTIVE	改版致しま &3次元座 り対応です 00 31iB and for mil 変換中&*	□素材 □組立 す。(198755A17 標変換中&サ 。 ling cutting pro サブ主軸側加工 g process on 3	□加コ □部占 →18) ブ主軸側加 つcess on 3-c 2 & D2369. -dimension r	L & D2369.0 I & D2369.0 dimension mode a 0=1のときのエ mode & sub-spin	E更 = 1のときの工具補正指 & sub-spindle side & 具補正指令を修正しま dle side & D2369.0=1.
内容/CONTENTS NTX1000/2000 31ii 変更内容: NTX10 令を修正します。20 Standard macro ex: Contents: Modificat 02369.0=1 for NTX 02369.0=1 for NTX 02369.0=1 for NTX 000におけるミー の TX1000におけるミー の の NTX1000. 作業指示/PROCES	3対応標準マクロエグゼを DOIにおけるミーリング工具 12年5月25日出荷機。 acutor for NTX1000/200 ion of tool offset comm 1000. ay. 2012 for NTX1000 -リング工具&3次元座標 offset command for mil S DIRECTIVE	改版致しま &3次元座 り対応です 00 31iB and for mil 変換中&4	□素材 □組立 す。(198755A17 標変換中&サ 。 ling cutting pro サブ主軸側加工 g process on 3	□加コ □部占 →18) ブ主軸側加コ ocess on 3-c 3-c 3-c 3-c 3-c 3-c 3-c 3-c 3-c 3-c	L L L & D2369.0= dimension mode & 0=1のときのエ mode & sub~spin	E更 = 1のときの工具補正指 & sub-spindle side & 具補正指令を修正しま dle side & D2369.0=1.
内容/CONTENTS NTX1000/2000 31ii 変更内容: NTX10 令を修正します。20 Standard macro exi Contents: Modificat 02369.0=1 for NTX Applied: From 25 M 原因/CAUSE NTX1000におけるミー 。 Modification of tool or NTX1000. 作業指示/PROCES	B対応標準マクロエグゼを DOIにおけるミーリング工具 12年5月25日出荷機。 Decutor for NTX1000/200 ion of tool offset comm 1000. ay. 2012 for NTX1000 -リング工具 & 3次元座欄 offset command for mil S DIRECTIVE	改版致しま &3次元座 り対応です 00 31iB and for mil	□素材 □組立 す。(198755A17 標変換中&サ 。 ling cutting pro サブ主軸側加工 g process on 3	□加コ □部占 →18) ブ主軸側加 ocess on 3-c : & D2369. -dimension r	L 仕事分類 設計す L & D2369.0= dimension mode a 0=1のときのエ mode & sub~spin	E更 = 1のときの工具補正指 & sub-spindle side & 具補正指令を修正しま dle side & D2369.0=1.
内容/CONTENTS NTX1000/2000 31ii 変更内容: NTX10 令を修正します。20 Standard macro ex. Contents: Modificat Contents: Modification NTX Applied: From 25 M Modification of tool or NTX1000におけるミー の Modification of tool or NTX1000. 作業指示/PROCES	B対応標準マクロエグゼを DOIにおけるミーリング工具 12年5月25日出荷機。 ecutor for NTX1000/200 ion of tool offset comm 1000. ay. 2012 for NTX1000 -リング工具&3次元座標 offset command for mil S DIRECTIVE	改版致しま &3次元連 り対応です 00 31iB and for mil 変換中&4 lling cutting	□素材 □組立 す。(198755A17 標変換中&サ 。 ling cutting pro サブ主軸側加工 g process on 3	□加コ □部占 →18) ブ主軸側加: ∞cess on 3-c 3.&D2369. -dimension r	L L & D2369.0= dimension mode & 0=1のときのエ mode & sub~spin	E更 = 1のときの工具補正指 & sub-spindle side & 具補正指令を修正しま dle side & D2369.0=1.
内容/CONTENTS NTX1000/2000 31ii 変更内容: NTX100 令を修正します。20 Standard macro exi Contents: Modificat D2369.0=1 for NTX Applied: From 25 M 原因/CAUSE NTX1000におけるミー か。 Modification of tool for NTX1000. 作業指示/PROCES	B対応標準マクロエグゼを DOIにおけるミーリング工具 12年5月25日出荷機。 accutor for NTX1000/200 ion of tool offset comm 1000. ay. 2012 for NTX1000 -リング工具&3次元座構 offset command for mil S DIRECTIVE	改版致しま &3次元座 り対応です 00 31iB and for mil (変換中&1 lling cutting	□素材 □組立 す。(198755A17 標変換中&サ 。 ling cutting pro サブ主軸側加工 g process on 3	□加コ □部占 →18) ブ主韓側加 ocess on 3-c :&D2369. -dimension r	L 世事分類 設計す L & D2369.0= fimension mode a 0=1のときのエ mode & sub~spin	E更 = 1のときの工具補正指 & sub-spindle side & 具補正指令を修正しま dle side & D2369.0=1.
内容/CONTENTS NTX1000/2000 31ii 変更内容: NTX10 令を修正します。20 Standard macro ex: Contents: Modificat 02369.0=1 for NTX Applied: From 25 M 原因/CAUSE NTX1000におけるミー す。 Modification of tool or NTX1000. 作業指示/PROCES	B対応標準マクロエグゼを DOIにおけるミーリング工具 12年5月25日出荷機。 Docutor for NTX1000/200 ion of tool offset comm 1000. ay. 2012 for NTX1000 -リング工具&3次元座棚 offset command for mil S DIRECTIVE	改版致しま &3次元座 り対応です 00 31iB and for mil 変換中&4 lling cutting	□素材 □組立 す。(198755A17 標変換中&サ 。 ling cutting pro サブ主軸側加工 g process on 3	□加コ □部店 718) 7主軸側加: 0cess on 3-0 3. 4. D2369. -dimension r	L AD2369.0= dimension mode a 0=1のときのエ mode & sub~spin	E更 = 1のときの工具補正指 & sub-spindle side & 具補正指令を修正しま dle side & D2369.0=1.
内容/CONTENTS NTX1000/2000 31ii 変更内容: NTX100 令を修正します。20 Standard macro exi Contents: Modificat 2369.0=1 for NTX Applied: From 25 M 原因/CAUSE NTX1000におけるミー た。 Modification of tool or NTX1000. 作業指示/PROCES	B対応標準マクロエグゼを DOIにおけるミーリング工具 12年5月25日出荷機。 accutor for NTX1000/200 ion of tool offset comm 1000. ay. 2012 for NTX1000 -リング工具&3次元座標 offset command for mil S DIRECTIVE	改版致しま &3次元座 り対応です 00 31iB and for mil (変換中&1 lling cuttin)	□素材 □組立 す。(198755A17 標変換中&サ 。 ling cutting pro サブ主軸側加工 g process on 3	□加コ □部占 →18) ブ主軸側加 ocess on 3-c 3-c 3-c 3-c 3-c 3-c 3-c 3-c 3-c 3-c	L L & D2369.0= fimension mode a 0=1のときのエ mode & sub-spin	E更 = 1のときの工具補正指 & sub-spindle side & 具補正指令を修正しま dle side & D2369.0=1.

21.3. Setting the Sub Spindle work offset value in the machine control

Since the sub spindle (A axis) can be moved to different locations when machining on the NT1000 the post is setup so that program zero for the sub spindle MUST be set while the A axis is at the G330 position (A0). The G474 command is used to shift the Z values by the amount the A axis is moved from A0 when the part is machined.

If this is not done the NC code will not cut the part as desired on the sub spindle.

21.4. Motion Axis setup on ESPRIT for sub spindle operations on upper head

On any upper head MillTurn operation used on the sub spindle ESPRIT has 3 rotary axis to choose from to create the tool path for the operation, B, BW and C axis. One of these axis has to be fixed and the other two can be used for interpolation.

ESPRIT has 3 choices for the Motion Axis in the Rotary axis definition in the operations properties.

Key	Value	
Work Coordinate	G55_Baxis	*
Rotary Angles		
- Motion Axes	B & C	
B	65.000000	
ToolSpindleAxis	0.000000	=
BW	-25.000000	-
C	-0.000000	
E Collinear Axes		
Total Cycle Time:	00:00:33	
<	III	•

The options are as follows:

- B&BW (C is fixed)
- B&C (BW is fixed)
- BW&C (B is fixed).

It is best to use the B&C option as the machine can only use the BW axis for positioning.

In 5 axis operations this MUST be set to B&C.

Since the B and BW axis are parallel axis leaving an operation set to B&BW can prevent ESPRIT's ability position the rotary axis to machine a feature. In addition this can cause irregular movements of the BW axis in simulation if BW&B are used.

When the Motion axis setup is changed you will also need to

21.5. Setting the Work Support/Sub Spindle angle (BW Axis)

As with operations performed on the main spindle the B axis will tilt the B axis to the angle determined by the work plane of the feature being machined. The default angle of the BW axis will be BW0

ESPRIT allows the user to change the angles of either the B or BW axis angles of any operation programed on the sub spindle in the property browser to allow for machining at any combination of B and BW axis angles that the machine can machine the part at. If the B axis angle is changed the BW will be changed to the complementary angle. If the BW axis angle is changed to the complementary B axis angle

With the desired operation highlighted in the Project Manager, the Properties Window will look like this:

		2
Key	Value	
Group Count	1	
🖻 General		
- Element Type	Tool Path	
- Element Number	72	
- Layer	Default	
Color		
- Line Type		_
- Line Weight		-
Technology		
Tool Path		
- Work Coordinate	G55_Baxis	
Rotary Angles		
- Motion Axes	8 & BW	
B	60.000000	
BW	-30.000000	_
C	0.000000	
Collinear Axes		
- Total Cycle Time:	00:00:17	
- Rapid Length	263.812909	
- Feed Length	67.812909	
Name	Peck Drilling Sub BW-30	

Once the operation grouped, all the user needs to do is expand the Rotary Angle section and change the B or BW angle (ESPRIT will automatically calculate the complementary angle) under Rotary Angles.

In addition to setting the BW and B axis angles, the user may also change the C axis angle if the operation is on the face of the part as long as the operation is not a "Wrap" or "Rotary Face" type

operation. The C axis angles for operations that are not on the face of the part will be determined by the work plane of the feature being machined.

ey	Value	
- Technology		
Tool Path		
- Work Coordinate	G55_Baxis	
Rotary Angles		
Motion Axes	B & BW	
B	65.000000	
ToolSpindleAxis	0.000000	
BW	-25.000000	
C	10.000000	
Collinear Axes		
Total Cycle Time:	00:55:56	
- Rapid Length	0.000000	
- Feed Length	277.761511	
- Name	Pocket Mill Face Sub BW-25	

Note that if you change the C axis angle the B and BW axis angle may change to a different value than previously set. If this happens they will need to be set back.

21.6. Conditions need to allow the Moving of the BW axis

	NT1000/WZ, NT1000/WZM	NT1000/SZ, NT1000/SZM	NT1000/W	NT1000/S
A-axis	−95.0 mm/−3.740 in	-128.0 mm/-5.039 in	-130.0 mm/-5.118 in	-245.0 mm/-9.646
	(G330 P4)	(G330 P4)	(G330 P4)	(G330 P4)
X2-axis	0mm ~ 1.0mm	0mm ~ 1.0mm	-	-
	0in ~ −0.039in	0in ~ −0.039in		
	(G28 U0)	(G28 U0)		
Z2-axis	−305.0mm ~ −305.5mm	−305.0mm ~ −305.5mm	-	-
	12.008 in ~ -12.008 in	12.008 in ~ -12.008 in		
	(G30 P4 W0)	(G30 P4 W0)		
B-axis	0° ~ 120°	0°~120°	0°~120°	0°~120°
X1-axis	0 mm ~ −1.0 mm	0 mm ~ −1.0 mm	0 mm ~ −1.0 mm	0 mm ~ −1.0 mm
	0 in ~ −0.039 in	0 in \sim –0.039 in	0 in ~ −0.039 in	0 in \sim –0.039 in
Z1-axis	0 mm ~ − <mark>230.5</mark> mm	0 mm ~ −230.5 mm	0 mm ~ -230.5 mm	0 mm \sim -230.5 mm
	0 in ~ −9.075 in	0 in \sim –9.075 in	0 in ~ −9.075 in	0 in \sim –9.075 in

Prior to moving the BW axis the machines axis must be in the following locations:

21.7. Conditions need to allow the Moving of the B axis

Prior to moving the B axis the machines axis must be in the following locations:

	NT1000/WZ, NT1000/WZM	NT1000/SZ, NT1000/SZM	NT1000/W	NT1000/S
A -axis	0 mm [~] −95.0 mm 0 in [~] −3.740 in	0 mm [~] −128.0 mm 0 in [~] −5.039 in	0 mm [~] −130.0 mm 0 in [~] −5.118 in	0 mm [~] −245.0 mm 0 in [~] −9.646 in
	(G330 P4 is Ok)	(G330 P4 is Ok)	(G330 P4 is Ok)	(G330 P4 is Ok)
X1-axis	G28 U0(0 mm)	G28 U0(0 mm)	G28 U0(0 mm)	G28 U0(0 mm)
Z1-axis	G28 W0(0 mm)	G28 W0(0 mm)	G28 W0(0 mm)	G28 W0(0 mm)

21.8. Setting A-axis Position with the AutoPark Addin

The A axis position for machining can be set with one of two methods, with the AutoPark Addin or with the A-axis position (custom setting 9) field.

The AutoPark addin was specifically designed to work with the NT1000 machines. Using this addin saves you from doing the math required to set the Co-linear value for the A axis position as well as ensuring that the simulation matches the NC code.

Once an operation has been setup with the AutoPark addin the values in the A-axis position (Custom setting 9) field will be ignored by the post.

Setting the A axis position:

- Create operation (Do not set any values in the A axis position field\(Custom Setting 9)
- 2. Press OK to create operation
- 3. Group operation
- 4. Look in the properties dialog box and find the "Mori Seki AutoPark" heading
- 5. Press "+" to expend the fields under the heading
- 6. Set the desired A axis position as follows:
 - a. If you want to position the A axis at either G330 or G330 P4 select one of these positions in the "Park Spindle Code field".
 - b. If you want to position the A axis to another position set the desired position (relative to the machine coordinate system) in the "Relative A" field.
- 7. The AutoPark addin will set the A axis collinear value to the same relative position ensuring that simulation will match the NC code.

Notes:

- If you select G330 the Relative A field will be set to 0 as this is A0
- If you select G330 P4 the A axis position will be set to the appropriate position for your machine.

If you change the Relative A axis field to a position other than your machines G330 or G330 P4 position after you park the A axis to either G330 or G30P4 the park spindle code will be set to None.

- If you have run the AutoPark addin on a given sub spindle operation, the A-axis position field (Custom setting 9) from legacy files will be ignored.
- Note the A axis position would always be a value between 0 and the machines negative travel limit.

Example of AutoPark with A axis set to G330 P4 on an NT1000-SZ:

Key	Value	
Technology		
⊕ Tool Path		
Work Coordinate	G55_Baxis	
🗄 Rotary Angles		
🕀 Collinear Axes		
Total Cycle Time:	00:00:16	
Rapid Length	99.000000	
Feed Length	828.266868	
Name	OD Contour Mill BW-25	
🖻 Mori Seiki AutoPark		
Park Spindle Code	G330 P4 (A-128)	
Relative A	-128.000000	

Example of AutoPark with A axis set to -150mm on an NT1000-SZ:

Key	Value	
• Geometry		
⊕ Tool Path		
Work Coordinate	G55	
• Rotary Angles		
🕀 Collinear Axes		
Total Cycle Time:	00:01:25	
Rapid Length	0.000000	
Feed Length	30.00000	=
Name	Face BW-25	
🖻 Mori Seiki AutoPark		
Park Spindle Code	None	
Relative A	-150.000000	

NOTES:

When machining with the B axis head on the sub spindle simulation will show the A-axis moving to the specified location in Collinear Axes Z1 BEFFORE the BW angle index move. The NC code will the machine move the A-axis return to G330 P4 in order to index the BW axis, then if the A axis specifed in the A axis collinear position is not equal to G330 P4 the A axis will be moved to the specifed location.

When beginning to machine with the lower turret with the sub spindle the simulatition will show the A axis positon after the tool change takes place. The NC code will move the A axis prior to the tool change.

For reference the Collinear positions for the G330 or G330 P4 locations are shown on the chart below:

	G330 (Metric)		G330 P4 (Metric)		G330 (Inch)		G330 P4 (Inch)	
NT1000 type	Machine Position	A axis Collinear Position						
NT-1000-W	0	995.0	-130	865.0	0	39.17	-5.11811	34.06
NT-1000-S	0	970.0	-245	725.0	0	38.19	-9.64567	28.54
NT-1000-SZ	0	888.5	-128	760.5	0	34.98	-5.03937	29.94
NT-1000-WZ	0	863.0	-95	768.0	0	33.98	-3.74016	30.24

21.9. Parking the Lower Turret

It is necessary to program park operations for the lower turret at the G28 U0; G30 P4 W0 whenever the BW is rotated or when a part transfer is made.

The lower turret park operation can be setup as shown below or the AutoPark addin can set this up automatically. See for <u>Setting up the park operation with the AutoPark addin</u> section to learn how to use the AutoPark addin.

To park the Lower Turret manually to the G28 U0; G30 P4 W0 location setup the park operation as follows:

General Tab:

~ (OK 🗙 Cancel 📲	•	🕜 Help	
neral	S Operation Nam	e		
g	Park G30 P4			
Park	S Tool Selection			
E	Select Tool	No	•	
Custo	S Work Setup			
	Spindle Name	MainSpindle	•	
	Turret Name	BottomTurret	•	
	Head Id	Head-1	•	
	Comment			

Notes:

- 1. The tool selection can be setup however you need it to be set to or the select tool option can be set to no.
- 2. The Spindle must be set to Main Spindle
Park Tab should be set as follows:

~	🛩 OK 🛪 Cancel 🛍 🗝 💿 Help							
neral		Park Position						
Ger		Park Position X	Home	•	0.000000	R		
¥		Park Position Y	Home	•	0.000000	R		
Pa		Park Position Z	Machine	•	-305.000000	R		
E Return Mode X Then Z						-		
Cus		Stop Code	op Code None					
		Rotary Axes						
		Park Axis 1	No			•		
		Park Axis 2	No			•		

Notes:

- 1. Setting the X axis to Home will have the post output G28 U0 in the NC code.
- 2. The G30 P4 W0 position is Z-305MM (-12.008 Inches) from machine zero.

21.10. Parking the A and BW axis Prior to Tool change

The machine requires that the A axis is between G330 (A0) and G330 P4 before making a tool change with the B axis head (refer to <u>Conditions need to allow the Moving of the B axis</u> section for the specific range of travel for your machine).

It is also recommended that the BW axis be rotated to BW0 before a tool change is made on the upper head to avoid possible collision but it is not always necessary.

Below are two examples of tool changes with the B axis head with BW and A at 0 and not at zero.



Tool change with BW<>0 and A@G330P4 Tool Change with BW@0 and A@G330 (0)

In both of these examples the tool change can be made but there is more clearance when the BW and A axis are at zero.

If needed, a park operation can be used to move the A axis BW axis to a safe location prior to a tool change.

The A axis position is set in the park operation with the AutoPark addin (refer to section <u>Setting</u> <u>A-axis Position with the AutoPark Addin</u>)

Parking the A and BW axis is always done with the upper turret's (B axis head) program.

Below is an example of B axis head tool changes\operations on the sub spindle with and without park operations between them:

				Park G30 P4	MainSpindle
		Sync 🗧		Sync 🗧	
		TI - 80DEG - U			
		Sync		Sync 😑	
No park		Face BW-25	SubSpindle	TI - 55DEG - OD - L	
operations		EM - 10MM - U		Face Turn Rough	MainSpindle
		PB OD Contour Mill BW-25	SubSpindle	OD Tum Rough	MainSpindle
between tool		Pocket Mill Face BW-25	SubSpindle	Park G30 P4	MainSpindle
changes and	-	Sync		Sync	
		EM - 10MM - U Copy			
operations		Sync		Sync	
		Pocket Mill Face BW-30	SubSpindle	EM - 10MM - F - L	
				Notary Contour Face	MainSpindle
	< l>			Park G30 P4	MainSpindle
Deule en enstiene	$) \subset$	Sync		Sync	
Park operations		EM - TOMM - U	0.1.0.1.1		
used between		Pocket Mill OD BVV0	SubSpindle		
oparations with		Park G330 P4 and BW0	SubSpindle		
operations with		Party C220 PA and PMO	SubSpindle		
BW axis angle		The OD Castave Mill DW 15, D15	SubSpindle		
changes and		Pade C220 PA and PW0	Sub Spindle		
changes and			Sub-Spindle		
tool changes.		THO Crocke BW/0	Sub Spindle		
-			Subspinule	- Suno	
	1	Jync		- Sync	
		- Release	SubSpindle		
		/B Release	SubSpindle	Svnc	

21.11. Parking the B axis head Prior to BW Angle Change

The NT1000 requires that the B axis between 0 and 120 degrees before the BW axis can be rotated. (refer to <u>Conditions need to allow the Moving of the BW axis</u> section).

The post will always move the machine as follows when the BW angle changes between operations:

- X axis is moved home
- Y and Z axis are moved home
- The B axis is moved to B0
- The A axis is moved to G440 P4
- The BW axis is moved to the new angle for the next operation
- The B axis is moved to the angle of the next operation
- The A axis is moved to the location set in the next operation.

For more accurate simulation of these movements a park operation can be added to the operation list prior to a BW axis angle change.

If accurate simulation is desired setup the park operation as follows:

- Spindle Name to Sub Spindle
- X Y and Z axis set to Home
- The B axis is parked at B0
- The A axis is Parked at G330 P4 (with AutoPark addin)

Refer to the Park Operation section for additional information



21.12. Parking the BW axis Prior to part transfer

Prior to part a transfer with the NT1000 the BW axis MUST be at BW0. The transfer section will not move the BW axis prior to the transfer. If the operations prior to the transfer have the BW axis at an angle other than BW0 it the BW axis must be moved to BW0 degrees with a park operation.

A typical transfer will include a part eject (release operation) prior to the workpiece transfer. The part eject cycle will move the BW axis to BW0.

If the transfer does not include a part eject and the BW axis is not at BW0 setup a park operation on the upper head as follows:

- Spindle Name to Sub Spindle
- X Y and Z axis set to Home
- The B axis is parked at B0
- The A axis is Parked at G330 P4 (Refer to <u>Setting A-axis Position with the AutoPark</u> <u>Addin</u> section)
- BW axis is set to BW0 (refer to <u>Setting the Work Support/Sub Spindle angle (BW Axis)</u> section)

Refer to the Park Operation section for additional information

21.13. Maximize clearance when on the face of the main spindle with the B axis head:

To maximize clearance while working on the main spindle face with the B axis head you have the ability to park the A axis to the G330 position and rotate the BW axis to -90 degrees



If the transfer does not include a part eject and the BW axis is not at BW0 setup a park operation on the upper head as follows:

- Spindle Name to Sub Spindle
- X Y and Z axis set to Home
- The B axis is parked at B0
- The A axis is Parked at G330 (Refer to <u>Setting A-axis Position with the AutoPark Addin</u> section)
- BW axis is set to BW-90 (refer to <u>Setting the Work Support/Sub Spindle angle (BW</u> <u>Axis)</u> section)

Refer to the Park Operation section for additional information

21.14. Part Release from Sub Spindle

The part release operation can be added to the Esprit file to remove the part from the sub spindle. The part can be removed automatically or manually.

To manually remove the part from the machine set the Part Chute field to "No"

To automatically remove the part from the machine set the Part Chute field to "Yes"

If the part chute field is set to Yes, the following custom settings will be used:

A-axis Position (Custom setting 9) = A axis location for part release Smart Loader Shutter (M37) (Custom setting 10) =37 and M37\M38 is output to open\close the work unloading conveyer shutter Notes:

The Custom settings are not currently set by the Mori Seiki Transfer utility therefore you will need to set it after the operations are created.

Prior to programming the part release cycle the lower turret should be parked to its G30 P4 location.

The post currently only supports a part release from the sub spindle.

Set the BW rotary axis angle to 0 in the release operation properties. Set the A axis position to the position of G330 P4 location.

ase	S Operation Name								
Rele	Part Eject								
stom	Work Setup								
Ö	Spindle Name	SubSpindle 👻							
	Turret Name	B-Axis Head 🔹							
	Head Id	Head-1 🔹							
	Feeds and Speeds								
	Speed RPM	0.000000							
	Feedrate PM	250.000000							
	Use Feed and Speed KB	No	=						
	Sync Spindles Off								
	😵 Release								
	Position X, Y, Z	-100.000000 0.000000 768.550000							
	Part Chute	Yes 🗸							
	Dwell Time 0.000000								

If part chute option is set to yes the NC code will output the following:

- 1. Upper head moves to its home position
- 2. A axis is moved to G330 P4
- 3. If Smart Loader Shutter =37 Open work unloading conveyer shutter with an M37
- 4. M705 is output (moves BW axis to second zero point)
- 5. If A-axis Position is not zero, move the A axis to location in the A-axis Position
- 6. If A-axis Position is zero, move the A axis to G330 P2
- 7. Open the chuck
- 8. Turn on the air blow for one second
- 9. Move the A axis back to G330 P4
- 10. Move the BW axis to BW0
- 11. If Smart Loader Shutter =37 close the work unloading conveyer shutter with an M38

If the part chute option is set to No the post will output an M0 to allow the operator to remove the part from the machine manually

21.15. Location of part release within the ESPRIT file:

If the part transfer is at the end of the program and the machine is machining parts in both spindles simultaneously, a part release must be done prior to the part transfer as shown below:



If the part transfer is in the middle of the operations therefore machining only the part one spindle at a time (for example: machining in main spindle, transfer to sub, machine in sub spindle) the part release is typically the last operation in the file as show below:



21.16. Turret synchronization requirements for programming NT1000-WZ and NT1000-SZ

Due to certain machine requirements on the location of the turret heads during movement of the BW rotary axis and to avoid unwanted machine collisions, it is necessary to program specific syncs in ESPRIT to allow the post processor to correctly determine the needed NC code output during these operations. Below are the guidelines for correctly programming the NT1000WZ and NT1000SZ in ESPRIT 2012 and newer to ensure good NC code.

21.16.1. Parking the lower turret before a BW axis angle change made by an upper turret operation:



Note: While the lower turret is parked the upper turret operations can change the BW axis without programing additional syncs or park operations

21.16.2. Machining on the Main Spindle with the B axis head while machining on sub spindle with lower turret:



6. Machining operations

Notice how that in this situation how the tool changes are synced. It is mandatory that the tool changes are synced in this manner to ensure the lower turret remains parked while tool change in upper head takes place.

21.16.3. Machining on the sub spindle with the B axis head while machining on the main spindle with the lower turret:



21.16.4. Changing BW axis angle with B axis head operation while machining with lower turret on the main spindle

The lower turret must be parked prior to the operation with the BW axis change.

If the same tool is used before and after the BW axis angle change you must make a duplicate\copy of that tool in Esprit and change the tool in the Esprit operation with the BW axis change the duplicate tool. This has to be done to allow the proper syncs to be setup with the operation list.

		Sync					1. Park Lower	
2. Tool change		EM - 10MM - U	SubSpindle	Face Turn Rough	MainSpindle	_	turret for BW	
in B axis head,		D Contour Mill BW-25	Sub Spindle	OD Tum Rough	MainSpindle		angle change	
BW and A axis		Sync	Jab Spindle		Mainopindie			
are moved		EM - 10MM - U Copy		Svnc				
during tool		A Pocket Mill Face BW-30	SubSpindle	EM - 10MM - F - L				
change section				Rotary Contour Face Park G30 P4	MainSpindle MainSpindle		2 Machinina	
of NC code)	A Supp		Cupo			5. Machining	
							operations	J

21.16.5. Changing A axis position between two lower turret operations in a row:

When changing A axis position in-between operations on the lower turret you must insert a sync.

	Supplinue	1		
Park G330 P4 BW0	SubSpindle			
Sync		🔴 Sync		
		GI-OD-L		A-128
		메르OD Groove Lower	SubSpindle	
Sync		Sync		
		EM - 5MM - OD - L		
		2 Contour Mill OD	Sub Spindle	A-158
		Park G30 P4	MainSpindle	
Sync		Sync		
EM - 20MM - U				
Pocket Mill Face BW-25	SubSpindle			

21.16.6. Machining with one head\ turret while other head\turret is waiting:

Machining with only one head\turret at a time is much the same as the larger NT machines with the exception of necessary park operations.



21.16.7. Part release on sub spindle

A Release operation MUST to be surrounded by syncs if just before a part transfer

Since the BW axis is rotated by the Release operation the B axis must be moved to a position between 0 and 120 degrees. If needed this can be done with a park operation.

SubSpindle		
SubSpindle		
Main Spindle		E
	Sync	
SubSpindle		
	🔴 Sync	
SubSpindle		
MainSpindle		
MainSpindle		
		•
	Sub Spindle Sub Spindle Sub Spindle Sub Spindle Main Spindle Main Spindle	Sub Spindle Sub Spindle Main Spindle Sub Spindle Sub Spindle Main Spindle Main Spindle Main Spindle Main Spindle

21.16.8. Part Transfer

If a part transfer is the last thing in the operation list, a sync must be before the transfer

	Pocket Mill OD BW0	SubSpindle		
	Park G330	SubSpindle		
	Pocket Mill OD	MainSpindle		E
	Sync		Sync	
	Part Release	SubSpindle		
	Sync		Sync 🗧	
Part transfer as	Pickup	SubSpindle		
	▶ Barpull	MainSpindle		
last operations in	<mark>7</mark> co-υ			
file	= ^R / _{2≫} Cutoff	MainSpindle		
	Eastures 🕅 Teals			

If the part transfer is in the middle of the operations in a given file a sync must be on both sides of the transfer section

	格H1-VLT DRILL .1015	MainSpindle		
	PARTOFF C5 .118W			
	H1 ROUGH P/O	MainSpindle		
	Sync		Sync	
Part transfer in	┣권 Pickup	SubSpindle		
the middle of the	har Pull	MainSpindle		
	PARTOFF C5 .118W			
operations	■ ^B / ₂ H1-PARTOFF118W	MainSpindle		
	Sync		Sync	
	TURN .008 x 80° B-HEAD			
	Re-FACE	SubSpindle		
	L2-CHAMFER	SubSpindle		
	CEM 1/2 4FL			Ξ
	La noucu a ma cinone e le	0.1.0.1.0		

21.16.9. When in Long Part Machining mode and changing from Turning to MillTurn operation on the same head\turret

When holding on to the part with both spindles (Long Part Marching mode) and going from Turing mode to milling mode a sync must be between the turning and milling operations



22. Custom Settings Index

22.1. Operations Page

Operation Page						
	Section	Use	Comments			
	10.10.1	Push Check (G38)	Pickup Operation			
	19.2	Tail Stock Position	Manual Turning Operation			
	9.5	Position with G53	Park Operation			
Custom Setting 1	6.4	Rapid Feedrate (G01 F)	Milling Operations			
	15	Nose Radius Offset	B Axis Contour Operation			
	20.2.3	Steady Rest Position	Steady Rest Operation			
	19.3.2	G479 V value	Tail Stock Operation			
	10.10.1	G38 K value	Pickup Operation			
Custom Setting 2		Percentage of Spinning Tool RPM	Turning Contour and Roughing Operation			
	10.10.1	G38 Feedrate	Pickup Operation			
Custom Setting 3	16.6	G84\G88 Q value	MillTurn Drill and Wrap Drill Operations			
Custom Setting 4	10.10.1	G38 Q Value	Pickup Operation			
	10.10.3	Stopper G53 X	Barfeed (By stopper) Operation			
	10.10.3	Stopper G53 Z	Barfeed (By stopper) Operation			
Custom Setting 5	6.5	G332 Setting Value	Milling and Turning Operations (Cutting)			
Custom Sotting 6	16.9	Simultaneous Operation	SolidTurn Drilling Operation			
Custom Setting 0	10.10.2	Spindle Orientation	Pickup Operation			
	16.9	Percentage of Spinning Tool RPM	SolidTurn Drilling Operation			
Custom Setting 7	5.3	Tool Nose Direction (1-9)	Turning Operation (Cutting but not Drilling)			
	10.10.2	C Index for M45	Pickup Operation			
	10.10.5	Work Coordinate	Pickup Operation			
Custom Setting 8	5.5	G361\G362	Turning Operations (Cutting) and all MillTurn Operations			
	20.2.4	G478 Synchronize	Steady Rest Operation (NT-S 6600 only)			
	10.10.4	Workpiece Ejector	Release Operation (NT-S & NT- SZ)			
Custom Setting 9	21.8	A Axis position	Turning Operations (Cutting) and MillTurn Operations (NT1000) Release Operation (NT1000)			
Custom Setting 10	5.4	G30 or G28	Turning Operations (Cutting) and MillTurn Operations and Park Operation			
	10.10.6	Convever Shutter (M37)	Release (NT1000)			

22.2. Tools Page

Tool Page						
	Section	Use	Comments			
Custom Setting 1	13.3	Second Coolant				
Custom Setting 2	5.6	Tool Change Position	NT-S 6600 only			
Custom Setting 3	5.4	G30 or G28				
Custom Setting 4		None				
Custom Setting 5	13.4	Coolant Pressure				
Custom Setting 6		None				
Custom Setting 7		None				
		Tool Life				
Custom Setting 8	5.2	Management				
Custom Setting 9		None				
Custom Setting 10		Spinning tool	Set to 999 by Spinning Tool Addin			