

POST PROCESSOR MANUAL FOR



NZX1500 – 2000ST / TY2 / TY3 NZX1500 – 2000SY / Y2 NZX1500 – 2000SDL / Y2DL



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. Introduct	ion	1
2. Required	add-ins	2
2.1. Hov	v to load an add-in ESPRIT	2
2.2. Aut	oSubStock (optional)	3
2.3. Tur	ning Work Coordinates	5
3. How to t	urn on Custom Pages / POST Output Configuration	9
4. Machine	Setup	10
4.1. Intr	oduction	10
4.2. Set	the program name, program number	11
4.3. Def	ine the turning stock	11
4.4. Mao	chine parameters	12
4.4.1.	Introduction	12
4.4.2.	Output of Program End (M02), Program End and Rewind (M30) (or Sub-
Program	End (M99)	13
4.4.3.	Work unloader on spindle 2 side: hand or receiver	13
4.4.4.	Tool station for part transfer and work unloader	14
4.4.5.	C-axis brake clamp/unclamp	14
4.4.6.	Set how to skip turning spindle control (for NZX-ST & NZX-S	1.4
configura	ations only)	14
4.4.7.	C-axis roll over	15
4.4.8.	Set position of optional stop code (MUI) in the NC code	15
4.4.1.	99, 155 of 200 1001 wear offset number (Option on machine)	10
4.4.2.	Set position of sequence numbers (N)	10
4.4.3. 5 Tools (T	function	10
5.10018(1)	inction specifications	17
5.1.1	Introduction	17
512	Specify the tool number: Method 1	17
513	Specify the tool number: Method 2	20
514	Additional information: Tool call when machining on spindle 2 si	ide (for
NZX-ST	& NZX-S configurations only)	22
5.1.5.	Restrictions	
5.1.6.	Tool life management	
5.1.7.	Second home position	25
5.2. Set	tool movements	27
5.2.1.	Set tool movements from tool change position	27
5.2.2.	Set tool movements to tool change position	
5.2.3.	Moves to tool change position optimization	29
5.3. Soli	dTurn Grooving: Control Edge Shift	33
5.4. Res	trictions of 99, 133 or 200 Tool wear offset number (Option on ma	chine).37
6. Spindle of	lirection, speed (S function) and feedrate (F function)	41
6.1. Tur	ning spindle direction and milling tool spindle direction	41
6.2. Spin	ndle speed output for CSS unit	42

6.3.	Feedrate for 4-axis milling operations	44
6.4.	Output of rapid positioning moves (G00) with linear interpolation moves	s (G01
F)		45
7. Hov	w to output coolant codes	46
7.1.	Introduction	46
7.2.	Set the first coolant code	47
7.3.	Set the second coolant code (optional)	49
7.4.	Set a special coolant code (optional)	50
8. Sim	ultaneous machining (for NZX-ST & NZX-S configurations only)	51
8.1.	Simultaneous turning operations: Turning on the same spindle with 2 dif	ferent
turrets		51
8.1.	1. Balanced roughing cycle	51
8.1.	2. Other turning cycles	52
8.1.	3. Other turning cycles in Balanced Cut Mode	54
8.2.	Simultaneous milling operations: Milling on the same spindle with 2 diff	ferent
turrets		57
9. Ope	eration synchronization	60
10. P	ark cycle	60
10.1.	Introduction	60
10.2.	How to park a tool	60
10.3.	Park a turret to its maximal travel position (G53)	63
10.4.	Park a turret prior to a transfer	65
10.5.	Other functionalities	66
11. H	Iow to program finished part catching and part transfer	67
11.1.	Introduction	67
11.2.	Finished part catching	67
11.2	2.1. Catching finished part on spindle 1 side after cut-off	67
11.2	2.2. Catching finished part on spindle 1 side after cut-off with stock	
repo	ositioning (programmed first)	70
11.2	2.3. Catching finished part on spindle 1 side after cut-off with stock	
repo	ositioning (programmed last)	75
11.2	2.4. Manually catching finished part on spindle 1 side (for NZXDL	
con	figuration only)	79
11.2	2.5. Catching finished part on spindle 2 side	80
11.3.	Part transfer	87
11.3	3.1. Part transfer from spindle 1 to spindle 2 with cut-off	87
11.3	3.2. Part transfer from spindle 1 to spindle 2 with cut-off (when machine	е
equi	ipped with a barfeeder)	95
11.3	3.3. Part transfer from spindle 1 to spindle 2 with cut-off and stock	
repo	ositioning (programmed last)	101
11.3	3.4. Part transfer from spindle 1 to spindle 2 without cut-off	107
11.3	3.5. Part transfer from spindle 2 to spindle 1 without cut-off	112
11.3	3.6. Stock repositioning	117
11.3	3.7. Machining of long parts with spindle 1 and 2 synchronized	123
11.3	3.8. Workpiece Pushing Check (G38)	130
11.3	3.9. Park turret for transfer	131

12. Multiple repetitive cycles	132
12.1. Introduction	132
12.2. Roughing cycle	133
12.2.1. Introduction	133
12.2.2. O.D./I.D. roughing	138
12.2.3. Face roughing	139
12.3. Grooving cycle	141
12.3.1. Introduction	141
12.3.2. Face grooving	143
12.3.3. O.D./I.D. grooving	144
12.4. Threading cycle	146
12.4.1. Introduction	146
12.4.2. Threading with canned cycle set to single path	147
12.4.3. Threading with canned cycle set to off	149
12.4.4. Threading with canned cycle set to multiple path	151
13. Hole machining canned cycles and other functionalities for drilling cycles	154
13.1. Introduction	154
13.2. Face and side high-speed deep hole drilling / Face and side deep hole drill	ling /
Deep hole drilling with G74	158
13.2.1. Face and side high-speed deep hole drilling (G83.5/G87.5)	158
13.2.2. Face and side deep hole drilling (G83.6/G87.6)	159
13.2.3. Deep hole drilling with G74	160
13.3. Face and side spot drilling (G83/G87)	162
13.4. Face and side tapping / Tapping at center of spindle	163
13.4.1. Face and side tapping (G84/G88)	163
13.4.2. Tapping at center of spindle (G32)	164
13.5. Face and side synchronized tapping / Face and side (high-speed) deep hol	e
synchronized tapping / Spindle synchronized tapping	165
13.5.1. Restriction of synchronized tapping	165
13.5.2. Face and side synchronized tapping (M329 G84/M329 G88)	165
13.5.3. Face and side (high-speed) deep hole synchronized tapping (M329	
G84/M329 G88)	167
13.5.4. Spindle synchronized tapping (M329 G84) (Option on machine)	170
13.6 Face and side boring / Boring in turning mode	172
13.6.1. Face and side boring (G85/G89)	172
13.6.2 Boring in turning mode	173
13.7. Spindle / Rotary Tool Spindle Simultaneous Operation Mode	174
14. 3D coordinate conversion (G68.1) (Option on machine)	176
14.1. Introduction	176
14.2. How to output 3D coordinate conversion.	177
15. 4-axis wrap milling cycles with interpolation	180
15.1. Introduction	180
15.2. Cylindrical interpolation	180
15.3 Polar coordinate interpolation (Notching)	182
16. Custom Settings index	184
16.1. On operation pages	184

16.2.	On tool pages	185
16.3.	In Machine Setup	185

1. Introduction

These notes are about the following post processors:

NZX1500 – 2000ST / TY2 / TY3 (formerly named NZ-T3Y3):

E12MSP_NZX-ST_2012-04-13_T1.pst: Turret 1 (upper left turret), machines on spindle 1 (main spindle) side only

E12MSP_NZX-ST_2012-04-13_T2.pst: Turret 2 (lower turret), machines on spindle 1 and spindle 2 (sub spindle) sides

E12MSP_NZX-ST_2012-04-13_T3.pst: Turret 3 (upper right turret), machines on spindle 2 side only

NZX1500 – 2000SY / Y2 (formerly named NZ-T2Y2):

E12MSP_NZX-S_2012-04-10_T1.pst: Upper turret, machines on spindle 1 and spindle 2 (sub spindle) sides

E12MSP_NZX-S_2012-04-10_T2.pst: Lower turret, machines on spindle 1 and spindle 2 (sub spindle) sides

NZX1500 – 2000SDL / Y2DL (formerly named NZ-T2Y2DL):

E12MSP_NZXDL_2012-01-11_T1.pst: Upper left turret, machines on spindle 1 side E12MSP_NZXDL_2012-01-11_T2.pst: Upper right turret, machines on spindle 2 (sub spindle) side

Note: NZX-ST has no Y-axis; NZX-STY2 has a Y-axis on upper left and lower turrets and on NZX-STY3 all turrets have a Y-axis.

The output will match your machine configuration based on the loaded NZX Machine Setup in ESPRIT. For example, if you work on a NZX-ST, no Y-axis movements will be output in the code.

In the following manual, you can ignore any reference to the Y-axis if your machine is not equipped with this axis.

In most cases, this programming manual will be referencing the most complex NZX configuration (NZX-ST). But the given instructions are also applicable to other NZX configurations mentioned above.

Post processors require ESPRIT 2012 (Build B19.0.3.1325 or above).

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

The mention **option on machine** refers to a nonstandard machine functionality. Please contact your Mori Seiki reseller for more details.

2. Required add-ins

2.1. How to load an add-in ESPRIT

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



You will then see the add-in window:

Add-In Manager		×
Available Add-Ins 3D Connexion Motion Controller AutoSubStock Citizen Cut20 Technology Add-In EDM Features Group Manager Monster MoriSeiki 3D Tool Simulation AddIn MoriSeiki AddIn Smart Toolbar Soft Syncing Add-In SpinningTurningTool Addin	Load Behavior Startup / Loaded Startup / Loaded Startup / Loaded Startup / Loaded	OK Cancel
Turning Work Coordinates Description Adds support for the 3D Connexion Motion (space ball)	Controller	Help I/Unloaded n Startup nd Line

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

Also check the box Load on Startup: the add-in will then be automatically loaded when you will start ESPRIT.

Add-In Mana	iger in the second s		×	
Available / 3D Connex AutoSubSt Citizen Cut20 Tec EDM Featu Group Mar Monster MoriSeiki 3	Add-Ins tion Motion Controller ock hnology Add-In ures hager D Tool Circulation AddIn tidlin 1	Load Behavior Startup / Loaded Startup / Loaded	3 OK Cancel	
Smart Tool Soft Syncir SpinningTu Turning W	bar Galan ng Add-In urningTool Addin ork Coordinates	Startup / Loaded	Help	
Description MoriSeiki A	ddin		2 Load Behavior Loaded/Unloaded Load on Startup Command Line	

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

- AutoSubStock (optional)
- MoriSeiki AddIn
- Turning Work Coordinates

2.2. AutoSubStock (optional)

The AutoSubStock add-in needs to be turned on prior to open a file. This add-in will allow you to correctly simulate the spindle 2 machining. It will also allow the simulation of production machining, when cutting on the spindle 1 and the spindle 2 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 spindle 2 and stop the simulation.



If you restart the simulation, you will be able to see the cut on the spindle 1 and spindle 2 sides all at the same time.

Important note: In ESPRIT, from the Tools menu, select Options...



On the Machining tab, if you check Enable Stock Automation, you will not need to turn off the AutoSubStock add-in since the Stock Automation will compute the state and shape of your stock present in both the spindle 1 and spindle 2:

Options	x
Attributes Input Advanced Machining Atkspace File Locations	
Tech Page Defaults	
Custom Page	
Stock	
Enable Stock Automation	
Stock Automation Tolerance 0.1	
Stock Transparency	

2.3. Turning Work Coordinates

The Turning Work Coordinate add-in will sort your operations in the appropriate work coordinates and also offset the NC code for the spindle 2 operations. It is necessary to run this add-in in order to generate correct NC code.

From the Create menu, select Turning Work Coordinates.

		· ·
iew	Create 1 ning Tools Window Help)
5	✓ Default <u>A</u> ttributes	
Z,		
	<u>F</u> eatures	
	Surfaces	
	Curves	
	Surface C <u>u</u> rves	
	✓ Solid Modeler	
	Form Tool	
	Engraving	
	Turning Work Coordinates	2

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	Spindle Z- Orientation				
Options					
Keep Z axis parallel with Tool axis With Tilted Orientation					
Reverse Z axis of WC if spindle Z axis is reversed NO					
Autorun Mode BEFORE POSTING					
	OK Cancel				

MainSpindle and SubSpindle information are directly coming from the Machine Setup.

• WC Name: It corresponds to the Work Coordinate name. Operations located on the spindle 1 side will be moved in the G54 work coordinate and on the spindle 2 side in G55. Please note that the name is just informative and will not affect the code. WC Numbers will.

• WC Numbers: The first field will be used in the NC code. 54 will output G54 at the beginning of an operation on the spindle 1 side, 55 will output G55 at the beginning of an operation on the spindle 2 side.

• **Z** Offset: It is used to correctly offset the work coordinate on the spindle 2 side. The value is read from the Part Stock Length field of the Machine Setup (on the General tab). The spindle 2 work coordinate will be offset by minus this amount from the spindle 1 work coordinate. That is why the value you enter in the Machine Setup is critical. A wrong value will cause a wrong offset and as a conclusion a part not cut correctly.

SolidTurn Machine Setup	x
General Assembly NC Output	
Machine Definition Machine Name NZ2000T3Y3 Start Position Z 1 Start Position X 0 Start Position Y 0	Stock Configuration Stock Type Bar Bar Diameter 60.000000 Total Bar Length 200 Inside Diameter 20 Casting Feature Image: Casting Feature
MillTurn Rotary Retract Movement Retract Position Clearance Rotary Clearance 254 Park Position X,Y,Z 0 0 0	Miscellaneous NC Offset Simulation Along Lead-in/out Dogleg Rapid Off Optimize Tool Path Off Rotary Index Time 0.18 Optimize Tolerance 0.254 5-Axis RTCP Coordinate Mode Fixed T Calculate Links Settings
	OK Cancel Help

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

On the Options field, Keep Z axis parallel with Tool axis and Reverse Z axis of WC if spindle Z axis is reversed will have no effect for this machine.

So, you basically do not need to change anything on this dialog. Once you click on ok, all your operations will be sorted for you in G54 (spindle 1 side) and G55 (spindle 2 side).

Note that if you want to output different work coordinate numbers, you can change the WC number to 56, 57, 58 or 59. This machine can work with work coordinates G54, G55, G56, G57, G58 or G59.

If you did already run the add-in, you can still edit the existing work coordinates and change the name and number to whatever you need:



On the Features tab, double click on an existing work coordinate. So you will be able to edit it. You can change on the Work Coordinate dialog the Work Coordinate Name (informative) and, for the NC code, the Standard Work Coordinate Number.

Please note that you can also create a new work coordinate with the desired number and then move any operation to it.

Finally, you can set an Autorun Mode. So, you will not need to think about running the add-in every time you need to output the NC code.

- None: No autorun mode selected, you have to manually run the add-in to sort operations.
- Before posting: Every time you will output the NC code, the add-in will be run.
- **Before simulation**: Every time you will start the simulation, the add-in will be run.

• **Before posting & simulation**: Every time you will output the NC code or you will start the simulation, the add-in will be run.

3. How to turn on Custom Pages / POST Output Configuration

In the following manual, you might need to use the Custom tab of an operation page or of the Machine Setup to be able to output specific code related to a special function.

To turn this tab on, in ESPRIT, from the Tools menu, select Options...



On the Machining tab, check the checkbox Custom Page, click on Default... (and Save current as user defaults) and finally on OK.

Options 🔀
Attributes Input Advanced Machining 1 kspace File Locations
Tech Page Defaults
Custom Page 2
Stock
Enable Stock Automation
Stock Automation Tolerance 0.1
Stock Transparency
OK 4 Cancel Default 3 Help

Now, you will have access to the custom page on which you will be able to set some flags. They will be detailed in the manual, when needed.

Note that with the MoriSeiki AddIn, you will be able to use the POST Output Configuration:

- Machine parameters can directly be set on the POST Output Configuration (see part **4.4. Machine parameters**)
- On operation and tool pages, the function of the required custom settings will be displayed

4. Machine Setup

4.1. 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:

J	ß	а. Ча		P	÷
---	---	----------	--	---	---

4.2. 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.

olidTurn Machin	e Setup							
General Assemb	NC Out	put 3						
NC Output	Order	Turret	Number of Heal	ads 1	т. Н	ead ID	Head-1	-
Sync String Co Number of S	odes Gyncs	4		Sync Codes Code Incremen	it 📘	0 🗧		
Sync ID an	d Name		•	Start and End M	Number 1	00	900	
C 10			Turret Pro	gram Output				_
Program Na	me	TEST	 Coordinate Mo	de Absolute	_			
Program Nu	mber [1000	Unit for NC Co	de Metric		4		
- Turret/Spin	dle Belation:	shins		,	_			
Spindle/Turi	et Names	MainSpindle	SubSpindle					
Upper	Left Turret	Shared	▼ Shared ▼	Shared 💌	Shared	Shared	~	
Lower	Turret	Shared	▼ Shared ▼	Shared 🗾	Shared	Shared	~	
Upper	Right Turr	Shared	💌 Shared 💌	Shared 💌	Shared	Shared	~	
		Shared	Shared 💌	Shared 💌	Shared	Shared	T	
		Shared	Shared 💌	Shared 🗾	Shared	Shared	V	

4.3. 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.

So	olidTurn Machine Setup								X
Γ	General Assembly NC 0	lutput Custom							
Г	Machine Definition			Stock Configuration-		_			
	Machine Name	NZ2000T3Y3		Stock Type	Bar 💌		Maximum Diameter	320	
	Start Position Z	2.5		Bar Diameter	100.000000		Total Bar Length	300	
	Start Position X	0	1	Inside Diameter	20		Part Stock Length	100	
	Start Position Y	0		Casting Feature					

Note: The stock configuration will be detailed when needed in this manual.

4.4. Machine parameters

4.4.1. Introduction

Machine parameters can be set on the POST Output Configuration (part of the MoriSeiki AddIn).

*	POST Output Configuration			×
9	Program End Code	M30 -	Custom String 1	
80	Work Unloader Type	Unload by Hand 👻	Custom String 2	
08	Custom Setting 3		Custom String 3	
8	Station on Turret 1 for Transfer	0	Custom String 4	
01	Station on Turret 2 for Transfer	0	Custom String 5	
	Station on Turret 3 for Transfer	0	Custom String 6	
ĩ	C Axis brake M Code	Output Brake M Code 👻	Custom String 7	
T	Block Skip Method	Comment Out 👻	Tool Wear Offset Limit	66 🔹
	C Axis Roll Over	Roll Over Off 🛛 👻	End of Bar Macro Call (/2 M98 P)	
8.8	Position of M01 Code	At Syncs 👻	Position of Sequence Numbers (N)	At Operations 👻
	ų			
	Extra Setting 1		Extra Setting 11	
0	Extra Setting 2		Extra Setting 12	
	Extra Setting 3		Extra Setting 13	
	Extra Setting 4		Extra Setting 14	
	Extra Setting 5		Extra Setting 15	
	Extra Setting 6		Extra Setting 16	
	Extra Setting 7		Extra Setting 17	
	Extra Setting 8		Extra Setting 18	
	Extra Setting 9		Extra Setting 19	
	Extra Setting 10		ESPRIT Version	12 🔹
				OK Cancel

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

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.

4.4.3. Work unloader on spindle 2 side: hand or receiver

On the POST Output Configuration, set **Work Unloader Type** to **Unload with Receiver** if your machine is equipped with a receiver; set it to **Unload by Hand** if it is equipped with an arm.

This will affect the NC output when you program your finished part catching. It will differ based on the type of work unloader present on your machine.

Please note that machine equipped with an arm is the default.

For additional information on how to program finished part catching, see part **11.2**. **Finished part catching**.

4.4.4. Tool station for part transfer and work unloader

By default, station 1 will be used as empty station during part transfer and work unloader.

It can be changed on the POST Output Configuration using the textboxes **Station on Turret 1 for Transfer, Station on Turret 2 for Transfer** and **Station on Turret 3 for Transfer** (for NZX-ST configuration only). You can enter a value between 1 and 16. Note that if you work on spindle 2 side, 30 will be added to the entered number for NZX-ST & NZX-S configurations only.

For example on NZX-ST, if Station on Turret 1 for Transfer is set to 4, Station on Turret 2 for Transfer is set to 7 and Station on Turret 3 for Transfer is set to 12, the tool call for transfer will be T0400 for head 1, T0700 for head 2 and T4200 for head 3.

Note: If the station number specified is out of range (value entered greater than 16 or less than 0), you will get the following error message in your NC code: ERROR: WRONG TRANSFER STATION NUMBER ENTERED IN MACHINE SETUP.

For additional information on finished part catching and part transfer, see part **11. How to program finished part catching and part transfer**.

4.4.5. C-axis brake clamp/unclamp

By default, C-axis brake clamp (M68 for spindle 1 and M268 for spindle 2) and unclamp (M69 for spindle 1 and M269 for spindle 2) M-codes will be output in the NC code.

If you do not want to output these M-codes in the NC code, set on the POST Output Configuration C-Axis Brake M Code to Do Not Output M Code.

4.4.6. Set how to skip turning spindle control (for NZX-ST & NZX-S configurations only)

When using spindle priority on turning cycles (see part **8.1.2. Other turning cycles**), you can specify using Block Skip Method on the POST Output Configuration how to skip the turning spindle control (spindle start and spindle stop) on the slave turret.

By default, the turning spindle control on the slave turret will be commented out.

Set on the POST Output Configuration **Block Skip Method** to **Block Skip** to skip turning spindle control with block skip (/).

Set on the POST Output Configuration **Block Skip Method** to **Block Skip 2** to skip turning spindle control with block skip 2 (/2).

4.4.7. C-axis roll over

On the POST Output Configuration, set C-Axis Roll Over to Roll Over On to turn on C-axis roll over.

Note that you will need to turn it on on the machine control as well.

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, C-Axis Roll Over on the POST Output Configuration is set to 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, C-Axis Roll Over on the POST Output Configuration is set to Roll Over On.

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

Note: During 4-axis milling (wrap and rotary face milling) operations with C-axis roll over on, if C-axis value exceeds a revolution (value output greater than 360° or less than - 360°), you will get the following error message in your NC code: ERROR: C-AXIS OVER LIMIT: ROLL-OVER MUST BE TURNED OFF.

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

By default, optional stop codes (M01) will be output on each head after synchronization codes between all heads in the NC code. This way, optional stop can be turned on on the machine while running the NC code on all heads at the same time (production mode).

Set on the POST Output Configuration **Position of M01 Code** to **At Operations and Syncs** to output optional stop codes after each operation and each programmed wait code. If optional stop is turned on on the machine, this mode can only be used when running NC code on one head at a time (NC code proofing mode).

Set on the POST Output Configuration **Position of M01 Code** to **At Tool Cancels and Syncs** to output optional stop codes at tool cancellation and after each programmed wait code. If optional stop is turned on on the machine, this mode can only be used when running NC code on one head at a time (NC code proofing mode).

4.4.1. 99, 133 or 200 Tool wear offset number (Option on machine)

By default, 66 tool wear offset number will be used as a limit when defining your tools in ESPRIT: the maximum tool wear offset number you will be able to output in the NC code will be 66.

If your machine has the 99, 133 or 200 tool wear offset number option (200 for NZXDL configuration only), set on the POST Output Configuration **Tool Wear Offset Limit** to **99, 133** or **200**. 99, 133 or 200 Tool wear offset number will be used as a limit when defining your tools in ESPRIT: the maximum tool wear offset number you will be able to output in the NC code will be 99, 133 or 200. Tool calls will be done with five digits instead of four when using the 133 or 200 option: tool wear offset numbers have three digits instead of only two.

Note: See part **5. Tools (T function)** for programming details.

4.4.2. Output of the end of bar macro call (/2 M98 Pxxxx)

If your machine is equipped with a barfeeder, you can output in the NC code the end of bar macro call with /2 M98 Pxxxx (xxxx is the program number). This way, when the bar (stock) reaches the end, the automatic barfeeder will load a new one.

On the POST Output Configuration, enter in the **End of Bar Macro Call** (/2 M98 P) textbox the program number you desire to call to check for the end of bar. It will then be output as entered in the NC code.

4.4.3. Set position of sequence numbers (N)

By default, sequence numbers (N) will be output at the beginning of each operation in the NC code.

Set on the POST Output Configuration **Position of Sequence Numbers (N)** to **At Tool Changes** to output sequence numbers at tool change only.

This setting will only affect regular cutting operations: sequence numbers for transfer operations will always be output.

Sequence numbers for SolidTurn Roughing operations with canned cycle on will also always be output. This is to avoid sequence number mismatch since sequence numbers will be used at the beginning and at the end of profile description.

5. Tools (T function)

5.1. T function specifications

5.1.1. Introduction

T[][][];

The first two digits of a T number specify the tool number and the tool geometry offset number.

The last two digits of a T number specify the tool wear offset number.

$\underline{NZX1500-2000ST\ /\ TY2\ /\ TY3:}$



	Turret and Us	Spindle to e	HEAD and T Code to Spec- ify		
1	Turret 1	Spindle 1	HEAD1	T0101 - T1616	
2	Turret 2	Spindle 1	HEAD2	T0101 - T1616	
3	Turret 2	Spindle 2	HEAD2	T3131 - T4646	
4	Turret 3	Spindle 2	HEAD3	T3131 - T4646	

<u>NZX1500 – 2000SY / Y2:</u>



	Turret and Spindle to Use		HEAD and	d T Code to Spec- ify
1	Turret 1	Spindle 1	HEAD1	T0101 - T1616
2	Turret 2	Spindle 1	HEAD2	T0101 - T1616
3	Turret 1	Spindle 2	HEAD1	T3131 - T4646
4	Turret 2	Spindle 2	HEAD2	T3131 - T4646

<u>NZX1500 - 2000SDL / Y2DL:</u>



	Turret and Us	Spindle to e	HEAD and T Code to Spec- ify	
1	Turret 1	Spindle 1	HEAD1	T0101 - T1616
2	Turret 2	Spindle 2	HEAD2	T0101 - T1616

Note: It is recommended to use the same number for the tool number and tool wear offset number when specifying a T command in a program to avoid operator's errors.

In ESPRIT, you have two different ways to enter the tool call number that will be output in the NC code. They will be explained below.

5.1.2. Specify the tool number: Method 1

A tool number in ESPRIT shall be entered as three- (101, for example) or four-digit numbers (1414, for example). It will be entered in the Tool Number dialogue box on the respective tool page. It will then be output in the NC code.

If, for example on turret 1, the Tool Number entered is 101, the tool call output will be T0101. If 1414 is entered, the tool call output will be T1414.

Turning tools:

ning Tools - Turning	Insert		2
nsert Holder Genera	l]		
- Common Parameters-			
Tool ID	Tool 14	Orientation	3V 💌
Tool Number	101	Orientation Angle	0.000000
Turret Name	(Unmounted)	Compensation	Corner
Station Name	Station:1	Length Register	101
Spindle Direction	CW	•	
Coolant	On	▼ Touch Off Angle	0.000000
Simulation Cut Color			
Tool Change			
Movement XZ,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	ancel Help

Milling tools:

Milling To	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 14 1414 0n CW 50.000000 Metric			
			 ОК	Cancel	Help

5.1.3. Specify the tool number: Method 2

If the Tool Number is entered as a one-digit number (1, for example) or a two-digit number (14 for example) in ESPRIT, the post processor will combine this number to the number entered in the Length Register dialogue box.

If, for example on turret 1, the Tool Number entered is 1 and the Length Register number entered is 3, the tool call output will be T0103.

Turning tools:

rning Tools - Turning	Insert		
nsert Holder Genera	al]		
Common Parameters-			
Tool ID	Tool 14	Orientation	3V 💌
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			
Tool Change			
Movement XZ,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	ncel Help

Milling tools:

Milling To	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 14 14 14 0n CW 50.000000 Metric		
			OK Cancel Help	

5.1.4. Additional information: Tool call when machining on spindle 2 side (for NZX-ST & NZX-S configurations only)

When machining on spindle 1 with turret 1 or turret 2, the tool number will be output directly as entered in the Tool Number dialogue box if method 1 is used or as a combination of the entered Tool Number and Length Register number if method 2 is used.

When machining on spindle 2 with turret 1 (for NZX-S configuration only) or turret 2, the post processor will add **3030** to the tool number entered in the Tool Number dialogue box if method 1 is used or it will add **30** to the Tool Number and **30** to Length Register number if method 2 is used.

When machining on spindle 2 with turret 3 (for NZX-ST configuration only), the post processor will add **3000** to the tool number entered in the Tool Number dialogue box if method 1 is used or it will add **30** to the Tool Number if method 2 is used.

See the examples below:

• *Example 1 for method 1 (on head 2):*

If 101 is entered in the Tool Number dialogue box, T3131 will be output in the NC code.

• *Example 2 for method 1 (on head 3):*

If 1444 is entered in the Tool Number dialogue box, T4444 will be output in the NC code.

• *Example 1 for method 2 (on head 2):*

If Tool Number it set to 1 and Length Register number is set to 3, T3133 will be output in the NC code.

• *Example 2 for method 2 (on head 3):*

If Tool Number it set to 16 and Length Register number is set to 60, T4660 will be output in the NC code.

5.1.5. Restrictions

Tool number and tool geometry offset T[][]— range of values:

The two first digits of a tool call entered in ESPRIT have to be contained **between 1 and 16**. When working on spindle 2, you do not need to add 30 to the number since the post processor will do it depending on your machine.

Note: If the Tool Number is out of range, you will get the following error message in your NC code: ERROR: WRONG TOOL NUMBER ENTERED IN ESPRIT.

Tool wear offset T—[][] range of values for NZX-ST configuration:

- Head 1: The entered Length Register number has to be contained between 1 and 66.
- Head 2 machining on spindle 1 side: The entered Length Register number has to be contained between 1 and 30.
- Head 2 machining on spindle 2 side: The entered Length Register number has to be contained between 1 and 30, 30 will be added to this number.
- Head 3: The entered Length Register number has to be contained between 1 and 66.

Tool wear offset T—[][] range of values for NZX-S configuration:

- Head 1 machining on spindle 1 side: The entered Length Register number has to be contained between 1 and 30.
- Head 1 machining on spindle 2 side: The entered Length Register number has to be contained between 1 and 30, 30 will be added to this number.
- Head 2 machining on spindle 1 side: The entered Length Register number has to be contained between 1 and 30.

• Head 2 machining on spindle 2 side: The entered Length Register number has to be contained between 1 and 30, 30 will be added to this number.

Tool wear offset T—[][] range of values for NZXDL configuration:

• Head 1: The entered Length Register number has to be contained between 1 and 66.

• Head 2: The entered Length Register number has to be contained between 1 and 66.

Note: If the entered Length Register number is out of range, you will get the following error message in your NC code: ERROR: WRONG LENGTH REGISTER NUMBER ENTERED IN ESPRIT.

5.1.6. Tool life management

On the Custom tab of the tool page, set Tool Life Management (Custom Setting 8) to 300 to turn on the tool life management.

Turning tool page:

Turn	ing Tools - Turning Ins	ert			X
Ins	sert Holder General	Custom			
	Values Second Coolant	0.000000			
	Custom Setting 2	0.000000			- 11
	G30 or G28	0.000000			- 11
	Custom Setting 4	0.000000			
	Custom Setting 5	0.000000			
	Special Coolant ON	0.000000			
	Special Coolant OFF	0.000000			
	Tool Life Management	300.000000			
	Custom Setting 9	0.000000			
	Custom Setting 10	0.000000			
			1		
			ОК	Cancel	Help

Milling tool page:

Milling To	ols - End Mill				X
	Values Second Coolant Custom Setting 2 G30 or G28 Custom Setting 4 Custom Setting 5 Special Coolant ON Special Coolant OFF Tool Life Management Custom Setting 9 Custom Setting 10	0.000000 0.000000 0.000000 0.000000 0.000000			
Commen	t		 		
			ОК	Cancel	Help

This will trigger the output of G336 (group command) and M300 (tool life count): The tool call will be made with G336 and at tool cancel M300 will be output.

If you omit (or enter a wrong value) to enter the value in the Tool Life Management field, the tools will be called with G00; G336 and M300 will not be output in the NC code.

5.1.7. Second home position

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

Turning tool page:

Turning Tools - Turning Ins	ert			×
Insert Holder General	Custom			
Values Second Coolant	0.000000			
Custom Setting 2	0.000000			
G30 or G28	30.000000			
Custom Setting 4	0.000000			
Custom Setting 5	0.00000			
Special Coolant ON	0.000000			
Special Coolant OFF	0.00000			
Tool Life Management	0.00000			
Custom Setting 9	0.00000			
Custom Setting 10	0.00000			
		ОК	Cancel	Help

Milling tool page:



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

If you omit (or enter a wrong value) to enter the value in the G30 or G28 field, turrets will be sent to the machine zero point with G28; G30 will not be output in the NC code.

5.2. Set tool movements

5.2.1. Set tool movements from tool change position

In ESPRIT, on the Assembly tab of the Machine Setup, select a turret from the list on the left hand side of the window. You will then see the pull-down Moves From Tool Change. It will let you specify how the tool moves away from its position to the start of the next operation once indexed during a tool change.

SolidTurn Machine Setup					×
General Assembly NC Output Custo	M New Turret X Turnet Properties Name Number Radial Output Turret Style Start Station Home Position X Y Z STL Ref. Point X Y Z TL Ref. Point X Y Z	New Spindle New Axis	New Steady Rest New Solid Rapid Rate Max Feedrate Tool Change Time Moves to Tool Change [357.00000 [0.000000] ar	Delete New Station 29000.000000 29000.000000 0.180000 X First YZ Next X XY Next Z XYZ XZ Next Y YZ Next X	
				OK Cancel	Help

Moves From Tool Change can be set to:

- **XY Next Z:** The tool moves along X and Y only to the entry location, then moves into position along the Z-axis.
- **XYZ:** Linear interpolation is used to move the tool from the tool change position to the start of the cut in a straight line.
- XZ Next Y: The tool moves along X and Z only, then moves along the Y-axis.
- YZ Next X: The tool moves along Y and Z only, then moves along the X-axis.

Notes:

- Based on your selection, the movements output in the NC code will change accordingly.
- When loading a Machine Setup of the Mori Seiki NZX series, the default selection for Moves From Tool Change is YZ Next X.

5.2.2. Set tool movements to tool change position

In ESPRIT, on the Assembly tab of the Machine Setup, select a turret from the list on the left hand side of the window. You will then see the pull-down Moves To Tool Change. It will let you specify how the tool moves away from the end of the previous operation to the tool change position of the next tool.

SolidTurn Machine Setup	
General Assembly NC Output Custom Imachine Base New Turret New Spindle New Steady Rest Development Imachine Base New Turret New Axis New Steady Rest Development Imachine Base New Turret New Axis New Steady Rest Development Imachine Base New Fight Turret New Axis New Solid New Imachine Base Turret Repeties New Axis New Solid New Imachine Base Turret Repeties Name Upper Left Turret Rapid Rate 29000.00 Imachine Base Name Upper Left Turret Rapid Rate 29000.00 Max Feedrate 29000.00 Imachine Base Name Upper Left Turret Rapid Rate 29000.00 Max Feedrate 29000.00 Imachine Base Name Upper Left Turret Rapid Rate 29000.00 Max Feedrate 29000.00 Imachine Base Name Upper Left Turret Rapid Rate 29000.00 Name New Store Tool Change X First Imachine Base Stat Station Statistation Statiston Stat	elete Station
ОК	Cancel Help

Moves To Tool Change can be set to:

- **X** First: The tool moves away from the part along the X-axis first, then moves along Y and Z to the tool change position of the next tool.
- **XYZ:** Linear interpolation is used to move the tool in a straight line from the part to the tool change position of the next tool.
- **Y** First: The tool moves along the Y-axis first, then along X and Z.
- **Z** First: The tool moves along the Z-axis first, then along X and Y.

Notes:

• Based on your selection, the movements output in the NC code will change accordingly.

• The selection of Moves To Tool Change movement will override the selection made for Moves From Tool Change. For example, if both Moves To & From Tool Change are set to XYZ and Moves To Tool Change is changed to X First, Moves From Tool Change will automatically be changed to YZ Next X. This is to maintain consistency in the tool movements before and after an operation.

• When loading a Machine Setup of the Mori Seiki NZX series, the default selection for Moves To Tool Change is X First.

5.2.3. Moves to tool change position optimization

Using the Tool Change Movement X, Y, Z settings on the turning and milling tool pages, you can optimize your tool movements to its tool cancel position. These settings will allow you to fully utilize the production capabilities of your machine.

Note that the tool cancel position of the current tool is also the tool change position of the next tool that will be used.

Turning tools:

Turning Tools - Turning Ir	isert		×
Insert Holder General	Custom		
Common Parameters			
Tool ID	Turning Insert	Orientation	3V 💌
Tool Number	1	Orientation Angle	0.000000
Turret Name	Upper Left Turret 💌	Compensation	Comer 💌
Station Name	Station:1	Length Register	1
Spindle Direction	CW		
Coolant	On 💌	Touch Off Angle	0.000000
Simulation Cut Color			
Tool Change			
Movement X,Z,Y	Home 💌	Home 💌	Home 💌
Position XZ,Y	None Home	125.000000	0.000000
Tool Shift	Position Machine	1	
Tool Shift X, Z, Y	0.000000	0.000000	0.000000
		OK Ca	ncel Help

Milling tools:

Milling Tool	ls - End Mill				×
	Fool Change Movement X,Y,Z Home None Home Position Mounting Turret Name Station Name Axis Orientation ·Vector X, Y, Z 1.000000 O.000000 0.000000	▼ Home ▼ 125.000000 125.000000 Upper Left Turret ▼ Station:1 ▼ X + ▼ 0.000000 0.000000			
			 ОК	Cancel	Help

Movement X, Y, Z can be set to:

• None: The tool will not move along the selected axis. If you set for example Movement X to None, the tool will not move along the X-axis when going to the tool cancel 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 **5.1.7. Second home position** for additional information. This is the default when creating a new tool.

• 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.

Tool Change				
Movement X,Z,Y	Machine	Home	✓ Home	·
Position X,Z,Y	-100.000000	125.000000	0.000000	

• **Position:** It cannot be used.
Note: The X; Y & Z positions entered in ESPRIT (for Machine mode) are absolute in YZX. So the post processor will compute and output the correct values from this absolute position based on the turret and the spindle in use.

Once the tool movement has been optimized, the movements sequence will be given by the Moves To Tool Change setting in the ESPRIT Machine Setup (see part **5.2.2. Set tool movements to tool change position**). See the examples below:

Example 1: If Moves To Tool Change is set to X First and Movement X, Y, Z is set to Machine (Position -100), Home, Home, the NC code output at tool cancel will be: G53 x-200.0 G28 v0 w0

• Example 2:

If Moves To Tool Change is set to XYZ and Movement X, Y, Z is set to Home, Home, Home, the NC code output at tool cancel will be: G28 U0 V0 W0

• Example 3:

If Moves To Tool Change is set to Z First and Movement X, Y, Z is set to None, Machine (Position 10), Home, the NC code output at tool cancel will be: G28 W0 G53 Y10.0

Note that if your Moves To Tool Change and Movements X, Y, Z combination would result in impossible movement (movement that would trigger an alarm on the machine for example), you will get an error message at the beginning of your NC code.

Here is a list of impossible movements:

- Trying to output G28 & G53 on the same line in the NC code: Home & Machine cannot be used on combined axes movements.
- Position cannot be used: if any axis is set to position, an error message will be output for that tool.

• None cannot be set for all three axes at the same time: the tool needs to be moved away at the end of an operation before indexing of the next tool for safety reasons.

The output error message will list all the incorrectly set tools to help you find what tools need to be modified.

Here is an example of an impossible movement that will trigger the output of an error message: If Moves To Tool Change is set to XYZ and Movement X, Y, Z is set to Home, Machine, None, you will get this error message in the NC code: ERROR: TOOL CHANGE MOVEMENT NOT DEFINED CORRECTLY FOR TOOL: TURNING INSERT OD UL; OPERATION: CONTOURING FACE.

Notes:

• The very first tool of a program will always be called from home (G28).

• Tool cancel position of current tool will match tool change position of the next tool.

• Movements X, Y, Z set to None, None, None can only be used for dummy tools created for park cycle.

• If your machine configuration does not have a Y-axis, Movement Y will not have any effect on the error checking routine when defining your optimized tool change position.

• By default, tools will be sent home at tool change and at tool cancel with X-axis moved first.

5.3. SolidTurn Grooving: Control Edge Shift

When you finish a groove with control edge shift, you can output the NC code controlling the left corner of the grooving insert for the left side of the groove and its right corner for the right side of the groove. You can assign to your grooving insert two different tool wear offset data. One register will be for the left edge of the tool and another will be for the right edge.



Control Edge Shift = YES



To enable Control Edge Shift in ESPRIT, on the Finish tab of the SolidTurn Grooving operation, set Control Edge Shift to Yes:

Project Manager 🛛 🗶						
₩ 0	K 🗙 Cancel 🛐 🗸		🕜 Help			
eral	Stock Allowance					
Gen	Finish Stock Z, X	0.000000 0.000000				
teg						
Stral	Finish RPM, CSS	0				
Чġ	Finish PM, PR	0.000000 0.000000				
Bot	Use Feed and Speed KB	No	•			
inish	😵 Finish Pass					
ш	Finish Direction	Forward	•			
stom	Finish Mode	Per Side	•			
3	Ö Control Edge Shift Yes					
	S Compensation					
	Cutter Comp NC	No	•			
	😵 Lead-In/Out					
	Lead-In Type	Normal	•			
	Normal Distance	0.000000				
	Lead-Out Type Normal					
	Normal Distance	0.000000				
	🚝 Features 📑 Tools 🖽 Operations 📲 SolidTurn - Grooving					

Project Manager 🛛 🕅					
heral	Cutting Strategy				
gei	Type of Work	OD 🔹			
<u>B</u>					
Strat					
_	Rough Pass	Yes			
loug	Finish Pass	Yes			
-	😵 Rapid Approach/Exit				
Finish	Entry Mode	None			
stom	Exit Mode	None			
Ö	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	No			
	Features 🚺 Tools 🔜	Operations 12 SolidTurn - Grooving			

Notes:

- This is only effective for the finish pass of the groove.
- The Finish Mode has to be set to Per Side.

The second tool wear offset data can be specified on the General tab of the Grooving Insert tool page in the Edge Shift Register field:

Tur	ning Tools - Grooving	Insert		×			
Ir	Insert Holder General						
	- Common Parameters						
	Tool ID	Tool 2	Orientation	3∨ ▼			
	Tool Number	1	Orientation Angle	0.000000			
	Turret Name	(Unmounted)	Compensation	Left Edge 💌			
	Station Name	Station:1	Length Register	1			
	Spindle Direction	CW	Edge Shift Register	10			
	Coolant	On 💌	-				
	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 Car	ncel Help			

Tool wear offset range of values for NZX-ST configuration:

- Head 1: The entered Edge Shift Register number has to be contained between 1 and 66.
- Head 1 with tool life management on: The entered Edge Shift Register number has to be contained between 1 and 66.
- Head 2 machining on spindle 1 side: The entered Edge Shift Register number has to be contained between 1 and 30.
- Head 2 machining on spindle 2 side: The entered Edge Shift Register number has to be contained between 1 and 30, 30 will be added to this number.
- Head 2 with tool life management on: The entered Edge Shift Register number has to be contained between 1 and 66.
- Head 3: The entered Edge Shift Register number has to be contained between 1 and 66.
- Head 3 with tool life management on: The entered Edge Shift Register number has to be contained between 1 and 66.

Tool wear offset range of values for NZX-S configuration:

- Head 1 machining on spindle 1 side: The entered Edge Shift Register number has to be contained between 1 and 30.
- Head 1 machining on spindle 2 side: The entered Edge Shift Register number has to be contained between 1 and 30, 30 will be added to this number.
- Head 1 with tool life management on: The entered Edge Shift Register number has to be contained between 1 and 66.
- **Head 2 machining on spindle 1 side:** The entered Edge Shift Register number has to be contained **between 1 and 30**.
- **Head 2 machining on spindle 2 side:** The entered Edge Shift Register number has to be contained **between 1 and 30**, 30 will be added to this number.
- Head 2 with tool life management on: The entered Edge Shift Register number has to be contained between 1 and 66.

Tool wear offset range of values for NZXDL configuration:

- Head 1: The entered Edge Shift Register number has to be contained between 1 and 66.
- Head 1 with tool life management on: The entered Edge Shift Register number has to be contained between 1 and 66.
- Head 2: The entered Edge Shift Register number has to be contained between 1 and 66.
- Head 2 with tool life management on: The entered Edge Shift Register number has to be contained between 1 and 66.

Note: If the entered Edge Shift Register number is out of range, you will get the following error message in your NC code: ERROR: WRONG EDGE SHIFT REGISTER NUMBER ENTERED IN ESPRIT.

5.4. Restrictions of 99, 133 or 200 Tool wear offset number (Option on machine)

If your machine has the 99, 133 or 200 tool wear offset number option (200 for NZXDL configuration only), the following restrictions apply.

<u>Regular operations - Tool wear offset range of values for NZX-ST configuration:</u> With the 99 option:

• Head 1: The entered Length Register number has to be contained between 1 and 99.

- Head 2 machining on spindle 1 side: The entered Length Register number has to be contained between 1 and 49.
- Head 2 machining on spindle 2 side: The entered Length Register number has to be contained between 1 and 49, 50 will be added to this number.
- Head 3: The entered Length Register number has to be contained between 1 and 99.

With the 133 option:

• Head 1: The entered Length Register number has to be contained between 1 and 133.

• Head 2 machining on spindle 1 side: The entered Length Register number has to be contained between 1 and 67.

• Head 2 machining on spindle 2 side: The entered Length Register number has to be contained between 1 and 67, 66 will be added to this number.

• Head 3: The entered Length Register number has to be contained between 1 and 133.

<u>Regular operations - Tool wear offset range of values for NZX-S configuration:</u> With the 99 option:

• Head 1 machining on spindle 1 side: The entered Length Register number has to be contained between 1 and 49.

• Head 1 machining on spindle 2 side: The entered Length Register number has to be contained between 1 and 49, 50 will be added to this number.

• Head 2 machining on spindle 1 side: The entered Length Register number has to be contained between 1 and 49.

• Head 2 machining on spindle 2 side: The entered Length Register number has to be contained between 1 and 49, 50 will be added to this number.

With the 133 option:

- Head 1 machining on spindle 1 side: The entered Length Register number has to be contained between 1 and 67.
- Head 1 machining on spindle 2 side: The entered Length Register number has to be contained between 1 and 67, 66 will be added to this number.
- Head 2 machining on spindle 1 side: The entered Length Register number has to be contained between 1 and 67.
- Head 2 machining on spindle 2 side: The entered Length Register number has to be contained between 1 and 67, 66 will be added to this number.

<u>Regular operations - Tool wear offset range of values for NZXDL configuration:</u> With the 99 option:

• Head 1: The entered Length Register number has to be contained between 1 and 99.

• Head 2: The entered Length Register number has to be contained between 1 and 99.

With the 133 option:

• Head 1: The entered Length Register number has to be contained between 1 and 133.

• Head 2: The entered Length Register number has to be contained between 1 and 133.

With the 200 option:

• Head 1: The entered Length Register number has to be contained between 1 and 200.

• Head 2: The entered Length Register number has to be contained between 1 and 200.

Note: If the entered Length Register number is out of range, you will get the following error message in your NC code: ERROR: WRONG LENGTH REGISTER NUMBER ENTERED IN ESPRIT.

With the 133 and 200 options, you now have five digits instead of four output at tool call. So if you are using method 1 to enter your tool and tool wear offset numbers when creating your tools in ESPRIT, make sure to enter a five-digit number. For example, where before you would enter 101 to output in the NC code T0101, you now have to enter 1001 to output T01001.

<u>SolidTurn Grooving with Control Edge Shift - Tool wear offset range of values for NZX-</u> <u>ST configuration:</u>

With the 99 option:

• Head 1: The entered Edge Shift Register number has to be contained between 1 and 99.

• Head 1 with tool life management on: The entered Edge Shift Register number has to be contained between 1 and 99.

• Head 2 machining on spindle 1 side: The entered Edge Shift Register number has to be contained between 1 and 49.

• Head 2 machining on spindle 2 side: The entered Edge Shift Register number has to be contained between 1 and 49, 50 will be added to this number.

• Head 2 with tool life management on: The entered Edge Shift Register number has to be contained between 1 and 99.

• Head 3: The entered Edge Shift Register number has to be contained between 1 and 99.

• Head 3 with tool life management on: The entered Edge Shift Register number has to be contained between 1 and 99.

With the 133 option:

• Head 1: The entered Edge Shift Register number has to be contained between 1 and 133.

• Head 1 with tool life management on: The entered Edge Shift Register number has to be contained between 1 and 133.

• Head 2 machining on spindle 1 side: The entered Edge Shift Register number has to be contained between 1 and 67.

• Head 2 machining on spindle 2 side: The entered Edge Shift Register number has to be contained between 1 and 67, 66 will be added to this number.

• Head 2 with tool life management on: The entered Edge Shift Register number has to be contained between 1 and 133.

• Head 3: The entered Edge Shift Register number has to be contained between 1 and 133.

• Head 3 with tool life management on: The entered Edge Shift Register number has to be contained between 1 and 133.

SolidTurn Grooving with Control Edge Shift - Tool wear offset range of values for NZX-S configuration:

With the 99 option:

• **Head 1 machining on spindle 1 side:** The entered Edge Shift Register number has to be contained **between 1 and 49**.

• Head 1 machining on spindle 2 side: The entered Edge Shift Register number has to be contained between 1 and 49, 50 will be added to this number.

• Head 1 with tool life management on: The entered Edge Shift Register number has to be contained between 1 and 99.

• **Head 2 machining on spindle 1 side:** The entered Edge Shift Register number has to be contained **between 1 and 49**.

• Head 2 machining on spindle 2 side: The entered Edge Shift Register number has to be contained between 1 and 49, 50 will be added to this number.

• Head 2 with tool life management on: The entered Edge Shift Register number has to be contained between 1 and 99.

With the 133 option:

• **Head 1 machining on spindle 1 side:** The entered Edge Shift Register number has to be contained **between 1 and 67**.

• **Head 1 machining on spindle 2 side:** The entered Edge Shift Register number has to be contained **between 1 and 67**, 66 will be added to this number.

• Head 1 with tool life management on: The entered Edge Shift Register number has to be contained between 1 and 133.

• **Head 2 machining on spindle 1 side:** The entered Edge Shift Register number has to be contained **between 1 and 67**.

• **Head 2 machining on spindle 2 side:** The entered Edge Shift Register number has to be contained **between 1 and 67**, 66 will be added to this number.

• Head 2 with tool life management on: The entered Edge Shift Register number has to be contained between 1 and 133.

<u>SolidTurn Grooving with Control Edge Shift - Tool wear offset range of values for</u> <u>NZXDL configuration:</u>

With the 99 option:

• Head 1: The entered Edge Shift Register number has to be contained between 1 and 99.

• Head 1 with tool life management on: The entered Edge Shift Register number has to be contained between 1 and 99.

• Head 2: The entered Edge Shift Register number has to be contained between 1 and 99.

• Head 2 with tool life management on: The entered Edge Shift Register number has to be contained between 1 and 99.

With the 133 option:

• Head 1: The entered Edge Shift Register number has to be contained between 1 and 133.

• Head 1 with tool life management on: The entered Edge Shift Register number has to be contained between 1 and 133.

• Head 2: The entered Edge Shift Register number has to be contained between 1 and 133.

• Head 2 with tool life management on: The entered Edge Shift Register number has to be contained between 1 and 133.

With the 200 option:

• Head 1: The entered Edge Shift Register number has to be contained between 1 and 200.

• Head 1 with tool life management on: The entered Edge Shift Register number has to be contained between 1 and 200.

• Head 2: The entered Edge Shift Register number has to be contained between 1 and 200.

• Head 2 with tool life management on: The entered Edge Shift Register number has to be contained between 1 and 200.

Note: If the entered Edge Shift Register number is out of range, you will get the following error message in your NC code: ERROR: WRONG EDGE SHIFT REGISTER NUMBER ENTERED IN ESPRIT.

6. Spindle direction, speed (S function) and feedrate (F function)

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.

Turning Tools - Turning Insert	Turning Tools - Turning Insert
Turning Tools - Turning Insert X Insert Holder Holder ID Lead Angle (LA) Holder Type Turning Holder Type Turning Holder Properties Relief Angle (RA) Style J - 3 Lead Shank AxB Custom Clearance Angle N 0*	Turning Tools - Turning Insert Insert Insert Holder Common Parameters Tool ID Tool Number 3 Orientation Angle 0.000000 Turret Name Upper Left Turret Station Name Station CW Coolant On Simulation Cut Color
Hand of Tool Right Image: Contract of the second s	Movement XZ,Y Home Home Home Home Movement Home </td

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.

General Tool ID SPOT 04 Lower Tool Number 12 Length Comp Register 12 Coolant Dn Spindle Direction CW Initial Clearance	
Unit Simulation Cut Color Feeds and Speeds Tool Material	

6.2. Spindle speed output for CSS unit

ojec 2 Ol	ct Manager K 🗙 Cancel 🕅 🛪		🖉 Helr
acrici	Uperation Name		
	J		
ar c dh	Tool Selection		
) D	Tool	-?????-	I
uñno	😻 Work Setup		
Ē	Spindle Name	MainSpindle	•
nish	HeadId	Head-1	-
L	Feeds and Speeds		
Istom	Speed RPM, CSS	0	0
5	Feedrate PM, PR	0.000000	0.000000
	Reference Diameter	60.000000	
	Speed Unit	CSS	
	Feed Unit	Per Revolution	▼
	Maximum RPM	5000	
	Spindle Range	Off	
	Spindle Priority	Off	
	Use Feed and Speed KB	No	
	S Comment		
		H	
e	Features 🚺 Tools 🔜	Operations 🔤 🏪	SolidTurn - Grooving

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 is turned 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, Rotary Face Pocketing and Rotary Face Contouring operations with Polar Interpolation set to No or 5-axis operations with 5th axis locked with Z-axis (5-axis operation becomes a 4-axis operation), a rotary feedrate will be computed by the post processor, based on the linear and rotary (C-axis) 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 mm/min (or inch/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 🛛 🛛						
₩ 0	K 🗡 Cancel 🛛 🖄 🗸		🕡 Help			
eral	S Operation Name					
Gen						
-						
×.						
а		-?????-				
Itrate	😵 Work Setup					
0	Spindle Name	MainSpindle				
- l g u o	Head Id	Head-1	•			
	Feeds and Speeds					
Links	Cut Speed RPM, SPM	0	0			
E	XY Feedrate PM PT	0.000000	0.000000			
Custor	Z Feedrate <mark>PM</mark> . PT	0.000000	0.000000			
	Feed Unit	Per Revolution	· · · · · · · · · · · · · · · · · · ·			
	Const. Removal Rate	No				
	Corner Slow Down	No				
	Use Feed and Speed KB	No				
	🛗 Features 🛛 🕅 Tools 🛛 🔚 Operations 🛛 💑 SolidMillTurn - Wrap Pocketing					

6.4. Output of rapid positioning moves (G00) with linear interpolation moves (G01 F)

CAUTION !!! Works with **milling** operations

By default, rapid positioning moves will be output with G00 in the NC code.

If you want to output your rapid positioning moves with linear interpolation (G01 F), enter the desired rapid feedrate value in the Rapid Feedrate (G01 F) (Custom Setting 1) field of the Custom tab of the milling operation page.

Project Manager 🛛							
l	🛩 OK 🗡 Cancel 🖄 🗝 🔞 Help						
	Ieral	S Values					
	Ger	Rapid Feedrate (G01 F)	5000.000000				
l	26	Custom Setting 2	0.000000				
	trate	Custom Setting 3	0.000000				
	S	Custom Setting 4	0.000000				
	Custom Setting 5		0.000000				
	vdvar	Custom Setting 6	0.000000				
	4	Custom Setting 7	0.000000				
	inks	Custom Setting 8	0.000000				
	_	Custom Setting 9	0.000000				
	storr	Tool Nose Detect	0.000000				
	õ						
	Features Tools Operations SolidMillTurn - Contouring						

7. How to output coolant codes

7.1. Introduction

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

For NZX-ST & NZX-S configurations:

Code	Function
M08	Coolant ON
M09	Coolant OFF
M278	Through-spindle coolant ON (spindle 2)
M279	Through-spindle coolant OFF (spindle 2)
M382	Shower coolant ON
M383	Shower coolant OFF
M478	Through-spindle coolant ON (spindle 1)
M479	Through-spindle coolant OFF (spindle 1)
M621	Super high pressure coolant ON
M622	Super high pressure coolant OFF
M651	Chuck top coolant ON (spindle 1)
M652	Chuck top coolant OFF (spindle 1)

For NZXDL configuration:

Code	Function		
M08	Coolant ON		
M09	Coolant OFF		
M382	1382 Shower coolant ON		
M383	Shower coolant OFF		
M478	VI478 Through-spindle coolant ON		
M479	Through-spindle coolant OFF		
M621	Super high pressure coolant ON		
M622	Super high pressure coolant OFF		
M651	Chuck top coolant ON		
M652	Chuck top coolant OFF		

The post processor will handle the output of two different coolant codes per tool.

7.2. Set the first coolant code

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

Turning tool page:

Turning Tools - Turning I	nsert		×		
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 💌	ouch Off Angle	0.000000		
Simulation Cut Color	Off On				
Tool Change	Mist Flood				
Movement X,Z,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		
,					
		OK Ca	ncel Help		

Milling tool page:



Set Coolant to:

- **On** to output M08 (M09)
- **Flood 2 Through Spindle** to output M278 (M279) for NZX-ST & NZX-S configurations only
- **Flood** to output M382 (M383)
- Flood Through Spindle to output M478 (M479)
- **On Through Spindle** to output M621 (M622)
- **Mist** to output M651 (M652)

7.3. Set the second coolant code (optional)

If you desire to output a second coolant code (which is optional), enter the coolant code value in the Second Coolant (Custom Setting 1) field of the Custom tab of the tool page.

Turning tool page:

Turn	ing Tools - Turning Ins	ert			×
In	sert Holder General	Custom			
	Values				
	Second Coolant	382.000000			
	Custom Setting 2	0.000000			
	G30 or G28	0.000000			
	Custom Setting 4	0.000000			
	Custom Setting 5	0.000000			
	Special Coolant ON	0.00000			
	Special Coolant OFF	0.000000			
	Tool Life Management	0.00000			
	Custom Setting 9	0.00000			
	Custom Setting 10	0.000000			
			ОК	Cancel	Help

Milling tool page:

Milling Too	ols - End Mill				x
	Values Second Coolant Custom Setting 2 G30 or G28 Custom Setting 4 Custom Setting 5 Special Coolant ON Special Coolant OFF Tool Life Management Custom Setting 9 Custom Setting 10	8.000000 0.000000 0.000000 0.000000 0.000000 0.000000 0.000000 0.000000 0.000000 0.000000 0.000000 0.000000 0.000000			
ĺ					
			ОК	Cancel	Help

Set Second Coolant to:

- **8** to output M08 (M09)
- 278 to output M278 (M279) for NZX-ST & NZX-S configurations only
- **382** to output M382 (M383)
- **478** to output M478 (M479)
- **621** to output M621 (M622)
- **651** to output M651 (M652)

7.4. Set a special coolant code (optional)

If you desire to output a special coolant code (which is optional, like high pressure coolant code for example), enter the coolant code ON value in the Special Coolant ON (Custom Setting 6) field of the Custom tab of the tool page and the coolant code OFF value in the Special Coolant OFF (Custom Setting 7) field of the Custom tab of the tool

page. Note that both values will need to be entered for the M-codes to be output in the NC code.

The codes will then be output in NC code: special coolant code ON will be output before turning spindle or live tool speed output; special coolant code OFF will be output after turning spindle or live tool stop code (M05).

8. Simultaneous machining (for NZX-ST & NZX-S configurations only)

8.1. Simultaneous turning operations: Turning on the same spindle with 2 different turrets

8.1.1. Balanced 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
heral	🗵 Cutting Strategy		
Ger	Type of Work	OD	
trategy			
S	Balanced Mode	Trailing Tool	
Rough			
g	Trailing Distance	0.000000	
Cus	Rough Pass	Yes	▼
	Finish Pass	No	

Proje	ect Manager		×
🧼 (DK 🛛 🗙 Cancel 🛛 🟦 👻		🕜 Help
ieral	S Cutting Strategy		
Ger	Type of Work	OD	
trategu			
S C	Balanced Mode	Simultaneous	<u> </u>
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 channel 1 for head 1 / head 2 balanced turning operation (or in channel 3 for head 2 / head 3 balanced turning operation for NZX-ST configuration), not in channel 2.

8.1.2. 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.

For not having this problem, 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:

Proj	ect Manager	×					
1	OK 🗙 Cancel 🛛 🔛 👻	🕢 Help					
eral	S Operation Name						
Ger							
B	Tool Selection						
Strate	Tool						
5							
ontor	Vork Setup						
Ŭ	Spindle Name	MainSpindle					
to the	HeadId	Head-1					
Ous	Feeds and Speeds						
	Speed RPM, CSS						
	Feedrate PM, PR	0.000000 0.000000 🛆					
	Reference Diameter	60.000000					
	Speed Unit	CSS					
	Feed Unit	Per Revolution					
	Transverse Feedrate %	100.000000					
	Maximum RPM	5000					
	Spindle Range	Off					
	Spindle Priority	Off 🗾					
	Use Feed and Speed KB	Off Da					
	S Comment	Other Operation					
]						
	🖞 Features 🛛 🚺 Tools 🛛 🗉	🗄 Operations 🛛 🗳 SolidTurn - Contouring					

Set Spindle Priority to:

- Off or On to output the spindle speed
- Other Operation: the other head is controlling the spindle and the spindle speeds are output (by default) as comments (or with a block skip (/) or block skip 2 (/2), see part 4.4.6. Set how to skip turning spindle control).

Note: Make sure, if needed, to activate the block delete function on the control.



8.1.3. Other turning cycles in Balanced Cut Mode CAUTION !!! Do not use with SolidTurn Drilling.

To program simultaneous turning operations in Balanced Cut Mode (G68), you will need to synchronize head 1 & head 2 (or head 2 & head 3 for NZX-ST configuration) prior to the turning operations with the **SimMach** Sync Code. It will trigger the simultaneous turning in Balanced Cut Mode: G68 will be output to turn on Balanced Cut Mode at the beginning of the operations and G69 will be output to turn it off at the end of the operations.

This functionality is typically used to perform pinch grooving operations.

🔴 Sync		🔴 Sync				
		🖊 Tuming I				
		🚅 OD Rou	MainS			
Sync		🔴 Sync				
Groovin		7 Groovin				
SimMach		🔴 SimMach		1		
THOD Groo	MainS	¶불OD Groo	MainS	1		
Sync		🔴 Sync		e		
Cutoff In						
= <mark>₽</mark> Cutoff	MainS					\overline{v}
Features Tools Perations						

To program the SimMach Sync Code, you can in ESPRIT on the Operations tab of the Project Manager create a Sync above your two operations and then double click on it and change the Sync Code to SimMach. After clicking on OK, your Sync will become a SimMach Sync Code.

Project Manager	X
SimMach	Edit Sync
Upper Left Turret	Sync Code
Name S. Spin Nar	Sub Codes < </th
Sync	Lower Turret
- Sinc	
Turning MainS	
Sync	
Sync	
Groovin	Since 2
마음OD Gro Main S 마음(DD Groo Main
Sync Sync Sync Sync Sync Sync Sync Sync	Sync
= [₽] _{1≫} Cutoff MainS	_ _
Features Tools	Operations 1

Or, you can also directly select SimMach from the pull-down on top of the Operations tab of the Project Manager. Once selected, simply program your sync: it will be a SimMach Sync Code.

Project Manager						x
SimMach		▼ 100%	💽 🖬			
< <sync>> SimMach</sync>		Lower T	Turret	Upper Rig	ht Turret	-
Name 9	Spin	Name	S. Spin	Name	S. Spin	
Sime		Suno		Dummy	SubS	
Jync		Pickup	SubS	Joyne		
		≝+Barpull	Main			
Sinc		/ 🖶 Release	SubS	Sinc		
Tuming		U Sync		U Uyrio		
Facing	MainS					
Sync Sync		Sync Tuming I				
		OD Rou	Main			
Sync		Sync				
SimMach	-	SimMach				
T OD Gro	MainS	DD Groo	Main	8		
Sync		Sync				
J Cutoff In	MainS					-
Features	Tools	💽 Operat	ions			,

Note: The SimMach Sync Code will not output an actual wait code. Its only functionality is to trigger simultaneous turning in Balanced Cut Mode.

This mode will be turned off at the next programmed wait code. So if you program multiple synchronized turning operations in a row, make sure to follow this routine to avoid possible machine crash:

SimMach Sync Code followed by Turning operation followed by Regular wait code...

Project Manag	er					×
SimMach		▼ 100%		∓ ∎ 😲		
Upper Lef	t Turret	Lower 1	Furret	Upper Rig	ht Turret	_
Name	S. Spin	Name	S. Spin	Name	S. Spin	
Groovin		Groovin				
SimMach	_	😑 SimMach		1		
📲 Sim. Grv 1	MainS	n면 Sim. Grv 1	MainS			
🔴 Sync		🔴 Sync				
😑 SimMach		😑 SimMach				
📲 Sim. Gr 2	MainS	፹ ^분 Sim. Grv 2	MainS			
😑 Sync		😑 Sync				-
Features	Tool:	s 🔜 Operat	tions			

Note that since two heads are cutting at the same time on the same spindle, **the head with the longest operation must have the control of the spindle speed**. For additional information on how to set the spindle priority, see part **8.1.2. Other turning cycles**.

Notes:

For NZX-ST configuration, the SimMach Sync Code has to be between head 1 & 2 (P12) or head 2 & 3 (P23). If you program the SimMach Sync Code all across head1, 2 & 3 (P123), you will get the following error message in your NC code: ERROR: P12 SYNC EXPECTED (in head1), ERROR: P12 OR P23 SYNC EXPECTED (in head 2) and ERROR: P23 SYNC EXPECTED (in head 3).
If you have an operation type mismatch (milling on one head while turning on the other) below the SimMach Sync Code, you will get the following error message in your NC code: ERROR: OPERATIONS TYPE DIFFERS BETWEEN HEAD 1 AND HEAD 2 (in head 1 for NZX-ST and in head 1 & 2 for NZX-S configurations), ERROR: OPERATIONS TYPE DIFFERS BETWEEN HEAD 1 AND HEAD 2 OR HEAD 2 AND HEAD 3 (in head 2 for NZX-ST configuration only) and ERROR: OPERATIONS TYPE DIFFERS BETWEEN HEAD 1 AND HEAD 3 (in head 3 for NZX-ST configuration only).

8.2. Simultaneous milling operations: Milling on the same spindle with 2 different turrets

CAUTION !!! Use with SolidMillTurn Facing, SolidMillTurn Pocketing, SolidMillTurn Trochoidal Pocketing, SolidMillTurn Contouring, SolidMillTurn Rest Machining, SolidMillTurn Drilling, SolidMillTurn Spiraling, SolidMillTurn Threading, SolidMillTurn Manual Milling or SolidMillTurn Wire Frame Milling.

Simultaneous milling works for C-axis indexing milling operations and not 4-axis wrap or rotary milling operations.

For NZX-ST configuration:

Turret 1 and turret 2 can simultaneously mill on spindle 1 side. In this case, the C-axis is controlled by head 1.

Turret 2 and turret 3 can simultaneously mill on spindle 2 side. In this case, the C-axis is controlled by head 3.

For NZX-S configuration:

Turret 1 and turret 2 can simultaneously mill on spindle 1 or spindle 2 side. The C-axis is controlled by head 1.

To program simultaneous milling in ESPRIT, you will need to synchronize head 1 & head 2 (or head 2 & head 3 for NZX-ST configuration) prior to the 3-axis milling operations with the **SimMach** Sync Code. It will trigger the simultaneous milling mode.

🔴 Sync		🔴 Sync 👘			
💆 12.7mm E		💆 12.7m			
🔴 SimMach		🔴 Sim Mach		1	
🖓 Simultane	MainSpindle	🖓 Simult	MainSpin	1	
🔴 Sync		🔴 Sync		🔴 Sync 👘	
禍 Milling XYZ	MainSpindle			💆 12.7m	
		SimMach		SimMach	
		🖓 Simult	SubSpindle	🖓 Simult	SubSpindle
🔴 Sync		🔴 Sync		🔴 Sync	

To program the SimMach Sync Code, you can in ESPRIT on the Operations tab of the Project Manager create a Sync above your two operations and then double click on it and change the Sync Code to SimMach. After clicking on OK, your Sync will become a SimMach Sync Code.

Sync Sync Sync Sync Sync Sync MainSpindle Sync MainSpindle Milling XYZ MainSpindle	Sync Sync Sync Simult MainSpin Edit Sync X
Sync	Sync Code < <sync>> indle Sub Codes <sync>> 3 e SimMach 3 e Lower Turret</sync></sync>
Sync Cutoff Ins Sync Sync Sync ■ Sync ■ Sync ■ Sutoff MainSpindle	OK 4 ancel
Features 🚺 Tools	Operations 1

Or, you can also directly select SimMach from the pull-down on top of the Operations tab of the Project Manager. Once selected, simply program your sync: it will be a SimMach Sync Code.

Project Manag	er					×
SimMach		100%	- = =	• 🕚		
< <sync>> SimMach</sync>		Lower	Turret	Upper Rig	ght Turret	
Name	Spindle ID	Name	Spindle ID	Name	Spindle ID	
		攝 Threa	MainSpin			
Sync		Sync				
SimMach		SimMach				
Simultane	MainSpindle	6 Simult	MainSpin	-		
 Sync 		Sync	in an opiniti	Sync		
🐴 Milling XYZ	MainSpindle			💋 12.7m		
		SimMach		SimMach		
		禮 Simult	SubSpindle	砲 Simult	SubSpindle	
Sync		Sync Zila Milling	MainSpin	Sync 28 Million	SubSpindle	
		Sunc	Mainspin	Per Milling	Subspindle	
		袖 Milling	SubSpindle	• 0,110		
		Sync		🔴 Sync 👘		
		🕂 Part ej	SubSpindle			-
Features	Tools	🔝 Operatio	ns			

Note: The SimMach Sync Code will not output an actual wait code. Its only functionality is to trigger simultaneous milling mode.

This mode will be turned off at the next programmed wait code. So if you program multiple synchronized milling operations in a row, make sure to follow this routine to avoid possible machine crash:

SimMach Sync Code followed by Milling operation followed by Regular wait code...

Upper Left Turret		Lower	ver Turret 🔰 Upper Righ		ght Turret	
Name	Spindle ID	Name	Spindle ID	Name	Spindle ID	
12.7mm E		💆 12.7m				1
🔴 SimMach		🔴 Sim Mach		1		
袎 Sim. Mill. 1	MainSpindle	🆄 Sim. M	MainSpin			
🔴 Sync		🔴 Sync				
😑 SimMach		SimMach				
裆 Sim. Mill. 2	MainSpindle	🆓 Sim. M	MainSpin			
🔴 Sync		🔴 Sync				

Notes:

• For NZX-ST configuration, the SimMach Sync Code has to be between head 1 & 2 (P12) or head 2 & 3 (P23). If you program the SimMach Sync Code all across head1, 2 & 3 (P123), you will get the following error message in your NC code: ERROR: P12 SYNC EXPECTED (in head1), ERROR: P12 OR P23 SYNC EXPECTED (in head 2) and ERROR: P23 SYNC EXPECTED (in head 3).

• If you have an operation type mismatch (milling on one head while turning on the other) below the SimMach Sync Code, you will get the following error message

in your NC code: ERROR: OPERATIONS TYPE DIFFERS BETWEEN HEAD 1 AND HEAD 2 (in head 1 for NZX-ST and in head 1 & 2 for NZX-S configurations), ERROR: OPERATIONS TYPE DIFFERS BETWEEN HEAD 1 AND HEAD 2 OR HEAD 2 AND HEAD 3 (in head 2 for NZX-ST configuration only) and ERROR: OPERATIONS TYPE DIFFERS BETWEEN HEAD 2 AND HEAD 3 (in head 3 for NZX-ST configuration only).

9. Operation synchronization

Operation synchronization M-codes start at M101 and can incrementally reach M197. In the NC code, when M197 is reached or exceeded, the next output wait code will be output in the (101; 197) range: M197 will never be exceeded.

10. Park cycle

10.1. Introduction

With the Park cycle of ESPRIT, you will have different possibilities to park a tool located on head 1, 2 or 3 (for NZX-ST configuration). It is a great way to park a tool above a spindle to a clearance position, so you can freely machine on the same spindle with another turret. You can also send a tool home along the X, Y (Y-axis specifications only) and/or Z axis. You finally can park a turret to its maximal positions (left or right) or anywhere in-between using the machine work coordinate system (G53). You can for example, for NZX-ST configuration, park the upper left turret all the way to the left and park the upper right turret on its maximal left position: upper right and lower turret are aligned in the middle.

10.2. 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							
🖌 🗸)K 🗙 Cancel 🖞			🕜 Help			
reral	Park Position						
<u>G</u>	Park Position X	None	0.000000				
ž	Park Position Y	None	0.000000	R			
Ъ	Park Position Z	Machine	0.000000	R			
stom	Stop Code	Position		•			
Ő	😵 Rotary Axes						
	Park Axis 1	No		•			
	Park Axis 2	No		•			
	Features	ols 🔛 Operations	2 Solid Tur	n - 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 **5.1.7. Second home position** for additional information. This is the default when creating a new tool.

• 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	R		
Park Position Y	None	Ŧ	0.000000	ß		
Park Position Z	None	Ŧ	0.000000	R		
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).

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

Note: The X; Y & Z positions entered in ESPRIT (for Machine and Position modes) are absolute in YZX. So the post processor will compute and output the correct values from this absolute position based on the turret and the spindle 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.

Projec	Project Manager								
🛩 O	🛩 OK 🗡 Cancel 📳 🗝 🕜 Help								
Park Position									
Ger	Park Position X	Home 🔻	0.000000	R					
ž	Park Position Y	Home 💌	0.000000						
Pa	Park Position Z	Home 🔻	0.000000	L3					
E	Return Mode	None		-					
Cus	Stop Code	None X First							
	Park Axis 1	X Then Y X Then Z							
	Park Axis 2	Y Then X Y Then Z							
	Features Too	Z Then X Is Z Then Y							

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, any Y-axis movement programmed with the Park cycle will be ignored.

10.3. Park a turret to its maximal travel position (G53)

You will need to set Position with G53 (Custom Setting 1) of the Park operation to 53 or -53 to enable the output of the G53 code.

If you set Position with G53 to -53, the active turret will be parked to its left maximal travel position.

Proje	Project Manager					
₩ 0	🛩 OK 🗙 Cancel 🛛 🛃 👻					
heral	© Settings					
ge	Position with G53	-53.000000				
×	Output M09	0.000000				
L P	CustomSetting3	0.000000				
ε	CustomSetting4	0.000000				
Custo	CustomSetting5	0.000000				
Sec.	CustomSetting6	0.000000				

Set Position with G53 to 53 to park it to its right maximal travel position.

Project Manager 🛛 🛛 🖄						
🥪 OI	👽 OK 🗙 Cancel 🛛 🚽 🔹 🔞 Help					
neral	ि Settings					
Gei	Position with G53	53.000000				
×	Output M09	0.000000				
La La	CustomSetting3	0.000000				
E.	CustomSetting4	0.000000				
Custo	CustomSetting5	0.000000				
-	CustomSetting6	0.000000				

Turrets maximum travel position values (in the machine coordinate system G53) for

NZX-ST configuration: *Upper left turret T1:* Left position: -295 mm Right position: 95 mm *Lower turret T2:* Left position: -400 mm Right position: 400 mm *Upper right turret T3:* Left position: -95 mm Right position: 295 mm Turrets maximum travel position values (in the machine coordinate system G53) for NZX-S configuration: *Upper turret T1:* Left position: -400 mm Right position: 400 mm *Lower turret T2:* Left position: -400 mm Right position: 400 mm Turrets maximum travel position values (in the machine coordinate system G53) for NZXDL configuration: *Upper left turret T1:* Left position: -280 mm Right position: 0 mm *Upper right turret T2:*

Note that if the tool you are parking is the same as the previous tool in use, the turret will be sent home in Y (Y-axis specifications only) then X before the G53 output as a safety.

Left position: 0 mm Right position: 280 mm

10.4. Park a turret prior to a transfer

By default, for all turrets, tool station 1 (or 31 when needed for NZX-ST & NZX-S configurations only) will be called prior to a transfer. As seen in part **4.4.4. Tool station for part transfer and work unloader**, this tool station can be changed using the Station for Transfer textboxes on the POST Output Configuration.

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

You will first need to create a dummy tool on the turret you need to park, in the right station. Set the Tool Number to 1 (or whatever you set up in the Machine Setup) and the Length Comp Register to 0, or simply set the Tool Number to 100 (or whatever you set up in the Machine Setup time 100) and the Length Comp Register to the same number.



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 (or -53, see part **10.3. Park a turret to its maximal travel position (G53)**) on the Park operation Custom tab. The turret will be parked using G53.

😑 Sync		O Sync			
		💋 DUM		1	
		🛃 Park	MainS		
🔴 Sync	•	🗢 Sync		🔴 Sync 🛛	
7 Cutoff I		뵭 🖥 Pickup	SubSpi		
🔴 Sync		🔴 Sync 👘			
= <mark>≞</mark> Cutoff	MainSpin				

Note: If you omit to enter 53 in the Position with G53 field of the Park operation, you will get the following error message in your NC code: ERROR: NO TOOL WEAR OFFSET: PARK WITH POSITION (USING G00) CANNOT BE USED.

10.5. Other functionalities

If you set Output M09 (Custom Setting 2) to 1 on the Custom tab of the Park operation, the coolants will be stopped.

Proje	Project Manager 🛛 🛛 🖄					
🛩 OK 🗡 Cancel 🛛 🚽 🔹 🕜 He						
heral	Settings					
Ger	Position with G53	0.000000				
×	Output M09	1.000000				
Ъ.	CustomSetting3	0.000000				
Ę.	CustomSetting4	0.000000				
Custo	CustomSetting5	0.000000				
The second	CustomSetting6	0.000000				

On the Park tab of the operation, if you set Stop Code to Optional Stop or Stop, a Spindle Rotation Stop Code (M05) will be output.

Project Manager								
🛩 O	🛩 OK 🗙 Cancel 🖄 ▾ 🔞 Hel							
Ieral	Park Position							
Ger	Park Position X	None	•	0.000000				
ž	Park Position Y	None	•	0.000000				
Pa	Park Position Z	None	•	0.000000	L3			
E	Stop Code	Optional St	ор		-			
ISI		None						
	😵 Rotary Axes	Optional Sto	ор					
	Park Axis 1	Stop						
	Park Axis 2	No			•			
Features Tools E Operations Solid Turn - Park								

11. How to program finished part catching and part transfer

11.1. Introduction

In this following part, you will be given detailed instructions on how to manually program various catching of finished part and various part transfers in ESPRIT. A **correct program in ESPRIT is necessary to have a correct NC code.** Please note that the following part catching and part transfer scenarios can be programmed automatically using the Workpiece Transfer add-in.

11.2. Finished part catching

11.2.1. Catching finished part on spindle 1 side after cut-off

This type of part release comes at the end of the operation list. It can be performed if your machine is equipped with a receiver on the spindle 1 side.

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*
5	SolidTurn Machine Setup					
	General Assembly NC Output Custom					
1	Machine Definition					
		Machine Name		NZ2000T3Y3		
		Start F	Position Z	1	-	
		Start Position X		0		
		Start F	Position Y	0		

The regular steps to follow are:

- 1. Machining the part on the spindle 1 side
- 2. Perform a cut-off operation

Steps to program this part ejection type in ESPRIT:

- 1. Machining the part on the spindle 1 side.
- 2. Cut-off:

Use operation **Cutoff** using **Upper** (**Left** for NZX-ST & NZXDL configurations) **Turret** tool on **MainSpindle** (**Spindle Name**).

Cut-off tool is always loaded in the upper left turret for Mori Seiki NZX-ST & NZXDL configurations. The post processors handle cut-off operation programmed in upper left turret only for these configurations. It is always loaded in the upper turret for Mori Seiki NZX-S configuration. The post processors handle cut-off operation programmed in upper turret only for this configuration.

To trigger the part catching output, set Use Part Catcher to Yes on the Strategy tab.

Proje	ct Manager	×
🛩 C	Ж 🗙 Cancel 🏦 🗸	🕜 Help
ieral	S Cutting Strategy	
Ger	Rough Pass	Yes
eg	Finish Pass	No
Strat	😵 Rapid Approach/Exit	
hgu	Entry Mode	None
Bo	Exit Mode	None
stom	Sharp Edges	
5	Tool Blend	No
	S Machine Functions	
	Use Part Catcher	Yes
	Length Comp Register	0
	Canned Cycle	No
	Features 🛛 🛄 Tools 🛛 🔛 Operati	ons SolidTurn - Cutoff

Sample operation list for NZX-ST configuration:

Project Manag	jer					×
< <sync>></sync>		▼ 100%	_ ±	7		
Upper Le	eft Turret	Lower	Turret	Upper Right Turret		*
Name	Spindle ID	Name	Spindle	Name	Spindle	
🖉 Turning I						
🐔 Facing	MainSpindle					
🔴 Sync		😑 Sync				
		💋 Turnin				
		🛫 OD Ro	MainSpi			
Sync		🔴 Sync				
Cutoff Ins		1				
📲 🖓 Cutoff	MainSpindle					
P		e				Ţ.
<u> </u>				I		
E Features	Tools	🔣 Operati	ions			

Sample operation list for NZX-S configuration:

Project Manag	jer			×
SimMill		100%	💌 😫 🛼	•
Upper	Turret	Lower	Turret	
Name	Spindle ID	Name	Spindle ID	
🖉 Turning I				
🕰 Facing	MainSpindle			
🔴 Sync		🔴 Sync		
		📿 Turning I		
		🛫 OD Rou	MainSpindle	
😑 Svnc		🔴 Sync		
Cutoff In		1		
= <mark>≓</mark> a Cutoff	MainSpindle			-
📇 Features	🚺 Tools	🖪 Operation	ns	

Sample operation list for NZXDL configuration:

Project Manager 🗾						
< <sync>></sync>		T 100	% 🔽 🖆	•		
Upper Le	eft Turret	Upper Right Turret		~		
Name	Spindle ID	Name	Spindle ID			
Tuming						
🕰 Facing	MainSpindle					
COD Ro	MainSpindle					
Cutoff I		1				
= <mark>¤</mark> Cutoff	MainSpindle			\overline{v}		
Features Tools Decrations						

11.2.2. Catching finished part on spindle 1 side after cut-off with stock repositioning (programmed first)

If your machine is equipped with a barfeeder, you can reposition the stock, machine the part, catch the finished part on spindle 1 side and finally loop for the next part.

Note: On the General tab of the Machine Setup, set the Start Position Z: Start Position Z = - (Length of the finished part (Part Stock Length) + Cut-off tool width + Length of stock for facing the back side of the part (if needed))

Solie	SolidTurn Machine Setup					
G	General Assembly NC Output Custom					
	-Mach Mach	ine Definition- ine Name	NZ2000T3Y3			
	Start I	Position Z	-103			
	Start I	Position X	0			
	Start I	Position Y	0			

The steps to follow are the same as previous point. The only difference is that you need to program a barfeed (by stopper) operation first:

- 1. Perform a barfeed operation
- 2. Machining the part on the spindle 1 side
- 3. Perform a cut-off operation
- 4. Looping for next parts

Steps to program this part ejection type in ESPRIT:

1. Perform a barfeed operation:

Use operation **Bar Feed By Stopper** (**Bar Feed Type**) using **Upper** (**Left** for NZX-ST & NZXDL configurations) **Turret** tool on **MainSpindle** (**Spindle Name**).

On Bar Feed tab, set the Feed Length and Reposition Distance: Feed Length (=Barfeed distance) = Length of the finished part (Part Stock Length) + Cut-off tool width + Length of stock for facing the front side and, if needed, the back side of the part Reposition Distance = 0

With Position X, Y, Z, set the barfeed reference point. Its Z coordinate value should be equal to the length of stock for facing the front side of the part (if needed).

😵 Bar Feed	≫ BarFeed				
Position X, Y, Z	0.000000 0.000000 1.000000 📘				
Feed Length	104.000000				
Reposition Distance	0.000000				
Bar Diameter	100.000000				
Total Bar Length	300.000000				
Length Comp Register	0				
Dwell Time	0.000000				

The stopper tool will either be positioned in the G53 work coordinate or the current machine work coordinate.

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

Proje	ct Manager	×
🖌 🗸	K 🗡 Cancel 🛛 🛃 👻	🕡 Help
teral	Settings	
Ger	CustomSetting1	0.000000
l p	CustomSetting2	0.000000
r Fe	CustomSetting3	0.000000
ä	Stopper G53X	-50.000000
щ.	Stopper G53 Z	-100.000000
Crist	CustomSetting6	0.000000
Sec.	CustomSetting7	0.000000
	CustomSetting8	0.000000
	CustomSetting9	0.000000
	CustomSetting10	0.000000
	Features 🚺 Tools	Dperations

To use current machine work coordinate defined in ESPRIT (see part 2.3. Turning Work Coordinates): If no value is specified on the Custom tab in the Stopper G53 X and Stopper G53 Z fields (fields left equal to 0), the stopper tool will be positioned at the programmed position (Position X, Y, Z specified on Bar Feed tab) in the current active machine work coordinate.

Note that using End of Bar Macro Call (/2 M98 P) on the POST Output Configuration, you can output the end of bar macro call (/2 M98 Pxxxx); see part **4.4.2. Output of the end of bar macro call (/2 M98 Pxxxx).**

- 2. Machining the part on the spindle 1 side.
- 3. Cut-off:

Use operation **Cutoff** using **Upper** (**Left** for NZX-ST & NZXDL configurations) **Turret** tool on **MainSpindle** (**Spindle Name**).

Cut-off tool is always loaded in the upper left turret for Mori Seiki NZX-ST & NZXDL configurations. The post processors handle cut-off operation

programmed in upper left turret only for these configurations. It is always loaded in the upper turret for Mori Seiki NZX-S configuration. The post processors handle cut-off operation programmed in upper turret only for this configuration.

To trigger the part catching output, set Use Part Catcher to Yes on the Strategy tab.

Proje	Project Manager 🛛 🛛 🔤					
₩ 0)K 🗙 Cancel 🛛 🟦 👻	🕜 Help				
teral	S Cutting Strategy					
Ger	Rough Pass	Yes 💌				
teg	Finish Pass	No				
Stral	😵 Rapid Approach/Exit					
Чĝи	Entry Mode	None				
B	Exit Mode	None				
stom	😵 Sharp Edges					
13	Tool Blend	No				
	S Machine Functions					
	Use Part Catcher	Yes				
	Length Comp Register	0				
	Canned Cycle	No				
		ons Solia l urn - Cutoff				

Sample operation list for NZX-ST configuration:

Project Manager						×
< <sync>></sync>		▼ 100% ▼	+ -	•		
Upper Left Tu	urret	Lower Tum	et	Upper Right T	urret	
Name	Spindl	Name	Spindl	Name	Spindl	
BAR STOPP		1				
at Har Feed By	MainS					
Turning Insert						
Facing	MainS					
Sync		😑 Sync				
		Turning Insert				
		20D Roughing	MainSp			
Sync 🗧		😑 Sync				
Cutoff Insert		1				
=to Cutoff	MainS					~
Features	Tools	Operations				

Sample operation list for NZX-S configuration:

Project Manager				
< <sync>></sync>	▼ 100	% 💽 🚼 🌄 🕚		
Upper Turret	t	Lower Turre	t	-
Name	Spindle ID	Name	Spindle ID	
BAR STOPPER Z+		1		
Har Feed By Stopper	MainSpindle			
Turning Insert OD 3V		e		
🕰 Facing	MainSpindle			
😑 Sync		🔴 Sync		
		Turning Insert OD 2V		
		🛫 OD Roughing	MainSpindle	
Sync		😑 Sync		
Cutoff Insert 3V		1		
=ros Cutoff	MainSpindle			Ŧ
Features 📑 Too	ols 🔃 Oper	ations		

Sample operation list for NZXDL configuration:

Project Manager 🛛 💽					
< <sync>></sync>		▼ 100%	· 🔽 🖆	•	
Upper Le	Upper Left Turret		Upper Right Turret		
Name	Spindle ID	Name	Spindle ID		
BAR S		1			
≝+Bar Fee	MainSpindle			=	
Turning		e		-	
🕰 Facing	MainSpindle				
OD Ro	MainSpindle				
Cutoff I		1			
= <mark>¤</mark> Cutoff	MainSpindle			Ŧ	
Feature	s 🚺 Tools	s 🔃 Opera	ations		

11.2.3. Catching finished part on spindle 1 side after cut-off with stock repositioning (programmed last)

If your machine is equipped with a barfeeder, you can after catching the finished part on spindle 1 side reposition the stock. This way, you can loop for the next part.

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*

S	SolidTurn Machine Setup				
[General		Assembly	NC Out	put Custom
	Machine Definition				
		Machine Name		ſ	NZ2000T3Y3
		Start Position Z		[1
		Start Position X			D
		Start F	Position Y	[0

The steps to follow are the same as previous point. The only difference is that you need to program a barfeed (by stopper) operation last:

- 1. Machining the part on the spindle 1 side
- 2. Perform a cut-off operation
- 3. Perform a barfeed operation
- 4. Looping for next parts

Steps to program this part ejection type in ESPRIT:

- 1. Machining the part on the spindle 1 side.
- 2. Cut-off: Use operation **Cutoff** using **Upper** (**Left** for NZX-ST & NZXDL configurations)

Turret tool on MainSpindle (Spindle Name).

Cut-off tool is always loaded in the upper left turret for Mori Seiki NZX-ST & NZXDL configurations. The post processors handle cut-off operation programmed in upper left turret only for these configurations. It is always loaded in the upper turret for Mori Seiki NZX-S configuration. The post processors handle cut-off operation programmed in upper turret only for this

post processors handle cut-off operation programmed in upper turret only for this configuration.

To trigger the part catching output, set Use Part Catcher to Yes on the Strategy tab.

Proje	Project Manager 🛛 🛛 🛛					
₩ 0	🛩 OK 🗙 Cancel 💼 👻					
leral	Strategy					
Ger	Rough Pass	Yes				
(eg)	Finish Pass	No				
Strat	😵 Rapid Approach/Exit					
hgu	Entry Mode	None				
Ba	Exit Mode	None				
stom	😵 Sharp Edges					
6	Tool Blend	No				
	S Machine Functions					
	Use Part Catcher	Yes				
	Length Comp Register	0				
	Canned Cycle	No				
	Features 🚺 Tools 🖪 Operatio	ns 🔤 📲 SolidTurn - Cutoff				

3. Perform a barfeed operation:

Use operation **Bar Feed By Stopper** (**Bar Feed Type**) using **Upper** (**Left** for NZX-ST & NZXDL configurations) **Turret** tool on **MainSpindle** (**Spindle Name**).

On Bar Feed tab, set the Feed Length and Reposition Distance: Feed Length (=Barfeed distance) = Length of the finished part (Part Stock Length) + Cut-off tool width + Length of stock for facing the front side and, if needed, the back side of the part Reposition Distance = 0 With Position X, Y, Z, set the barfeed reference point. Its Z coordinate value should be equal to the length of stock for facing the front side of the next part (if needed).

😸 Bar Feed	≫ Bar Feed			
Position X, Y, Z	0.000000 0.000000 1.000000 📘			
Feed Length	104.000000			
Reposition Distance	0.000000			
Bar Diameter	100.000000			
Total Bar Length	300.000000			
Length Comp Register	0			
Dwell Time	0.000000			

The stopper tool will either be positioned in the G53 work coordinate or the current machine work coordinate.

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

Project Manager 🛛 🕹						
🥪 OF	🛩 OK 🗙 Cancel 🛛 🛃 👻 🔞 Help					
teral	Settings					
Ger	CustomSetting1	0.000000				
ba	CustomSetting2	0.000000				
ar Fe	CustomSetting3	0.000000				
B.	Stopper G53X	-50.000000				
tom.	Stopper G53 Z	-100.000000				
Cus	CustomSetting6	0.000000				
Sec. 1	CustomSetting7	0.000000				
	CustomSetting8	0.000000				
	CustomSetting9	0.000000				
	CustomSetting10	0.000000				
	Features 🚺 Tools	Dperations				

To use current machine work coordinate defined in ESPRIT (see part 2.3. Turning Work Coordinates): If no value is specified on the Custom tab in the Stopper G53 X and Stopper G53 Z fields (fields left equal to 0), the stopper tool will be positioned at the programmed position (Position X, Y, Z specified on Bar Feed tab) in the current active machine work coordinate.

Note that using End of Bar Macro Call (/2 M98 P) on the POST Output Configuration, you can output the end of bar macro call (/2 M98 Pxxxx); see part **4.4.2. Output of the end of bar macro call (/2 M98 Pxxxx).**

Sample operation list for NZX-ST configuration:

Project Mana	Project Manager 🛛					
< <sync>></sync>		100%	. ₹			
Upper L	eft Turret	Lower	Turret	Upper Ri	ght Turret	*
Name	Spindle ID	Name	Spindle	Name	Spindle	
🖉 Turning I						
🕰 Facing	MainSpindle					
Sync		🔴 Sync 👘				
		💋 Turnin				
		耄 OD R	MainSpin			
🔴 Sync		🔴 Sync				
Cutoff In		1				
= 🖧 Cutoff	MainSpindle					
BAR ST						
∰ +Bar Feed	MainSpindle					$\overline{\mathbf{v}}$
Features	📑 Tools	📰 Opera	tions			

Sample operation list for NZX-S configuration:

Project Manag	ger			×
SimMill		100%	💌 😫 🎫	•
Upper	Turret	Lower	Turret	
Name	Spindle ID	Name	Spindle ID	
💋 Turning I				
🕰 Facing	MainSpindle			
🔴 Sync		🔴 Sync		
		📿 Turning I		
		🛫 OD Rou	MainSpindle	
😑 Sync		😑 Sync		
ל Cutoff In		1		
= 🖧 Cutoff	MainSpindle			
BAR ST				
Harfeed	MainSpindle	1		$\overline{\mathbf{v}}$
📇 Features	[🖁 Tools	🖪 Operation	ns	

Sample operation list for NZXDL configuration:

Project Manager 🛛 🔂				
< <sync>></sync>		▼ 100	% 🔽 🖆	• =
Upper Le	ft Turret	Upper Rig	ght Turret	
Name	Spindle ID	Name	Spindle ID	
🐉 Wrap D	MainSpindle			
EM 06				
🐉 Wrap D	MainSpindle			
Cutoff I		1		
= ^B _{L≫} Cutoff	MainSpindle			-
BAR S				
🚔 + Bar Fee	MainSpindle			-
Features Tools Operations				

11.2.4. Manually catching finished part on spindle 1 side (for NZXDL configuration only)

This type of part release is performed when the part is completely machined on spindle 1 side. To be able to catch the finished part, head 1 needs to be stopped with M00.

The regular steps to follow are:

- 1. Machining the part on the spindle 1 side
- 2. Manually catching the finished part in the spindle 1

Steps to program this part ejection type in ESPRIT:

- 1. Machining the part on the spindle 1 side.
- 2. Manually catching the finished part in the spindle 1: To be able to manually catch the finished part in the spindle 1, head 1 needs to be stopped. To do so, the M-code M00 (program stop) needs to be output in head 1.

Use operation **Release** with **MainSpindle** (**Spindle Name**) on **Upper Left Turret** (**Turret Name**).

To output the stop code M00 in head 1, set Part Chute to No on the Release tab.

😵 Release	
Position X, Y, Z	-150.0000 0.00000 600.00000
Part Chute	No
Dwell Time	0.000000

Sample operation list of manual finished part catching:

Project Manager 🗾					
< <sync>></sync>	•	100% 💌 🕇	* =• (!	$-\ $	
Upper Le	ft Turret	Upper Rig	ht Turret	~	
Name	Spindle ID	Name	Spindle ID		
Turning Inse		🕖 Turning Ins			
Kacing	MainSpindle	🕰 Facing Sub	SubSpindle		
OD Roughin	MainSpindle	🚽 OD Roughi	SubSpindle		
/🗄 Part Eject	MainSpindle	1			
Sync		🔴 Sync			
Pickup	MainSpindle				
Sync		🔴 Sync			
		🕂 🔁 Release	SubSpindle	Ŧ	
Features	📲 Tools	Operations			

11.2.5. Catching finished part on spindle 2 side

This type of part release is performed when the part is completely machined on spindle 2 side.

The regular steps to follow are:

- 1. Machining the part on the spindle 2 side
- 2. Catching the finished part in the spindle 2

Steps to program this part ejection type in ESPRIT:

1. Machining the part on the spindle 2 side:

Note: The operations on the spindle 2 side are output with the origin shifted to the other face of the part (back face). During operations, only positive Z coordinates are output.

The setting "Part Stock Length" from the Machine Setup is used as default value for Turning Work Coordinates Sub Spindle Z Offset.

CAUTION !!! Part Stock Length = Finished Part Length. You must run the Turning Work Coordinates add-in prior to posting NC Code in order to have correct NC output.

2. For NZX-ST configuration:

Program a wait code between head 2 and head 3 (P23 sync code) or across the three heads (P123 sync code): A sync across the three heads is necessary if you want to manually catch the

A sync across the three heads is necessary if you want to manually catch the finished part in the spindle 2 (machine is stopped with M00).

Note: For manual part catching, if you do not program a P123 sync above your part release, you will get the following error message in your NC code: ERROR: PROGRAM A P123 SYNC ABOVE PART EJECT OPERATION TO

OUTPUT M00.

For NZX-S configuration:

To be able to manually catch the finished part in the spindle 2 (machine is stopped with M00), it is necessary to program a wait code across the two heads.

Note: For manual part catching, if you do not program a sync above your part release, you will get the following error message in your NC code (in head 2): ERROR: PROGRAM A SYNC ABOVE PART EJECT OPERATION TO OUTPUT M00.

3. Catching the finished part in the spindle 2:

Use operation **Release** with **SubSpindle** (**Spindle Name**) on **Lower** (**Upper Right** for NZXDL configuration) **Turret** (**Turret Name**).

The release operation has to be programmed with lower (upper right for NZXDL configuration) turret because it controls the B-axis (spindle 2).

Based on the setting set for Work Unloader Type on the POST Output Configuration (see part **4.4.3. Work unloader on spindle 2 side: hand or receiver**), the NC output will differ to match your machine specifications: hand or receiver equipped.

Catching the finished part in the spindle 2 on a machine equipped with a hand or a receiver:

To trigger the part catching output, set Part Chute to Yes on the Release tab.

😻 Release		
Position X, Y, Z	150.0000 0.000000 600.00000	R
Part Chute	Yes	•
Dwell Time	0.000000	

In the Catch Position (Custom Setting 8) field of the SolidTurn Release operation page, enter the part catching position. The spindle 2 will rapid to this position. Note that you can enter a positive or negative value, the output position will always be negative since it is output in the machine work coordinate (G53).

On a machine equipped with a receiver, with Ejector Type (Custom Setting 9), you can specify how you want to eject the finished part: set Ejector Type to 47 if you have the workpiece ejector OUT (M47) **option on machine**, to 360 if you have the workpiece ejector OUT (M360), IN (M361) **option on machine**. If you omit to enter a value in the Ejector Type field or enter a wrong value, no part ejection code will be output in the NC code.

Project Manager 🛛 🛛 🔀					
₩ 0	K 🟋 Cancel 🛛 🛃 👻	🕢 Help			
ease	Settings				
Relé	CustomSetting1	0.000000			
ε	CustomSetting2	0.000000			
Custo	CustomSetting3	0.000000			
Р	CustomSetting4	0.000000			
	CustomSetting5	0.000000			
	CustomSetting6	0.000000			
	CustomSetting7	0.000000			
	Catch Position	-50.000000			
	Ejector Type	360.000000			
	CustomSetting10	0.000000			
	Features 🚺 Tools 🗄	Operations BolidTurn - Release			

Manually catching the finished part in the spindle 2 (machine is stopped with M00 for NZX-ST & NZX-S configurations and head 2 is stopped with M00 for NZXDL configuration):

To be able to manually catch the finished part in the spindle 2 for NZX-ST & NZX-S configurations, the machine needs to be stopped. To do so, the M-code M00 (program stop) needs to be output in all heads after the same sync code across all heads (this is the reason why a sync code across all heads needs to be programmed prior to the part release operation).

To be able to manually catch the finished part in the spindle 2 for NZXDL configuration, head 2 needs to be stopped. To do so, the M-code M00 (program stop) needs to be output in head 2.

😵 Release	
Position X, Y, Z	-150.0000 0.000000 600.00000
Part Chute	No
Dwell Time	0.000000

To output the stop code M00, set Part Chute to No on the Release tab.

4. In head 3 for NZX-ST configuration, to avoid possible collisions, no operation should be programmed after the P23 (or P123) wait code until the next programmed wait code.

Sample operation list of finished part catching with arm or receiver for NZX-ST configuration:

Project Mana	iger					×
< <sync>></sync>		100%	💌 昔 🖡	• 🕚		
Upper Le	eft Turret	Lower	Turret	Upper Rig	ght Turret	
Name	Spindle ID	Name	Spindle ID	Name	Spindle ID	
😑 Sync		😑 Sync		표 ^표 Grooving	SubSpindle	
		12.7mm				
		💫 Rotary F	MainSpindle	-		
		🔴 Sync		🔴 Sync		
		/랲 Part Eject	SubSpindle			
Sync		Sync		1		
12.7mm						
Kotary F	MainSpindle					
2/mm In	M 1 0 1 1					
osa Drilling	Mainspindle	Come				
U Sync		Sync				
		- グ Furning	MainSpindle			
O Supp			Mainspinule	Cupo		
Cutoff In		Pickup	SubSpindle	Jync		
I Caton man			MainSpindle			1 1
Sunc		Sunc	in all to pindlo			
= Cutoff	MainSpindle	J				T
E Features		Uperat	ions			

Sample operation list of manual part catching for NZX-ST configuration:

Project Mana	iger					×
< <sync>></sync>		100%	- 🛨 🗄	• 🕚		
Upper Le	eft Turret	Lower	Turret	Upper Rig	ght Turret	
Name	Spindle ID	Name	Spindle ID	Name	Spindle ID	
🔴 Sync		🔴 Sync		📲 Grooving	SubSpindle	
		💆 12.7mm				
		🔊 Rotary F	MainSpindle			
😑 Sync		😑 Sync		😑 Sync		
-		/記 Part Eject	SubSpindle			
Sync		🔴 Sync		1		
12.7mm						
Rotary F	MainSpindle					
2/mm in	M 1 0 1 1					
• Drilling العني الم	MainSpindle	A Come				
U Sync						
		Z / Turning ≂Ø Bouchin	MainSpindle			
Supe		Supe	Mainspindle	Sunc		
5 Cutoff In		Pickup	SubSpindle	U Uyric		
1 Color III		Barpull	MainSpindle			
Sync		Svnc				
= <mark>ª</mark> Cutoff	MainSpindle					•
E Features	s 🚺 Tools	🔣 Operat	ions			

Sample operation list of finished part catching with arm or receiver for NZX-S configuration:

Project Manager 🗾 🛛					
SimMill		100%	- = -	•	
Upper	Turret	Lower	Turret		
Name	Spindle ID	Name	Spindle ID		
📲 Grooving	MainSpindle	ר Grooving			
况 DR 04 X+		📲 Grooving	SubSpindle		
🐉 Wrap Dril	MainSpindle	😫 TD M04			
🔴 Sync		🔴 Sync			
况 DR 20 Z-		🐉 Wrap Dri	MainSpindle	1	
🚱 🖣 Drilling	SubSpindle				
况 DR 04 X+					
🐉 Wrap Dril	SubSpindle			1	
Bore Bar				1	
🐉 Wrap Dril	SubSpindle				
🛟 TD M05					
🐉 Wrap Dril	SubSpindle				
Sync		🔴 Sync			
		7 Grooving			
		🖬 🖞 Grooving	SubSpindle		
		🐉 TD M05			
		🐉 Wrap Dri	SubSpindle		
		🔁 Part Eject	SubSpindle		
		况 DR 18 L			
		🚱 ID Drillin	MainSpindle	-	
Features	Tools	🖪 Operation	ns		

Sample operation list of manual part catching for NZX-S configuration:

Project Mana	ger			×
< <sync>></sync>		100%	- 🗄 🗧	• 🤇
Upper	Upper Turret		Turret	
Name	Spindle ID	Name	Spindle ID	
7 Groovin		ਜੁ ^{ਸ਼} Groovin	SubSpindle	
📲 Groovin	MainSpindle	Groovin		
况 DR 04 🛛 +		n ^뵨 Groovin	SubSpindle	
🐉 Wrap Dri	MainSpindle	🛟 TD M04		
🔴 Sync		🔴 Sync		
况 DR 20 Z-		🐉 Wrap Dri	MainSpindle	
🚱 🖣 Drilling	SubSpindle			
况 DR 04 X+				
🐉 Wrap Dri	SubSpindle			
Bore Bar				
🐉 Wrap Dri	SubSpindle			
👪 TD M05				
🐉 Wrap Dri	SubSpindle			
🔴 Sync		🔴 Sync		
		ר Groovin		
		📲 Groovin	SubSpindle	
		😫 TD M05		
		🐉 Wrap Dri	SubSpindle	
🔴 Sync		🔴 Sync		
·		📇 Part Eject	SubSpindle	
		况 DR 18 L		
		🚱 ID Drillin	MainSpindle	-
📑 Features	Tools	🔢 Operatio	Ins	

Sample operation list of finished part catching (with arm, receiver or manual) for NZXDL configuration:

Project Manag	ger					
< <sync>></sync>		• 100%	🛨 🔽 ،	+		
Upper Le	ft Turret	Upper Rig	ght Turret	-		
Name	Spindle ID	Name	Spindle ID			
		EM 05				
		🐉 Wrap	SubSpindle			
		🖸 BM 5 Z				
		🙀 Lathe	SubSpindle			
		🔁 Part Ej	SubSpindle			
🔴 Sync		Sync				
Cutoff I		┣월 Pickup	SubSpindle			
		<u>≱</u> +Barpull	MainSpindle	-		
Sync 🖉		Sync		-		
= <mark>≞</mark> Cutoff	MainSpindle			Ŧ		
Features Tools Operations						

11.3. Part transfer

11.3.1. Part transfer from spindle 1 to spindle 2 with cut-off

To improve production times, this type of transfer should come at the end of the operation list (documented below). But note that it can also be programmed after complete machining on spindle 1 side before start of machining on spindle 2 side.

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

Soli	SolidTurn Machine Setup						
G	General Assembly NC Output Custom						
1	- Mach	ine Definitio	n				
	Machine Name		NZ2000T3Y3	}			
	Start F	Position Z	1				
	Start I	Position X	0				
	Start I	Position Y	0				

The regular 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. Perform a cut-off and then the spindle 2 goes back home
- 6. Looping for next parts

Steps to program this transfer type in ESPRIT:

1. Machining the parts on the spindle 1 and spindle 2 sides:

Note: The operations on the spindle 2 side are output with the origin shifted to the other face of the part (back face). During operations, only positive Z coordinates are output.

The setting "Part Stock Length" from the Machine Setup is used as default value for Turning Work Coordinates Sub Spindle Z Offset.

CAUTION !!! Part Stock Length = Finished Part Length. You must run the Turning Work Coordinates add-in prior to posting NC Code in order to have correct NC output.

 Catching the finished part in the spindle 2: Spindle 2 needs to be empty before transferring the workpiece from spindle 1 to spindle 2 (see part 11.2.5. Catching finished part on spindle 2 side for programming details).

- 3. Program a wait code across all heads.
- Picking the part in the spindle 1 with the spindle 2: Use operation Pickup with SubSpindle (Spindle Name) on Lower (Upper Right for NZXDL configuration) Turret (Turret Name). With Position X, Y, Z, set the pickup point.

The pickup operation has to be programmed with lower (upper right for NZXDL configuration) turret because it controls the B-axis (spindle 2).

Set Sync Spindles to Speed and Direction Only (or to Off) for speed synchronization code output (M35) or to Oriented for phase synchronization code output (M34). The spindle speed entered on the SolidTurn Pickup page will be output on head 1 after synchronization of the spindles.

Enter the Feedrate PM of the spindle 2 B-axis and its clearance (from the pickup point); it will be output in head 2 program (since head 2 controls the B-axis).

Project Manager 🛛 🛛 🖄								
🛩 (K 🗙 Cancel 📳 🗸 🔞 Help							
dny	S Operation Name							
Pic								
đ	Work Setup							
Cns	Spindle Name SubSpindle							
	Turret Name							
	Head Id Head-1							
	× Feeds and Speeds							
	Speed BPM 500 000000							
	Feedrate PM 250.000000							
	Use Feed and Speed KB No							
	Off							
	Speed and Direction Only							
	Position X, Y, Z 0.000000 0.000000 20.000000							
	Dwell Time JU.000000							
	Comment							
	Features 📑 Tools 📰 Operations 📴 SolidTurn - Pickup							

5. Pulling the bar with the spindle 2 (if needed): Use operation Bar Feed by Spindle (Bar Feed Type) on MainSpindle (Spindle Name) by SubSpindle (Barfeed Spindle Name). Set Turret Name to Lower (Upper Right for NZXDL configuration) Turret.

On Bar Feed tab, set the Feed Length and Reposition Distance: Feed Length (= Barpull distance = Reposition Distance) = Length of the finished part (Part Stock Length) + Cut-off tool width + Length of stock for facing the front side and the back side of the part

With Position X, Y, Z, set the same point as for the pickup operation (If the point is different, chucks will unclamp and clamp before pulling the bar).

😸 Bar Feed	
Position X, Y, Z	0.000000 0.000000 20.000000 📘
Feed Length	41.000000
Reposition Distance	41.000000
Bar Diameter	60.000000
Total Bar Length	100.000000
Length Comp Register	0
Dwell Time	0.000000

Programming a pickup before the barpull makes the spindle 2 stay at the same location after the barpull. Without pickup operation, spindle 2 goes home after the barpull.

On the General tab, enter the Feedrate PM of the spindle 2 B-axis, it will be output in head 2 program (since head 2 controls the B-axis).

- Program a wait code between upper left turret and lower turret (P12 sync code) for NZX-ST configuration.
 Program a wait code across the two heads for NZX-S & NZXDL configurations.
- 7. Cut-off:

Use operation **Cutoff** using **Upper** (**Left** for NZX-ST & NZXDL configurations) **Turret** tool on **MainSpindle** (**Spindle Name**).

Cut-off tool is always loaded in the upper left turret for Mori Seiki NZX-ST & NZXDL configurations. The post processors handle cut-off operation programmed in upper left turret only for these configurations. It is always loaded in the upper turret for Mori Seiki NZX-S configuration. The post processors handle cut-off operation programmed in upper turret only for this configuration.

To output a dwell (G04 U) above workpiece cut-off detection (M80), enter in the Dwell Time field on the Rough tab of the Cutoff operation the dwell time in s.

F	Project Manager 🛛 🔊						
	≁ 0	K 🏋 Cancel 🎒 🗸	🕜 Help				
l	eral	Stock Definition					
l	Gen	Туре	Diameter -				
l	ategy.						
	h Str	Stock Diameter	60.000000				
l	Boug	😵 Rough Passes					
l	=	Rough Stock Allowance	0.000000				
	ustor	Rough to Diameter	0.000000				
	0	Stock Clearance	2.000000				
		Dwell Time	0.000000 G04 U				
		S Pecking					
		Peck Increment	0.000000				
-		Features 🚺 Tools 🖪 Operatio	ons = 🖧 Solid Turn - Cutoff				

The spindle 2 automatically goes home after cut-off completion.

CAUTION!!! Make sure to move the cut-off toolchange above the last programmed sync.

Sample operation list with transfer (with barpull) and cut-off for NZX-ST configuration:

Project Mana	ger					×
< <sync>></sync>		100%	💽 🛨	7		
Upper L	eft Turret	Lower	Turret	Upper Rig	ght Turret	
Name	Spindle ID	Name	Spindle	Name	Spindle	
		💋 12.7m		📲 Groov	SubSpin	
		🔊 Rotary	MainSpin			
		🔴 Sync		🔴 Sync		
_		🕂 Part Ej	SubSpin			
😑 Sync		🔴 Sync				
12.7mm						
🔊 Rotary F	MainSpindle					
27mm In						
Drilling هو)	MainSpindle					
Sync		Sync				
		//Turnin				
		C Houg	MainSpin	• •		
Sync		Sync	0.10.1	🗢 Sync		
7 Cutoff In		Pickup	SubSpin			
		큵+Barpull	MainSpin			
Sync		💛 Sync				
Pr S Lutorr	MainSpindle					
📇 Features	Tools	🔢 Operal	tions			

Sample operation list with transfer (without barpull) and cut-off for NZX-ST configuration:

Project Mana	ger					×
< <sync>></sync>		100%	▼ ¹	7		
Upper Lo	eft Turret	Lower	Turret	Upper Rij	ght Turret	
Name	Spindle ID	Name	Spindle	Name	Spindle	
🔴 Sync		🔴 Sync 👘				
况 SPOT 0		💋 DUM				
🐉 Wrap Dri	MainSpindle	👤 Park	MainSpin			
🔴 Sync		🔴 Sync 👘				
		况 DR 0				
		🐉 Wrap	MainSpin			
Sync		🔴 Sync —		🔴 Sync —		
7 Cutoff In		뵭 🖥 Pickup	SubSpindle			
🔴 Sync		🔴 Sync 👘				
= <mark>≞</mark> Cutoff	MainSpindle					-
📇 Features	🚺 Tools	🔚 Opera	tions			

Sample operation list with transfer (with barpull) and cut-off for NZX-S configuration:

Project Manager 🛛 🛛 🛛						
SimMill		100%	💌 😫 🎫	•		
Upper Turret		Lower Turret				
Name	Spindle ID	Name	Spindle ID			
😫 TD M03						
🐉 Wrap Dril	SubSpindle					
况 DRILL 1						
👍 Tilted Drill	SubSpindle					
况 DRILL 1						
🔴 Sync		🔴 Sync				
🖓 Tilted Drill	MainSpindle	况 DRILL 1				
		🖓 Tilted Drill	SubSpindle			
		📲 Part Eiect	SubSpindle			
🔴 Sync		🔴 Sync				
7 Cutoff In		🗦 🖥 Pickup	SubSpindle			
		🚔 + Barpull	MainSpindle			
🔴 Sync		🔴 Sync				
= <mark>#</mark> Cutoff	MainSpindle			•		
📇 Features	[🖁 Tools	🖽 Operation	ns			

Sample operation list with transfer (without barpull) and cut-off for NZX-S configuration:

Project Mana	Project Manager				
SimMill		100%	💌 😫 🎫	٢	
Upper	Turret	Lower	Turret		
Name	Spindle ID	Name	Spindle ID		
🔴 Sync		🔴 Sync			
🐉 Wrap Dril	MainSpindle	🛃 Park	MainSpindle		
🔴 Sync		🔴 Sync			
		🐉 Wrap Dri	MainSpindle		
		🐉 Wrap Dri	MainSpindle		
		📥 Bore Bar			
		🐉 Wrap Dri	MainSpindle		
		🛟 TD M06			
		🐉 Wrap Dri	MainSpindle		
		🕂 Part Eject	SubSpindle		
🔴 Sync		🔴 Sync			
Cutoff In		🗦 🖥 Pickup	SubSpindle		
🔴 Sync		😑 Sync			
= 🖧 Cutoff	MainSpindle			•	
🔚 Features	[🖁 Tools	🔢 Operation	ns		

Sample operation list with transfer (with barpull) and cut-off for NZXDL configuration:

Project Manager				
< <sync>></sync>	•	100% 💌 🕇	🕈 🕶 🚺	
Upper Le	ft Turret	Upper Rig	ght Turret	
Name	Spindle ID	Name	Spindle ID	
		🐉 Wrap Drilli	SubSpindle	
		TAP X+ UR		
		🐉 Wrap Drilli	SubSpindle	
		🐉 Wrap Drilli	SubSpindle	
		EM 05.0 X		
		🖑 Wrap Drilli	SubSpindle	
		🖸 BM 5 Z- UR		
		Lathe Mold	SubSpindle	
		/B Part Eject	SubSpindle	
Sync		🔴 Sync		
Cutoff Insert		Pickup	SubSpindle	=
		∰+Barpull	MainSpindle	-
Sync		😑 Sync		
= <mark>≓</mark> ‰Cutoff	MainSpindle			-
Features Tools E Operations				

Sample operation list with transfer (without barpull) and cut-off for NZXDL configuration:

Project Manager					
< <sync>></sync>	•	100% 💌 🕇	* = 0		
Upper Le	ft Turret	Upper Rig	jht Turret	^	
Name	Spindle ID	Name	Spindle ID		
26.5mm Ins		26.5mm Ins			
Main ID Drilli	MainSpindle	Gan Sub ID Drilli	SubSpindle		
BB 27 Z+ UL		📙 BB 27 Z- UR			
Drilling - Bor	MainSpindle	🚰 Drilling - Bo	SubSpindle		
Grooving In		12.7mm E			
Grooving ID	MainSpindle	禍 Milling XYZ	SubSpindle		
12.7mm EM		/BPart eject	SubSpindle		
禍 Milling XYZ	MainSpindle				
Sync		Sync			
Cutoff Insert		Pickup	SubSpindle		
😑 Sync		🔴 Sync			
= [™] _{Cutoff}	MainSpindle			-	
Features	📲 Tools 🔛	Operations			

11.3.2. Part transfer from spindle 1 to spindle 2 with cut-off (when machine equipped with a barfeeder)

If your machine is equipped with a barfeeder, instead of pulling the part from spindle 1, you can use the barfeeder to position the bar before pickup and cut-off.

To improve production times, this type of transfer should come at the end of the operation list (documented below). But note that it can also be programmed after complete machining on spindle 1 side before start of machining on spindle 2 side.

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

So	SolidTurn Machine Setup					
	General Assembly NC Output Custom					
		Mach	ine Definitio	n		
	ł	Machine Name		1	1Z2000T3Y3	
	2	Start F	Position Z	1		
		Start Position X		0)	
	Start Po		Position Y	C)	

The regular 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. Perform a barfeed operation
- 4. Picking the part in the spindle 1 with the spindle 2
- 5. Perform a cut-off and then the spindle 2 goes back home
- 6. Looping for next parts

Steps to program this transfer type in ESPRIT:

1. Machining the parts on the spindle 1 and spindle 2 sides:

Note: The operations on the spindle 2 side are output with the origin shifted to the other face of the part (back face). During operations, only positive Z coordinates are output.

The setting "Part Stock Length" from the Machine Setup is used as default value for Turning Work Coordinates Sub Spindle Z Offset.

CAUTION !!! Part Stock Length = Finished Part Length. You must run the Turning Work Coordinates add-in prior to posting NC Code in order to have correct NC output.

 Catching the finished part in the spindle 2: Spindle 2 needs to be empty before transferring the workpiece from spindle 1 to spindle 2 (see part **11.2.5. Catching finished part on spindle 2 side** for programming details).

- 3. Program a wait code across all heads.
- 4. Perform a barfeed operation:

Use operation **Bar Feed By Stopper** (**Bar Feed Type**) using **Upper** (**Left** for NZX-ST & NZXDL configurations) **Turret** tool on **MainSpindle** (**Spindle Name**).

On Bar Feed tab, set the Feed Length and Reposition Distance: Feed Length (=Barfeed distance = Reposition Distance) = Length of the finished part (Part Stock Length) + Cut-off tool width + Length of stock for facing the front side and the back side of the part

With Position X, Y, Z, set the barfeed reference point. Its Z coordinate value should be equal to the entered feed length.

Sar Feed					
Position X, Y, Z	0.000000 0.000000 41.000000				
Feed Length	41.000000				
Reposition Distance	41.000000				
Bar Diameter	60.000000				
Total Bar Length	300.000000				
Length Comp Register	0				
Dwell Time	0.000000				

The stopper tool will either be positioned in the G53 work coordinate or the current machine work coordinate.

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

Proje	ct Manager	×
🔶 O	K 🟋 Cancel 🛛 🛃 👻	🕜 Help
heral	Settings	
Ger	CustomSetting1	0.000000
g	CustomSetting2	0.000000
ar Fee	CustomSetting3	0.000000
ä	Stopper G53×	-50.000000
ų.	Stopper G53 Z	-100.000000
Cust	CustomSetting6	0.000000
Sec.	CustomSetting7	0.000000
	CustomSetting8	0.000000
	CustomSetting9	0.000000
	CustomSetting10	0.000000
	Features 🚺 Tools	Dperations

To use current machine work coordinate defined in ESPRIT (see part 2.3. Turning Work Coordinates): If no value is specified on the Custom tab in the Stopper G53 X and Stopper G53 Z fields (fields left equal to 0), the stopper tool will be positioned at the programmed position (Position X, Y, Z specified on Bar Feed tab) in the current active machine work coordinate.

Note that using End of Bar Macro Call (/2 M98 P) on the POST Output Configuration, you can output the end of bar macro call (/2 M98 Pxxxx); see part **4.4.2. Output of the end of bar macro call (/2 M98 Pxxxx).**

- Program a wait code between upper left turret and lower turret (P12 sync code) for NZX-ST configuration.
 Program a wait code across the two heads for NZX-S & NZXDL configurations.
- 6. Picking the part in the spindle 1 with the spindle 2: Use operation Pickup with SubSpindle (Spindle Name) on Lower (Upper Right for NZXDL configuration) Turret (Turret Name). With Position X, Y, Z, set the pickup point. Note that the pickup point will be shifted by the reposition distance amount entered on Bar Feed tab of Bar Feed operation.

The pickup operation has to be programmed with lower (upper right for NZXDL configuration) turret because it controls the B-axis (spindle 2).

Set Sync Spindles to Speed and Direction Only (or to Off) for speed synchronization code output (M35) or to Oriented for phase synchronization code output (M34). The spindle speed entered on the SolidTurn Pickup page will be output on head 1 after synchronization of the spindles.

Enter the Feedrate PM of the spindle 2 B-axis and its clearance (from the pickup point); it will be output in head 2 program (since head 2 controls the B-axis).

roject I	Manager					
V OK	🗙 Cancel 🛛 📓	<u> </u>				🕜 Help
d 🛛	Operation Nar	ne				
۳ ۲						
5						
ten [₹	Work Setup					
S	pindle Name		SubSpindle	•		-
Т	urret Name		Lower Turr	et		•
ŀ	lead Id		Head-1			⊡
×	Feeds and Sp	eeds				
S	peed RPM		500.00000)		
F	eedrate PM		250.00000)		
1	Jse Feed and Sp	oeed KB	No			•
s	Sync Spindles	M35	Off			
Ī) Pickup	-	Off Speed and	Direction Or	hlu	
C	learance M3	4 🔶	Oriented	010000101101		
F	Position X, Y, Z	_	0.000000	0.000000	20.00000)0 📘
C)well Time		0.000000			
×	Comment					
Fe	eatures 🛛 🖪 T	ools 🔳	Operations	┣권 Sol	idTurn - I	Pickup

- Program a wait code between upper left turret and lower turret (P12 sync code) for NZX-ST configuration.
 Program a wait code across the two heads for NZX-S & NZXDL configurations.
- 8. Cut-off:

Use operation **Cutoff** using **Upper** (**Left** for NZX-ST & NZXDL configurations) **Turret** tool on **MainSpindle** (**Spindle Name**).

Cut-off tool is always loaded in the upper left turret for Mori Seiki NZX-ST & NZXDL configurations. The post processors handle cut-off operation programmed in upper left turret only for these configurations. It is always loaded in the upper turret for Mori Seiki NZX-S configuration. The post processors handle cut-off operation programmed in upper turret only for this configuration.

To output a dwell (G04 U) above workpiece cut-off detection (M80), enter in the Dwell Time field on the Rough tab of the Cutoff operation the dwell time in s.

Project Manager 🛛 🔊						
₩ 0	🛩 OK 🗡 Cancel 📳 🗸 🕜 Help					
eral	Stock Definition					
Gen	Туре	Diameter -				
trategy						
S	Stock Diameter	60.000000				
Bough	🛞 Rough Passes					
E	Rough Stock Allowance	0.000000				
ustor	Rough to Diameter	0.000000				
0	Stock Clearance	2.000000				
	Dwell Time	0.000000 G04 U				
	Pecking					
	Peck Increment	0.000000				
	Features Tools E Operations Solid Turn - Cutoff					

The spindle 2 automatically goes home after cut-off completion.

CAUTION!!! Make sure to move the cut-off toolchange above the last programmed sync.

Sample operation list for NZX-ST configuration:

Project Manager 🛛 🛛 🛛						
< <sync>></sync>		100%	_ ±	7		
Upper Le	eft Turret	Lower	Turret	Upper Rig	ght Turret	
Name	Spindle ID	Name	Spindle	Name	Spindle	
		💋 Turnin				
		🛫 Roug	SubSpindle			
		🔴 Sync 👘		😑 Sync 👘		
				🦨 Dummy	SubSpin	
		🔴 Sync 👘		😑 Sync 👘		
		🕂 Part E	SubSpindle			
		💋 Turnin				
		Roug	MainSpin			
🔴 Sync		🔴 Sync 👘		🔴 Sync 👘		
BAR ST						
🚔 + Bar Feed	MainSpindle					
🔴 Sync		🔴 Sync				
ל Cutoff In		₿ 3 Pickup	SubSpindle			
🔴 Sync		🔴 Sync 👘				
= <mark>‡%</mark> Cutoff	MainSpindle					Ψ.
📇 Features	Features Tools E Operations					

Sample operation list for NZX-S configuration:

Project Manager				
SimMill		100%	💌 불 🛼	•
Upper	Turret	Lower	Turret	*
Name	Spindle ID	Name	Spindle ID	
		💋 Turning I		
		🛫 Roughin	SubSpindle	
		🕂 Part Eject	SubSpindle	
🔴 Sync		🔴 Sync		
💋 BAR ST				
≝r + Barfeed	MainSpindle			
🔴 Sync		🔴 Sync		
7 Cutoff In		🗦 🖥 Pickup	SubSpindle	
🔴 Sync		🔴 Sync		
= <mark>≞</mark> Cutoff	MainSpindle			-
📇 Features	[🖁 Tools	🖽 Operation	ns	

Sample operation list for NZXDL configuration:

Project Manager				
< <sync>></sync>	•	100% 💌 🕇	* =• (!	
Upper Le	ft Turret	Upper Rig	ht Turret	
Name	Spindle ID	Name	Spindle ID	
[} ID Drilling	MainSpindle			
EM 0.3in Z+				
🔊 Rotary face	MainSpindle			
🔀 Drill 0.3in Z+				
🐇 Wrap drilling	MainSpindle			
Sync		🔴 Sync		m.
BAR STOP				
∰+Bar Feed By	MainSpindle			_
Sync		🔴 Sync		=
Cutoff Insert		Pickup	SubSpindle	
😑 Sync		😑 Sync		
= <mark>≓</mark> ‰Cutoff	MainSpindle			Ŧ
Features Tools E Operations				

11.3.3. Part transfer from spindle 1 to spindle 2 with cut-off and stock repositioning (programmed last)

If your machine is equipped with a barfeeder, you can after transferring the finished part from spindle 1 to spindle 2 reposition the stock. This way, you can loop for the next part.

To improve production times, this type of transfer should come at the end of the operation list (documented below). But note that it can also be programmed after complete machining on spindle 1 side before start of machining on spindle 2 side.

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*

9	SolidTurn Machine Setup						
	General Assembly NC Output Custo				Custom		
		Machine Definition					
	Machine		ine Name	NZ2	000T3Y3		
	Start P		Position Z	1			
	Start Pos Start Pos		Position X	0			
			Position Y	0			

The regular 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. Perform a cut-off and then the spindle 2 goes back home
- 5. Perform a barfeed operation
- 6. Looping for next parts

Steps to program this transfer type in ESPRIT:

1. Machining the parts on the spindle 1 and spindle 2 sides: *Note:* The operations on the spindle 2 side are output with the origin shifted to the other face of the part (back face). During operations, only positive Z coordinates are output.

The setting "Part Stock Length" from the Machine Setup is used as default value for Turning Work Coordinates Sub Spindle Z Offset.

CAUTION !!! Part Stock Length = Finished Part Length. You must run the Turning Work Coordinates add-in prior to posting NC Code in order to have correct NC output.

- Catching the finished part in the spindle 2: Spindle 2 needs to be empty before transferring the workpiece from spindle 1 to spindle 2 (see part 11.2.5. Catching finished part on spindle 2 side for programming details).
- 3. Program a wait code across all heads.
- 4. Picking the part in the spindle 1 with the spindle 2: Use operation Pickup with SubSpindle (Spindle Name) on Lower (Upper Right for NZXDL configuration) Turret (Turret Name). With Position X, Y, Z, set the pickup point.

The pickup operation has to be programmed with lower (upper right for NZXDL configuration) turret because it controls the B-axis (spindle 2).

Set Sync Spindles to Speed and Direction Only (or to Off) for speed synchronization code output (M35) or to Oriented for phase synchronization code output (M34). The spindle speed entered on the SolidTurn Pickup page will be output on head 1 after synchronization of the spindles.

Enter the Feedrate PM of the spindle 2 B-axis and its clearance (from the pickup point); it will be output in head 2 program (since head 2 controls the B-axis).

Project Manager 🛛 🛛 🕅			
🛩 OK 🗙 Cancel 🗐 👻			
🖯 🗵 Operation Name			
Pict			
Custo	Work Setup		
	Spindle Name SubSpindle		
	Turret Name	Lower Turret	
	Head Id	Head-1	
	Feeds and Speeds		
	Speed RPM	500.000000	
	Feedrate PM 250.000000		
	Use Feed and Speed KB	No	
	Sync Spindles M35	M35 Off	
		Speed and Direction Unly	
		100.000000	
	Position X, Y, Z	0.000000 0.	000000 20.000000 📘
	Dwell Time 0.000000		
	S Comment		
	Features 🚺 Tools 🗄	Operations	🗦 🖥 SolidTurn - Pickup 🗌

- Program a wait code between upper left turret and lower turret (P12 sync code) for NZX-ST configuration.
 Program a wait code across the two heads for NZX-S & NZXDL configurations.
- 6. Cut-off:

Use operation **Cutoff** using **Upper** (**Left** for NZX-ST & NZXDL configurations) **Turret** tool on **MainSpindle** (**Spindle Name**).

Cut-off tool is always loaded in the upper left turret for Mori Seiki NZX-ST & NZXDL configurations. The post processors handle cut-off operation programmed in upper left turret only for these configurations. It is always loaded in the upper turret for Mori Seiki NZX-S configuration. The post processors handle cut-off operation programmed in upper turret only for this
configuration.

To output a dwell (G04 U) above workpiece cut-off detection (M80), enter in the Dwell Time field on the Rough tab of the Cutoff operation the dwell time in s.

Project Manager				
V OK 🗙 Cancel 🛍 🗸 🔞 Help				
eral	Stock Definition			
Gen	Туре	Diameter -		
trategy				
S	Stock Diameter	60.000000		
Rough	HBno SRough Passes			
E	Rough Stock Allowance	0.000000		
ustor	Rough to Diameter	0.000000		
0	Stock Clearance	2.000000		
	Dwell Time	0.000000 G04 U		
	Expecting			
	Peck Increment	0.000000		
	Features 🔣 Tools 🖽 Opera	ations = 🂫 Solid Turn - Cutoff		

The spindle 2 automatically goes home after cut-off completion.

CAUTION!!! Make sure to move the cut-off toolchange above the last programmed sync.

7. Perform a barfeed operation:

Use operation **Bar Feed By Stopper (Bar Feed Type)** using **Upper (Left** for NZX-ST & NZXDL configurations) **Turret** tool on **MainSpindle (Spindle Name)**.

On Bar Feed tab, set the Feed Length and Reposition Distance: Feed Length (=Barfeed distance = Reposition Distance) = Length of the finished part (Part Stock Length) + Cut-off tool width + Length of stock for facing the front side and the back side of the part

With Position X, Y, Z, set the barfeed reference point. Its Z coordinate value should be equal to the length of stock for facing the front side (of the next part).

🕉 Bar Feed			
Position X, Y, Z	0.000000 0.000000 1.000000 📘		
Feed Length	105.000000		
Reposition Distance	105.000000		
Bar Diameter	100.000000		
Total Bar Length	300.000000		
Length Comp Register	0		
Dwell Time	0.000000		

The stopper tool will either be positioned in the G53 work coordinate or the current machine work coordinate.

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

Proje	Project Manager 🛛 🕅				
₩ 0	K 🗙 Cancel 🛛 🛃 👻	🕡 Help			
teral	Settings				
Ger	CustomSetting1	0.000000			
l p	CustomSetting2	0.000000			
ar Fee	CustomSetting3	0.000000			
ä	Stopper G53X	-50.000000			
M.	Stopper G53 Z	-100.000000			
Cust	CustomSetting6	0.000000			
-	CustomSetting7	0.000000			
	CustomSetting8	0.000000			
	CustomSetting9	0.000000			
	CustomSetting10	0.000000			
	🚰 Features 🛛 🚺 Tools 🔛 Operations 🚔 - SolidTurn - Bar Feed				

To use current machine work coordinate defined in ESPRIT (see part 2.3. Turning Work Coordinates): If no value is specified on the Custom tab in the Stopper G53 X and Stopper G53 Z fields (fields left equal to 0), the stopper tool will be positioned at the programmed position (Position X, Y, Z specified on Bar Feed tab) in the current active machine work coordinate.

Note that using End of Bar Macro Call (/2 M98 P) on the POST Output

Configuration, you can output the end of bar macro call (/2 M98 Pxxxx); see part **4.4.2. Output of the end of bar macro call (/2 M98 Pxxxx).**

Project Mana	ger					×
< <sync>></sync>		100%	. ₹			
Upper Le	eft Turret	Lower	Turret	Upper Rij	ght Turret	
Name	Spindle ID	Name	Spindle	Name	Spindle	
🖉 Turning I				💋 Turni		
🕰 Facing	MainSpindle			🕰 Facin	SubSpin	
		🔴 Sync 👘		🔴 Sync		
		🕂 Part ej	SubSpindle			
		💋 Turnin				
		🗳 OD R	MainSpin			
🔴 Sync		🔴 Sync 👘		🔴 Sync 🚽		
7 Cutoff In		뵭 🖥 Pickup	SubSpindle			
🔴 Sync		🔴 Sync 👘				
= <mark>≞</mark> Cutoff	MainSpindle					
💋 BAR ST						
i∰+Bar Feed	MainSpindle					-
📇 Features	Features 📑 Tools 🔛 Operations					

Sample operation list with transfer, cut-off and barfeed (last) for NZX-ST configuration:

Sample operation list with transfer, cut-off and barfeed (last) for NZX-S configuration:

Project Manag	Project Manager				
< <sync>></sync>		100%	💌 😫 🛼	•	
Upper	Turret	Lower	Turret	^	
Name	Spindle ID	Name	Spindle ID		
💋 Turning I		💋 Turning I			
🕰 Facing	MainSpindle	🕰 Facing	SubSpindle		
🔴 Sync		🔴 Sync			
		🕂 Part eject	SubSpindle		
		💋 Turning I			
		🛫 OD Rou	MainSpindle		
🔴 Sync		🔴 Sync			
Cutoff In		🗦 🖥 Pickup	SubSpindle		
🔴 Sync		🔴 Sync			
= 🖧 Cutoff	MainSpindle				
BAR ST					
∰+Bar Feed	MainSpindle			Ψ.	
📇 Features	🚺 Tools	📰 Operation	ns		

Sample operation list with transfer, cut-off and barfeed (last) for NZXDL configuration:

Project Manager 🗾				
< <sync>></sync>	•	100% 💌 🕇	🕈 🎫 🕚	
Upper Le	ft Turret	Upper Rig	ht Turret	
Name	Spindle ID	Name	Spindle ID	
		TAP X+ UR		
		🖑 Wrap Drillin	SubSpindle	
		🖉 Wrap Drillin	SubSpindle	
		EM 04.5 X		
		🖑 Wrap Drillin	SubSpindle	
		🕂 Part Eject	SubSpindle	
🔴 Sync		🔴 Sync		
Cutoff Insert		┣월 Pickup	SubSpindle	E
😑 Sync		🔴 Sync		
= <mark>≓</mark> Cutoff	MainSpindle			
BAR STOP				
≝+Bar Feed By MainSpindle 🗸 👻				
Features Tools E Operations				

11.3.4. Part transfer from spindle 1 to spindle 2 without cutoff

The stock is manually loaded in the spindle 1; the part is not pulled. The finished part in the spindle 2 has to be released. The entire part is transferred to the spindle 2 when the work on spindle 1 is finished.

To improve production times, this type of transfer should come at the end of the operation list (documented below). But note that it can also be programmed after complete machining on spindle 1 side before start of machining on spindle 2 side.

Note: On the General tab of the Machine Setup, set the Start Position Z and Total Bar Length:

Start Position Z = Length of stock for facing the front side of the part

Soli	SolidTurn Machine Setup				
G	General Assembly NC Output Custom				
	Machine Definition Machine Name Start Position Z Start Position X Start Position Y		n NZ2	000T3Y3	
			1		
			0		
			0		

Total Bar Length = Length of the finished part (Part Stock Length) + Length of stock for facing the front side and the back side of the part

Stock Configuration—			
Stock Type	Bar 💌	Maximum Diameter	320
Bar Diameter	100.000000	Total Bar Length	102
Inside Diameter	20	Part Stock Length	100
Casting Feature	R		

The regular 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. Releasing the part from the spindle 1 and then the spindle 2 goes back home
- 5. Stop for loading a new stock in the spindle 1

Steps to program this transfer type in ESPRIT:

1. Machining the parts on the spindle 1 and spindle 2 sides:

Note: The operations on the spindle 2 side are output with the origin shifted to the other face of the part (back face). During operations, only positive Z coordinates are output.

The setting "Part Stock Length" from the Machine Setup is used as default value for Turning Work Coordinates Sub Spindle Z Offset.

CAUTION !!! Part Stock Length = Finished Part Length. You must run the Turning Work Coordinates add-in prior to posting NC Code in order to have correct NC output.

- Catching the finished part in the spindle 2: Spindle 2 needs to be empty before transferring the workpiece from spindle 1 to spindle 2 (see part 11.2.5. Catching finished part on spindle 2 side for programming details).
- 3. Program a wait code across all heads.
- Picking the part in the spindle 1 with the spindle 2: Use operation Pickup with SubSpindle (Spindle Name) on Lower (Upper Right for NZXDL configuration) Turret (Turret Name). With Position X, Y, Z, set the pickup point.

V Pickup				
Clearance	30.000000			
Position X, Y, Z	0.000000 0.000000 20.000000 💫			
Dwell Time	0.000000			

The pickup operation has to be programmed with lower (upper right for NZXDL configuration) turret because the lower turret controls the B-axis (spindle 2).

Enter the Feedrate PM of the spindle 2 B-axis and its clearance (from the pickup point); it will be output in head 2 program (since head 2 controls the B-axis).

During this type of transfer, the turning spindles are stopped. Using Spindle Orientation (Custom Setting 6) on the Custom tab of the Pickup operation, you can specify how you want to orient your spindles. Set Spindle Orientation to 19 if you want to use spindle orientation (M19 / M219) or set Spindle Orientation to 45 if you want to use the milling C-axis mode (M45 / M245). If you use milling C-axis mode, using C Index for M45 (Custom Setting 7), you can specify the C angle value you want to index to. The angle specified will be output in both heads. Make sure to enter a correct value for C. For example, if C-axis roll-over is on (see part **4.4.7. C-Axis roll over**), C has to be contained between -359.999° and 359.999°.

Project Manager 🛛 🛛 🖄				
₩ 0	K 🟋 Cancel 🛛 🛃 👻 🚽		🕡 Help	
ckup	Settings			
1.2	Push Check (G38)	0.000000		
Ę	G38 K Value	0.000000		
Cust	G38 Feedrate	0.000000		
-	G38 Q Value 0.000000			
	G38 R Value	0.000000		
	Spindle Orientation	19.000000		
	C Index for M45	0.000000		
	CustomSetting8	0.000000		
CustomSetting9 0.0		0.000000		
	CustomSetting10	0.000000		
	🔚 Features 🔢 Tools 📰 Operations 📴 SolidTurn - Pickup			

Proje	Project Manager 🛛 🛛 🔀				
~	OK 🟋 Cancel 🛛 🛃 👻		🕜 Help		
kup	Settings				
Ρi	Push Check (G38)	0.000000			
ш	G38 K Value	0.000000			
Cust	G38 Feedrate	0.000000			
Sec.	G38 Q Value	0.000000			
	G38 R Value	0.000000			
	Spindle Orientation	45.000000			
	C Index for M45	0.000000			
	CustomSetting8	0.000000			
	CustomSetting9	0.000000			
	CustomSetting10	0.000000			
	Features 🚺 Tools	🔛 Operations	📴 SolidTurn - Pickup		

Note that the milling C-axis mode is the default. So if you omit to enter a value in the Spindle Orientation field or enter a wrong value, the spindles will be indexed using this mode.

- Program a wait code between upper left turret and lower turret (P12 sync code) for NZX-ST configuration.
 Program a wait code across the two heads for NZX-S & NZXDL configurations.
- Releasing the part from the spindle 1: Use operation Release with MainSpindle (Spindle Name) on Upper (Left for NZX-ST & NZXDL configurations) Turret (Turret Name). With Position X, Y, Z, set the release point.

😵 Release				
Position X, Y, Z	0.000000 0.000000 130.0000 📐			
Part Chute	No			
Dwell Time	0.000000			

Sample operation list for NZX-ST configuration:

Project Mana	ger					×
< <sync>></sync>		100%	▼ 🖆			
Upper L	eft Turret	Lower	Turret	Upper Right Turret		
Name	Spindle ID	Name	Spindle	Name	Spindle	
🖉 Turning I				💋 Turni		
🕰 Facing	MainSpindle			🕰 Facin	SubSpin	
🔴 Sync		🔴 Sync 👘		🔴 Sync 👘		
		🕂 Part ej	SubSpindle			
		💋 Turnin				
		耄 OD R	MainSpin			
🔴 Sync		🔴 Sync 👘		🔴 Sync 👘		
		🗦 🖥 Pickup	SubSpindle			
🔴 Sync		🔴 Sync 👘				
/ 🗄 Part Rel	MainSpindle					-
Features	Features Tools Derations					

Sample operation list for NZX-S configuration:

Project Manag	ger			×
< <sync>></sync>		100%	- = -	•
Upper	Turret	Lower	Turret	A
Name	Spindle ID	Name	Spindle ID	
🖉 Turning		💋 Turning I		
🕰 FaceTurn	SubSpindle	🕰 FaceTurn	MainSpindle	
🔴 Sync		🔴 Sync		
		🕖 Park Tool		
		👤 Park Lo	MainSpindle	
		🕂 Part Eject	SubSpindle	
🔴 Sync		🔴 Sync		
		🗦 🖥 Pickup	SubSpindle	
🔴 Sync		🔴 Sync		
🕂 Release	MainSpindle			Ψ.
📇 Features	[🖁 Tools	🖽 Operation	ns	

Sample operation list for NZXDL configuration:

Project Manager				
< <sync>></sync>	•	100% 💌 🕇	? 🕶 .	_
Upper Le	ft Turret	Upper Rig	ht Turret	
Name	Spindle ID	Name	Spindle ID	
Thread Mill		[}⊶ ID Drilling	SubSpindle	
Threading	MainSpindle	🗱 TD M11 x		
况 DR 28 Z+ UL		🚱 Drilling - Ta	SubSpindle	
🚱 ID Drilling	MainSpindle	🖏 DR 19 Z- UR		
TD M30 x 3		🚰 ID Drilling	SubSpindle	
[}⊶ Drilling - Tap	MainSpindle	11 TD M22 x		
		🚱 Drilling - Ta	SubSpindle	
		Part Eject	SubSpindle	Ξ
Sync		😑 Sync		
		┣·· Pickup	SubSpindle	
Sync		🔴 Sync		
/昰 Release	MainSpindle			-
Features	🐻 Tools 📃 📰	Operations		

11.3.5. Part transfer from spindle 2 to spindle 1 without cutoff

The stock is manually loaded in the spindle 2; the part is not pulled. The finished part has to be manually unloaded from the spindle 1. The entire part is transferred to the spindle 1 when the work on spindle 2 is finished.

To improve production times, this type of transfer should come at the end of the operation list (documented below). But note that it can also be programmed after complete machining on spindle 2 side before start of machining on spindle 1 side.

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*

So	olidTurn Machine Setup				
	General	Assembly	NC Output	Custom	
	Machi	ine Definitio	n		
	Machi	ine Name	NZ2	000T3Y3	
	Start F	Position Z	1		
	Start F	Position X	0		
	Start F	Position Y	0		

Total Bar Length = Length of the finished part (Part Stock Length) + Length of stock for facing the front side and the back side of the part

Stock Configuration—			
Stock Type	Bar 💌	Maximum Diameter	320
Bar Diameter	100.000000	Total Bar Length	102
Inside Diameter	20	Part Stock Length	100
Casting Feature	R		

The regular steps to follow are:

- 1. Machining the parts on the spindle 1 and spindle 2 sides
- 2. Manually unloading the finished part in the spindle 1
- 3. Picking the part in the spindle 2 with the spindle 1
- 4. Releasing the part from the spindle 2 and then the spindle 2 goes back home
- 5. Stop to load a new stock in the spindle 2

Steps to program this transfer type in ESPRIT:

1. Machining the parts on the spindle 1 and spindle 2 sides:

Note: The operations on the spindle 2 side are output with the origin shifted to the other face of the part (back face). During operations, only positive Z coordinates are output.

The setting "Part Stock Length" from the Machine Setup is used as default value for Turning Work Coordinates Sub Spindle Z Offset.

CAUTION !!! Part Stock Length = Finished Part Length. You must run the Turning Work Coordinates add-in prior to posting NC Code in order to have correct NC output.

- Manually unloading the finished part in the spindle 1: Spindle 1 needs to be empty before transferring the workpiece from spindle 2 to spindle 1 (see part 11.2.4. Manually catching finished part on spindle 1 side for programming details for NZXDL configuration). *Note for NZX-ST & NZX-S configurations:* A Release operation with MainSpindle (Spindle Name) on Upper (Left for NZX-ST configuration) Turret (Turret Name) can be programmed for simulation purpose (part eject). It will not affect the NC code.
- 3. Program a wait code across all heads.
- 4. Picking the part in the spindle 2 with the spindle 1: Use operation Pickup with MainSpindle (Spindle Name) on Upper (Left for NZX-ST & NZXDL configurations) Turret (Turret Name). With Position X, Y, Z, set the pickup point: point that you would like the back of spindle 1 to reach while picking up the part (usually the zero point of the part in the spindle 2).

🕙 Pickup	
Clearance	45.000000
Position X, Y, Z	0.000000 0.000000 -125.0000 📐
Dwell Time	0.000000

Enter the Feedrate PM of the spindle 2 B-axis and its clearance (from the pickup point); it will be output in head 2 program (since head 2 controls the B-axis).

During this type of transfer, the turning spindles are stopped. Using Spindle Orientation (Custom Setting 6) on the Custom tab of the Pickup operation, you can specify how you want to orient your spindles. Set Spindle Orientation to 19 if you want to use spindle orientation (M19 / M219) or set Spindle Orientation to 45 if you want to use the milling C-axis mode (M45 / M245). If you use milling C-axis mode, using C Index for M45 (Custom Setting 7), you can specify the C angle value you want to index to. The angle specified will be output in both heads. Make sure to enter a correct value for C. For example, if C-axis roll-over is on (see part **4.4.7. C-Axis roll over**), C has to be contained between -359.999° and 359.999°.

Ρ	roje	ct Manager		×
ſ	🔶 OI	K 🟋 Cancel 🛛 🛃 👻		🕜 Help
	kup	Settings		
I	ä	Push Check (G38)	0.000000	
I	m.	G38 K Value	0.000000	
I	Cust	G38 Feedrate	0.000000	
ľ	-	G38 Q Value	0.000000	
		G38 R Value	0.000000	
		Spindle Orientation	19.000000	
I		C Index for M45	0.000000	
		CustomSetting8	0.000000	
		CustomSetting9	0.000000	
		CustomSetting10	0.000000	
Ī				
		Features 8 Tools	⊡ Operations 🔤 📴 S	olidTurn - Pickup

Proje	ect Manager		×
~	OK 🟋 Cancel 🛛 🛃 👻		🕜 Help
kup	Settings		
Ρi	Push Check (G38)	0.000000	
ш	G38 K Value	0.000000	
Cust	G38 Feedrate	0.000000	
Sec.	G38 Q Value	0.000000	
	G38 R Value	0.000000	
	Spindle Orientation	45.000000	
	C Index for M45	0.000000	
	CustomSetting8	0.000000	
	CustomSetting9	0.000000	
	CustomSetting10	0.000000	
	Features 🚺 Tools	🔛 Operations	📴 SolidTurn - Pickup

Note that the milling C-axis mode is the default. So if you omit to enter a value in the Spindle Orientation field or enter a wrong value, the spindles will be indexed using this mode.

- Program a wait code between upper left turret and lower turret (P12 sync code) for NZX-ST configuration.
 Program a wait code across the two heads for NZX-S & NZXDL configurations.
- Releasing the part from the spindle 2: Use operation Release with SubSpindle (Spindle Name) on Lower (Upper Right for NZXDL configuration) Turret (Turret Name). With Position X, Y, Z, set the release point.

😵 Release		
Position X, Y, Z	0.000000 0.000000 0.000000	R
Part Chute	No	•
Dwell Time	0.000000	

Sample operation list for NZX-ST configuration:

Project Mana	ger					×
< <sync>></sync>		100%	💌 🖆			
Upper L	eft Turret	Lower	Turret	Upper Ri	ght Turret	
Name	Spindle ID	Name	Spindle	Name	Spindle	
💋 Turning I				💋 Turni		
🕰 Facing	MainSpindle			🕰 Facin	SubSpin	
🔴 Sync		🔴 Sync 👘				
		💋 Turnin				
		💐 OD R	MainSpin			
Sync		🔴 Sync —		🔴 Sync 🚽		
🗦 🖥 Pickup	MainSpindle					
🔴 Sync		🔴 Sync 👘				
		🔁 Part R	SubSpindle			-
Features Tools Derations						

Sample operation list for NZX-S configuration:

Project Manager				
< <sync>></sync>		100%	- = -	•
Upper	Turret	Lower	Turret	
Name	Spindle ID	Name	Spindle ID	
🖉 Turning I]
🕰 Facing	MainSpindle			
🖉 Turning I				
🕰 Facing	SubSpindle			
💋 Turning I				
🚅 OD Rou	MainSpindle			
💋 Turning I				
🛫 OD Rou	SubSpindle			
🔴 Sync		🔴 Sync		
		🕖 Turning I		
		🛫 OD Rou	MainSpindle	
🔴 Sync		🔴 Sync		
🕂 Part Eject	MainSpindle			
😑 Sync		🔴 Sync		
뵭 🖥 Pickup	MainSpindle			
🔴 Sync		🔴 Sync		
		🕂 Release	SubSpindle	-
Features 🕅 Tools 🛄 Operations				

Sample operation list for NZXDL configuration:

Project Manager 🗾					
< <sync>></sync>	•	100% 🔽 🕇	? = 0		
Upper Le	ft Turret	Upper Rig	ht Turret	~	
Name	Spindle ID	Name	Spindle ID		
Turning Inse		🖉 Turning Ins			
🕰 Facing	MainSpindle	🕰 Facing Sub	SubSpindle		
OD Roughin	MainSpindle	🛫 OD Roughi	SubSpindle		
/∄ Part Eject	MainSpindle				
Sync		Sync			
┣丞 Pickup	MainSpindle				
🔴 Sync		🔴 Sync			
		🕂 🔁 Release	SubSpindle	Ŧ	
Features Tools Derations					

11.3.6. Stock repositioning

If your machine is not equipped with a barfeeder, using this type of transfer, you will be able to reposition the stock in spindle 1 using the spindle 2. This type of transfer can be programmed anywhere in the program.

Note: Spindle 2 needs to be empty before repositioning since spindle 2 will pick up the part in spindle 1.

The regular steps to follow are:

- 1. Picking the part in the spindle 1 with the (empty) spindle 2
- 2. Pulling the bar with the spindle 2
- 3. Releasing the part from the spindle 2 and then the spindle 2 goes back home

Steps to program this transfer type in ESPRIT:

- 1. Program a wait code across all heads.
- Picking the part in the spindle 1 with the (empty) spindle 2: Use operation Pickup with SubSpindle (Spindle Name) on Lower (Upper Right for NZXDL configuration) Turret (Turret Name). With Position X, Y, Z, set the pickup point.

🔇 Pickup	
Clearance	30.000000
Position X, Y, Z	0.000000 0.000000 20.000000 📘
Dwell Time	0.000000

The pickup operation has to be programmed with lower (upper right for NZXDL configuration) turret because it controls the B-axis (spindle 2).

Enter the Feedrate PM of the spindle 2 B-axis and its clearance (from the pickup point); it will be output in head 2 program (since head 2 controls the B-axis).

During this type of transfer, the turning spindles are stopped. Using Spindle Orientation (Custom Setting 6) on the Custom tab of the Pickup operation, you can specify how you want to orient your spindles. Set Spindle Orientation to 19 if you want to use spindle orientation (M19 / M219) or set Spindle Orientation to 45 if you want to use the milling C-axis mode (M45 / M245). If you use milling C-axis mode, using C Index for M45 (Custom Setting 7), you can specify the C angle value you want to index to. The angle specified will be output in both heads. Make sure to enter a correct value for C. For example, if C-axis roll-over is on (see part **4.4.7. C-Axis roll over**), C has to be contained between -359.999° and 359.999°.

Proje	ect Manager		×
🛹 C)K 🗙 Cancel 🛛 🛃 👻		🕜 Help
kup	Settings		
Ĕ	Push Check (G38)	0.000000	
ŝ	G38 K Value	0.000000	
Cust	G38 Feedrate	0.000000	
-	G38 Q Value	0.000000	
	G38 R Value	0.000000	
	Spindle Orientation	19.000000	
	C Index for M45	0.000000	
	CustomSetting8	0.000000	
	CustomSetting9	0.000000	
	CustomSetting10	0.000000	
	Features 🚺 Tools	📰 Operations	📴 SolidTurn - Pickup

Proje	ect Manager		×
🔶 C)K 🗙 Cancel 🛛 🛃 👻		🕜 Help
kup	Settings		
μ	Push Check (G38)	0.000000	
Ш,	G38 K Value	0.000000	
Cus	G38 Feedrate	0.000000	
-	G38 Q Value	0.000000	
	G38 R Value	0.000000	
	Spindle Orientation	45.000000	
	C Index for M45	0.000000	
	CustomSetting8	0.000000	
	CustomSetting9	0.000000	
	CustomSetting10	0.000000	
	Features 🚺 Tools	🔛 Operations	📲 SolidTurn - Pickup

Note that the milling C-axis mode is the default. So if you omit to enter a value in the Spindle Orientation field or enter a wrong value, the spindles will be indexed using this mode.

3. Pulling the bar with the spindle 2:

Use operation **Bar Feed by Spindle** (**Bar Feed Type**) on **MainSpindle** (**Spindle Name**) by **SubSpindle** (**Barfeed Spindle Name**).

Set Turret Name to Lower (Upper Right for NZXDL configuration) Turret.

On Bar Feed tab, set the Feed Length and Reposition Distance: Feed Length = Barpull distance = Length of stock that spindle 2 will pull out of spindle 1 for repositioning Reposition Distance = Set it to 0 if you do not want to shift your Z values after repositioning or set it to Feed Length if you want to shift your Z values by the repositioning distance value

With Position X, Y, Z, **set the same point as for the pickup operation** (If the point is different, chucks will unclamp and clamp before pulling the bar).

😸 Bar Feed	
Position X, Y, Z	0.000000 0.000000 20.000000 📘
Feed Length	41.000000
Reposition Distance	41.000000
Bar Diameter	60.000000
Total Bar Length	100.000000
Length Comp Register	0
Dwell Time	0.000000

Programming a pickup before the barpull makes the spindle 2 stay at the same location after the barpull. Without pickup operation, spindle 2 goes home after the barpull.

On the General tab, enter the Feedrate PM of the spindle 2 B-axis, it will be output in head 2 program (since head 2 controls the B-axis).

4. Releasing the part from the spindle 2:

Use operation **Release** with **SubSpindle** (**Spindle Name**) on **Lower** (**Upper Right** for NZXDL configuration) **Turret** (**Turret Name**). With Position X, Y, Z, set the release point.

😵 Release				
Position X, Y, Z	0.000000	0.000000	20.000000	B
Part Chute	No			•
Dwell Time	0.000000			

5. Program a wait code across all heads.

Sample operation list for NZX-ST configuration:

Project Manage	er								×
< <sync>></sync>			▼ 100%		-				
Upper Left	Tum	et	Lower Tu	urret		Upper Righ	t Tu	irret	
Name	S.	Spin	Name	S.	Spi	Name	S.	Spi	
			-			Dummy c		Sub	
Sync			Sync			Sync 🗧			1
	_		Pickup		Sub				
	-				Sub				
Svnc			Svnc		500	Svnc			
Tuming I		_		_	_		-		8
Racing		Main							
Sync			Sync						
	_		Turning I						
O Sumo			CD Roug		Main				
SimMach			SimMach						
¶ ^뵨 OD Groo…		Main	n면OD Groo		Main				
Sync			🔴 Sync						
Cutoff In									
=t‰Cutoff		Main							Ŧ
Features		8 Tools	🛛 🔛 Operati	ons	.J				

Sample operation list for NZX-S configuration:

Project Manager						
< <sync>></sync>		▼ 100%	: 🔄 불 🚍 🕚)		
Upper 1	urret		Lower T	Turret		_
Name	S	Spindle ID	Name	S	Spindle ID	
Sync			Sync			
			Fickup	4	SubSpindle	
			Release	9 2	Sub Spindle	
Sync			Sync		Sabopinaic	
Turning Insert O						
Facing	٩	MainSpindle				
Sync			Sync			
				æ	Main Spindle	
Svnc			Svnc	*	Mainopinaie	
Grooving Insert			Grooving Insert			
SimMach			😑 SimMach			
마루 OD Grooving	٩	MainSpindle	PBOD Grooving	4	MainSpindle	
Cutoff loset To			Sync Sync			
= Cutoff	٨	MainSpindle				-
Eesturee 8	Tool				1	
	100					

Sample operation list for NZXDL configuration:

Project Manager				×
< <sync>></sync>	•	100% 💌 🕇	* =. 🕚	
Upper Le	ft Turret	Upper Rig	jht Turret	-
Name	Spindle ID	Name	Spindle ID	
Sync		Sync		1
		Pickup	SubSpindle	
		≝+ Barpull	MainSpindle	
		🕂 Release	SubSpindle	
Sync		Sync		
Turning Inse				e .
Facing	MainSpindle			
OD Roughing	MainSpindle			
Cutoff Insert				
= <mark>a</mark> Cutoff	MainSpindle			$\overline{\nabla}$
Features	Tools E	Operations		

11.3.7. Machining of long parts with spindle 1 and 2 synchronized

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 (only in head 1 for NZXDL configuration) and in the code will be output the necessary spindles synchronization and unsynchronization 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.

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

Soli	SolidTurn Machine Setup							
G	General Assembly NC Output Custom							
	-Mach Mach	ine Definitio ine Name	n NZ2000T3Y3	_				
	Start F	Position Z	1					
	Start F	Position X	0					
	Start F	Position Y	0					

The regular 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 (only with head 1 for NZXDL configuration) 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

Steps to program this transfer type in ESPRIT:

1. Machining the parts on the spindle 1 and spindle 2 sides: *Note:* The operations on the spindle 2 side are output with the origin shifted to the other face of the part (back face). During operations, only positive Z coordinates are output.

The setting "Part Stock Length" from the Machine Setup is used as default value for Turning Work Coordinates Sub Spindle Z Offset.

CAUTION !!! *Part Stock Length = Finished Part Length*. You must run the

Turning Work Coordinates add-in prior to posting NC Code in order to have correct NC output.

- Catching the finished part in the spindle 2: Spindle 2 needs to be empty before transferring the workpiece from spindle 1 to spindle 2 (see part 11.2.5. Catching finished part on spindle 2 side for programming details).
- 3. Program a wait code across all heads.
- 4. Picking the part in the spindle 1 with the spindle 2: Use operation Pickup with SubSpindle (Spindle Name) on Lower (Upper Right for NZXDL configuration) Turret (Turret Name). With Position X, Y, Z, set the pickup point.

The pickup operation has to be programmed with lower (upper right for NZXDL configuration) turret because it controls the B-axis (spindle 2).

Based on the following operation in head 1 (or 2 for NZX-ST & NZX-S configurations), the correct synchronization code will be output in the code: - M34 (or M35): Turning operation is next on head 1; the spindle speed entered on the SolidTurn Pickup page will be output on head 1 after synchronization of the spindles; the spindle direction code will be output on head 1 based on the direction of the following turning operation.

- M234 (or M235) for NZX-ST & NZX-S configurations only: Turning operation is next on head 2; the spindle speed entered on the SolidTurn Pickup page will be output on head 2 after synchronization of the spindles; the spindle direction code will be output on head 2 based on the direction of the following turning operation.
- M480: Milling operation is next on head 1 (or 2 for NZX-ST & NZX-S configurations); the spindles are stopped.

Enter the Feedrate PM of the spindle 2 B-axis and its clearance (from the pickup point); it will be output in head 2 program (since head 2 controls the B-axis).

Proje	ct Manager	×
₩ 0	K 🗙 Cancel 🛛 🛍 👻	🕜 Help
đ	S Operation Name	
Pioł		
E.		
Cluster	Vork Setup	
	Spindle Name	SubSpindle
	Turret Name	Lower Turret
	HeadId	Head-1
	Feeds and Speeds	
	Speed RPM	500.000000
	Feedrate PM	250.000000
	M35/M235	No
	Sync Spindles	Off 🗾
	Rickup	Off Speed and Direction Only
	M34/M234	Driented
	Position V V 7	
	Dwell Time	
		0.000000
	S Comment	
	L	
	Features [🖁 Tools 📃	Operations

Note: The synchronization mode selected (as shown on picture above) on the pickup operation page will be used for all the following turning operations while the spindles are both holding the part.

5. Pulling the bar with the spindle 2 (if needed): Use operation Bar Feed by Spindle (Bar Feed Type) on MainSpindle (Spindle Name) by SubSpindle (Barfeed Spindle Name). Set Turret Name to Lower (Upper Right for NZXDL configuration) Turret.

On Bar Feed tab, set the Feed Length and Reposition Distance: *Feed Length* (= *Barpull distance* = *Reposition Distance*) With Position X, Y, Z, **set the same point as for the pickup operation** (If the point is different, chucks will unclamp and clamp before pulling the bar).

😣 Bar Feed	
Position X, Y, Z	0.000000 0.000000 20.000000 📘
Feed Length	41.000000
Reposition Distance	41.000000
Bar Diameter	60.000000
Total Bar Length	100.000000
Length Comp Register	0
Dwell Time	0.000000

Programming a pickup before the barpull makes the spindle 2 stay at the same location after the barpull. Without pickup operation, spindle 2 goes home after the barpull.

On the General tab, enter the Feedrate PM of the spindle 2 B-axis, it will be output in head 2 program (since head 2 controls the B-axis).

- Program a wait code between upper left turret and lower turret (P12 sync code) for NZX-ST configuration.
 Program a wait code across the two heads for NZX-S & NZXDL configurations.
- 7. Machining the part on the spindle 1 side (only with head 1 for NZXDL configuration) while the spindles are synchronized and are both supporting the part:

The correct synchronization (and unsynchronization) codes will be output in the NC code based on the programmed operations.

Note that, for NZX-ST configuration, when switching from a turning operation in head 1 or 2 to a milling operation in head 1 or 2, a wait code across the three heads (P123 sync code) needs to be programmed.

Project Manag	er					×
< <sync>></sync>		100%	- =	? 📮 🕚		
Upper Left	Turret	Lower Tu	urret	Upper Right	t Turret	
Name	Spindl	Name	Spindl	Name	Spindl	
		🖉 Turning In		🖉 Turning In		
		🜊 Face Turn	MainSpi	🜊 Face Turn	SubSpi	
Sync		Sync		😑 Sync		
Turning I		🕂 🛱 Part Relea	SubSpi			- 11
CUD Roug	MainSp					
Sync 🗧		Sync	CubCai	Sync		1 1
		Fickup I 환 Daraull	SubSpi			1 1
Supe		Supe	Mainspi			1 1
Turning L		U Sync				1 1
OD Pre-Fi	MainSp					1 1
Sync	Preside pro-	Sync				1
		🕖 Turning In		1		1
		💐 OD Finish	MainSpi			
🔴 Sync		🔴 Sync		🔴 Sync		
EM 20 U						
🍯 Spiral Milli	MainSp					
😝 Spiral Milli	MainSp					
	14 : 0					
SolidMill	MainSp					
	Mainsp	Cupo				
Jync		Jync		L		
E Features	Too	ls <u>EB</u> Operat	ions			

Also note that, for NZX-ST & NZX-S configurations, in M480 mode, C-axis values of milling operations programmed on head 2 cannot be output in head 2 and so will be output in head 1.

- 8. Program a wait code across all heads.
- 9. Perform one of the previously described transfer scenarios to transfer the part to spindle 2 side.

Sample operation list for NZX-ST configuration:

Project Manager 🛛 🛛 🕅						
< <sync>> 🔽 100% 🔽 🚼 🎜 😲</sync>						
Upper Left Turret		Lower Turret		Upper Right Turret		
Name	Spindl	Name	Spindl	Name	Spindl	
		🖉 Turning In		🖉 Turning In		
		😤 Face Turn	MainSpi	😤 Face Turn	SubSpi	
Sync		Sync	CubCai	Sync 🗧		
- グ Turning I	MainSp	/Part Relea	SubSpl			
Sunc	Mainsp	Sunc		Sunc		
		Pickup 1	SubSpi	Initializa	tion	
		Starpull	MainSpi		uon	
🔴 Sync		🔴 Sync				
/ Turning I						
CD Pre-Fi	MainSp					
Sync		Sync				
		CD Finish	MainSpi	Synchror	nized	
Sync		Sync	Mainopill	spindle	es 👘	
💋 EM 20 U				machin	ing 🚺	
🍯 Spiral Milli	MainSp			L		
Sync		🔴 Sync				
		11 TP 8.0 Lo				
		K SolidMill	MainSpi			
		Wran Drilli	MainSpi			
Svnc		Svnc	mainspi	Svnc		
Cutoff Ins		Pickup 2	SubSpi		_	
Sync		Sync		Transf	er	
= <mark>≞</mark> Cutoff	MainSp					\mathbf{v}
🚰 Features 🛛 🗱 Tools 🛛 🖽 Operations						

Sample operation list for NZX-S configuration:

Project Manager 🛛 🛛 🛛 🛛				
< <sync>></sync>	• 11	00% 🔄 🚼 🏞	()	
Upper Turret		Lower Turret		<u>^</u>
Name	Spindle ID	Name	Spindle ID	
✓ Turning Insert 55 ✓ Face Turn	SubSpindle	✓ Turning Insert L ✓ Face Turn	MainSpindle	
🔴 Sync		● Sync 掃 Part Release	SubSpindle	
Sync		Sync		
✓ Turning Insert 55 ✓ OD Rough and fi	MainSpindle			
Sync Initializati	on	● Sync ┣∄ Pickup 1 ☵+Barpull	SubSpindle MainSpindle	
Sync		🔴 Sync		
	MainSpindle	Sunc		
Synchroni	zed	Turning Insert 5 CD Finish Turn Sync	MainSpindle	
Spi Spi machinir	s Ig	Sync		
		HP 8.0 Lower Milling - Threadi MR 10 Lower	MainSpindle	
		🐉 Wrap Drilling O	MainSpindle	
Cutoff I Transfe	r	 Sync Pickup 2 Sync 	SubSpindle	
= ^R Cutoff	MainSpindle			-
Features 📑 Tools 🔛 Operations				

Sample operation list for NZXDL configuration:



11.3.8. Workpiece Pushing Check (G38) $G38 B_K_F_Q_R_;$

The G38 command can be specified when transferring a workpiece from chuck 1 to chuck 2 (or from chuck 2 to chuck 1).

In the workpiece transfer operation, when the position error of the B-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 the workpiece and the program advances to the next block.

The output of the workpiece pushing check G38 is triggered on the Custom tab of the SolidTurn Pickup operation.

Set Push Check (G38) (Custom Setting 1) to 38 to trigger the output of the G38 command.

You can specify in the G38 K Value (Custom Setting 2) field the value of K (B-axis retraction stroke, unsigned value). If no value is entered, the B-axis is automatically retracted by the servo error amount in the uni-direction positioning mode.

You can specify in the G38 Feedrate (Custom Setting 3) field the value of F (B-axis feedrate (mm/min) from R). If no value is entered, 30 mm/min will be used by the machine.

You can specify in the G38 Q Value (Custom Setting 4) field the value of Q (Tolerance for workpiece transfer position). If no value is entered, 1 mm will be used by the machine.

You can specify in the G38 R Value (Custom Setting 5) field the value of R (B-axis feedrate changing position). If no value is entered, 1 mm will be used by the machine.

Proje	Project Manager 🛛 🛛 🛛 🛛			
🔶 O	K 🗙 Cancel 🛛 🛃 👻		🕜 Help	
ckup	😻 Settings			
Ē	Push Check (G38)	38.000000		
ш,	G38 K Value	1.000000		
Ous	G38 Feedrate	1000.000000		
Sec.	G38 Q Value	0.005000		
	G38 R Value	0.500000		
	Spindle Orientation	0.000000		
	C Index for M45	0.000000		
	CustomSetting8	0.000000		
	CustomSetting9	0.000000		
CustomSetting10		0.000000		
	Features 🚺 Tools	💶 Operations 🔄 🗦 🖥 Solid	furn - Pickup	

The spindle 2 will rapid to the programmed clearance location (entered on the Pickup tab of Pickup operation page) and then feed to the pickup location plus 10% of the programmed clearance. G38 will be performed on the remaining distance (10% of the clearance) to the pickup location.

11.3.9. Park turret for transfer

See part **10.4. Park a turret prior to a transfer** if you need to park a turret before transfer.

12. Multiple repetitive cycles

12.1. 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.



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

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



12.2. Roughing cycle

12.2.1. 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.

Proje	ct Manager		×
₩ 0	K 🗙 Cancel 🛛 🖄 👻		🕜 Help
ieral	S Cutting Strategy		
Gen	Type of Work	OD	
itrategy			
0,	Finish Pass	No	_
loug	S Feature Extension		
	Start Extension	0.000000	
ustom	End Extension	0.000000	
	🖲 Rapid Approach/Exit		
	Entry Mode	None	
	Exit Mode	None	
	S Collision Detection		
	Undercutting Mode	Yes	
	Collision Detection	Yes	
	Front Clearance Angle	0.000000	
	Back Clearance Angle	0.000000	
	Tool Blend	No	
	S Machine Functions		
	Cutter Comp NC	No	•
	Length Register	0	
	Canned Cycle	Yes	
	Features 🚺 Tools	Operations	≰ SolidTurn - Roughing

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.

Project Manager 🛛 🛛 🗙			
~ 0	K 🟋 Cancel 🛛 📆 👻	🕑 Help	
ieral	😵 Cutting Strategy		
Ger	Type of Work	OD 🔄	
n Strategy	Finish Pass		
lguof	S Feature Extension		
	Start Extension	0.000000	
ustom	End Extension	0.000000	
	🖲 Rapid Approach/Exit		
	Entry Mode	None	
	Exit Mode	None	
	S Collision Detection		
	Undercutting Mode	Yes 💌	
	Collision Detection	Yes	
	Front Clearance Angle	0.000000	
	Back Clearance Angle	0.000000	
	Tool Blend	No	
	S Machine Functions		
	Cutter Comp NC	No	
	Length Register	0	
	Canned Cycle	Yes	
	Features 🚺 Tools	🖪 Operations 🛛 🗲 SolidTurn - Roughing	

Note: Pockets on a turning profile will be cut by the roughing canned cycle, if Undercutting Mode on the Strategy tab is set to:

- Yes: All pockets on the O.D. or I.D. and the face will be cut.
- Below: Only pockets on the O.D. or I.D. will be cut.
- **Front**: Only pockets on the face will be cut.



The first block of the blocks used to define the finish shape must contain both X and Z axes: Even if either of them does not move in the first block, U0 or W0 will be specified in the NC code.

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

Proje	ct Manager		×
≁ 0	K 🗙 Cancel 🛛 📆 👻		🕜 Help
Teral	🗵 Cutting Strategy		
Ger	Type of Work	OD	
trategu			
S S	Finish Pass	Yes	▼
lough	S Feature Extension		
	Start Extension	0.000000	
Finish	End Extension	0.000000	
ε	😵 Rapid Approach/Exit		
Custo	Entry Mode	None	•
	Exit Mode	None	
	S Collision Detection		
	Undercutting Mode	Yes	
	Collision Detection	Yes	•
	Front Clearance Angle	0.000000	
	Back Clearance Angle	0.000000	
	Tool Blend	No	
	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 ${\bf F}$ (and the cut speed ${\bf S}$) can be specified on the General tab of the operation.

12.2.2. 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: \mathbf{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).

12.2.3. Face roughing

G72 W(1) R_; G72 P_ Q_ U_ W(2) F_;
Project Manager 🛛 🗡		
≁ 0	K 🗙 Cancel 🛛 🔛 👻	🕢 Help
eral	Stock	
Gen	Туре	Length
Ba		
tratei		
S	Stock Length	0.000000
hguo	Stock Allowance	W(2) U/2
ά.	Rough Stock Z.X	0.000000
stom		
3	Vasses	
	Rough Pattern	Parallel to Axis
	Dopth) (printion	Constant V
	Maximum Depth of Cut	5 000000 W/(1)
	Clearance Along Cut	2 \$0000
	Depth Clearance	2000
		Part R
	Setract	
	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 🔤 📽 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.

12.3. Grooving cycle

12.3.1. 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	
	K 🔨 Cancel 🛄 🔹	🕐 нер
neral	Cutting Strategy	
ge	Type of Work	
а		
trate		
S	Rough Pass	Yes
46mo	Finish Pass	Yes
Ĕ	S Bapid Approach/Exit	
hish	Entru Modo	Nama
L IT		
tom	Exit Mode	None
Cus	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 1 SolidTurn - 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.

Project Manager 🛛 🛛 🛛		
V OK X Cancel 🕙 -		
E Cutting Strategy		
C Type of Work		
Finish Pass		
Hapid Approach/Exit		
Entry Mode None		
Exit Mode None		
Ö Entry && Exit Clearance 2.0000000		
Sharp Edges		
Tool Blend No		
S Machine Functions		
Cutter Comp NC No		
Length Register 0		
Edge Shift Register 0		
Canned Cycle Yes		
Features Tools Tools Stations		

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

- The return amount $\mathbf{R}(1)$ 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 \mathbf{F} (and the cut speed \mathbf{S}) can be specified on the General tab of the operation.

• Note that for face grooving, the value **P** designates the **Step Over** and the value **Q** designates the **Peck Increment**. For O.D./I.D. grooving, these values are **REVERSED**, i.e. **P** is the **Peck Increment** and **Q** is the **Step Over**.

Notes: 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.

Grooving canned cycles can only be used when groove type is set to single plunge or multiple plunge. If you set groove type to zigzag or zigzag diagonal, you will get the following error message in your NC code: ERROR: SET GROOVE TYPE TO SINGLE PLUNGE OR MULTIPLE PLUNGE FOR GROOVE CANNED CYCLE.

Project Manager 🛛 🗶			
🔶 OI	K 🗡 Cancel 🛛 📆 🗸	🕜 Help	
leral	Stock		
Ger	Туре	Diameter	
e.			
Strate			
	Stock Diameter	60.000000	
loug	Stock Allowance		
	Rough Stock Z, X	0.000000 0.000000	
Finisł	Passes		
E E	Groove Type	Multiple Plunge	
Custo	-		
	Step Over, % of Tool Width		
	Step Over Mode		
	Pre-finish		
	Dwell Time		
	😵 Plunges		
	Plunge Pattern	Consecutive	
	Plunge Direction	Forward 🔽	
	Retract Feed Move	No	
	Pecking		
	Peck Increment	0.000000	
	Eastures		
. <u>NT</u>		Sperations	

12.3.2. Face grooving

G74 R(1); G74 X_ Z_ P_ Q_ F_ ;

Project Manager 🛛 🛛 🗡		
~ 0	K 🗙 Cancel 🛛 🔛 👻	🕜 Help
eral	Stock	
Gen	Туре	Length
Strategy		
	Stock Length	0.000000
lguol	Stock Allowance	
-Hail	Rough Stock Z, X	0.000000 0.000000
E.	😵 Passes	
Custom	Groove Type	Multiple Plunge
	Step Quer, % of Teel)) (idth	
	Step Over, % of 1001 width	Count -
	Des Geliek	
	Dural Time	
	Dweii 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
	Features	Operations Inthe Solid Turn - Grooving

12.3.3. O.D./I.D. grooving G75 R(1) ; G75 X_ Z_ P_ Q_ F_ ;

je	ct Manager	
 OI 	K 🗙 Cancel 🛄 🕶	🕜 Help
	😻 Stock	
ig A	Туре	Diameter
	Stock Diameter	60.000000
n 5 2	😒 Stock Allowance	
	Rough Stock Z, X	0.000000
-	😵 Passes	
0.000	Groove Type	Multiple Plunge
	Step Over, % of Tool Width	0.000000 Q %
	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 P
	Peck Order	Down Then Across
	Clearance	0.000000 R(1)
	Full Retract IDepth	0.000000
	Sub Groove Order	Region
ð	Features [🖁 Tools 💷	Operations 🛛 📅 SolidTurn - Grooving

12.4. Threading cycle

12.4.1. 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 🛛 🛛 💌		
₩ 0	K 🗡 Cancel 🛛 📆 👻	🕜 Help
eneral	S Cutting Strategy	
Ō	Type of Work	
Strategu		
P	😵 Rapid Approach/Exit	
Threa	Entry Mode	None
stom	Exit Mode	None
Ő	😵 Lead-In/Out	
	Clearance	2.000000
	Lead-In Type	Rapid
	Lead-Out Type	Rapid
	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 \mathbf{F} will be output as entered in ESPRIT.
- Specify the angle of shift of the thread cutting start angle \mathbf{Q} in °, 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	Project Manager 🛛 🕹		
₩ 0	K 🗙 Cancel 🛛 🔛 👻	🕢 Help	
eral	S Cutting Strategy		
Gen	Type of Work	ID 💌	
a			
itrate			
	🖲 Rapid Approach/Exit		
Thread	Entry Mode	None	
	Exit Mode	None	
ustom			
Ō	S Lead-In/Out		
	Clearance	2.000000	
	Lead-In Type	Rapid	
	Lead-Out Type	Chamfer 🗾	
	Chamfer Length	1.000000	
	S Machine Functions		
	Length Register	0	
	Canned Cycle	Off	
	Features 🛛 🔠 🖁 Tools 🖉 🔛	Uperations SolidTurn - Threading	

 $12.4.2. \ \ \ \ Threading \ with \ canned \ cycle \ set \ to \ single \ path \\ G76 \ P(1) \ Q(1) \ R(1) \ ; \\ G76 \ X_2 \ Z_R(2) \ P(2) \ Q(2) \ F_{-} \ ; \\$

Proje	ct Manager K 🗙 Cancel 🛛 🕀 🗸		×
10	Seconetru		
Gene	Thread Definition	From Profile	
	Thread Lead	2.000000	
rateg	Thread Depth	1.000000	P(2)
ŝ	Major Diameter	32.873700	(2)
ead	Minor Diameter	30.873700	
보	Lead Variation	0.000000	
Ę	Thread Angle	60.000000	P 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
	Sector 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

Project Manager 🛛 🛛 📉		
🛩 OK 🛛 X Cancel 🛛 🛍 👻	🕢 Help	
😨 区 Cutting Strategy		
Type of Work		
а		
Otrate		
🕞 🗵 Rapid Approach/Exit		
Entry Mode	None	
Exit Mode	None	
ී 💽 Lead-In/Out		
Clearance	2.000000	
Lead-In Type	Rapid	
Lead-Out Type	Chamfer	
Chamfer Length	5.000000	
🛛 Machine Fur	P	
Length Register Thread	d chamfering size	
Canned Cycle	Off	
Features	Operations 👹 SolidTurn - Threading	

 $\mathbf{R}(1)$ will be output as entered in ESPRIT. $\mathbf{Q}(1)$, $\mathbf{P}(2)$ and $\mathbf{Q}(2)$ 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 $\mathbf{Q}(1)$ in ESPRIT, the output will be in the NC code Q2000.

12.4.3. Threading with canned cycle set to off Threading with canned cycle set to off, lead variation set to 0: G32 $X_Z_F_Q_$;

Project Manager 🛛 🛛 🛛				
₩ 0	🛩 OK 🔭 Cancel 🖄 🕶 🔞 Help			
eral	S Geometry			
Gen	Thread Definition	From Profile		
в	Thread Lead	2.000000 F		
trate	Thread Depth	1.000000		
S	Major Diameter	40.100000		
rread	Minor Diameter	38.100000		
È	Lead Variation	0.000000 G32		
tom	Thread Angle	60.000000		
0°°	Start Angle	0.000000 Q		
	Threads / Lead	1		
	S Feature Extensions			
	Start Length, Leads	4.000000 2.000000		
	End Length, Leads	4.000000 2.000000		
	S 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				

Threading with canned cycle set to off, lead variation different from 0 (Option on machine):

G34 X_Z_F_K_Q_;

Project Manager 🛛 💌		
₩ 0	K 🗙 Cancel 🛐 🗸	🕡 Help
eral	S Geometry	
Gen	Thread Definition	From Profile
Ba	Thread Lead	2.000000 F
trateg	Thread Depth	1.000000
S	Major Diameter	40.100000
read	Minor Diameter	38.100000
É	Lead Variation	0.500000 G34; K
tom	Thread Angle	60.000000
0°°	Start Angle	0.000000 Q
	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
	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
Eestures Tools Conceptions SolidTure - Threading		

K will be output as entered in ESPRIT.

12.4.4. 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.

Note: If you set the Lead-In Type to Feed or Chamfer or the Lead-Out Type to Feed, you will get the following error message in your NC code: ERROR: WRONG LEAD-IN OR LEAD-OUT TYPE SPECIFIED.

😢 Lead-In/Out	
Clearance	2.000000
Lead-In Type	Rapid 🗾
Lead-Out Type	Rapid

Straight thread cutting: G92 X_ Z_ F_ Q_ ;

Proje	ect Manager	×
~	OK 🟋 Cancel 🛛 🔛 👻	🕜 Help
eral	S Geometry	
Gen	Thread Definition	From Profile
а	Thread Lead	2.000000 F
trate	Thread Depth	1.000000
0,	Major Diameter	40.100000
nead	Minor Diameter	38.100000
F	Lead Variation	0.000000
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
	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
		Uperations 50lid lurn - Threading

Tapered thread cutting: G92 X_ Z_ R_ F_ ;

Proje	ect Manager	2
🛹 C)K 🗙 Cancel 📆 🗸	🕜 Help
eral	S Geometry	
Gen	Thread Definition	From Profile
8	Thread Lead	2.000000 F
trate	Thread Depth	1.000000
S	Major Diameter	40.100000
rread	Minor Diameter	38.100000
È	Lead Variation	0.000000
tom	Thread Angle	60.000000
Cus	Start Angle	0.000000
	Threads / Lead	1
	😵 Feature Extensions	
	Start Length, Leads	4.000000 2.000000
	End Length, Leads	4.000000 2.000000
	Image: Second secon	
	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
		0
_ 8=		

13. Hole machining canned cycles and other functionalities for drilling cycles

13.1. 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 (Y-axis specifications only).

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 (Y-axis specifications only).

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:

Proje	ect Manager			×
~)K 🗙 Cancel 🛛 📆 🗸			🕜 Help
-				
nera	S Drill Cycle			
Ğ	Cycle Type	Drill		•
		Drill		_
ΞŪ.	Uepths	Peck		
-	Total Depth	Bore		
tom	Starting Depth	Drill 2		
Ő	Tin 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 - I	Drilling

On the Drill tab for the SolidMillTurn Drilling cycle:

Proje	ct Manager		×
🛩 0)K 🟋 Cancel 🛛 🛍 👻		🕜 Help
ieral	S Drill Cycle		
Ger	Cycle Type	Drill	
	Reverse	Drill Peck	
	S Depths	Tap	l l
stom	Total Depth	Drill 2	_
0	Starting Depth	Peck 2 Tap 2	
	Use Chamfer Diameter	Bore 2	<u> </u>
	Tip Already Included	No	_
	Clearances		
	Full Clearance	10.000000	B
	Clearance	2.000000	
	Return Plane	Clearance	•
	Last Pt Return Plane	Clearance	
	S Machine Functions		
	Canned Cycle	Yes	•
	Dwell	0.000000	
		_	84
	Features 🚺 Tools	Operations	👸 SolidMillTurn - Drilling

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

Proje	ect Manager		×
🥪 C	DK 🗙 Cancel 🖺 🗸 👘		🕜 Help
General	S Type Of Work		•
Deil D	S Drill Cycle		\equiv
Vrap	Cycle Type	Drill	•
- u	Reverse	Drill Peck	
Custo	🗵 Depths	Tap Bore	
	Total Depth	Drill 2 Pook 2	
	Use Chamfer Diameter	Tap 2	
	Tip Already Included	Bore 2	
	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	0.000000	
	Fortures Tools		Deilling
		Derations SolidMill um - Wrap I	Unling

Also make sure to set **Canned Cycle** to **Yes**:

Proje	ct Manager		×
₩ 0	K 🗡 Cancel 🛛 🖄 👻		🕡 Help
eral	😵 Drill Cycle		
Gene	Cycle Type	Drill	
TE O	🗵 Depths		
	Total Depth	0.000000	R
Istom	Starting Depth	0.000000	R
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	0.000000	
	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 \mathbf{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 \mathbf{F} (and the cut speed \mathbf{S}) can be specified on the General tab of the operation.

13.2. Face and side high-speed deep hole drilling / Face and side deep hole drilling / Deep hole drilling with G74

13.2.1. Face and side high-speed deep hole drilling (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_;

Proje	ct Manager		×
🛩 0	K 🗙 Cancel 🛛 📆 🗸	Q) Help
eral	🗵 Drill Cycle		
Gen	Cycle Type	Peck 2 G83.5/G87.5	┓
-	First Peck Increment	0.500000	=
ă	Peck Increment	0.250000 Q	
ε	Full Retract IDepth	0.000000	
Cust	Reverse	No	•
	S Depths		
	Total Depth	1.000000	R
	Starting Depth	0.000000	R
	Use Chamfer Diameter	No	•
	Tip Already Included	No	•
	Clearances		\equiv
	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 - Dril	ling

13.2.2. Face and side deep hole drilling (G83.6/G87.6) CAUTION !!! Use SolidTurn Drilling, SolidMillTurn Drilling or SolidMillTurn Wrap Drilling

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

Proje	ct Manager		×
₩ 0	K 🗙 Cancel 🛛 🔛 👻		🕜 Help
leral	S Drill Cycle		
Gen	Cycle Type	Peck G83.6/G87.6	•
-	First Peck Increment	0.500000	
	Peck Increment	0.250000 Q	
tom	Reverse	No	
Ous	S Depths		
	Total Depth	1.000000	L8
	Starting Depth	0.000000	L3
	Use Chamfer Diameter	No	•
	Tip Already Included	No	
	S Clearances		
	Full Clearance	10.000000	L3
	Clearance	2.000000	
	Return Plane	Clearance	-
	Last Pt Return Plane	Clearance	•
	S Machine Functions		
	Canned Cycle	Yes	•
	Dwell	0.100000 P	
	Features 🚺 Tools	🖪 Operations 🔄 쁍 SolidMillTurn -	Drilling

Settings for SolidTurn Drilling:

Proje	ect Manager	×
🔶 C)K 🗙 Cancel 🛛 🛍 👻	🕢 Help
eral	😒 Drill Cycle	
Gen	Cycle Type	Peck G83.6
	First Peck Increment	0.000000
ā	Peck Increment	0.250000 Q
tom	S Depths	
Cus	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
		Descriptions Deficiency Deficiency
		Derations _ 년째 Solid Furn - Drilling

13.2.3. Deep hole drilling with 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	ect Manager	×
🖌 🗸	DK 🗙 Cancel 🔡 🗸 👘	🕜 Help
la I	S Drill Cucle	
Gene	Cuele Ture	
	Cycle Type	
E O	😻 Depths	Peck
	Total Depth	Tap
stom	Starting Depth	Drill 2
Ő	Tip Already Included	Peck 2 Tap 2
	S Clearances	Bore 2
	Return Plane	Clearance
	Clearance	2.000000
	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 \mathbf{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:

😸 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 🛛 🗶						
🖌 🗸	✓ OK X Cancel 🖄 -					
🖁 😒 Drill Cycle						
Gen	Cycle Type	Peck 2				
-	First Peck Increment	0.000000				
ā	Peck Increment	0.250000				
E L	Full Retract IDepth	0.000000				
Cust	🗵 Depths					
	Total Depth	0.000000				
	Starting Depth	0.000000				
	Tip Already Included	No				
	Clearances					
	Return Plane	Clearance				
	Clearance	2.000000 R(1)				
	S Machine Functions					
	Length Comp Register	0				
	Canned Cycle	Yes				
	Dwell Time	0.000000				
	Features	Dperations (Sea SolidTurn - Drilling)				

13.3. Face and side spot drilling (G83/G87)

CAUTION !!! Use SolidMillTurn Drilling or SolidMillTurn Wrap Drilling

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

Project Manager 🛛 🛛 🛛					
≁ 0	🛩 OK 🗡 Cancel 🛍 👻				
eral	S Drill Cycle				
Gen	Cycle Type	Drill G83/G87	-		
-	Reverse	No	-		
	S Depths				
stom	Total Depth	1.000000	B		
3	Starting Depth	0.000000	L3		
	Use Chamfer Diameter	No	•		
	Tip Already Included	No	•		
	S Clearances				
	Full Clearance	10.000000	B		
	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		

13.4. Face and side tapping / Tapping at center of spindle

13.4.1. Face and side tapping (G84/G88) CAUTION !!! Use SolidMillTurn Drilling or SolidMillTurn Wrap Drilling

 $\begin{array}{c} G84 \; X_{-} \left(Y_{-} \right) C_{-} \; Z_{-} \; R_{-} \; P_{-} \; F_{-} \; ; \\ G88 \; Z_{-} \left(Y_{-} \right) C_{-} \; X_{-} \; R_{-} \; P_{-} \; F_{-} \; ; \end{array}$

Project Manager 🛛 🛛 🛛 🛛				
₩ 0	K 🗙 Cancel 🔡 🗸		🕜 Help	
ieral	S Drill Cycle			
Gen	Cycle Type	Tap G84/G88	•	
E	Reverse	No		
	🖲 Depths			
Istom	Total Depth	1.000000	R	
0	Starting Depth	0.000000	L2	
	Use Chamfer Diameter	No	•	
	Tip Already Included	No	•	
	Clearances			
	Full Clearance	10.000000	B	
	Clearance	2.000000		
	Return Plane	Clearance	•	
	Last Pt Return Plane	Clearance	•	
	Machine Functions			
	Canned Cycle	Yes	•	
	Dwell	0.100000 P		
	Features	🖪 Operations 🔤 🔠 SolidMillTurn -	Drilling	

13.4.2. 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.

Proje	ct Manager		×
₩ 0	K 🗙 Cancel 🔢 👻		🕜 Help
eral	S Drill Cycle		
Gen	Cycle Type	Tap G32	┓
TE O	😵 Depths		
	Total Depth	0.000000	B
Istom	Starting Depth	0.000000	R
G	Tip Already Included	No	•
	S Clearances		
	Return Plane	Clearance	•
	Clearance	2.000000	
	Machine Functions		
	Length Comp Register	0	
	Canned Cycle	Yes	•
	Dwell Time	0.000000	
		7	
	Features 8 Tools	🔛 Operations 🔤 🚰 SolidTuri	n - Drilling

Note: The direction of the tap is controlled by the tool spindle direction (clockwise or counterclockwise) set on the tool page.

13.5. Face and side synchronized tapping / Face and side (high-speed) deep hole synchronized tapping / Spindle synchronized tapping

13.5.1. Restriction 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.

13.5.2. Face and side synchronized tapping (M329 G84/M329 G88)

CAUTION !!! Use SolidMillTurn Drilling or SolidMillTurn Wrap Drilling

M329 S_;

G84 X_ (Y_) C_ Z_ R_ P_ F_; M329 S_; G88 Z_ (Y_) C_ X_ R_ P_ F_;

Project Manager 🛛 🛛 🔀			
₩ 0)K 🗙 Cancel 🛐 🗸	🕡 Help	
ieral	S Operation Name		
Ger			
E	S Tool Selection		
ε	Tool ID	??????	
Custo	😵 Work Setup		
	Spindle Name	MainSpindle	
	HeadId	Head-1	
	😵 Feeds and Speeds		
	Cut Speed RPM, SPM	0 M329 S	
	Z Feedrate PM, PR	0.000000 0.000000 🔥	
	% of Calculated Feed	0	
	Use Feed and Speed KB	No	
	S Comment		
	Features 🚯 Tools 🗄	Operations 🖁 SolidMillTurn - Drilling	

Project Manager 🛛 🛛 🔀				
🛩 0	K 🗙 Cancel 🔡 🗸	🕜 Help		
eral	🗵 Drill Cycle			
Gen	Cycle Type	Tap 2 G84/G88		
-	Reverse	No		
ā	Depths			
tom	Total Depth	1.000000		
Cus	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		
	Machine Functions			
	Canned Cycle	Yes		
	Dwell	0.100000 P		

13.5.3. Face and side (high-speed) deep hole synchronized tapping (M329 G84/M329 G88)

CAUTION !!! Use SolidMillTurn Drilling or SolidMillTurn Wrap Drilling

M329 S_; G84 X_(Y_) C_Z_R_P_Q_F_; M329 S_; G88 Z_(Y_) C_X_R_P_Q_F_;

Project Manager 🛛 🛛 🔀					
~ (
eral	S Operation Name				
Gen					
E	S Tool Selection				
	Tool ID	-??????			
Custom	S Work Setup				
 	Spindle Name	MainSpindle			
	Head Id	Head-1			
	Feeds and Speeds				
	Cut Speed RPM, SPM	0 M329 S			
	Z Feedrate PM, PR	0.000000 0.000000 🔥			
	% of Calculated Feed	0			
	Use Feed and Speed KB	No			
	S Comment				
	Features	🗄 Operations 🛛 🖓 SolidMillTurn - Drilling			

Project Manager 🛛 🛛 🛛						
🔶 C	🛩 OK 🗙 Cancel 🔛 👻					
eral	S Drill Cycle					
Gen	Cycle Type	Tap 2 G84/G88	•			
1	Reverse	No	•			
	🗵 Depths					
Istom	Total Depth	1.000000	R			
l d	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			

You can enter the depth of cut per pass \mathbf{Q} in the Depth of Pass Q (Custom Setting 3) field of the SolidMill Drilling (or Wrap Drilling) operation.

Pr	Project Manager				
Ī	🛹 OK 🗙 Cancel 🔛 🗸 🔞 Hel				
	eral	Values			
	Ger	Rapid Feedrate (G01F)	0.000000		
	=	Custom Setting 2	0.000000		
1	ā	Depth of Pass Q	1.000000		
	E.	Custom Setting 4	0.000000		
	Cust	Custom Setting 5	0.000000		
		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		
Ē	Features 📑 Tools 🖽 Operations 🖓 SolidMillTurn - Drilling				

13.5.4. Spindle synchronized tapping (M329 G84) (Option on machine)

CAUTION !!! Use SolidTurn Drilling

M329 S_; G84 X_ Z_ R_ P_ F_;

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.

If you set the Cycle Type to Tap 2 on the SolidTurn Drilling cycle, the synchronized tapping canned cycle will be output with M329 G84.

Proje	ect Manager			×
🛩 c	DK 🗙 Cancel 🛛 🔛 👻		(🖉 Help
ieral	S Operation Name			
Ger				
-=	S Tool Selection			\equiv
	Tool		•	
stom		1		
3	Work Setup			
	Spindle Name	MainSpindle		•
	HeadId	Head-1		∍
	Feeds and Speeds			
	Speed Unit	RPM		
	Speed RPM, CSS	0	M329 S	
	Feed Unit	Per Revolution	า	•
	Feedrate PM, PR	0.000000	0.000000	
	Maximum RPM	5000		
	Spindle Range	Off		•
	Spindle Priority	Off		•
	Use Feed and Speed KB	No		
	S Comment			
	Features	Operations	Can SolidTure - D	rilling [
_ 65			D-Solution - D	

Proje	Project Manager 🛛 🛛 🗡				
₩ 0	🛩 OK 🗙 Cancel 🔀 🗸 🔞 Help				
eral	S Drill Cycle				
Gen	Cycle Type	Tap 2 G84			
E.	😻 Depths				
	Total Depth	0.000000			
Istom	Starting Depth	0.000000			
3	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			
	Features 🚺 Tools	Operations GaldTurn - Drilling			

13.6. Face and side boring / Boring in turning mode

13.6.1. Face and side boring (G85/G89) CAUTION !!! Use SolidMillTurn Drilling or SolidMillTurn Wrap Drilling

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

Project Manager 🛛 🛛 🛛 🛛			
🛩 0	K 🗙 Cancel 🔛 👻		🕜 Help
leral	S Drill Cycle		
Ger	Cycle Type	Bore G85/G89	•
-2	Offset	0.000000	
	Reverse	No	
stom	S Depths		
3	Total Depth	1.000000	R
	Starting Depth	0.000000	L2
	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	
		#4	
	Features 🔀 Tools 🚦	🖪 Operations 🔤 🎁 SolidMillTurn	Drilling

13.6.2. 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.

Project Manager 🛛 🛛 🛛					
V OK 🗡 Cancel 🔛 👻					
heral	😒 Drill Cycle				
Ger	Cycle Type	Bore			
T	Offset	0.000000			
	S Depths				
Istom	Total Depth	0.000000	R		
0	Starting Depth	0.000000	R		
	Tip Already Included	No			
	Clearances				
	Return Plane	Clearance	•		
	Clearance	2.000000			
	Machine Functions				
	Length Comp Register	0			
	Canned Cycle	No	•		
	Dwell Time	0.000000			
	Eestures 10 Tools		Can SolidTurn - Drilling		

Note: If you omit to set Canned Cycle to No, you will get the following error message in your NC code: ERROR: CANNED CYCLE NOT SUPPORTED FOR LATHE BORING CYCLE.

13.7. Spindle / Rotary Tool Spindle Simultaneous Operation Mode

CAUTION !!! Use SolidTurn Drilling

M90 S_; M91; M290 S_; M291;

The M90 and M290 (for NZX-ST & NZX-S configurations) 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 Simultaneous Operation (Custom Setting 6) to 90 to trigger the output of the M90 (or M290 if machining on the spindle 2 side for NZX-ST & NZX-S configurations) command.

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

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

Example:

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

Project Manager				
•	🛩 OK 🗙 Cancel ∰] → 🔞 He			
- and	0	Settings		
C	5	Custom Setting 1	0.000000	
	=	Custom Setting 2	0.000000	
Ľ	2	Custom Setting 3	0.000000	
E		Custom Setting 4	0.000000	
		Custom Setting 5	0.000000	
	-	Simultaneous Operation	90.000000	
		Speed Percentage	30.000000	
		Custom Setting 8	0.000000	
		Custom Setting 9	0.000000	
		Custom Setting 10	0.000000	
Features Tools Derations Solid Turn - Drilling				

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 pull-down Spindle Direction.
Proje	ct Manager	×					
🔶 O	🛩 OK 🛪 Cancel 📳 👻						
General	S Operation Name						
E	Tool Selection						
E E	Tool	??????					
Custo	😻 Work Setup						
	Spindle Name	MainSpindle					
	Head Id	Head-1					
	Feeds and Speeds						
	Speed Unit	RPM 🔽					
	Speed RPM, CSS	6000 0					
6	Feed Unit	Per Minute G98					
	Feedrate PM, PR	0.000000 0.000000 🔥					
	Maximum RPM	6000					
Ľ	Spindle Range Tota	al speed					
	Use Feed and Speed KB	No					
	S Comment						
	Features	Operations					

14. 3D coordinate conversion (G68.1) (Option on machine)

14.1. Introduction

```
T0;
G68.1 X0 Y0 Z0 I0 J1 K0 R_;
T_;
T0;
G69.1;
```

By using the G68.1 command, it is possible to execute coordinate conversion for the program shape, created in the 3D coordinate system, around an optional axis.

The 3D coordinate conversion function is effective when machining with an angle holder.

By using the program created assuming that machining is executed on the ZX plane, the programmed machining can be executed on the plane after rotation by converting the coordinate system by specifying the center of rotation, the direction of the center axis of rotation, and the angle of rotation.

Note that the center of rotation will always be (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 tilted holder.

14.2. How to output 3D coordinate conversion

In ESPRIT, do a right click on the Features tab of the project manager and click on New Work Coordinate.

On the Work Coordinate window:

- Enter the Work Coordinate Name (for spindle 1 side: G54_Tilted and for spindle 2 side G55_Tilted, for example).
- Enter the Standard Work Coordinate Number: Usually 54 for spindle 1 side and 55 for spindle 2 side.
- Set the Work Plane to YZX if you machine on the spindle 1 side and to G55 if you machine on the spindle 2 side.
- Set Rotate With Part to Point Only.

			×		
Features	2				
			Work Coordinate	3	2
G54_Tilted	New Work Coordinate			L ²	
	Edit Work Coordinate		Define Work Loordinate		
Tilted Drilling Upper Left	Active Work Coordinate		Work Coordinate Name	G54_Tilted	
⊡ S 9 PTOP			Standard Additional Work F	Coordinate Number 54	
Tilted Drilling Lower Main	Rename		Standard, Additional Work C		
de+ G55	Cut	Ctrl+X	X, Y, Z Coordinate	0 0	5
G55_Tilted	Conv	Chrl+C	Offset Angle	Botaru Clearance 254	
E 11 PTOP	Decte	Chiev	Olisec Angle 0	Hotaly clearance 234	
Tilted Drilling Lower Sub	Accoriato to Activo WC	Sector	Work Plane YZX	 Rotate With Part Point Only 	-
□ 12 PTOP	Associate to Active WC		· · · · · ·		
• 1 Tilted Drilling Upper Right	Delete	Del			
	Select All				
Features Tools Tools	Invert Selection				
				4	
	File	→ <u>⊢</u>			
	Machining			OK Cancel	Help
Key	Value				
Group Coupt	1				

W	ork Coordinate						×
[- Define Work Coor	dinate —					
	Work Coordinate N	lame		G55_Tilted	ł		
	Standard, Addition	al Work Co	oordinate N	lumber	55	0	
	X, Y, Z Coordinate		0	0		0	
	Offset Angle	0	R	lotary Clear	ance	254	
	Work Plane	G55	▼ R	otate With	Part	Point Only	-
l							
				OK		Cancel	Help

Make sure to activate the work coordinate in which you want to output your tilted operation (G54_Tilted for spindle 1 side or G55_Tilted for spindle 2 side).

Your work coordinate is now correctly set for 3D coordinate conversion. If you **create a Tilted** tool in ESPRIT, the NC code output when using this tool will be with 3D coordinate conversion.

Milling T	ools - Drill				×
	Tool Change Movement X,YZ Home Home Position X,YZ 0.000000	▼ Home ▼ 125.000000			
<u>₩</u>	Mounting Turret Name Station Name	Upper Left Turret 💌			
8	Axis Orientation Vector X, Y, Z 1.000000 0.000000	Tilted		8	
		0.000000		Ø	
Comme	ent				
			ОК	Cancel	Help

If, for example, you program a SolidMillTurn Drilling operation using a tilted tool in the defined G54_Tilted work coordinate, the 3D coordinate conversion will be output in the NC code.

Note: If you programmed an operation with a tilted tool but forgot to set Rotate With Part to Point Only for your work coordinate, you will get the following error message in your NC code: ERROR: ROTATE WITH PART MUST BE SET TO POINT ONLY FOR 3D COORDINATE CONVERSION.

CAUTION !!! When working with G68.1 function, make sure to set Autorun Mode to NONE in Turning Work Coordinates add-in.

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	-20
Spindle Orientation	Z+	Spindle Orientation	Z- 1
Options			
Keep Z axis par	allel with Tool axis		With Tilted Orientation 📃 🔽
Reverse Z axis	NO		
Autorun Mode		NONE	
			OK Cancel

15. 4-axis wrap milling cycles with interpolation

15.1. 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.

15.2. 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	ct Manager	×			
🔶 OI	K 🗡 Cancel 🛛 🟦 👻	🕜 Help			
heral	🗵 Rotary Machining				
Ger	Working Diameter	25.000000			
융	Type of Work	OD 💌			
\geq	Wall Type	Radial Tool Axis			
- Bi	Cylindrical Interpolation	Yes			
1 2 1	Tolerance	0.100000			
	Features 🚺 Tools 🔜 🤃	Operations 🛛 🏧 SolidMillTurn - Wrap Pocketing			

It will be turned on in the NC code by **G07.1** C_{_} and turned off by **G07.1** 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	Ж 🟋 Cancel 🛐 🗸 👘		🕜 Help
heral	Rotary Machining		
Ge	Working Diameter	25.000000	C*2
₽	Type of Work	OD	
Ň	Cylindrical Interpolation	Yes	
ategy	😵 Transverse Move		
Stra	Моче Туре	Radial Tool A	xis 💌
	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).

15.3. Polar coordinate interpolation (Notching)

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.

Proj	ect Manager		×
~	OK 🗙 Cancel 🛛 🟦 👻		🕢 Help
eral	S Rotary Machining		
Gen	Polar Interpolation	Yes	
tegy	S Cutting Strategy		
Stra	# of Rough Passes	1	
ed	Finish Pass	No	_
hanc	Cutting Strategy	Climb	
Ad	Process Order	Width	
nks	Spiral Move	No	
	Stock Allowance		
Istom	Stock Allowance Walls	0.000000	
0	Stock Allowance Floors	0.000000	
	S Depths		
	Total Depth	3.000000	N
	Incremental Depth	0.000000	
	Starting Depth	0.000000	<u>L</u> e
	Tapered Wall Angle	0.000000	
	S Compensations		
	Offset Side Computer	Left	
	Offset Tool Radius	Yes	<u> </u>
	Cutter Comp NC	Left	_
	Offset Register NC	1	
	Offset Register Value	0.000000	
	Cashuan Tash	Onessiens	S Calidadut
		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.



16. Custom Settings index

16.1. On operation pages

	See page	Use	Comments
	63	Park turret with G53	Head 1, 2 & 3: Used for Park cycle
Custom Setting 1	117	Workpiece pushing check (G38)	Head 2: Used for Part pickup cycle
outloan ootting r	45	Output of rapid moves (G00) with linear interpolation moves (G01 F)	Head 1, 2 & 3: Used for all milling cycles

	66	Output stop coolant code (M09)	Head 1, 2 & 3: Used for Park cycle
Custom Setting 2	117	B-axis retraction stroke	Head 2: Used for Part pickup cycle / Goes with Custom Setting 1
Oustan Ostina 2	167	Depth of cut per pass Q	Head 1, 2 & 3: Used for SolidMillTurn Drilling cycle (TAP2)
Custom Setting 3	117	B-axis feedrate	Head 2: Used for Part pickup cycle / Goes with Custom Setting 1
	92	X position of bar stopper tool (G53)	Head 1: Used for Barfeed (By stopper) cycle
Custom Setting 4	117	Tolerance for workpiece transfer position	Head 2: Used for Part pickup cycle / Goes with Custom Setting 1
Custom Setting 5	92	Z position of bar stopper tool (G53)	Head 1: Used for Barfeed (By stopper) cycle
	117	B-axis feedrate changing position	Head 2: Used for Part pickup cycle / Goes with Custom Setting 1
Custom Setting 6	174	Spindle/Rotary Tool Spindle Simultaneous Operation Mode	Head 1, 2 & 3: Used for SolidTurn Drilling cycle
	107	C-Axis mode for spindle orientation	Head 1 & 2: Used for Part pickup cycle
Custom Setting 7	174	Percentage of the total speed you want to apply to the rotary tool	Head 1, 2 & 3: Used for SolidTurn Drilling cycle / Goes with Custom Setting 6
	107	C-Axis angle value	Head 1 & 2: Used for Part pickup cycle
Custom Setting 8	80	Spindle 2 catching position	Head 2: Used for Part release cycle
Custom Setting 9	80	Type of workpiece ejector	Head 2: Used for Part release cycle
Custom Setting 10		Not used	

Note: Head 3 in array above is only applicable for NZX-ST configuration.

16.2. On tool pages

	See page	Use	Comments
Custom Setting 1	49	Second coolant type	Used to set a second coolant (Optional)
Custom Setting 2		Not used	
Custom Setting 3	25	Second home position	Used to use second home position (G30)
Custom Setting 4		Not used	
Custom Setting 5		Not used	
Custom Setting 6	50	Special coolant ON	Used to turn on special coolant (Optional)
Custom Setting 7	50	Special coolant OFF	Used to turn off special coolant (Optional) / Goes with Custom Setting 6
Custom Setting 8	24	Tool life management	Used to turn on/off tool life management
Custom Setting 9		Not used	
Custom Setting 10		Not used	

16.3. In Machine Setup

	See page	Use	Comments
Custom Setting 1	13	Output of Program End (M02), Program End and Rewind (M30) or Sub-Program End (M99)	
Custom Setting 2	13	Work unloader on spindle 2 side: hand or receiver	
Custom Setting 3		Not used	

Custom Setting 4	14	Tool station for part transfer and work unloader (Turret 1)	
Custom Setting 5	14	Tool station for part transfer and work unloader (Turret 2)	
Custom Setting 6	14	Tool station for part transfer and work unloader (Turret 3)	For NZX-ST configuration only
Custom Setting 7	14	C-axis brake clamp/unclamp	
Custom Setting 8	14	Set how to skip spindle control using comment form, Block Skip or Block Skip 2	For NZX-ST & NZX-S configurations only
Custom Setting 9	15	C-axis roll over	
Custom Setting 10	15	Set position of optional stop code (M01) in the NC code	
Custom String 8	16	99, 133 or 200 Tool wear offset number (Option on machine)	200 is for NZXDL configuration only
Custom String 9	16	Output of the end of bar macro call (/2 M98 Pxxxx); Custom String 9 should be set to the program number (xxxx) called by M98	
Custom String 10	16	Set position of sequence numbers (N)	