



**POST PROCESSOR  
MANUAL  
FOR**

***MORI SEIKI***

**NZX2500 – 4000 – 6000**

**NLX1500 – 2000 – 2500**

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.	Introduction .....	1
2.	Required add-ins .....	1
2.1.	How to load an add-in ESPRIT .....	1
2.2.	AutoSubStock (optional) (for NLX configuration only) .....	3
2.3.	Turning Work Coordinates .....	6
3.	How to turn on Custom Pages / POST Output Configuration .....	10
4.	Machine Setup .....	11
4.1.	Introduction .....	11
4.2.	Set the program name, program number .....	12
4.3.	Define the turning stock .....	12
4.4.	Machine parameters .....	13
4.4.1.	Introduction .....	13
4.4.2.	Output of Program End (M02), Program End and Rewind (M30) or Sub- Program End (M99) .....	15
4.4.3.	Work unloader on spindle 1 or spindle 2 side (for NLX configuration only) . .....	15
4.4.4.	Type of tailstock installed on the machine.....	16
4.4.5.	Tool station for part transfer and work unloader (for NLX configuration only) .....	16
4.4.6.	C-axis brake clamp/unclamp.....	17
4.4.7.	Set how to skip turning spindle control (for NZX configuration only) .....	17
4.4.8.	C-axis roll over .....	17
4.4.9.	Set position of optional stop code (M01) in the NC code.....	18
4.4.10.	Set position of sequence numbers (N).....	18
5.	Tools (T function) .....	19
5.1.	T function specifications.....	19
5.1.1.	Introduction.....	19
5.1.2.	Specify the tool number: Method 1 .....	20
5.1.3.	Specify the tool number: Method 2 .....	22
5.1.4.	Additional information: Tool call when machining on spindle 2 side .....	24
5.1.5.	Restrictions.....	25
5.1.6.	Tool life management.....	25
5.1.7.	Second home position .....	29
5.2.	Set tool movements .....	31
5.2.1.	Set tool movements from tool change position .....	31
5.2.2.	Set tool movements to tool change position .....	32
5.2.3.	Moves to tool change position optimization .....	33
5.3.	SolidTurn Grooving: Control Edge Shift .....	37
5.4.	Automatic Determination of Tool Nose Radius (G46).....	41
6.	Spindle direction, speed (S function) and feedrate (F function) .....	43
6.1.	Turning spindle direction and milling tool spindle direction .....	43
6.2.	Turning spindle gear M-codes (for NZX configuration only).....	44
6.3.	Spindle speed output for CSS unit .....	46
6.4.	Feedrate for 4-axis milling operations .....	47

6.5.	Output of rapid positioning moves (G00) with linear interpolation moves (G01 F)	48
7.	How to output coolant codes	49
7.1.	Introduction	49
7.2.	Set the first coolant code	49
7.3.	Set the second coolant code (optional)	51
7.4.	Set a special coolant code (optional)	52
8.	Simultaneous turning operations: Turning on the same spindle with 2 different turrets (for NZX configuration only)	53
8.1.	Balanced roughing cycle	53
8.2.	Other turning cycles	54
8.3.	Other turning cycles in Balanced Cut Mode	56
9.	Operation synchronization (for NZX configuration only)	59
10.	Park cycle	59
10.1.	Introduction	59
10.2.	How to park a tool	59
10.3.	Park a turret to its maximal travel position (G53)	62
10.4.	Park a turret prior to a transfer (for NLX configuration only)	64
10.5.	Other functionalities	65
11.	How to program finished part catching and part transfer	66
11.1.	Introduction	66
11.2.	Finished part catching	66
11.2.1.	Catching finished part on spindle 1 side after cut-off (for NLX configuration only)	66
11.2.2.	Catching finished part on spindle 1 side after cut-off with stock repositioning (programmed first) (for NLX configuration only)	69
11.2.3.	Catching finished part on spindle 1 side after cut-off with stock repositioning (programmed last) (for NLX configuration only)	72
11.2.4.	Manually catching finished part on spindle 1 side	75
11.2.5.	Catching finished part on spindle 2 side (for NLX configuration only)	77
11.3.	Part transfer (for NLX configuration only)	79
11.3.1.	Part transfer from spindle 1 to spindle 2 with cut-off	79
11.3.2.	Part transfer from spindle 1 to spindle 2 with cut-off (when machine equipped with a barfeeder)	84
11.3.3.	Part transfer from spindle 1 to spindle 2 with cut-off and stock repositioning (programmed last)	89
11.3.4.	Part transfer from spindle 1 to spindle 2 without cut-off	94
11.3.5.	Part transfer from spindle 2 to spindle 1 without cut-off	98
11.3.6.	Stock repositioning	102
11.3.7.	Machining of long parts with spindle 1 and 2 synchronized	106
11.3.8.	Workpiece Pushing Check (G38)	110
11.3.9.	Park turret for transfer	111
12.	Multiple repetitive cycles	111
12.1.	Introduction	111
12.2.	Roughing cycle	113
12.2.1.	Introduction	113

12.2.2.	O.D./I.D. roughing.....	118
12.2.3.	Face roughing.....	119
12.3.	Grooving cycle .....	121
12.3.1.	Introduction.....	121
12.3.2.	Face grooving.....	123
12.3.3.	O.D./I.D. grooving.....	124
12.4.	Threading cycle .....	126
12.4.1.	Introduction.....	126
12.4.2.	Threading with canned cycle set to single path .....	127
12.4.3.	Threading with canned cycle set to off .....	129
12.4.4.	Threading with canned cycle set to multiple path .....	131
13.	Hole machining canned cycles and other functionalities for drilling cycles.....	134
13.1.	Introduction.....	134
13.2.	Face and side high-speed deep hole drilling / Face and side deep hole drilling / Deep hole drilling in turning mode / Deep hole drilling with G74.....	138
13.2.1.	Face and side high-speed deep hole drilling (G83/G87) (Parameter #8115 = 1) .....	138
13.2.2.	Face and side deep hole drilling (G83/G87) (Parameter #8115 = 0) ....	139
13.2.3.	Deep hole drilling in turning mode.....	139
13.2.4.	Deep hole drilling with G74.....	140
13.3.	Face and side spot drilling (G83/G87).....	142
13.4.	Tapping at center of spindle (G32) .....	143
13.5.	Face and side synchronized tapping / Face and side synchronized reverse tapping .....	144
13.5.1.	Restriction of synchronized tapping .....	144
13.5.2.	Face and side synchronized tapping (M329 G84/M329 G88).....	145
13.5.3.	Face and side synchronized reverse tapping (M329 G84.1/M329 G88.1) ..	146
13.6.	Face and side boring / Boring in turning mode .....	148
13.6.1.	Face and side boring (G85/G89).....	148
13.6.2.	Boring in turning mode .....	149
13.7.	Spindle / Rotary Tool Spindle Simultaneous Operation Mode .....	150
14.	4-axis wrap milling cycles with interpolation .....	152
14.1.	Introduction.....	152
14.2.	Cylindrical interpolation .....	152
14.3.	Polar coordinate interpolation (Notching) .....	154
15.	Other Functionalities.....	156
15.1.	Tailstock.....	156
15.1.1.	Introduction.....	156
15.1.2.	Defining the tailstock in the Machine Setup .....	157
15.1.3.	Creating the tailstock operation .....	158
15.2.	Independent steady rest .....	160
15.2.1.	Introduction.....	160
15.2.2.	Defining the steady rest.....	160
15.2.3.	Creating the steady rest operation.....	161
15.3.	Long Boring Bar (for NZX6000 configuration only) .....	164

16.	Custom Settings index .....	167
16.1.	On operation pages .....	167
16.2.	On tool pages .....	168
16.3.	In Machine Setup.....	168

# 1. Introduction

These notes are about the following post processors:

NZX2500 – 4000 – 6000 (formerly named NZL):

**E12MSP\_NZX\_2012-03-09\_T1.asc:** Upper turret, machines on spindle 1 side

**E12MSP\_NZX\_2012-03-09\_T2.asc:** Lower turret, machines on spindle 1 side

NLX1500 – 2000 – 2500:

**E12MSP\_NLX\_2012-04-03.asc:** Turret 1 (upper turret), machines on spindle 1 and spindle 2 (sub spindle) sides

*Note:*

- NZX machines with the letter Y in their model number have a Y-axis on upper turret only; others have no Y-axis. Some NLX configurations have no Y-axis. The output will match your machine configuration based on the loaded NZX or NLX Machine Setup in ESPRIT.

For example, if you work on a NZXD, no Y-axis (nor C-axis) movements will be output in the code.

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

- NZX & some NLX machines do not have a spindle 2. The output will match your machine configuration based on the loaded NZX or NLX Machine Setup in ESPRIT.

In the following manual, you can ignore any reference to the spindle 2 if your machine is not equipped with this spindle.

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

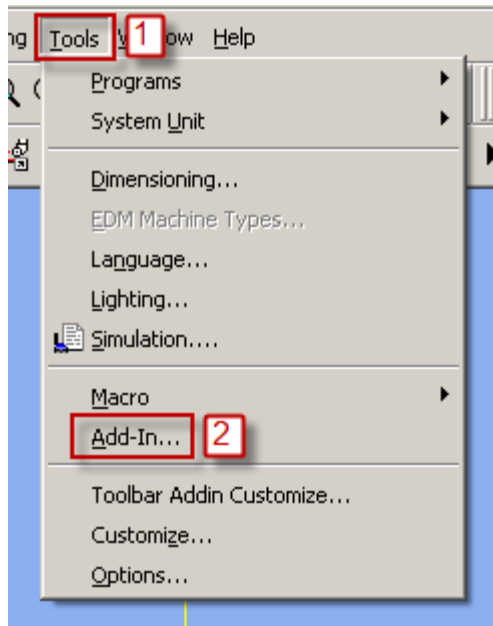
Please refer to NZX & NLX programming manuals for detailed information on the related G and M-codes.

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

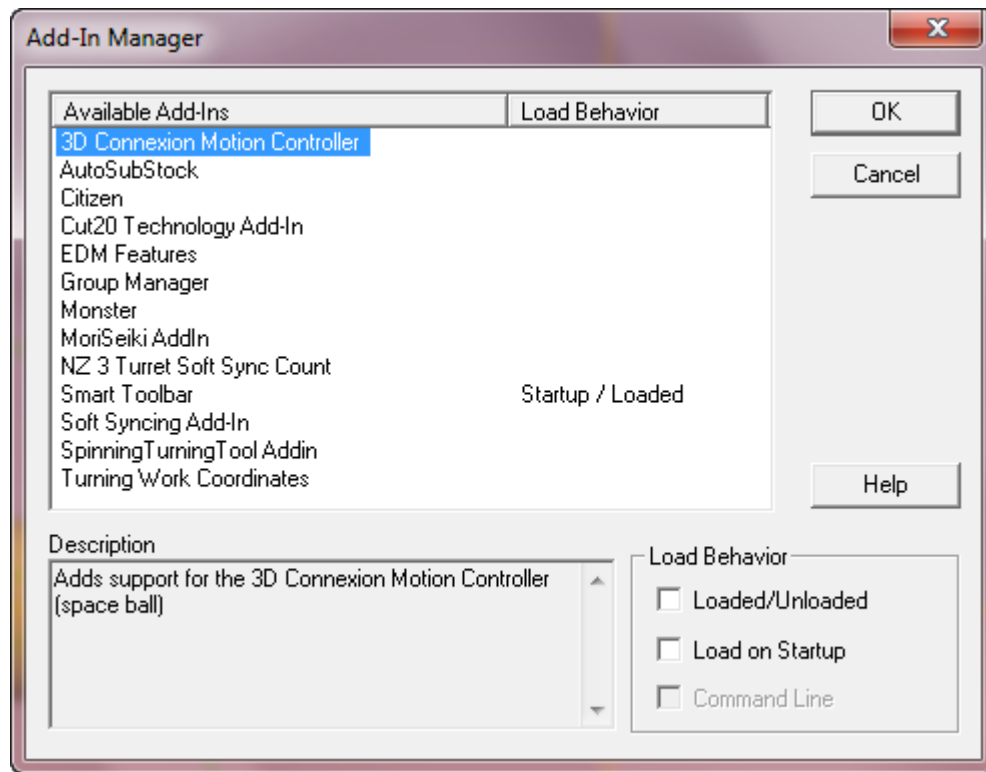
## 2. Required add-ins

### 2.1. *How to load an add-in ESPRIT*

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

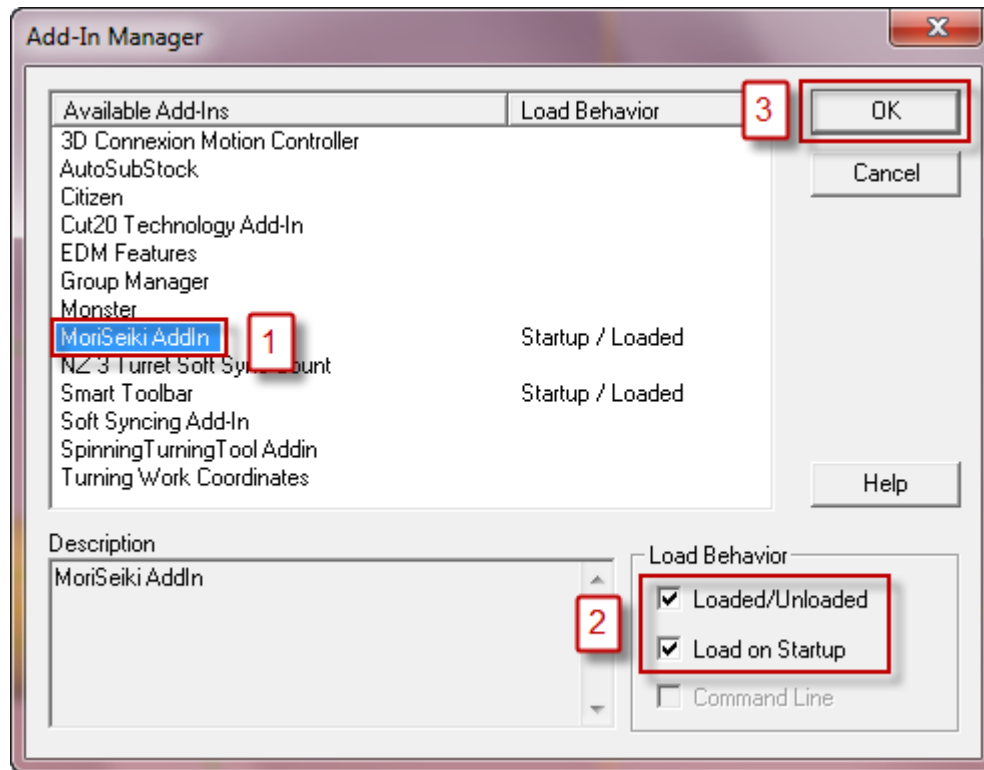


You will then see the add-in window:



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.



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

- **MoriSeiki AddIn**
- **Turning Work Coordinates**

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

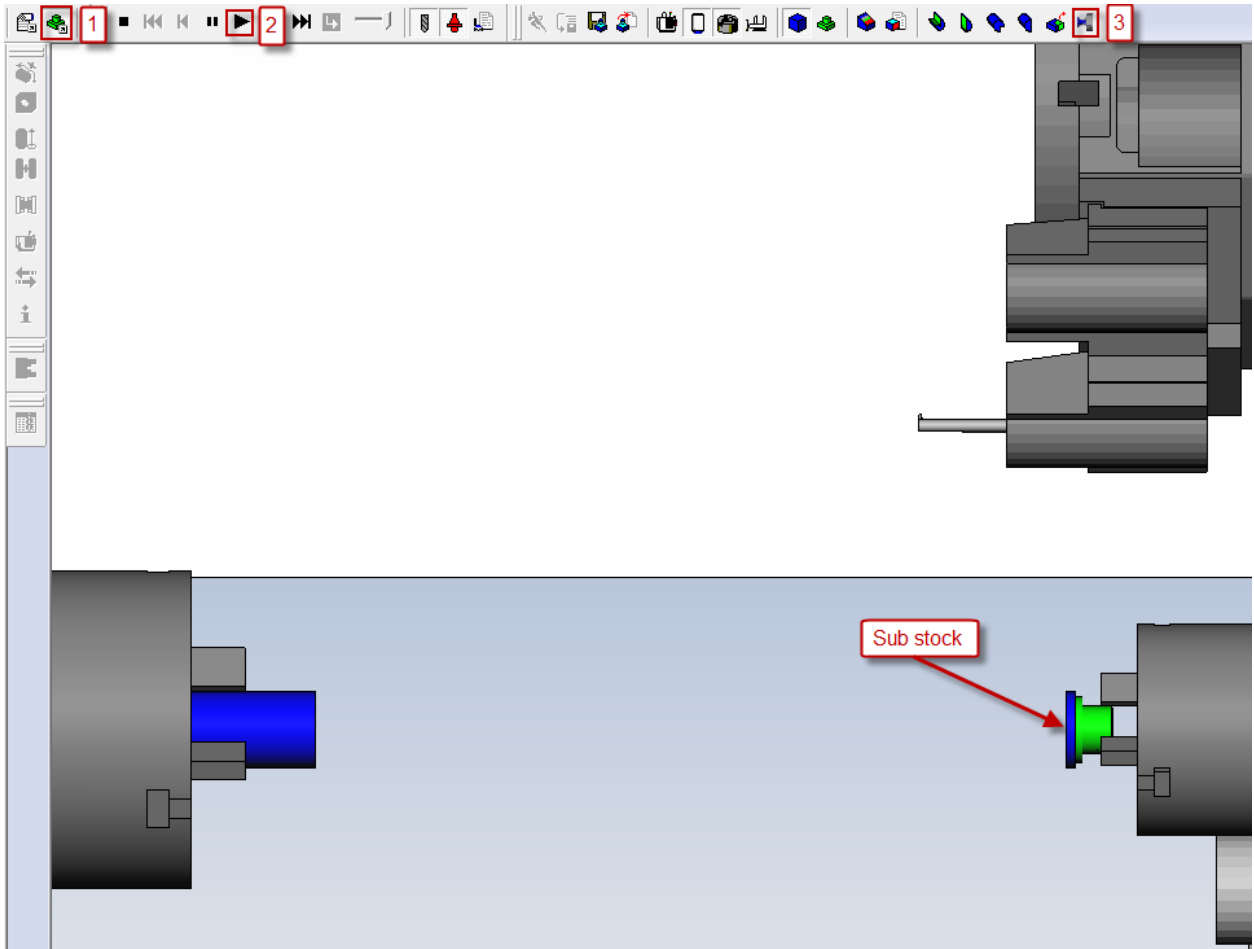
- **AutoSubStock (optional)**
- **MoriSeiki AddIn**
- **Turning Work Coordinates**

## ***2.2. AutoSubStock (optional) (for NLX configuration only)***

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.

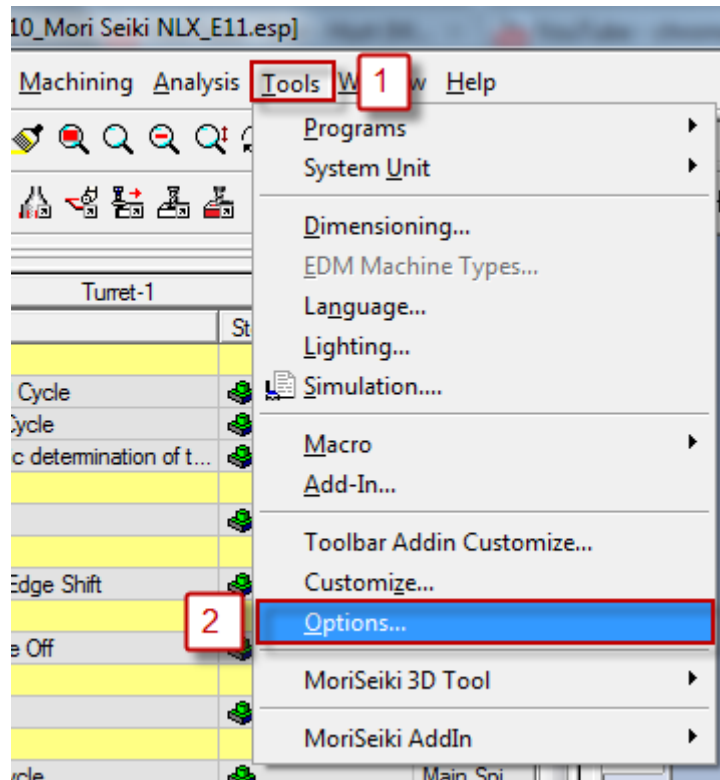
Once you are done programming a part, simply play the entire simulation and once complete click on Auto Sub Stock. This will save the sub stock in the spindle 2 and stop the simulation.



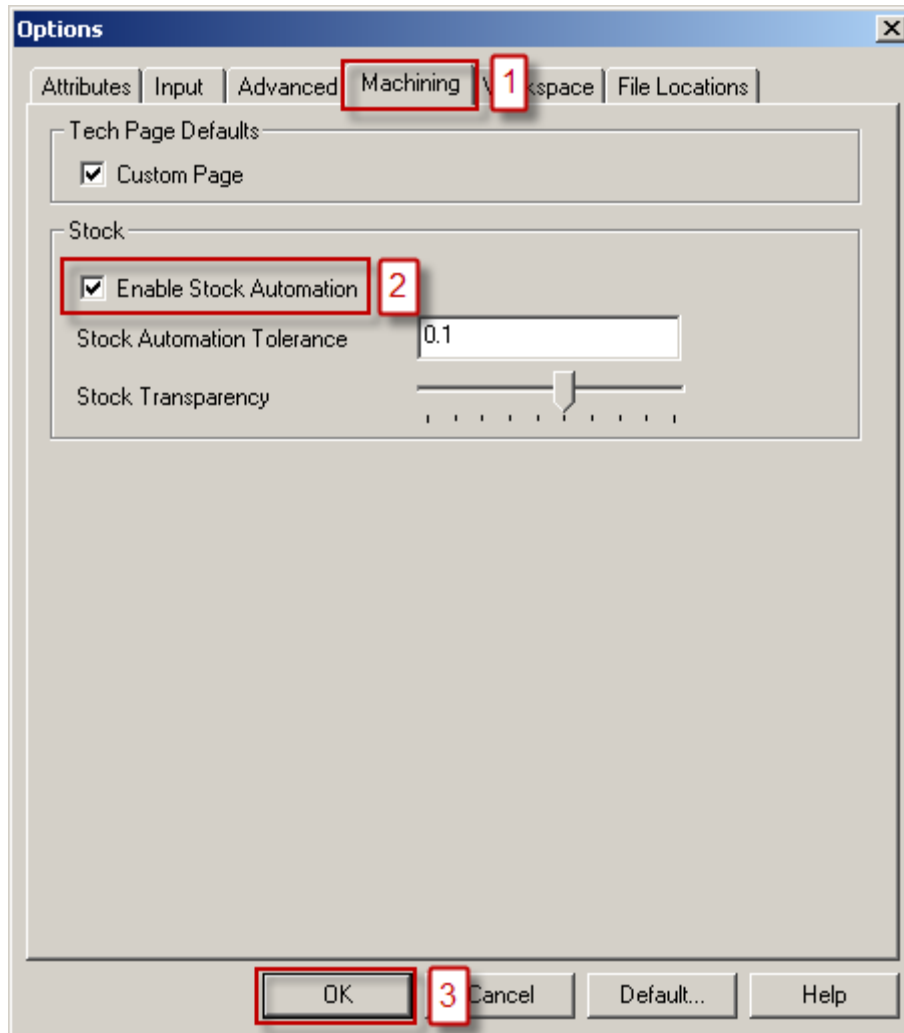


If you restart the simulation, you will be able to see the cut on the spindle 1 and spindle 2 sides all at the same time.

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



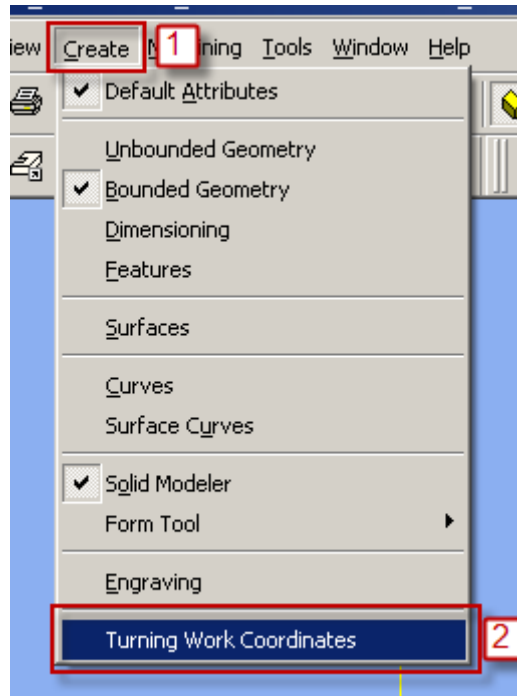
On the Machining tab, if you check Enable Stock Automation, you will not need to turn off the AutoSubStock add-in since the Stock Automation will compute the state and shape of your stock present in both the spindle 1 and spindle 2:



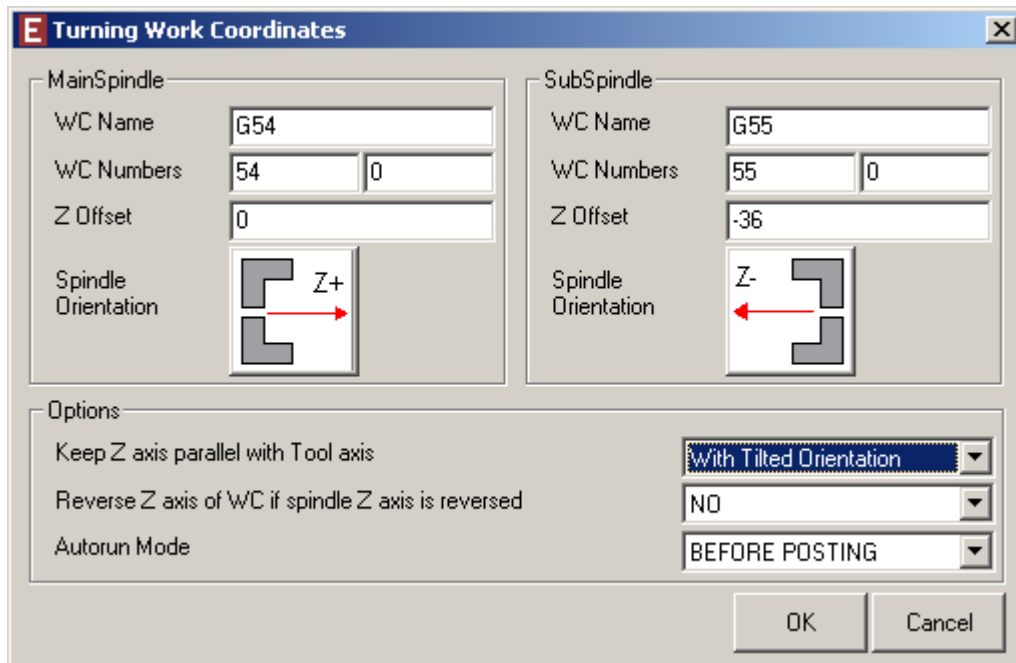
### **2.3. Turning Work Coordinates**

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.



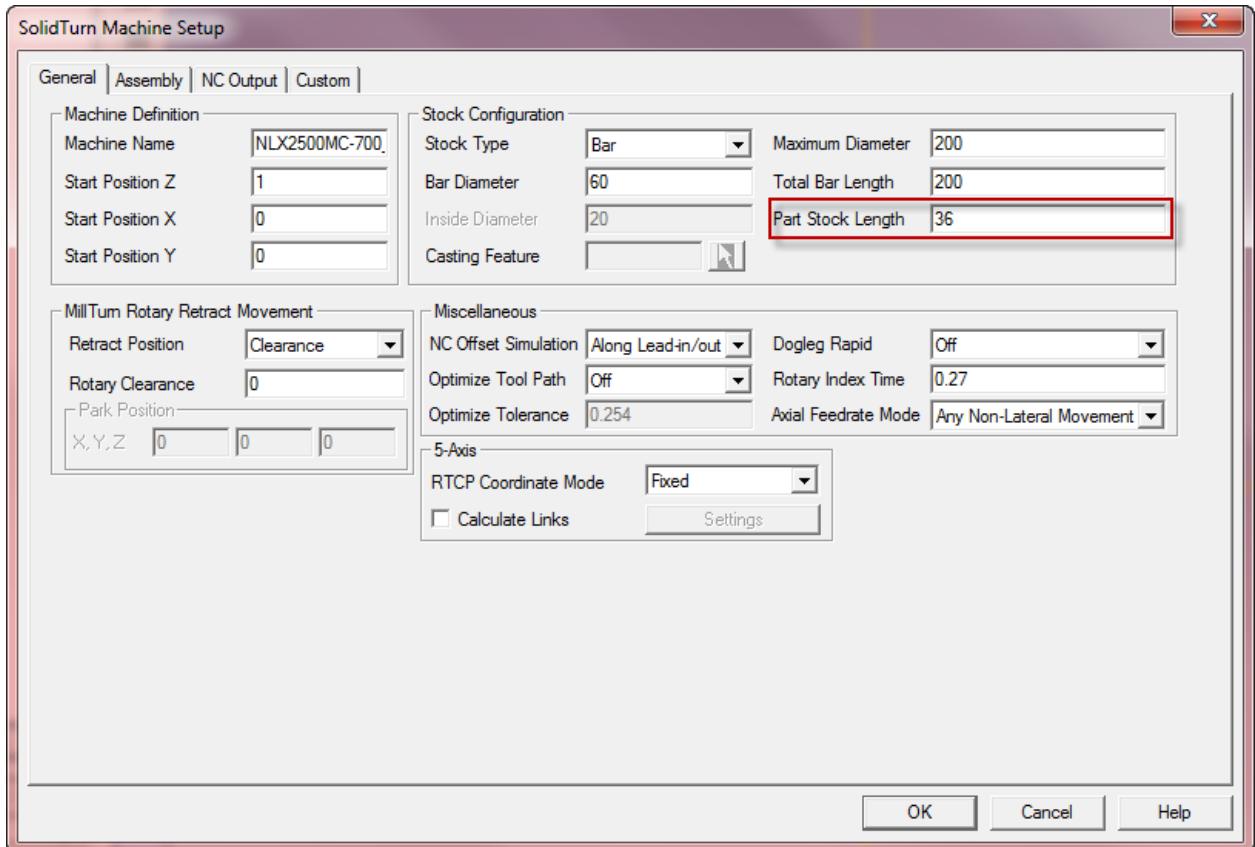
You will then see the Turning Work Coordinates dialog:



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.



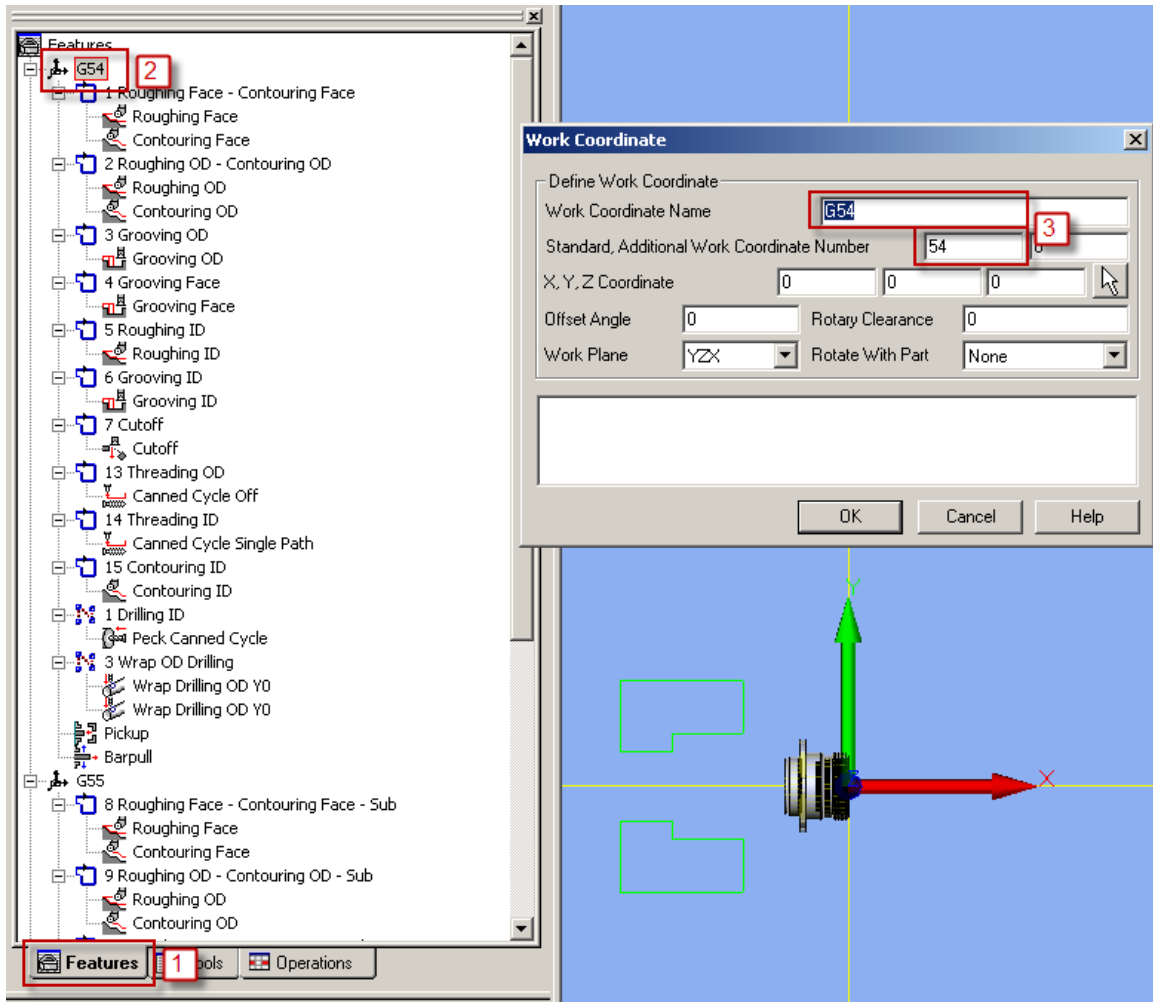
- **Spindle Orientation:** This information is coming from the Machine Setup and is just informative.

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.

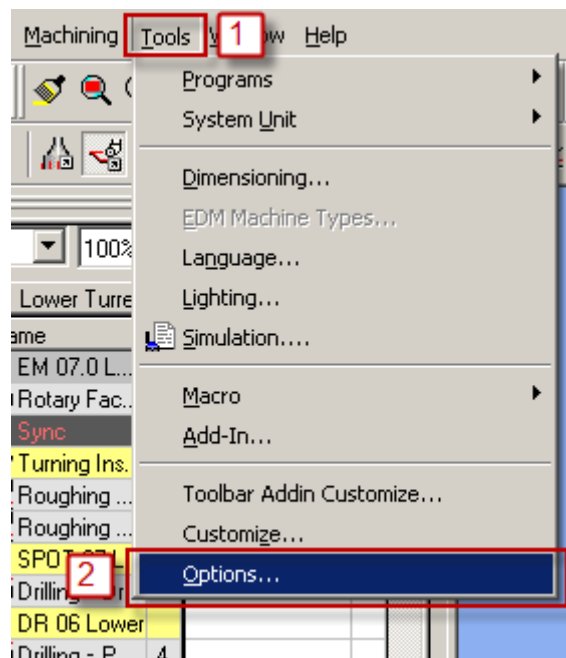
- **None:** No autorun mode selected, you have to manually run the add-in to sort operations.
- **Before posting:** Every time you will output the NC code, the add-in will be run.
- **Before simulation:** Every time you will start the simulation, the add-in will be run.

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

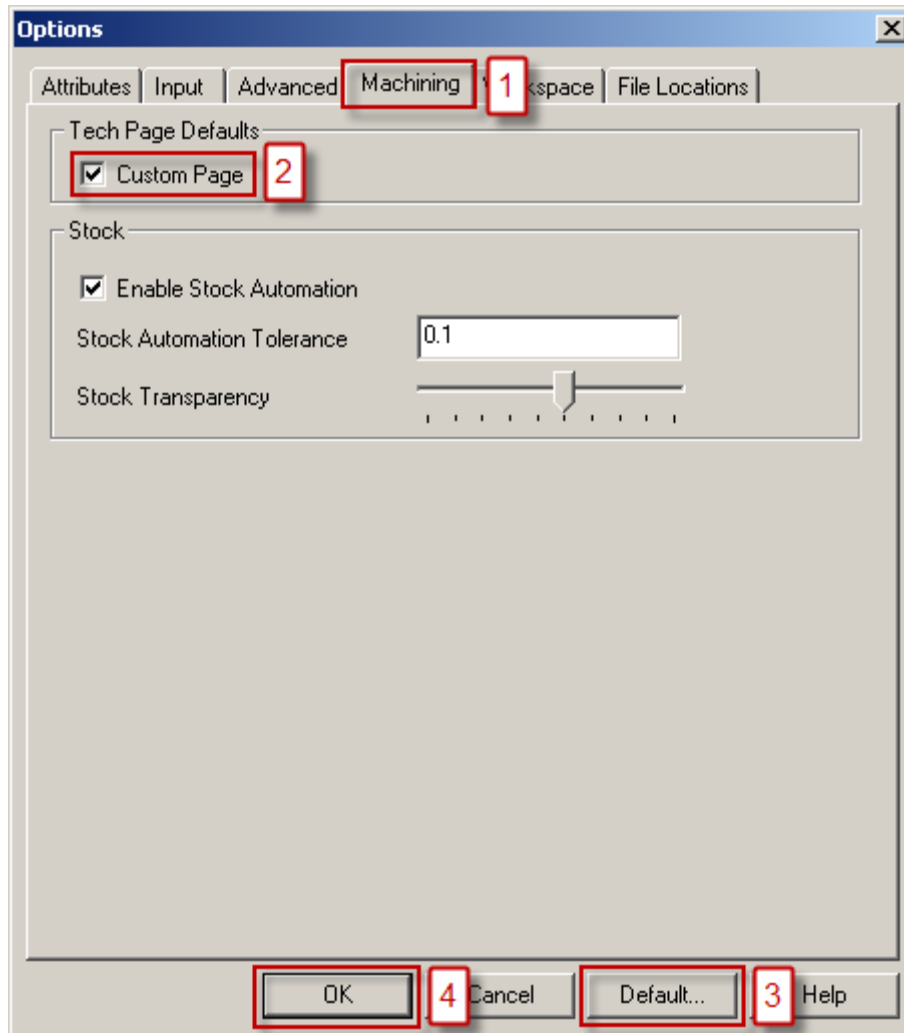
### 3. How to turn on Custom Pages / POST Output Configuration

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.



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 when the MoriSeiki AddIn is loaded, 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.

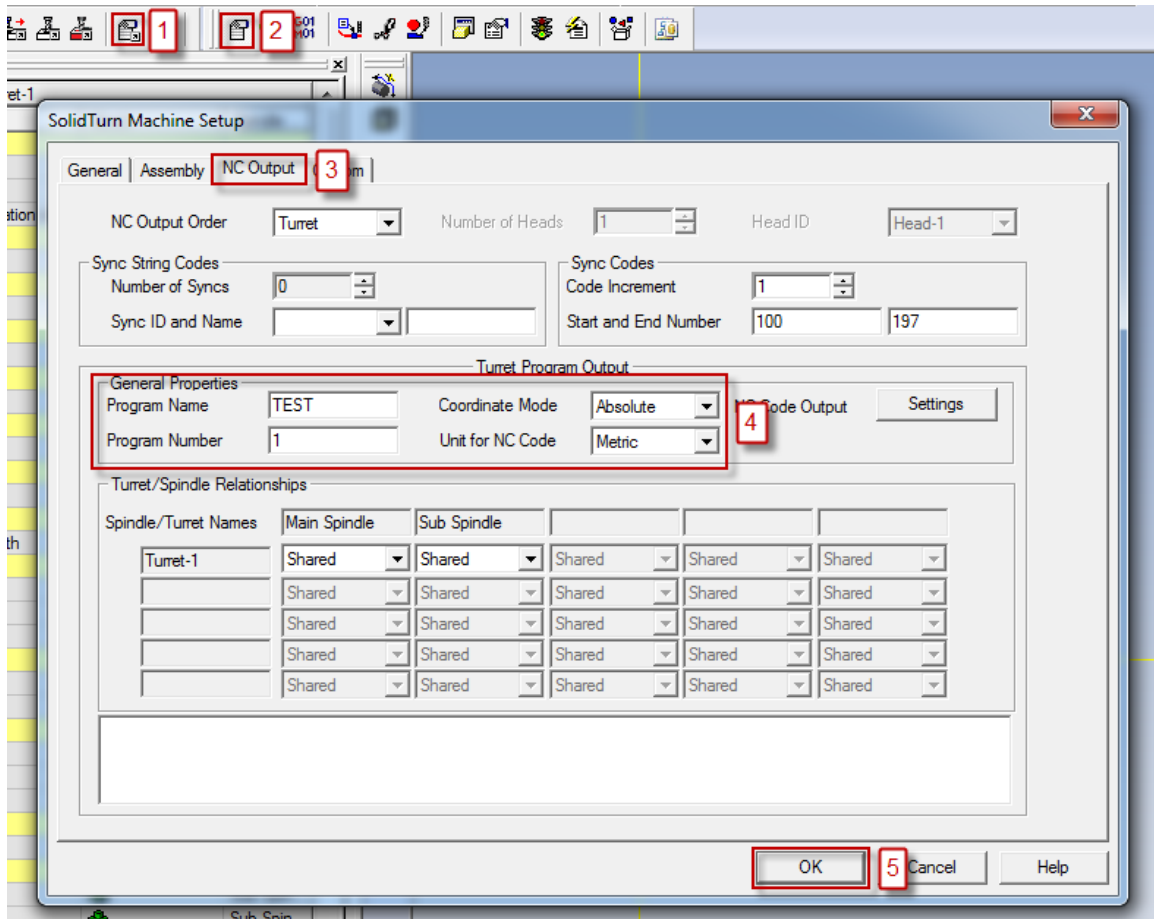


To open the Machine Setup, click on Common Machining and then on Setup:



#### 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.



#### 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.

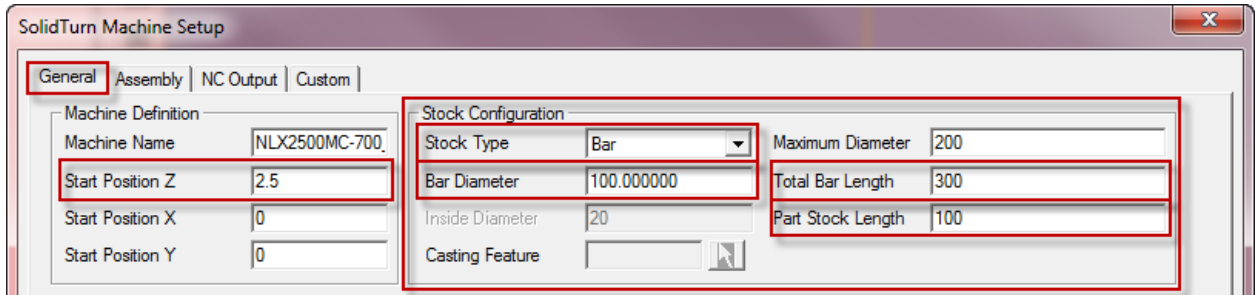
Use **Start Position Z** to position the stock along the Z axis. It will be used, for example, to define the front face facing amount.

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.



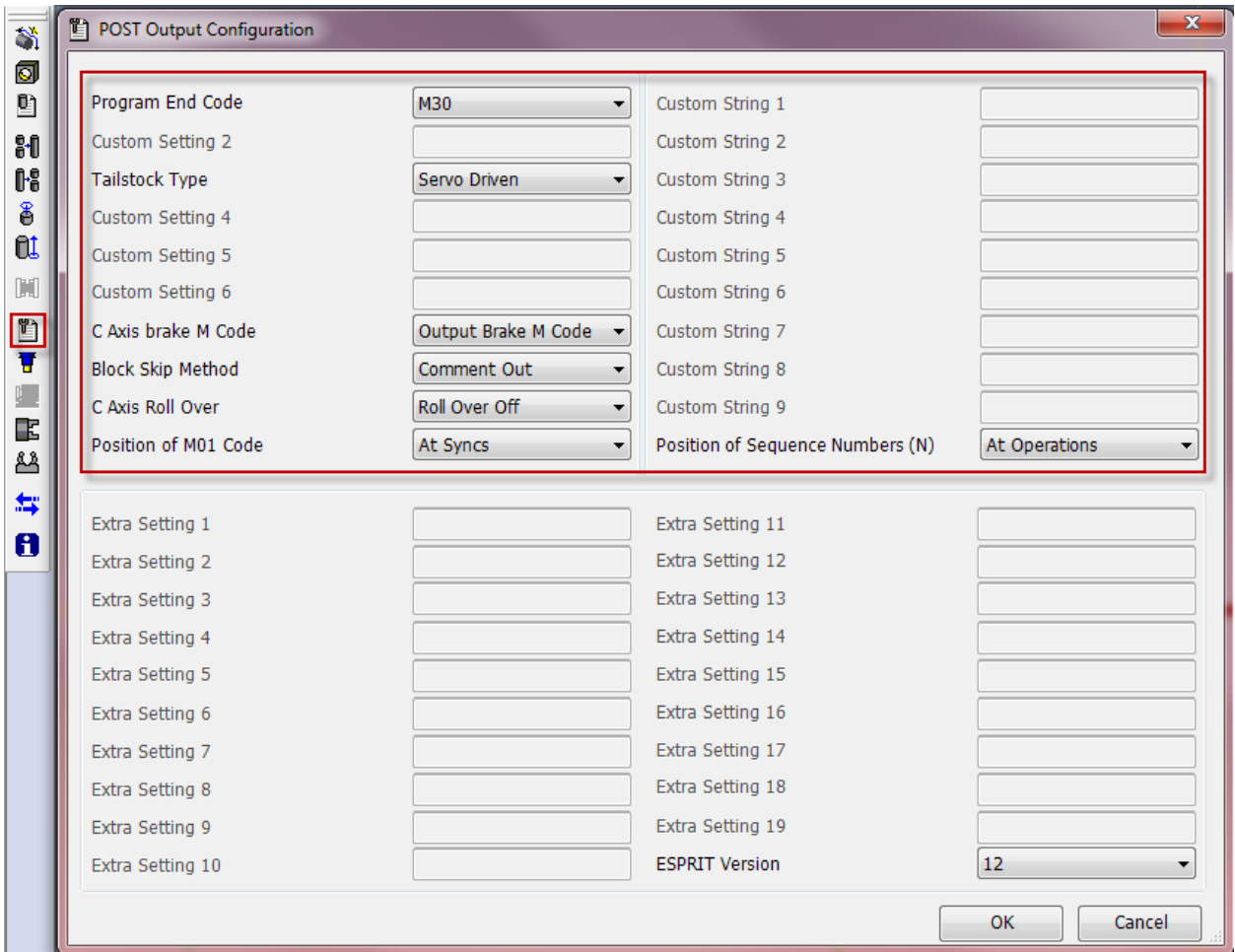
*Note:* The stock configuration will be detailed when needed in this manual.

## **4.4. Machine parameters**

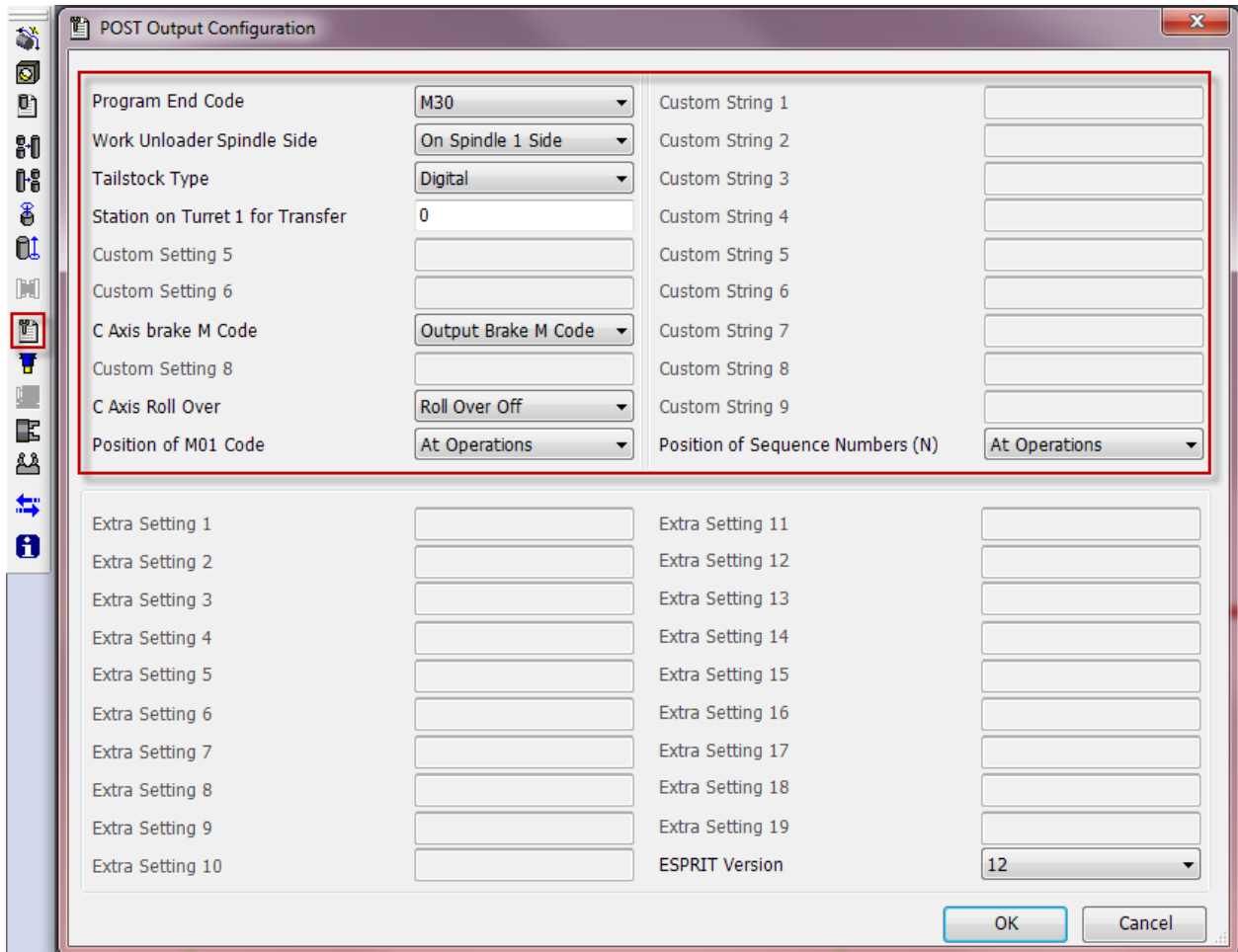
### *4.4.1. Introduction*

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

For NZX configuration:



For NLX configuration:



#### 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 1 or spindle 2 side (for NLX configuration only)

On the POST Output Configuration, set **Work Unloader Spindle Side** to **On Spindle 2 Side** if the work unloader of your machine is installed on the spindle 2 side; set it to **On Spindle 1 Side** if it is installed on the spindle 1 side.

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

Please note that machine with work unloader installed on spindle 1 side is the default.

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

#### 4.4.4. Type of tailstock installed on the machine

On the POST Output Configuration, set **Tailstock Type** to **Carriage Direct-Coupled** if your machine is equipped with a carriage direct-coupled tailstock; set it to **Servo Driven (Digital for NLX configuration)** if it is equipped with a servo driven (digital for NLX configuration) tailstock.

This will affect the NC output when you program your tailstock operations. It will differ based on the type of tailstock present on your machine. In addition for NZX configuration, based on the type of tailstock and machine, you will have to program your tailstock on head 1 or on head 2:

	NZX2500	NZX4000	NZX6000
Servo driven tailstock	HEAD1	-	HEAD1
Carriage direct-coupled tailstock	HEAD1	HEAD2	HEAD2

Please note that machine with servo driven (digital for NLX configuration) tailstock is the default.

For additional information on how to program tailstock, see part **15.1. Tailstock**.

#### 4.4.5. Tool station for part transfer and work unloader (for NLX configuration only)

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 textbox **Station on Turret 1 for Transfer**. You can enter a value between 1 and 10 (for turret with 10 tool stations), between 1 and 12 (for turret with 12 tool stations; option on machine) or between 1 and 20 (for turret with 20 tool stations; option on machine).

For example, if Station on Turret 1 for Transfer is set to 4, the tool call for transfer will be T0400.

*Note:* If the station number specified is out of range (value entered greater than 10 (for a turret with 10 tool stations for example) 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**.

#### 4.4.6. 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**.

#### 4.4.7. Set how to skip turning spindle control (for NZX configuration only)

When using spindle priority on turning cycles (see part **8.2. 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).

#### 4.4.8. 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.**

#### *4.4.9. 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 in the NC code for NZX configuration (this way, optional stop can be turned on on the machine while running the NC code on both heads at the same time (production mode)) and will be output after each operation in the NC code for NLX configuration.

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

#### *4.4.10. 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####;

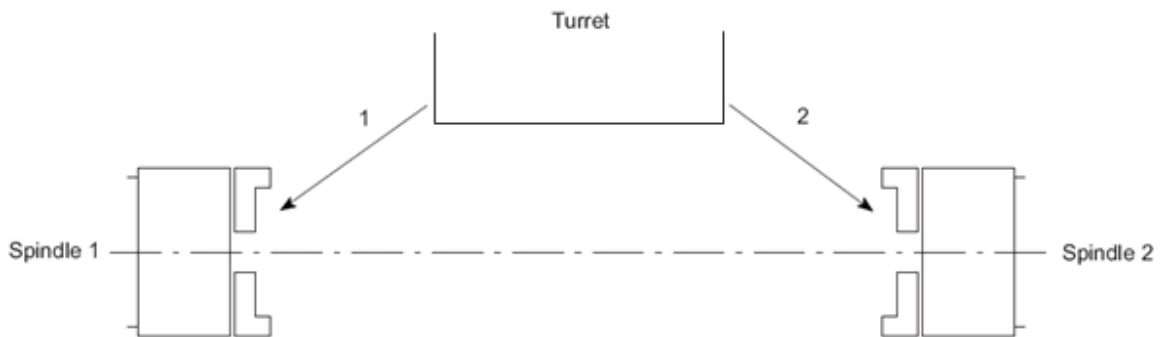
The first two digits of a T number specify the tool number and the tool geometry offset number.

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

NZX2500 – 4000 – 6000:

Machine	Turret to Use	T Code
NZL6000 Series	Turret 1	T0101 ~ T1212
	Turret 2	T0101 ~ T1010

NLX1500 – 2000 – 2500:



**For turret with 10 tool stations:**

	Spindle to use	T Code to specify
1	Spindle 1	T0101 - T1010
2	Spindle 2	T0131 - T1040

**For turret with 12 tool stations (option on machine):**



	<b>Spindle to use</b>	<b>T Code to specify</b>
1	Spindle 1	T0101 - T1212
2	Spindle 2	T0131 - T1242

**For turret with 20 tool stations (option on machine):**

	<b>Spindle to use</b>	<b>T Code to specify</b>
1	Spindle 1	T0101 – T2020
2	Spindle 2	T0131 – T2050

*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:**

**Turning Tools - Turning Insert** [X]

Insert | Holder | General

Common Parameters

Tool ID	Tool 14	Orientation	3V
<b>Tool Number</b>	<b>101</b>	Orientation Angle	0.000000
Turret Name	(Unmounted)	Compensation	Corner
Station Name	Station:1	Length Register	101
Spindle Direction	CW	Touch Off Angle	0.000000
Coolant	On		
Simulation Cut Color	[Red]		

Tool Change

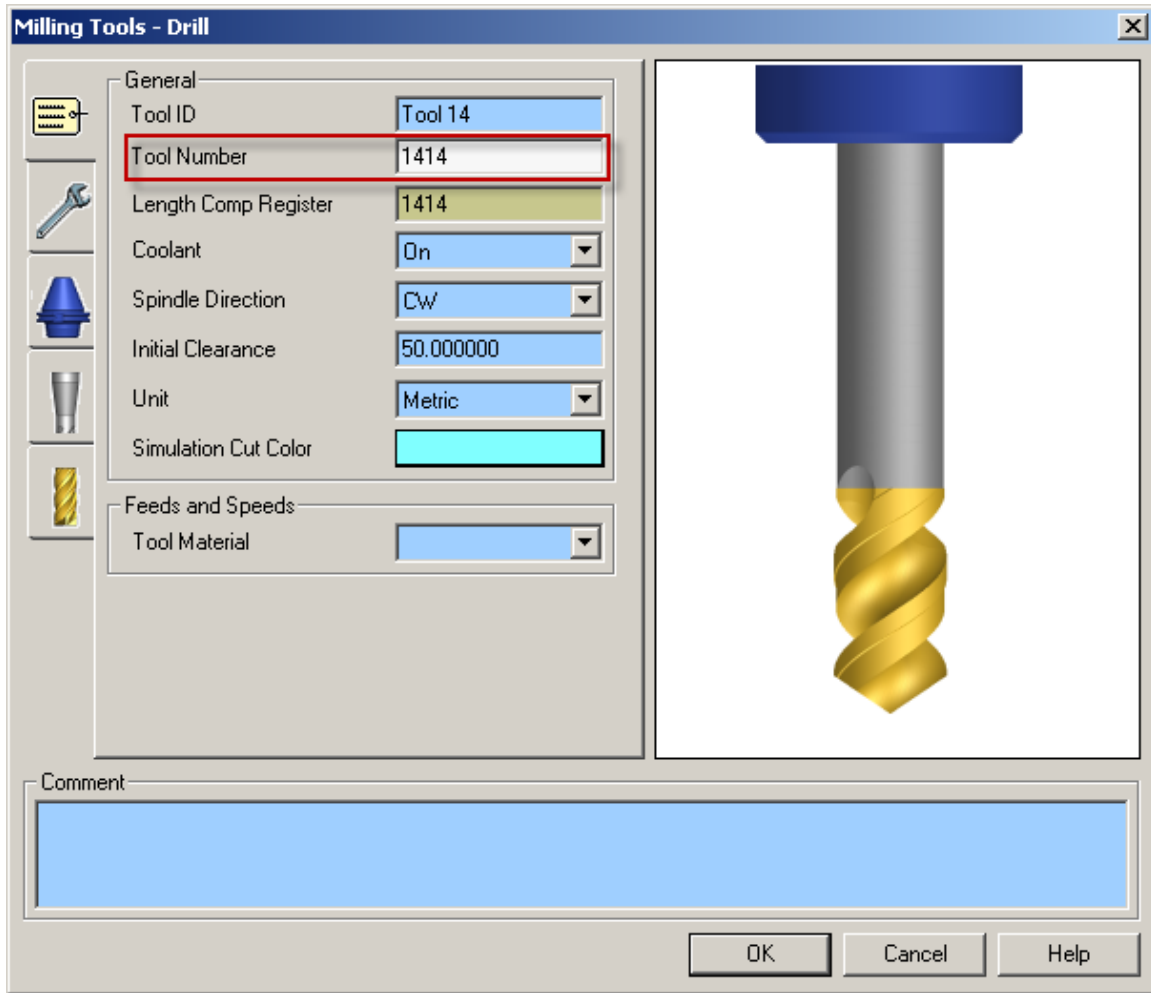
Movement X,Z,Y	Home	Home	Home
Position X,Z,Y	125.000000	125.000000	0.000000

Tool Shift

Tool Shift X, Z, Y	0.000000	0.000000	0.000000
--------------------	----------	----------	----------

[OK] [Cancel] [Help]

**Milling tools:**



### 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:**

**Turning Tools - Turning Insert** [X]

Insert | Holder | General

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	Touch Off Angle	0.000000
Coolant	On		
Simulation Cut Color	[Red]		

Tool Change

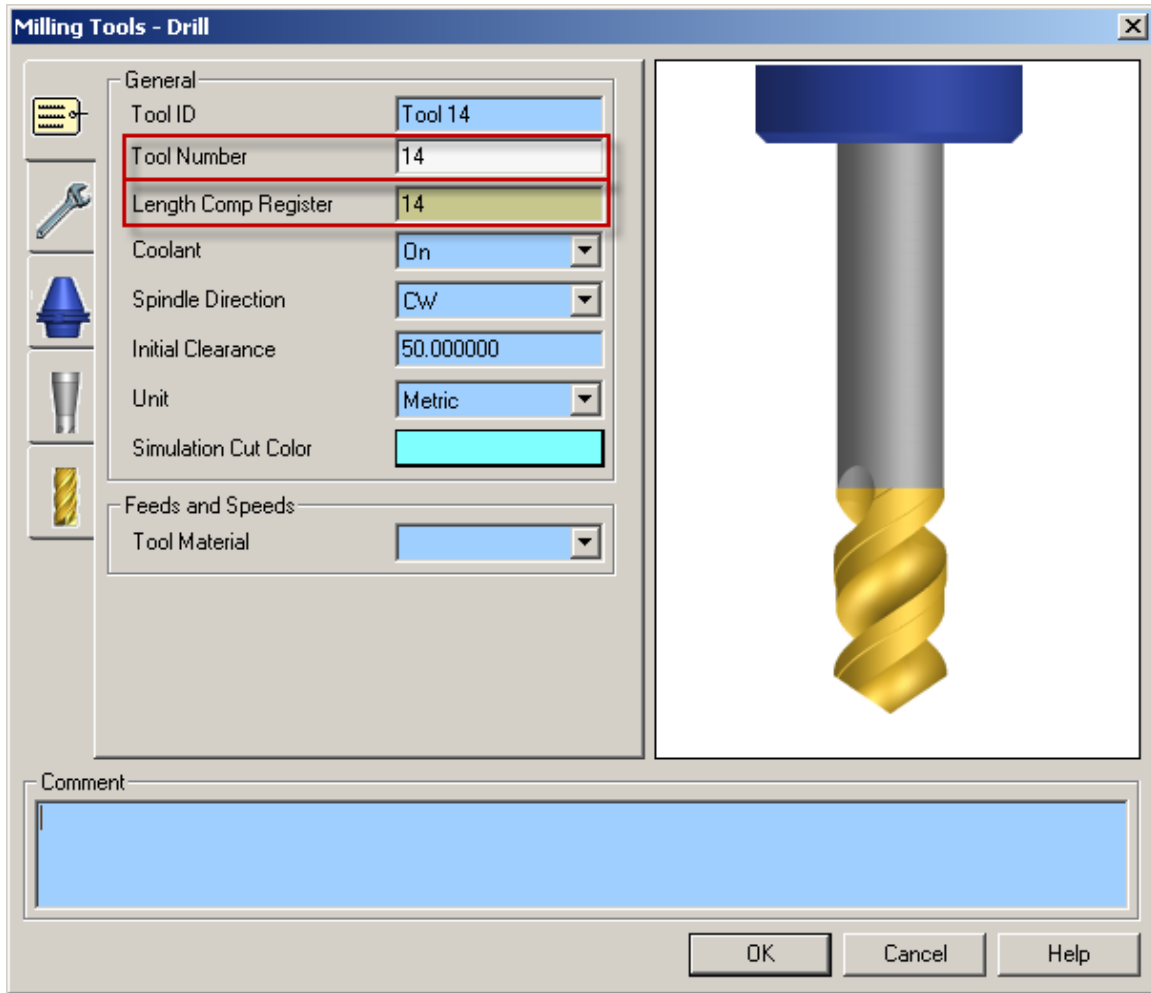
Movement X,Z,Y	Home	Home	Home
Position X,Z,Y	125.000000	125.000000	0.000000

Tool Shift

Tool Shift X, Z, Y	0.000000	0.000000	0.000000
--------------------	----------	----------	----------

[OK] [Cancel] [Help]

**Milling tools:**



#### 5.1.4. Additional information: Tool call when machining on spindle 2 side

When machining on spindle 1, 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, the post processor will add **30** to the tool number entered in the Tool Number dialogue box if method 1 is used or to Length Register number if method 2 is used.

See the examples below:

- *Example 1 for method 1:*

If 101 is entered in the Tool Number dialogue box, T0131 will be output in the NC code.

- *Example 2 for method 1:*

If 1212 is entered in the Tool Number dialogue box, T1242 will be output in the NC code.

- *Example 1 for method 2:*

If Tool Number is set to 1 and Length Register number is set to 3, T0133 will be output in the NC code.

- *Example 2 for method 2:*

If Tool Number is set to 10 and Length Register number is set to 10, T1040 will be output in the NC code.

### 5.1.5. Restrictions

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

The two first digits of a tool call entered in ESPRIT have to be contained **between 1 and 12 for turret 1** and **between 1 and 10 for turret 2**.

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

The two first digits of a tool call entered in ESPRIT have to be contained **between 1 and 10 (for turret with 10 tool stations), between 1 and 12 (for turret with 12 tool stations; option on machine) or between 1 and 20 (for turret with 20 tool stations; option on 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 configuration:

- **Head 1:** The entered Length Register number has to be contained **between 1 and 40**.
- **Head 2:** The entered Length Register number has to be contained **between 1 and 40**.

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

- **Machining on spindle 1 side:** The entered Length Register number has to be contained **between 1 and 80**.
- **Machining on spindle 2 side:** The entered Length Register number has to be contained **between 1 and 50**, 30 will be added to this number.

*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

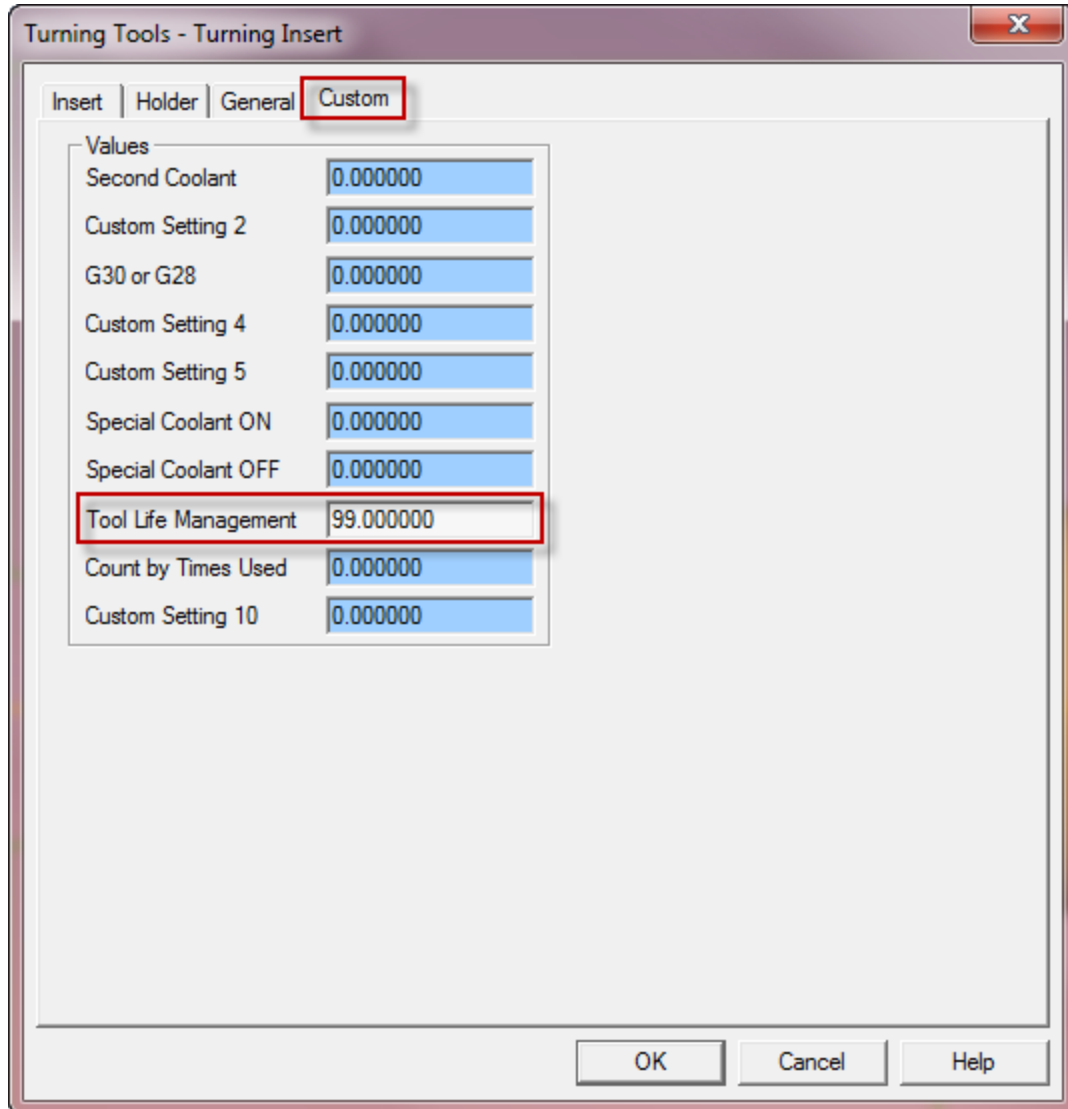
There are two types of tool life management for these machines: one is **counting by length of time**, the other is **counting by times used**.

**Counting by length of time:**

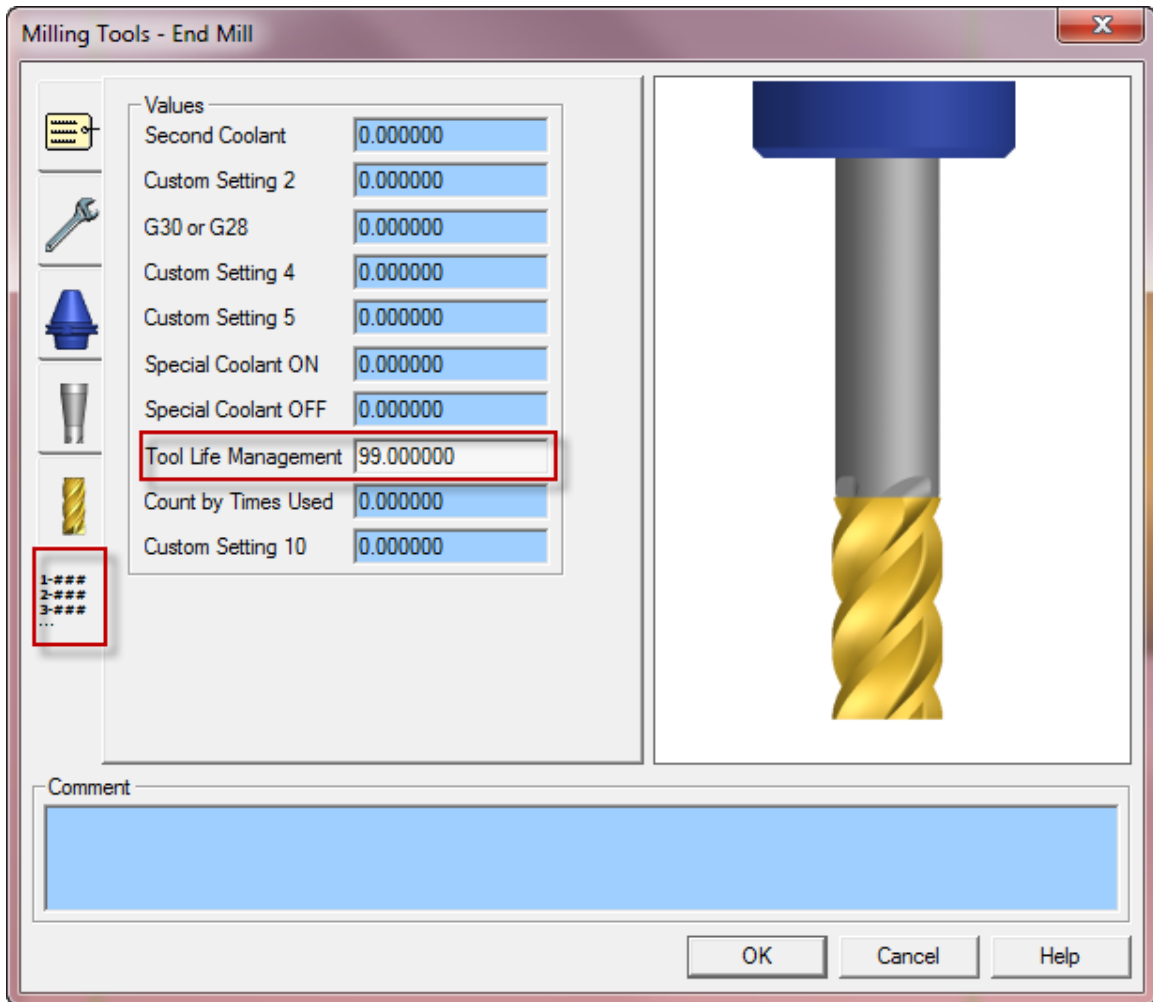
On the Custom tab of the tool page, set Tool Life Management (Custom Setting 8) to 99 to turn on this type of tool life management.

This will trigger the output of T\*\*\*\*99.

**Turning tool page:**



**Milling tool page:**

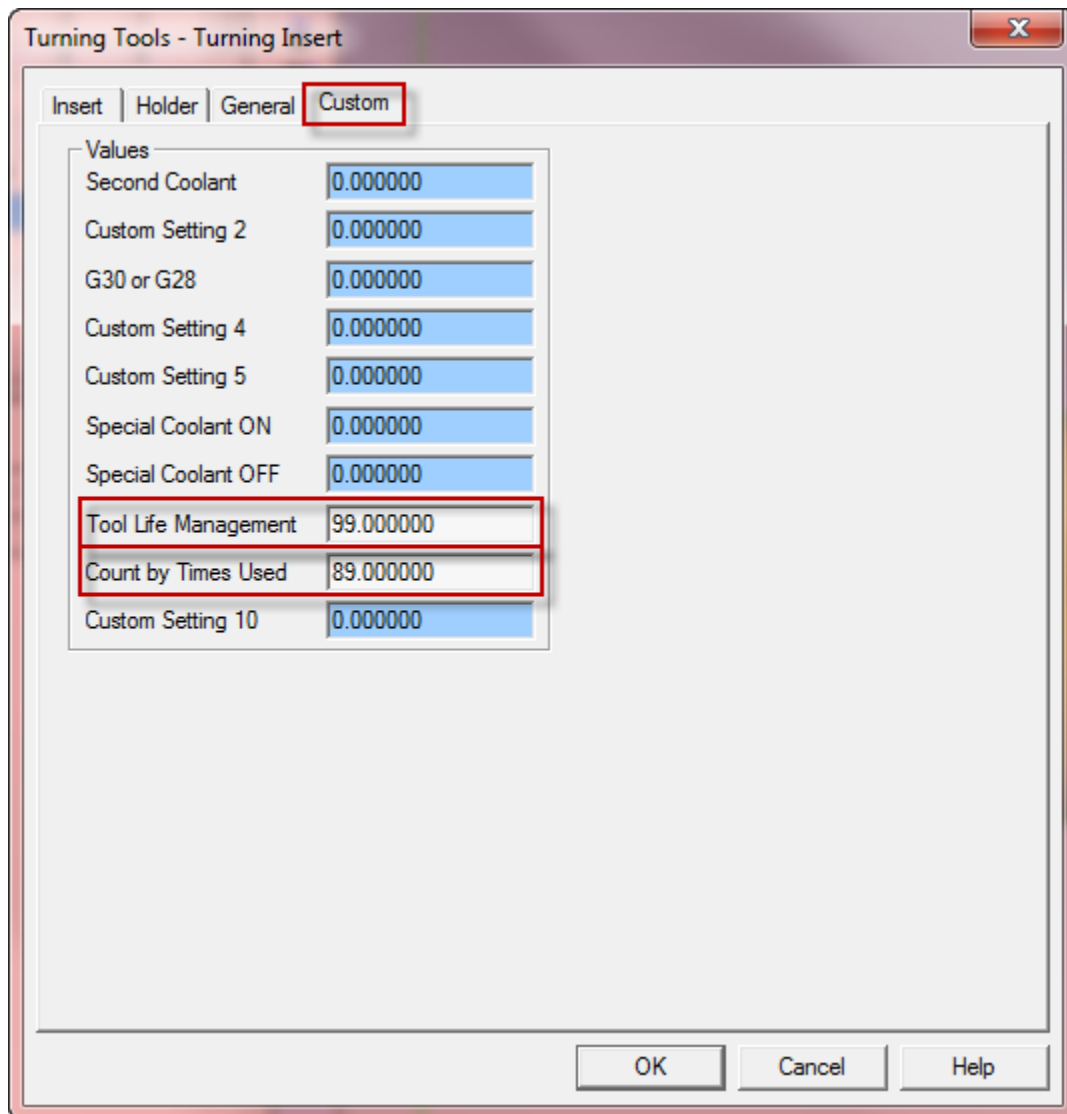


**Counting by times used:**

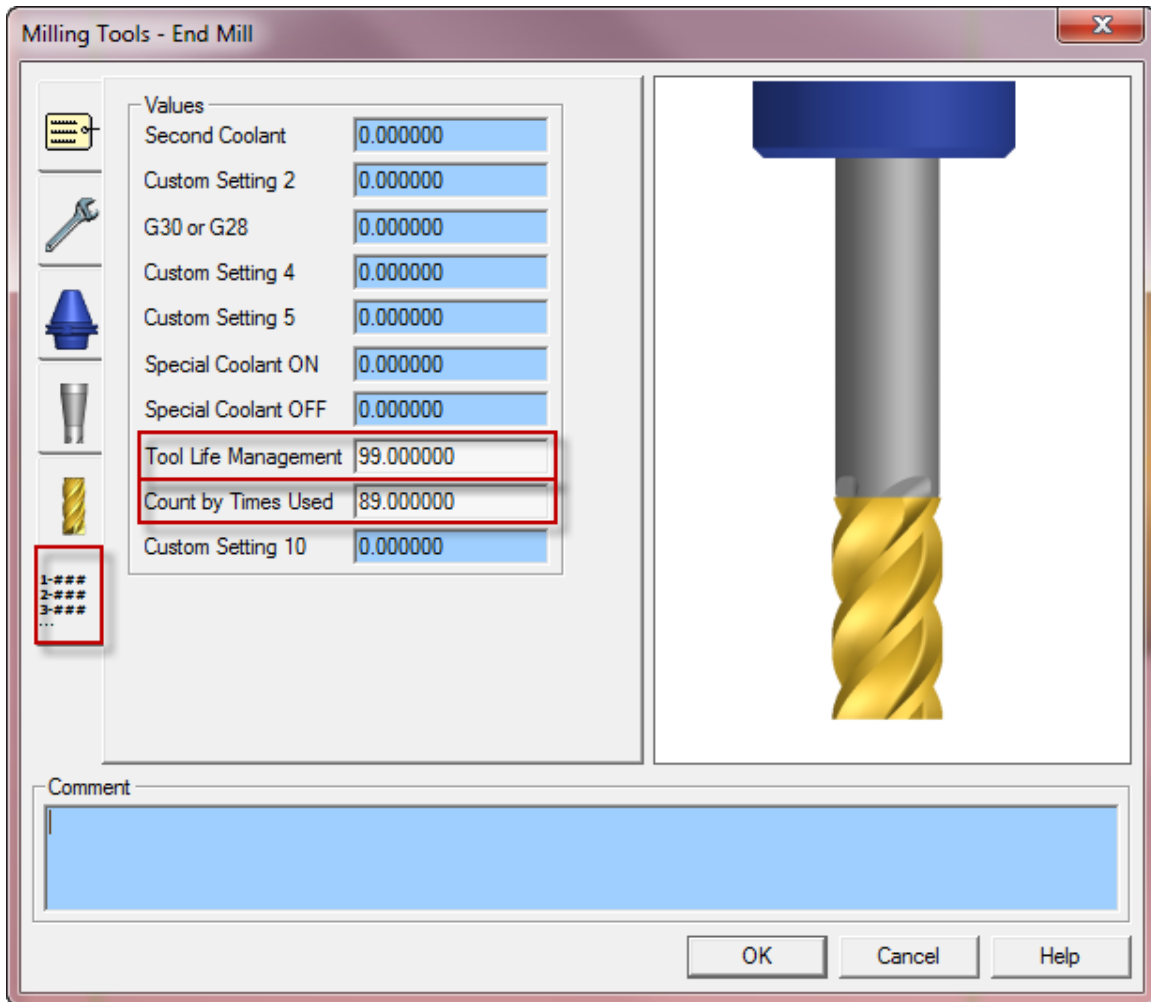
On the Custom tab of the tool page, set Tool Life Management (Custom Setting 8) to 99 and Count by Times Used (Custom Setting 9) to 89 to turn on this type of tool life management.

**Turning tool page:**





**Milling tool page:**

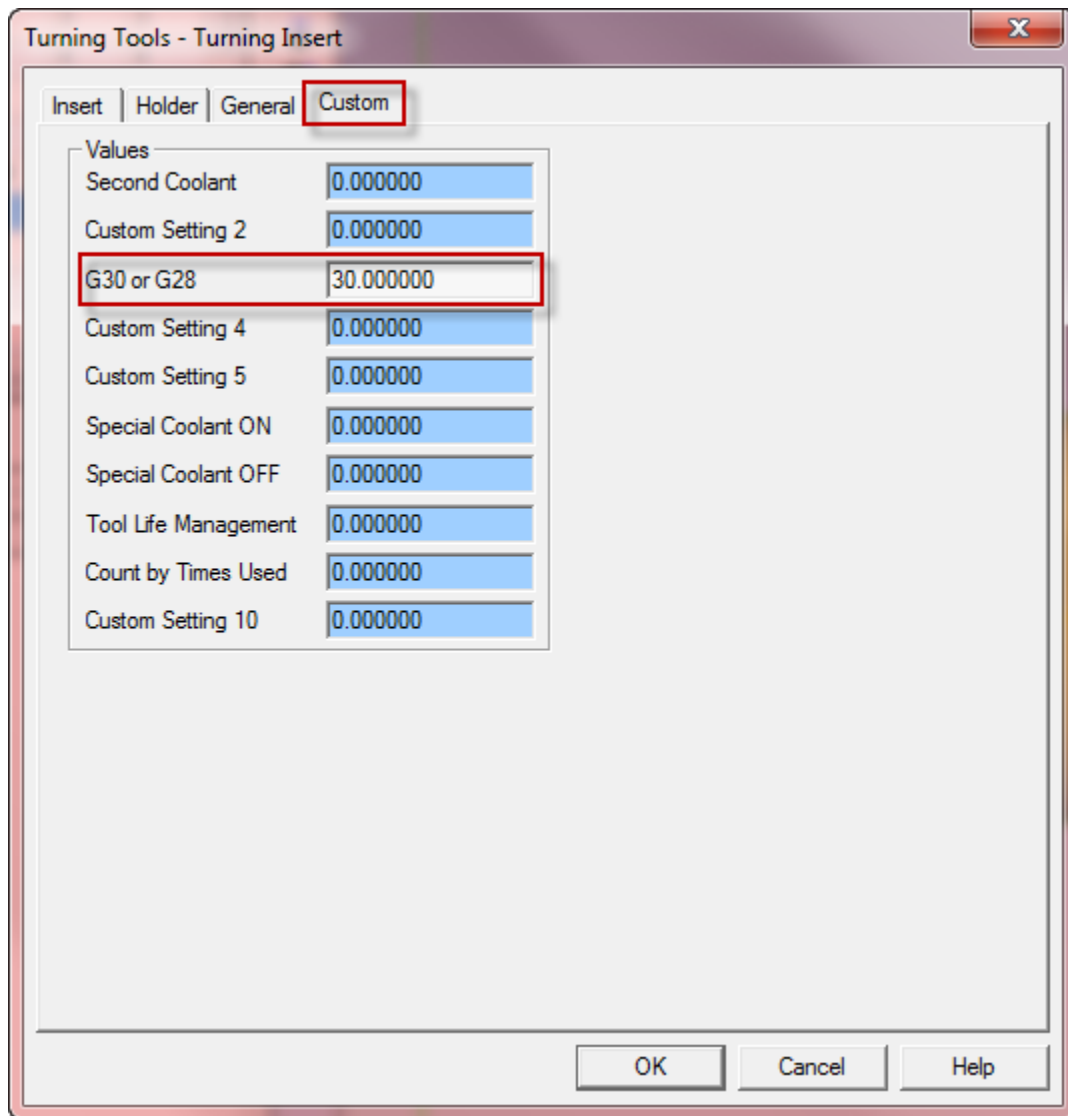


For both types of tool life management, if you omit (or enter a wrong value) to enter the value in the Tool Life Management field, the tools will be called with G00; T\*\*\*\*99 (and M89) will not be output in the NC code.

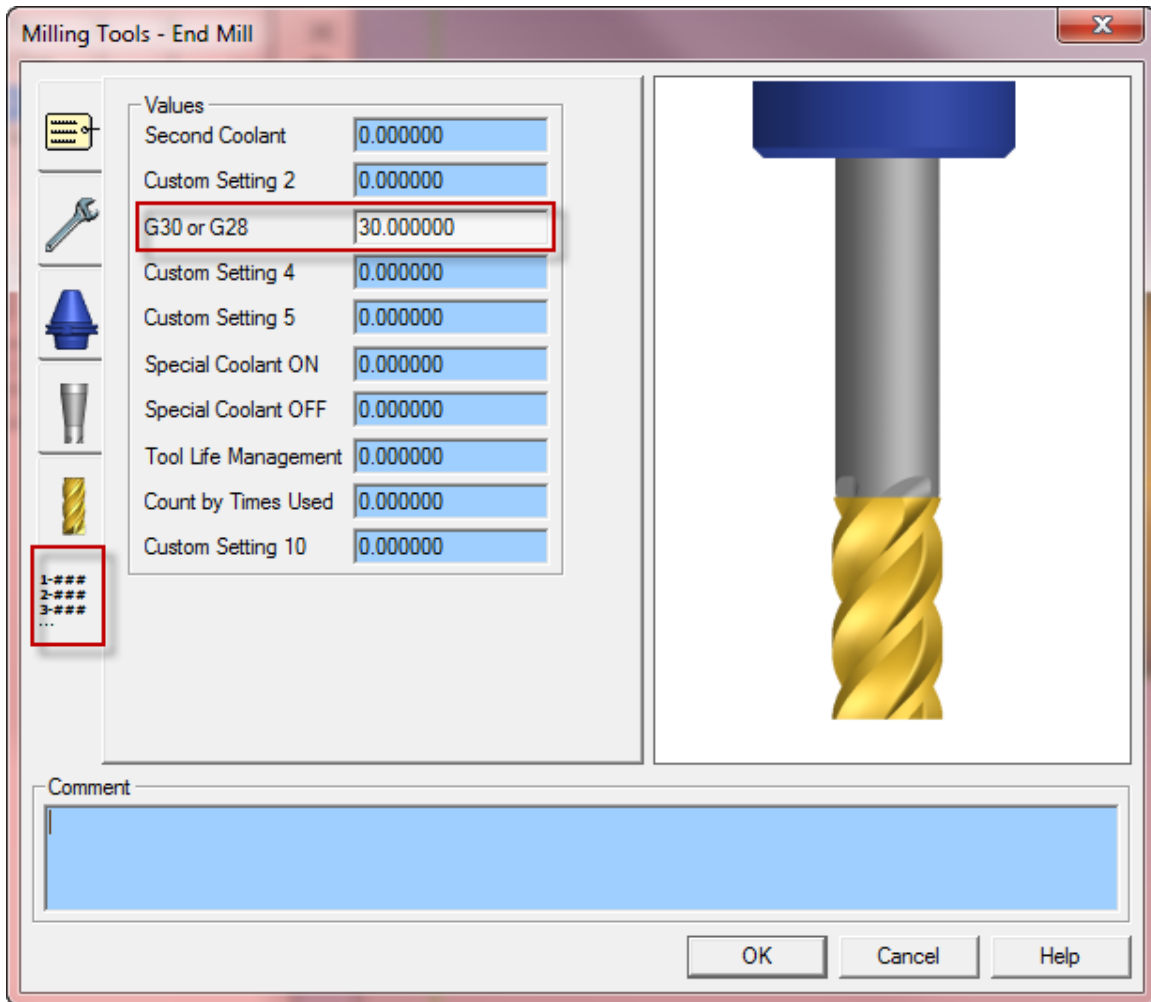
### 5.1.7. Second home position

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:**



**Milling tool page:**



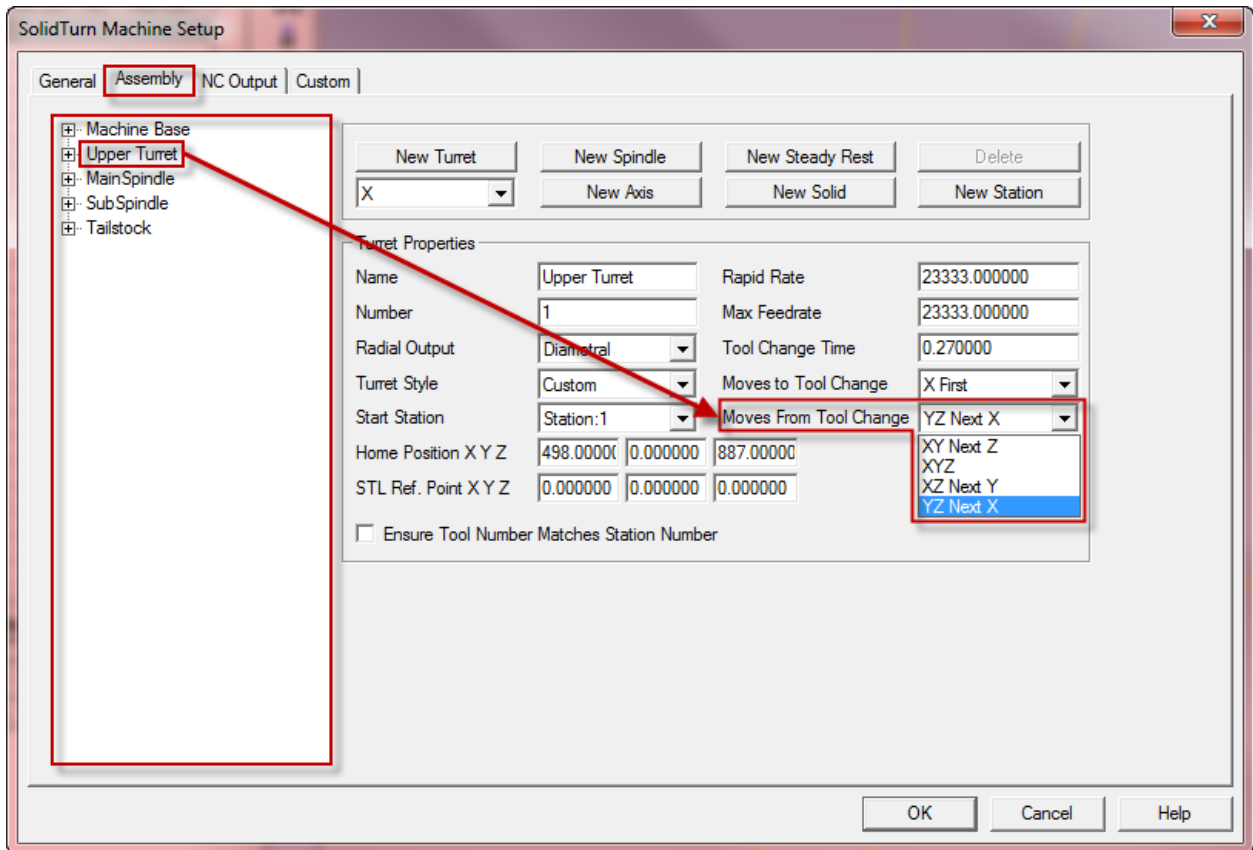
This will trigger the output of G30 (instead of G28) whenever a turret needs to be sent home.

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.



Moves From Tool Change can be set to:

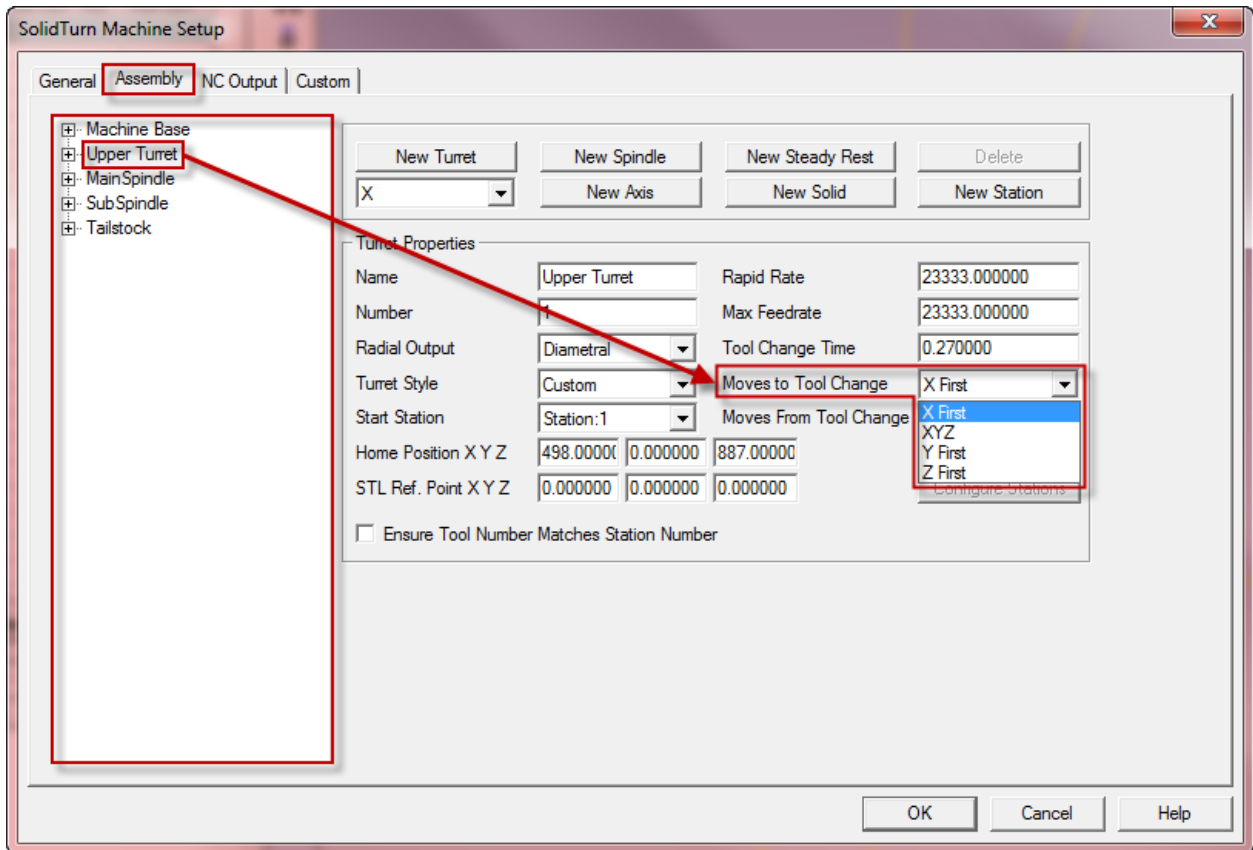
- **XY Next Z:** The tool moves along X and Y only to the entry location, then moves into position along the Z-axis.
- **XYZ:** Linear interpolation is used to move the tool from the tool change position to the start of the cut in a straight line.
- **XZ Next Y:** The tool moves along X and Z only, then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



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.

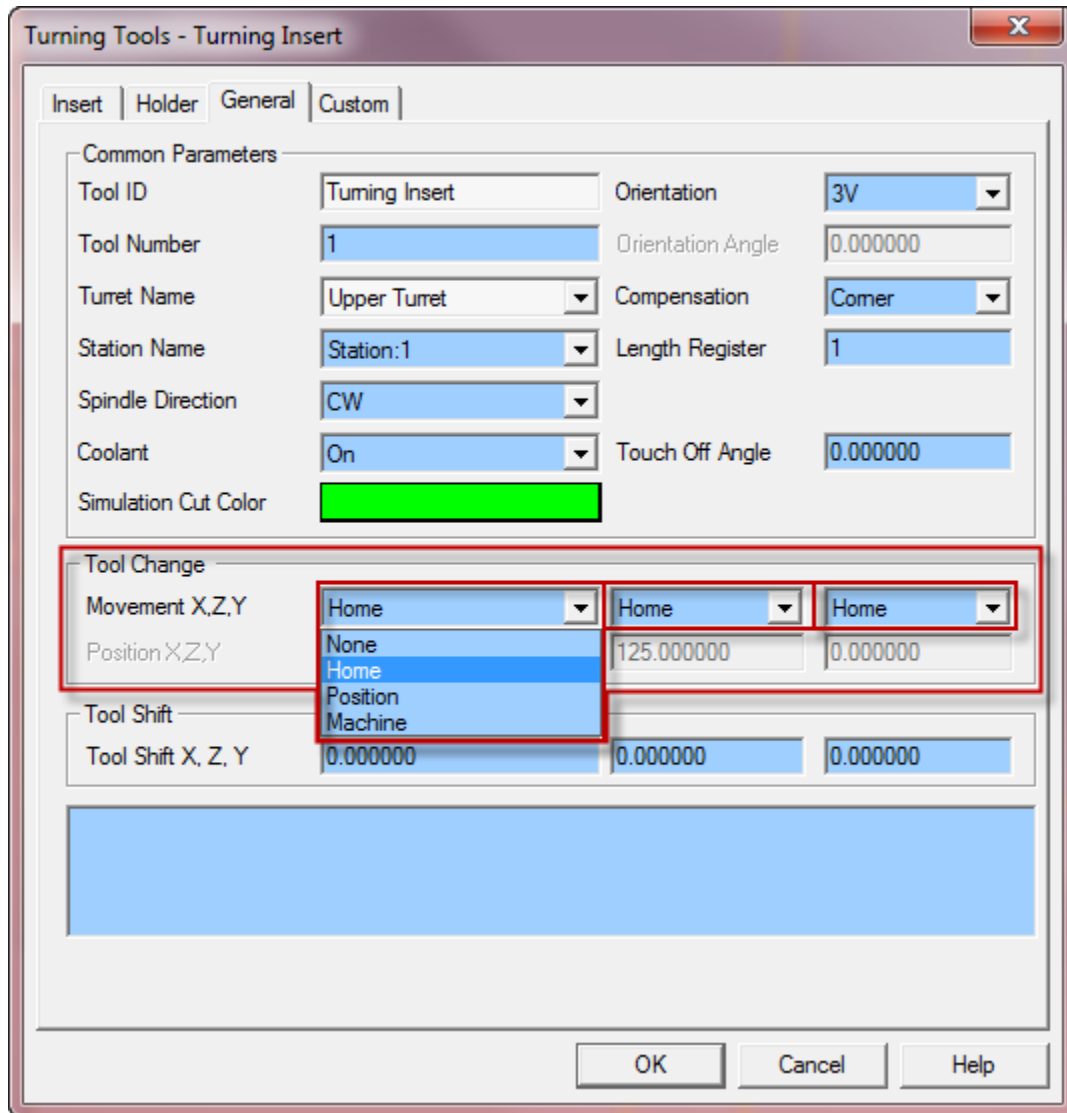
*Notes:*

- 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 or NLX series, the default selection for Moves To Tool Change is X First.

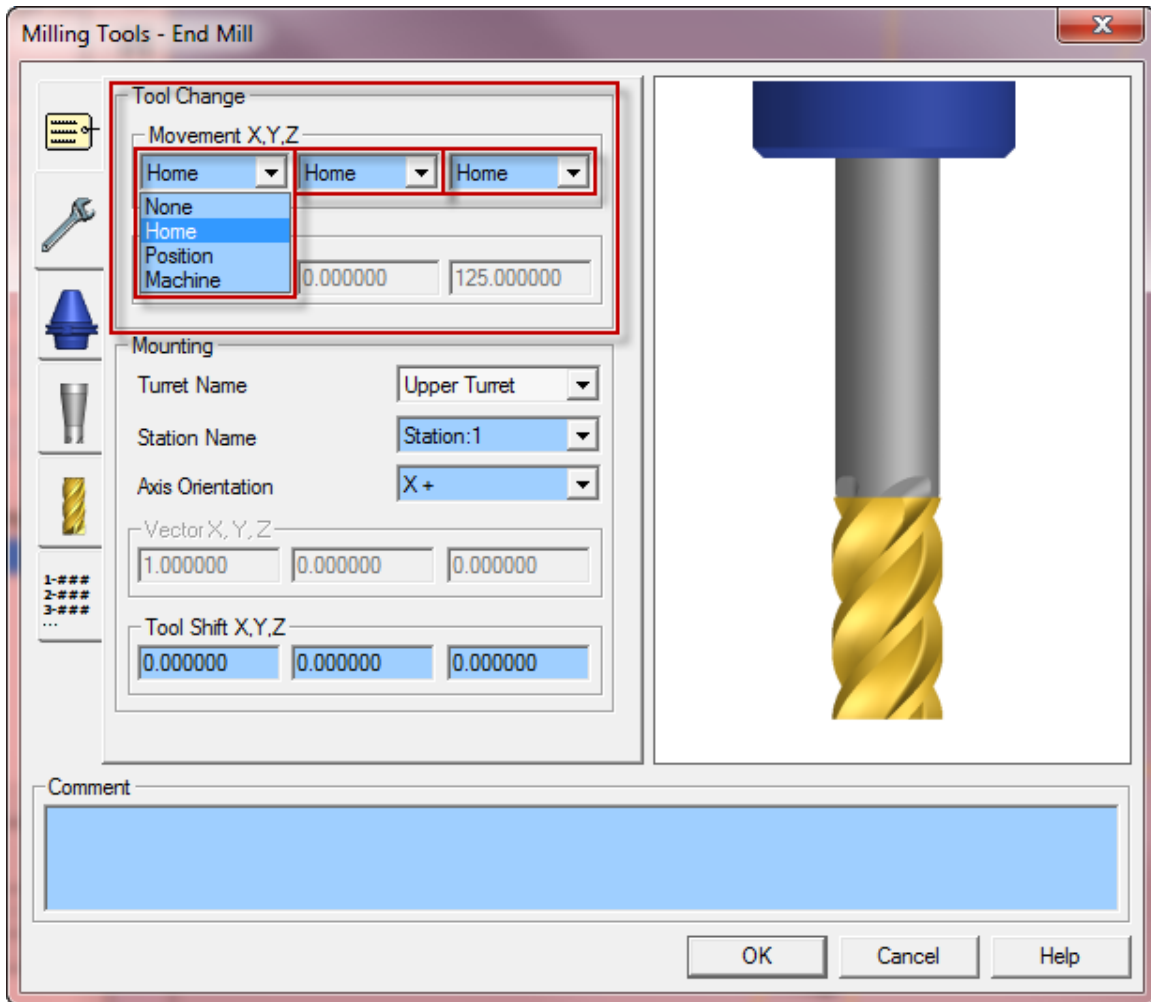
### 5.2.3. Moves to tool change position optimization

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:

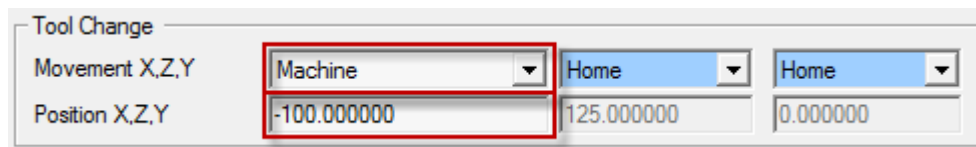


### Milling tools:



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.



- **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.**

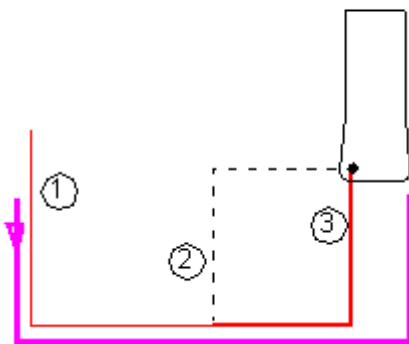
*Notes:*

- 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.

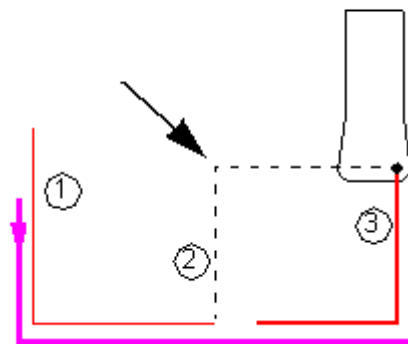
### 5.3. SolidTurn Grooving: Control Edge Shift

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.

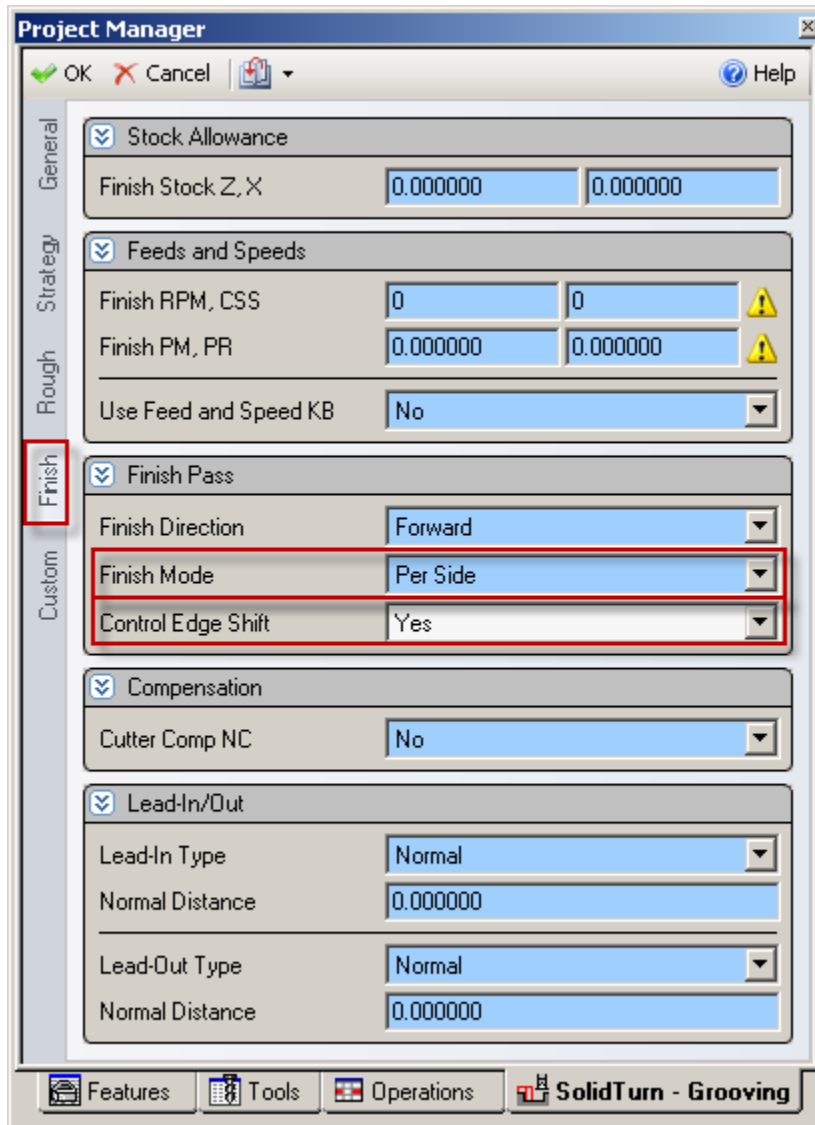
Control Edge Shift = NO

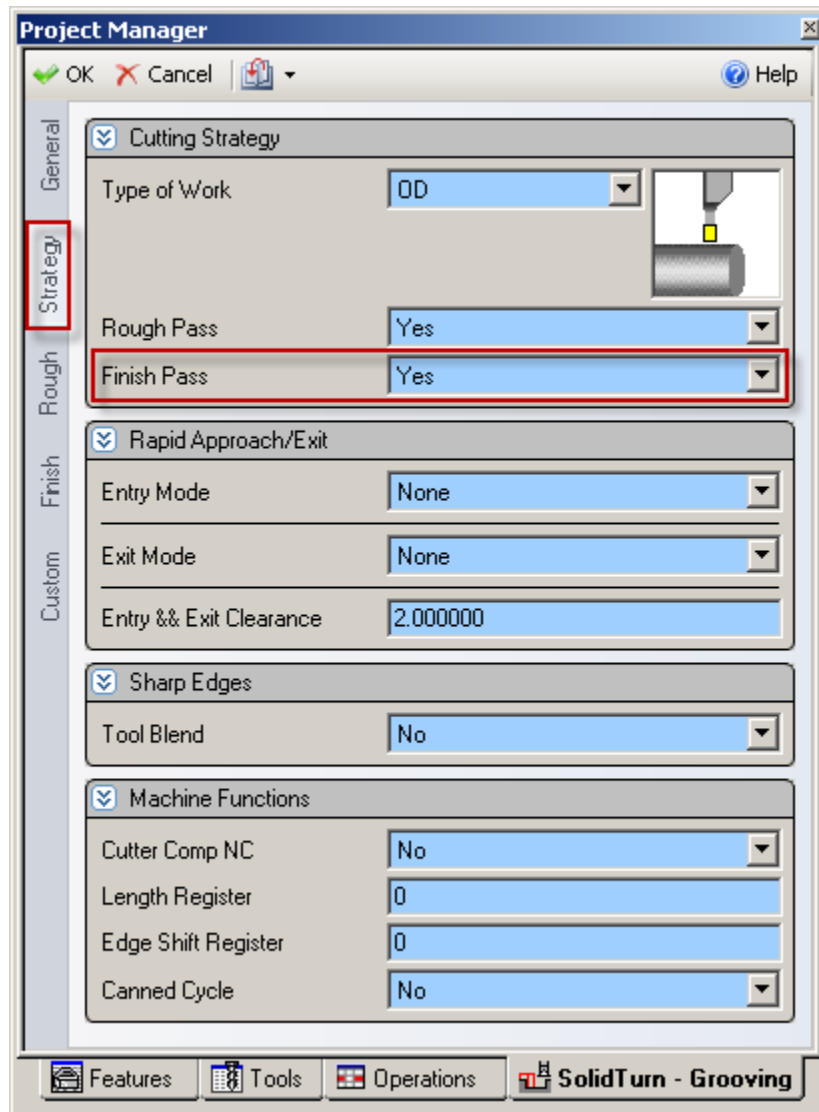


Control Edge Shift = YES



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

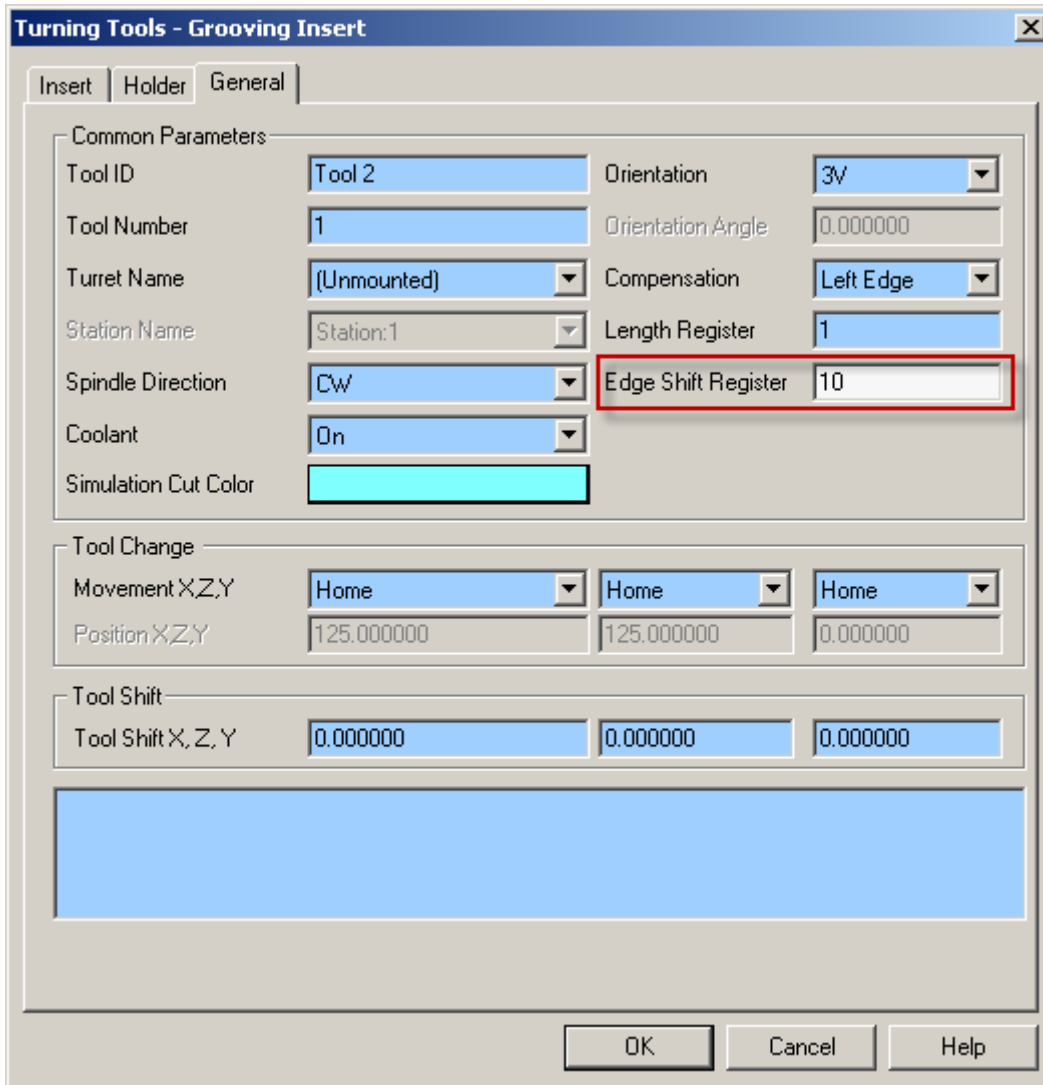




*Notes:*

- 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:



Tool wear offset range of values for NZX configuration:

- **Head 1:** The entered Edge Shift Register number has to be contained **between 1 and 40**.
- **Head 2:** The entered Edge Shift Register number has to be contained **between 1 and 40**.
- **Machining with tool life management on:** The entered Edge Shift Register number has to be contained **between 1 and 40**.

Tool wear offset range of values for NLX configuration:

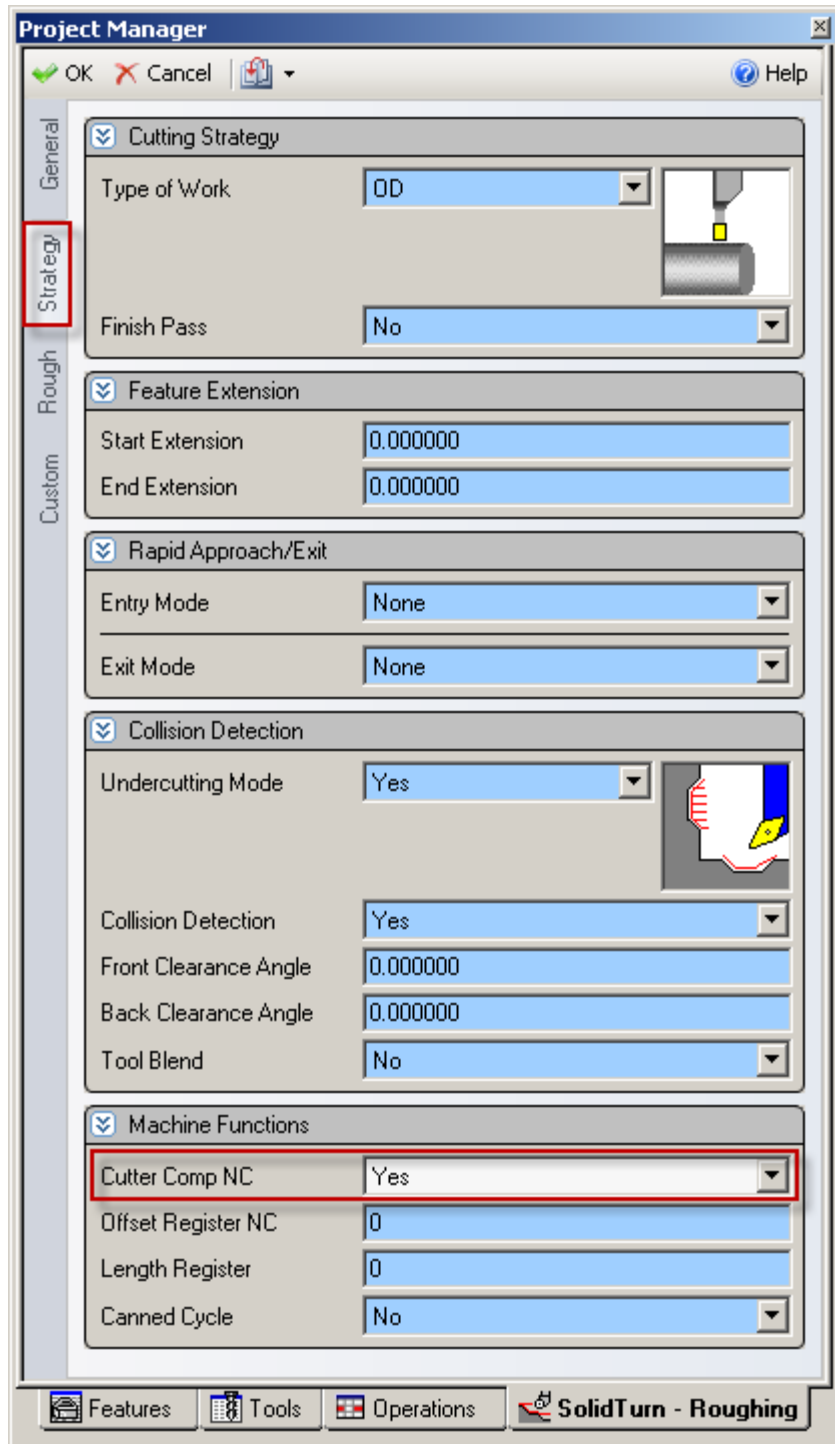
- **Machining on spindle 1 side:** The entered Edge Shift Register number has to be contained **between 1 and 80**.
- **Machining on spindle 2 side:** The entered Edge Shift Register number has to be contained **between 1 and 50**, 30 will be added to this number.
- **Machining with tool life management on:** The entered Edge Shift Register number has to be contained **between 1 and 80**.

*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.**

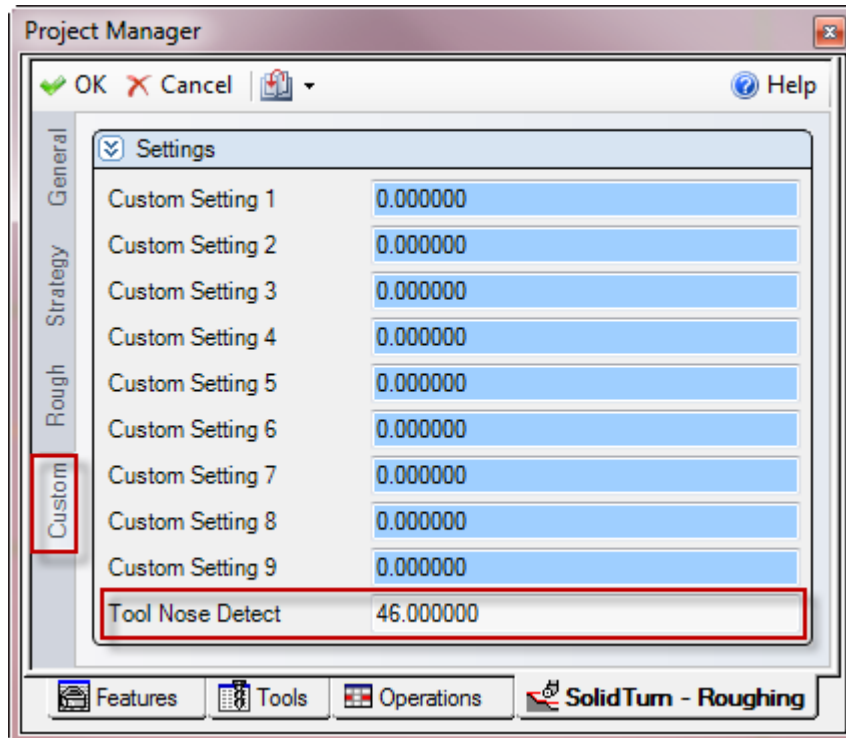
#### **5.4. Automatic Determination of Tool Nose Radius (G46)**

Instead of using G41 or G42 to determine left or right tool offset values, users may choose to use G46 and have their machine automatically determine the correct offset side for the tool using G46. To use G46 for any operation in ESPRIT, the user needs to set Tool Nose Detect (Custom Setting 10) on the operations page to 46. When Cutter Comp NC is set to Yes, the post will automatically output G46 in place of either G41 or G42.

**Operations page, Strategy tab:**



**Operations page, Custom tab:**

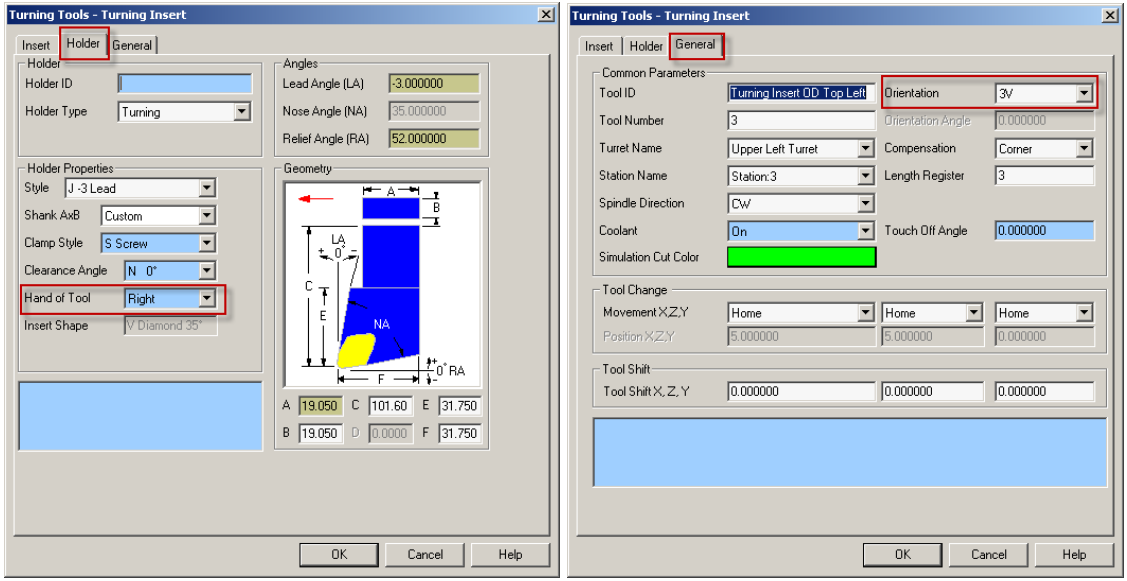


## 6. Spindle direction, speed (S function) and feedrate (F function)

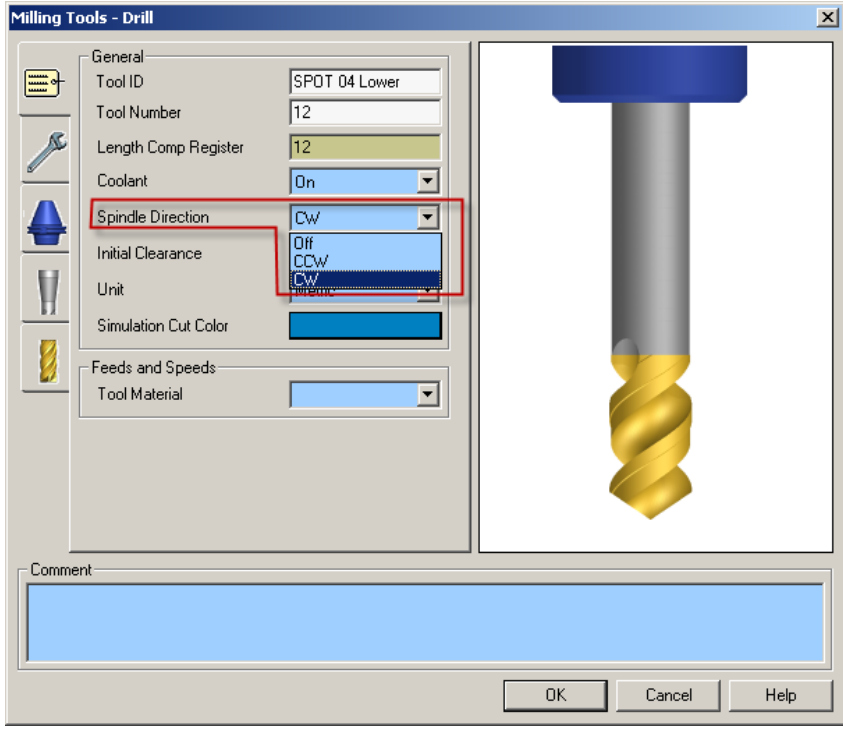
### 6.1. Turning spindle direction and milling tool spindle direction

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.





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.

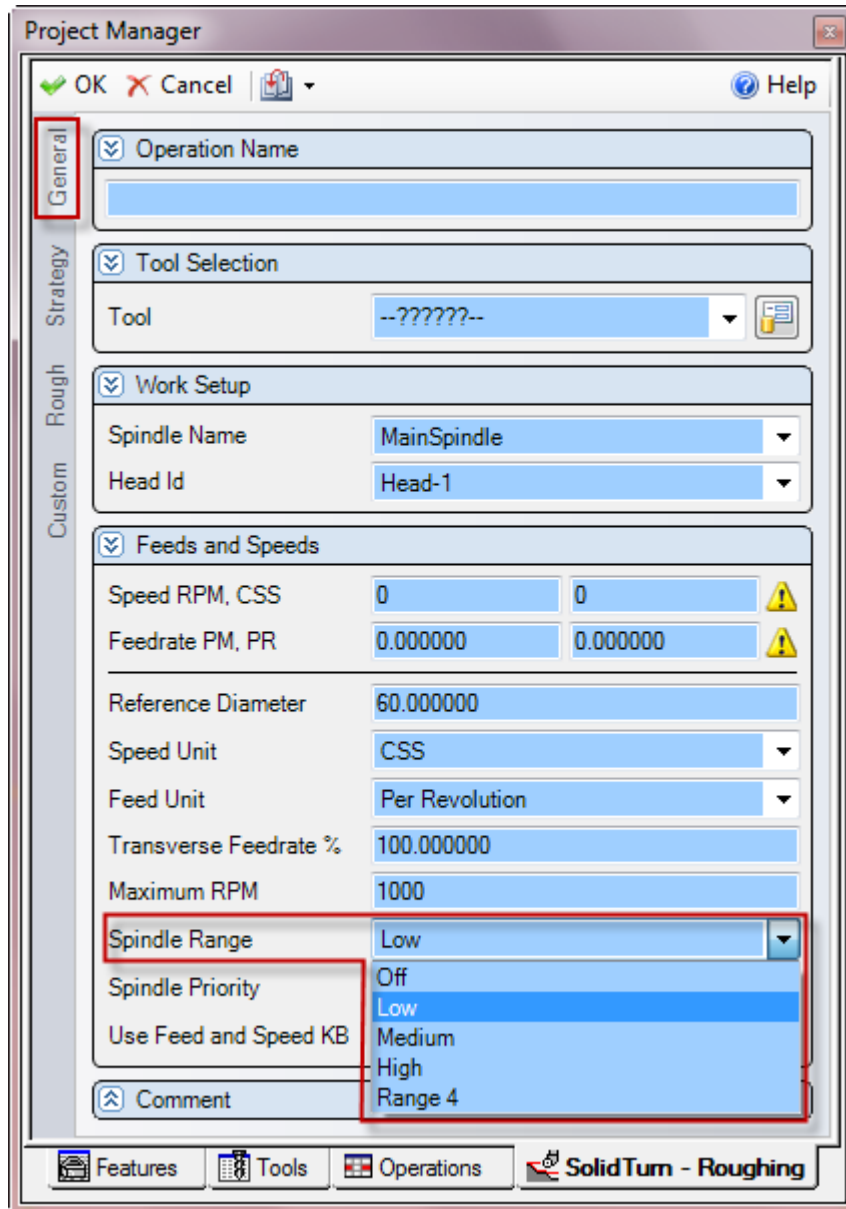


**6.2. Turning spindle gear M-codes (for NZX configuration only)**

The turning spindle on NZX6000 machines (B, C and D types) is equipped with gears, as is the spindle on all NZX4000 machines: M40 (neutral gear, can be used to rotate the

spindle by hand), M41 (low speed gear) and M42 (high speed gear) have to be output in NC code in upper turret for turning operations.

In ESPRIT, the gear M-code can be selected using the Spindle Range pull-down on the General tab of each turning operation page.



**NZX6000 (B and C types) and all NZX4000:**

Set **Spindle Range** to:

- **Off** to output M40
- **Low** to output M41
- **High (Medium or Range 4)** to output M42

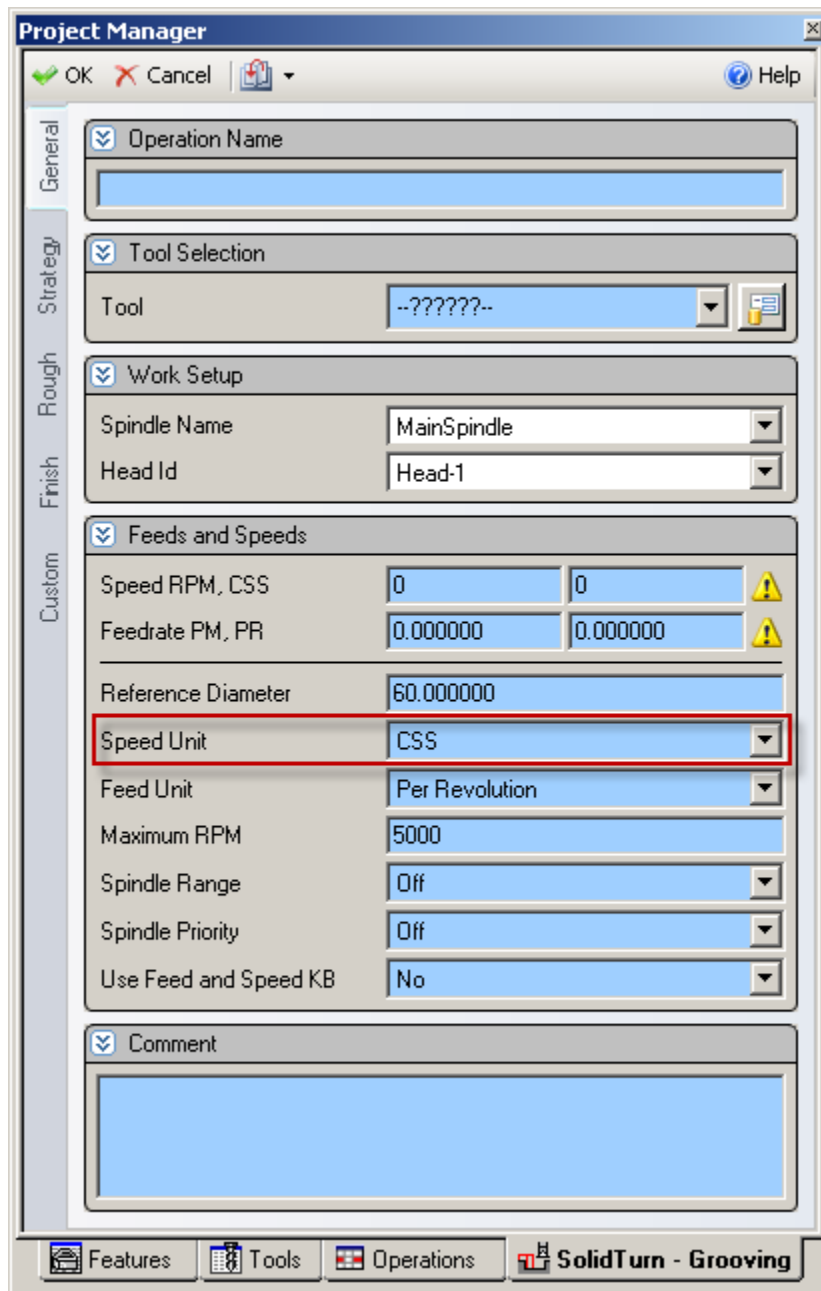
### NZX6000 (D type):

Set Spindle Range to:

- Off to output M40
- Low (Medium, High or Range 4) to output M41

Note that NZX6000 (E type) and all NZX2500 do not have gears: the above described M-codes are not necessary and so will not be output in NC code.

### 6.3. Spindle speed output for CSS unit



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

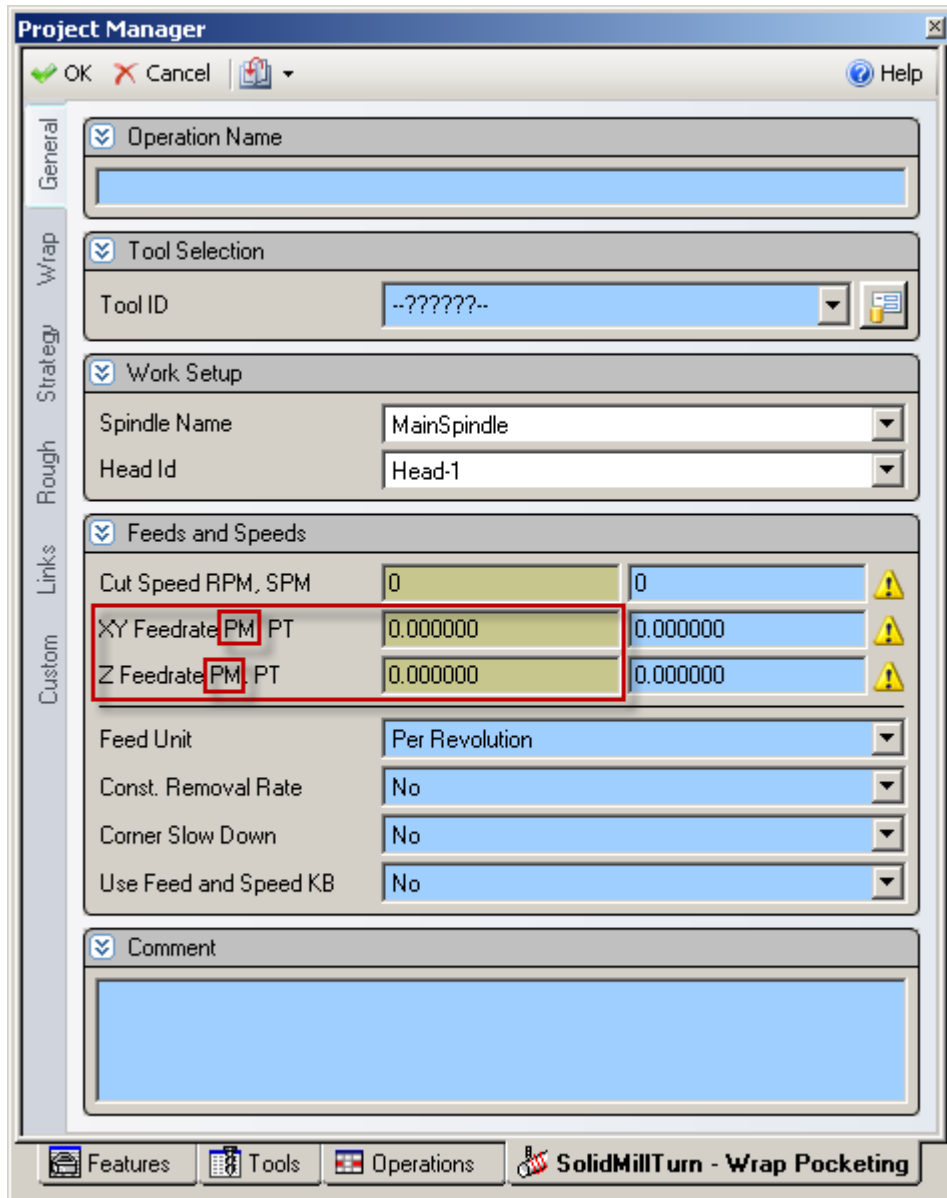
#### 6.4. Feedrate for 4-axis milling operations

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 5<sup>th</sup> 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.

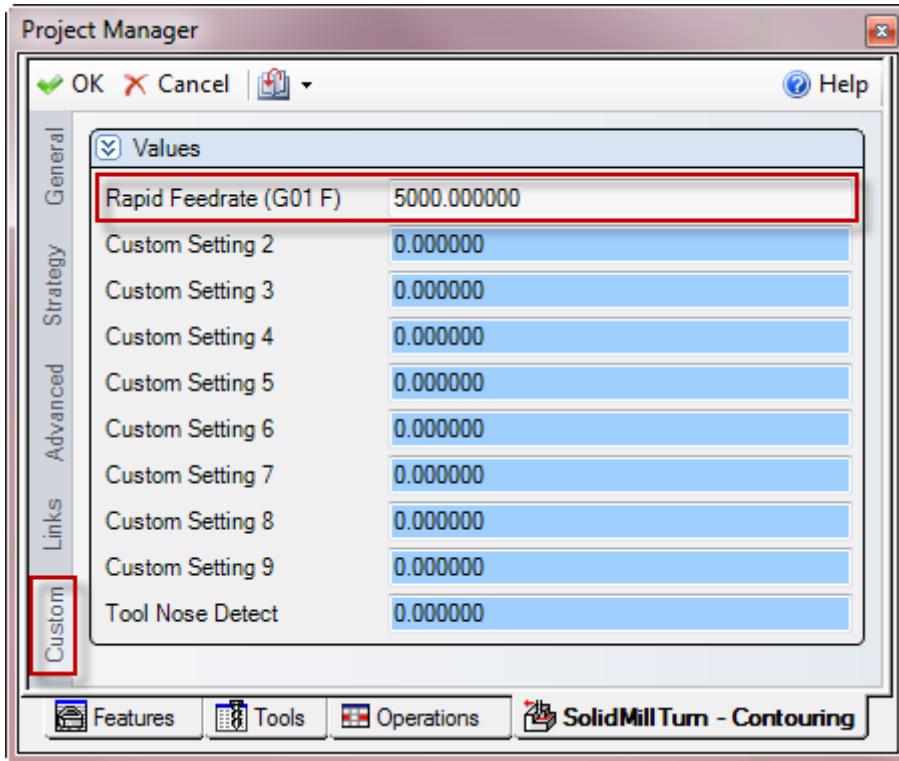


## 6.5. Output of rapid positioning moves (G00) with linear interpolation moves (G01 F)

**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.



## 7. How to output coolant codes

### 7.1. Introduction

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

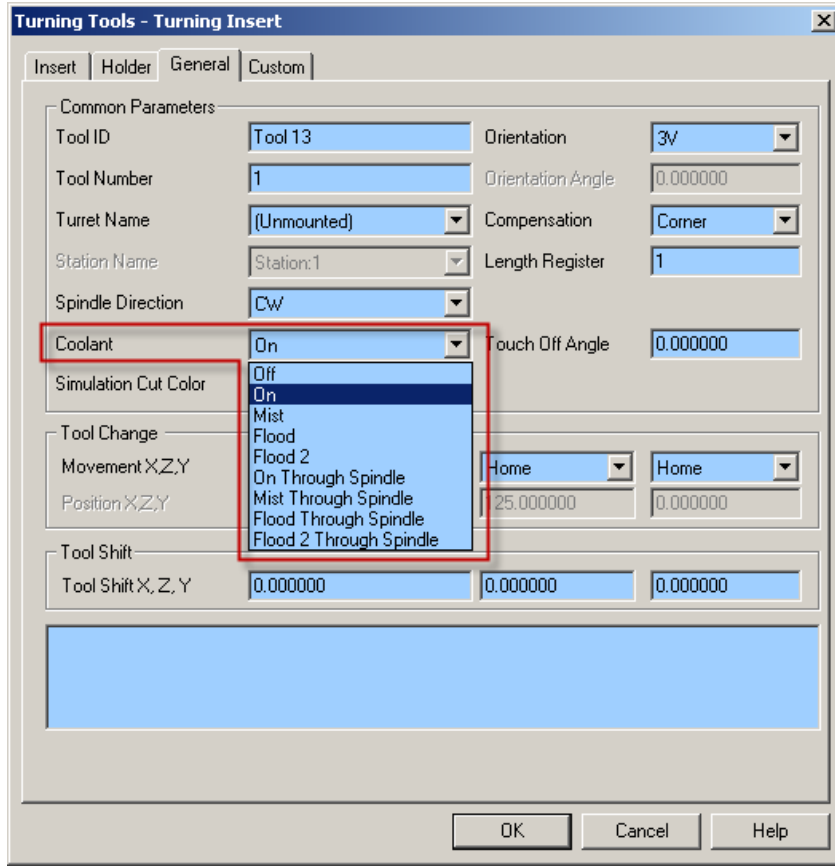
Code	Function
M08	Coolant ON
M09	Coolant OFF
M382	Bed cover chip removal coolant system ON
M383	Bed cover chip removal coolant system OFF
M478	Through-spindle coolant ON
M479	Through-spindle coolant OFF

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

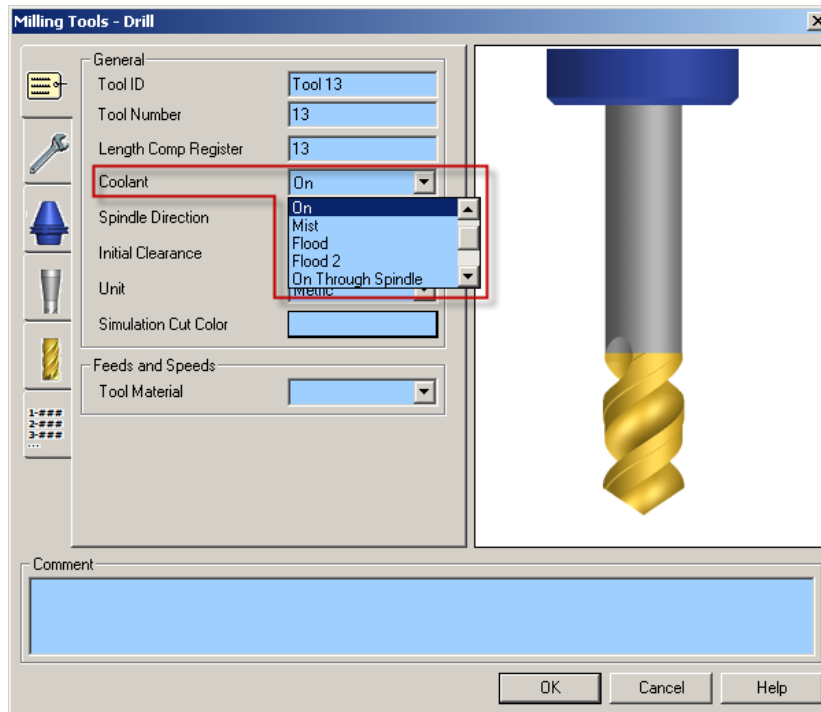
### 7.2. Set the first coolant code

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

**Turning tool page:**



**Milling tool page:**



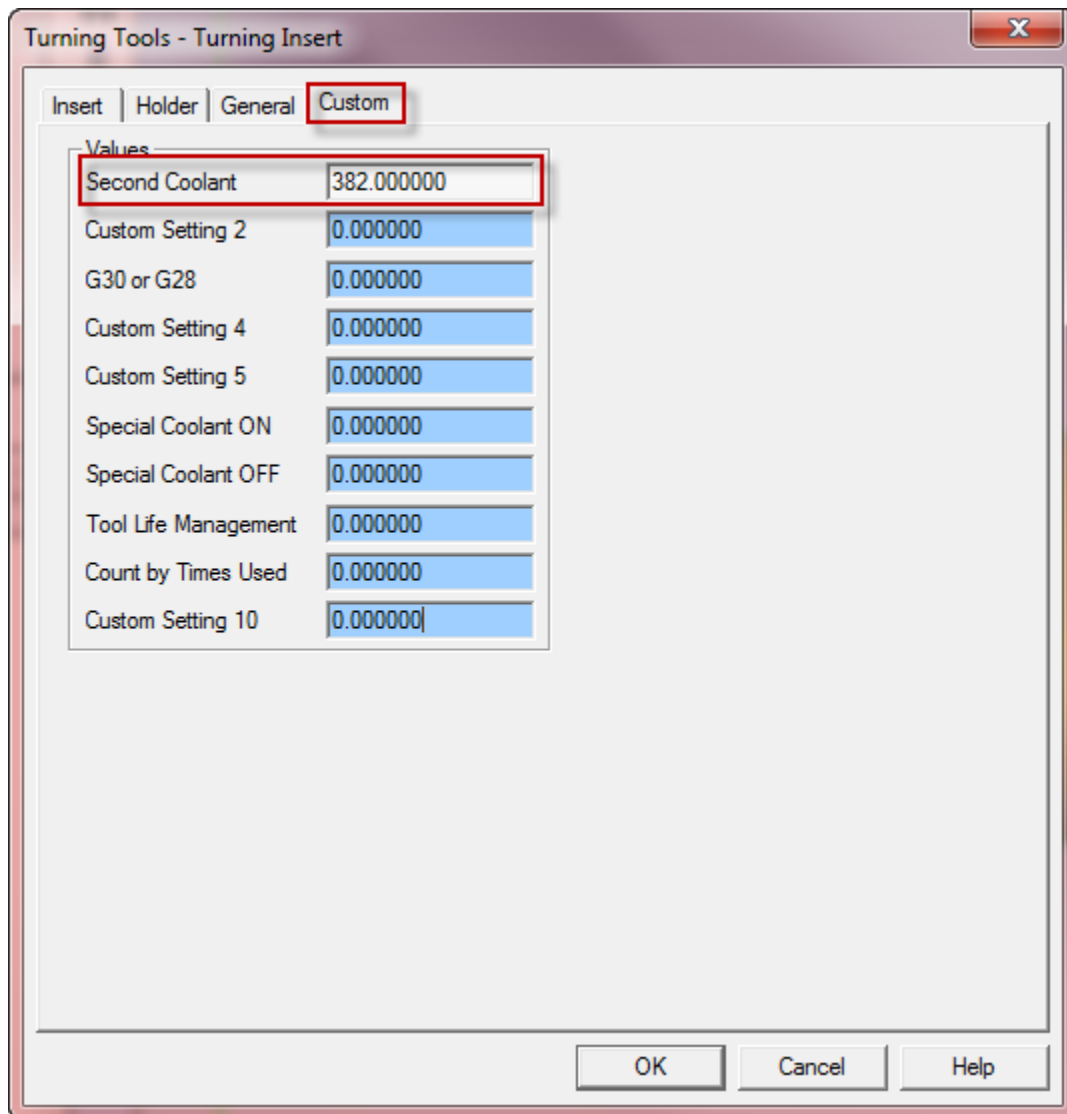
Set **Coolant** to:

- **On** to output M08 (M09)
- **Flood** to output M382 (M383)
- **Flood Through Spindle** to output M478 (M479)

### 7.3. Set the second coolant code (optional)

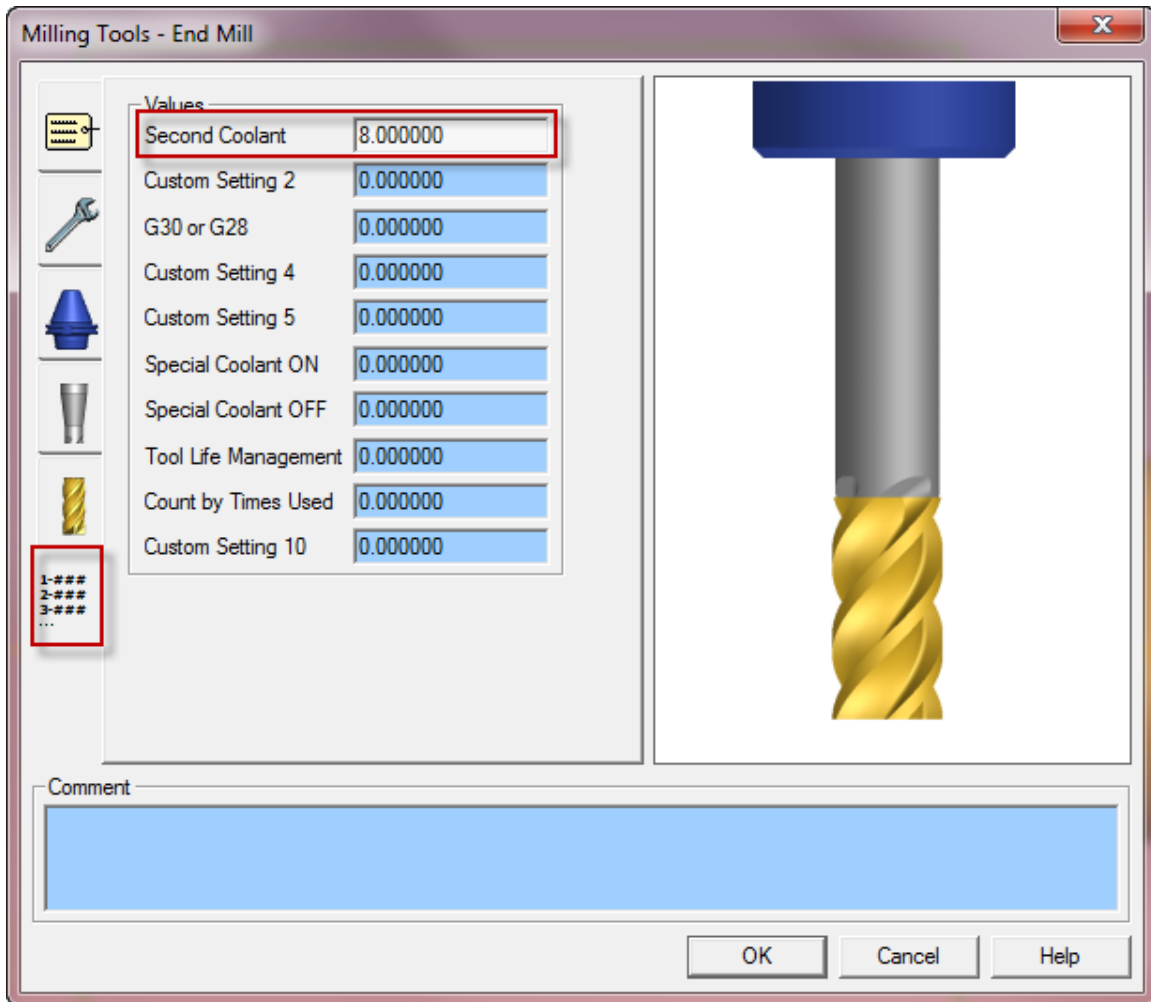
If you desire to output a second coolant code (which is optional), enter the coolant code value in the Second Coolant (Custom Setting 1) field of the Custom tab of the tool page.

**Turning tool page:**



**Milling tool page:**





Set **Second Coolant** to:

- **8** to output M08 (M09)
- **382** to output M382 (M383)
- **478** to output M478 (M479)

#### **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.

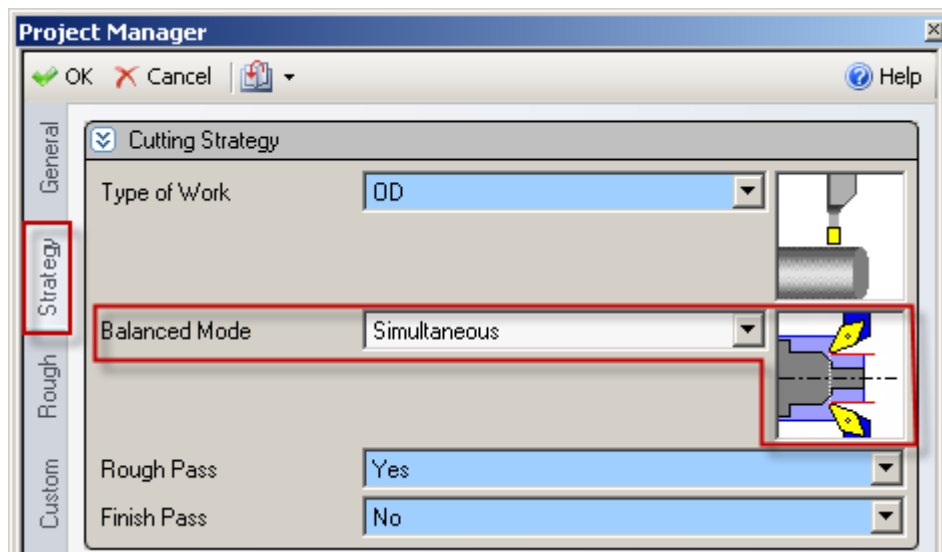
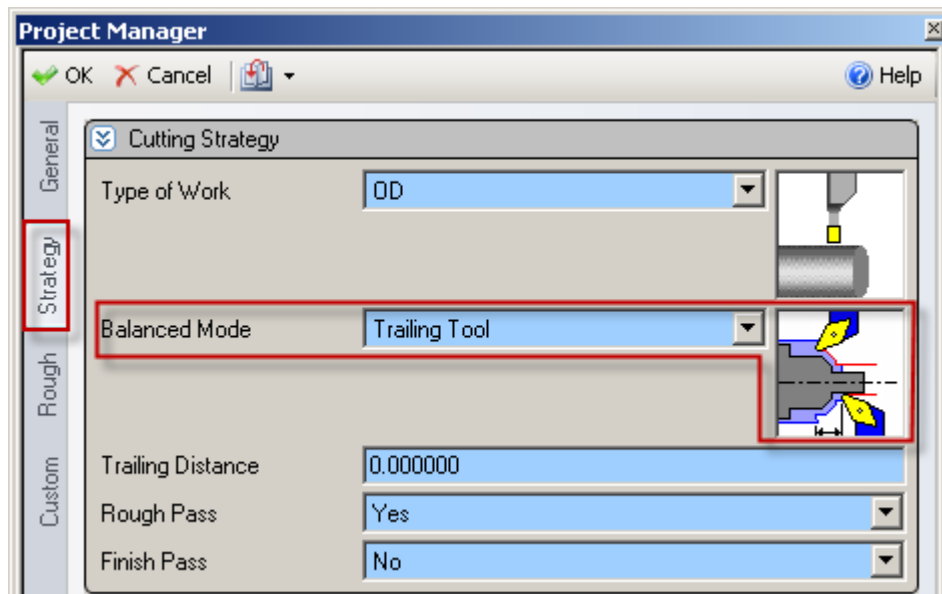
The codes will then be output in NC code: special coolant code ON will be output before turning spindle or live tool speed output; special coolant code OFF will be output after turning spindle or live tool stop code (M05).

## 8. Simultaneous turning operations: Turning on the same spindle with 2 different turrets (for NZX configuration only)

### 8.1. *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.



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).

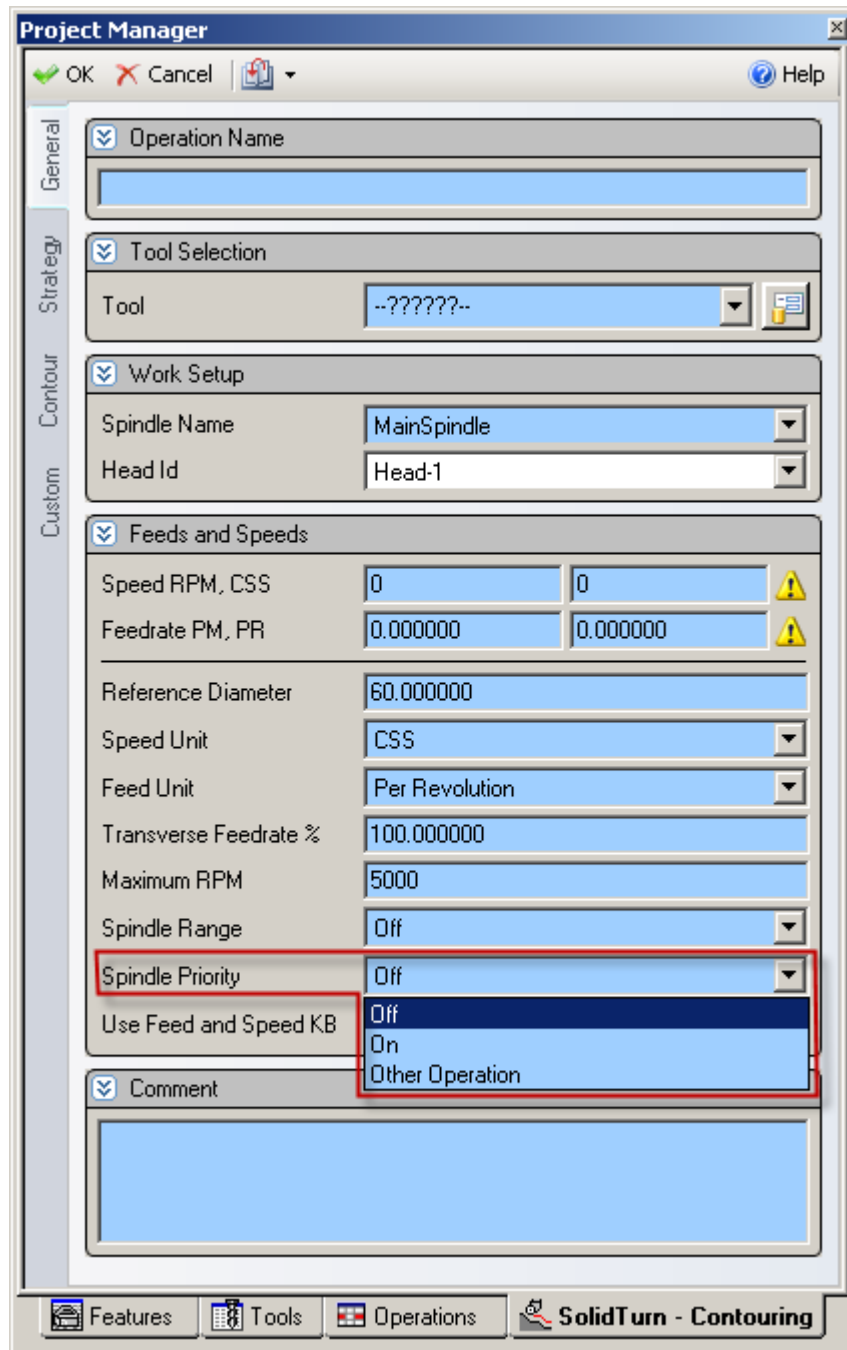
The spindle speed will only be output in channel 1, not in channel 2.

## **8.2. Other turning cycles**

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

For not having this problem, **the head with the longest operation must have the control of the spindle speed.**

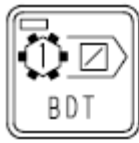
The head controlling the spindle and the head “slave” are set on the operation page of turning operations, using the Spindle Priority setting:



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 **4.4.7. Set how to skip turning spindle control**).

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



(Turret 1)



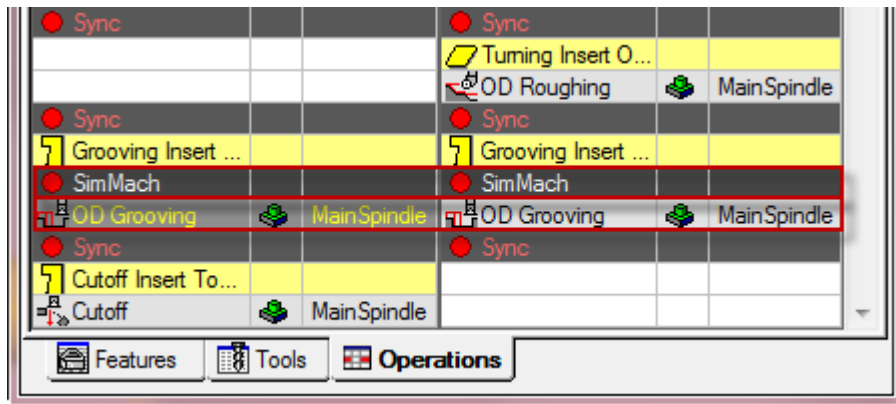
(Turret 2)

### 8.3. Other turning cycles in Balanced Cut Mode

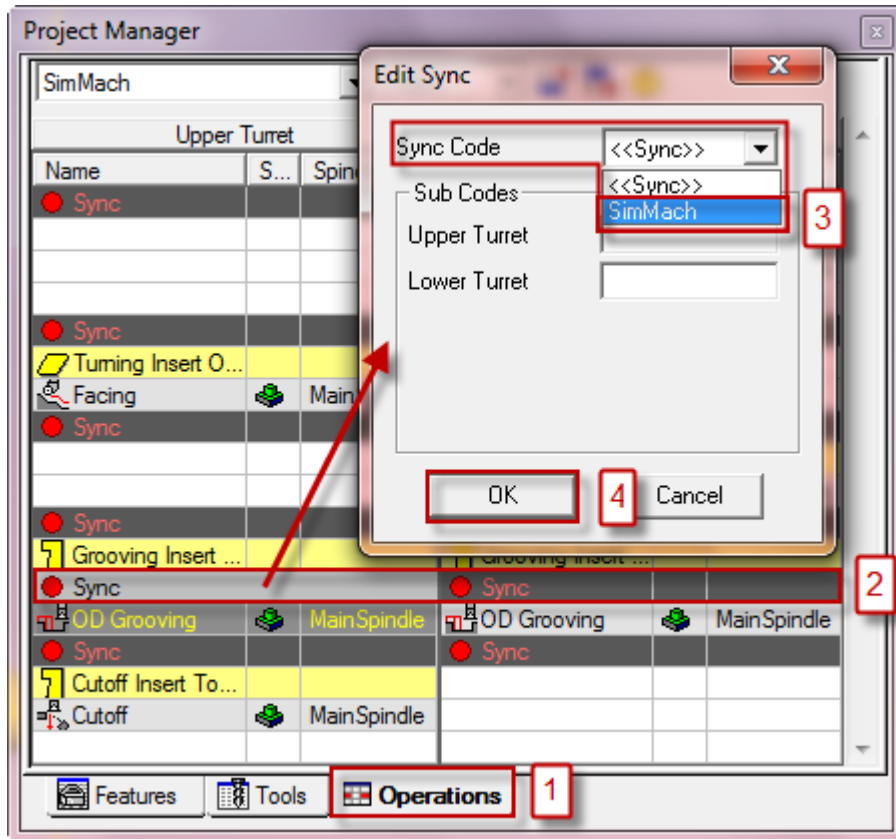
**CAUTION !!!** Do not use with **SolidTurn Drilling**.

To program simultaneous turning operations in Balanced Cut Mode (G68), you will need to synchronize head 1 & head 2 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.

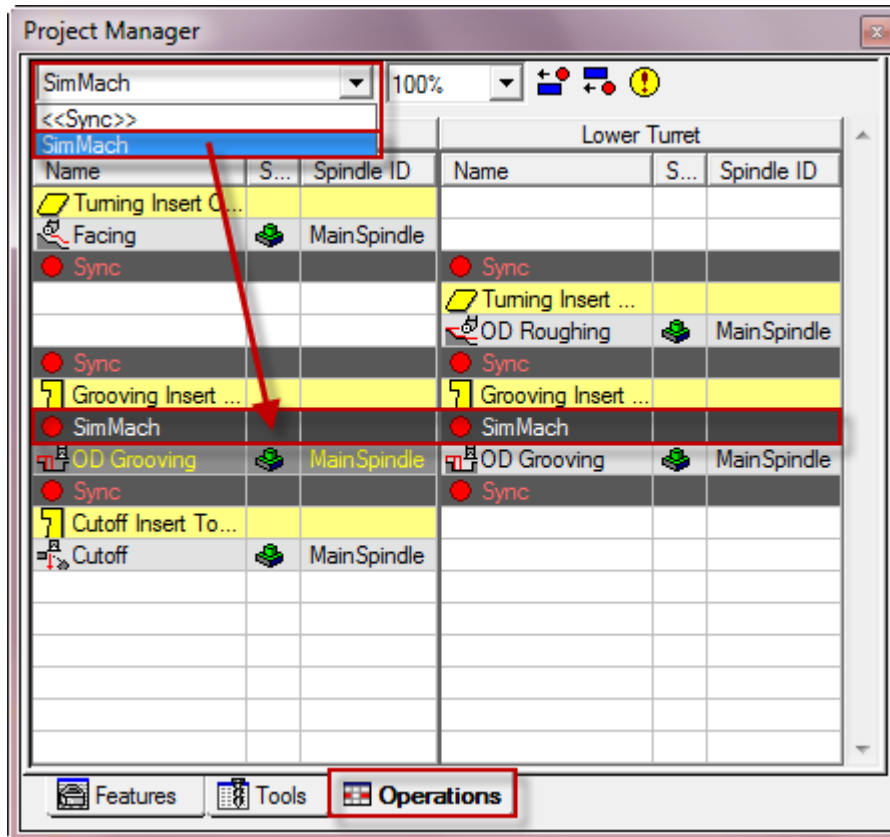
This functionality is typically used to perform pinch grooving operations.



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



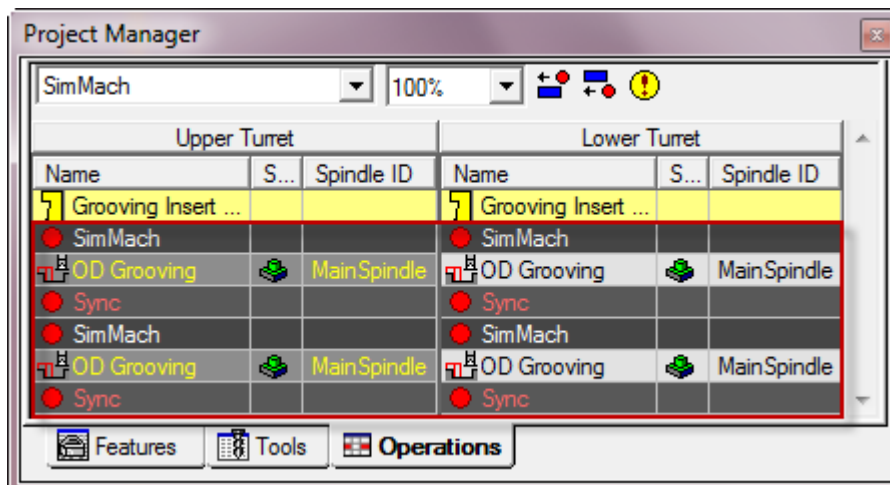
Or, you can also directly select SimMach from the pull-down on top of the Operations tab of the Project Manager. Once selected, simply program your sync: it will be a SimMach Sync Code.



*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...**



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 **8.2. Other turning cycles.**

*Notes:*

- If you have an operation type mismatch (milling on head 1 while turning on head 2) 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.**

## **9. Operation synchronization (for NZX configuration only)**

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**

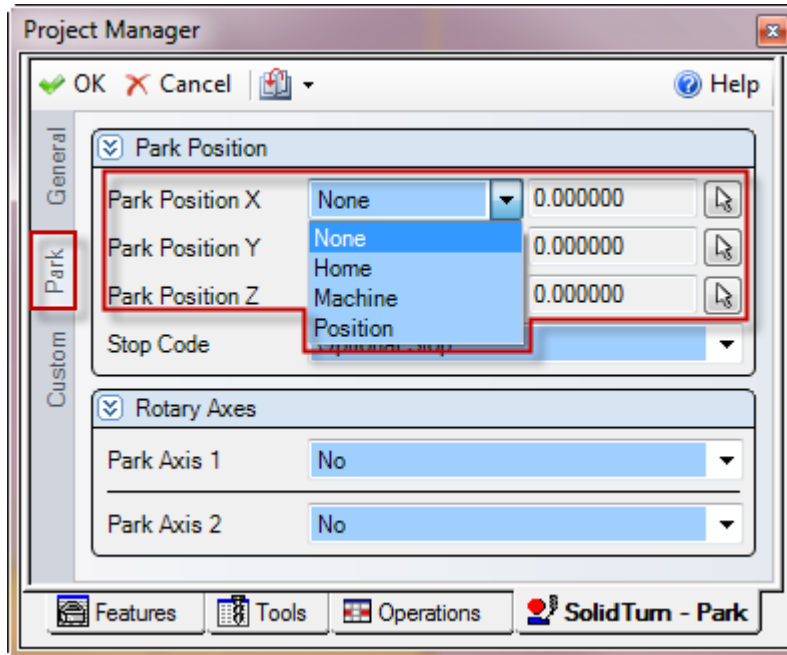
### ***10.1. Introduction***

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

### ***10.2. How to park a tool***

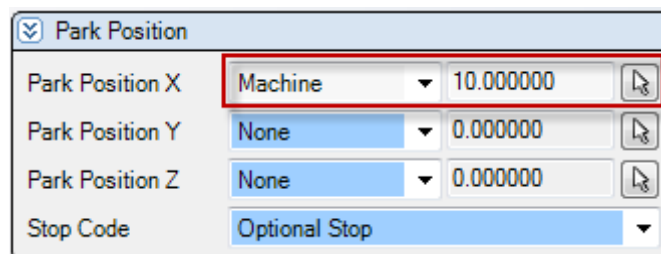
On the Park tab of the Park operation page, using the Park Position X, Y & pull-downs and fields, you can set how and where you want to park your tool.



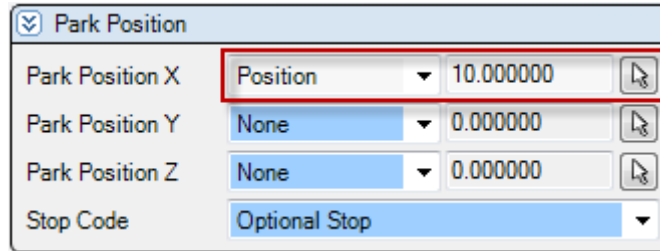


Set Park Position X, Y or Z to:

- **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.

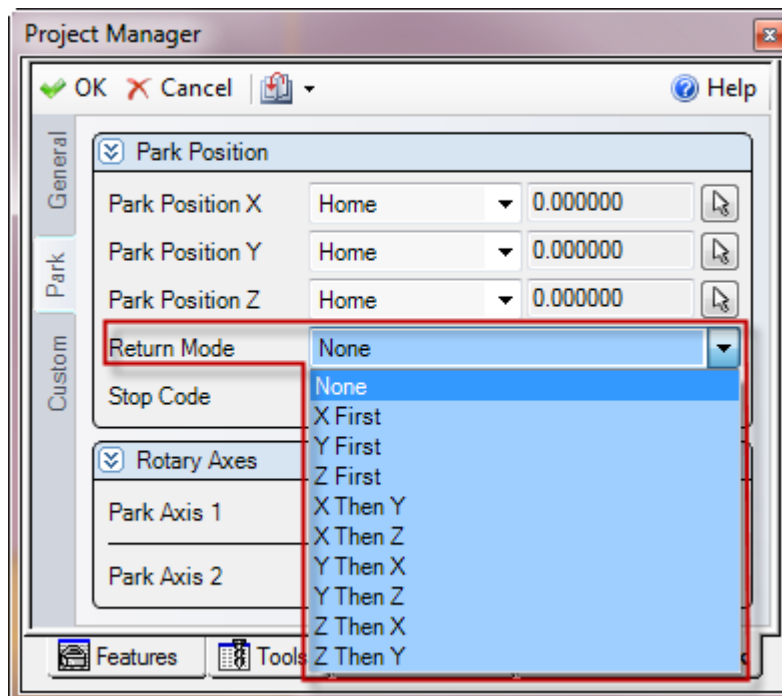


- **Position:** The tool will be parked at the specified position in Position X, Y, Z fields at rapid traverse rate (G00).



*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.



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:*

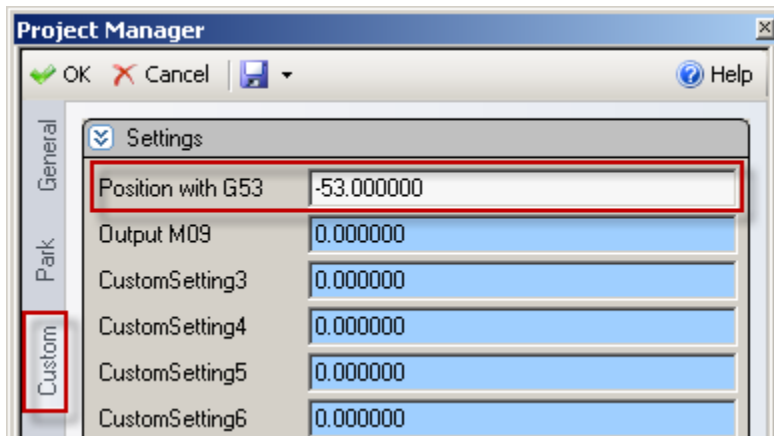
If Park Position X, Y & Z are all set to Home: all Return Modes become available.

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

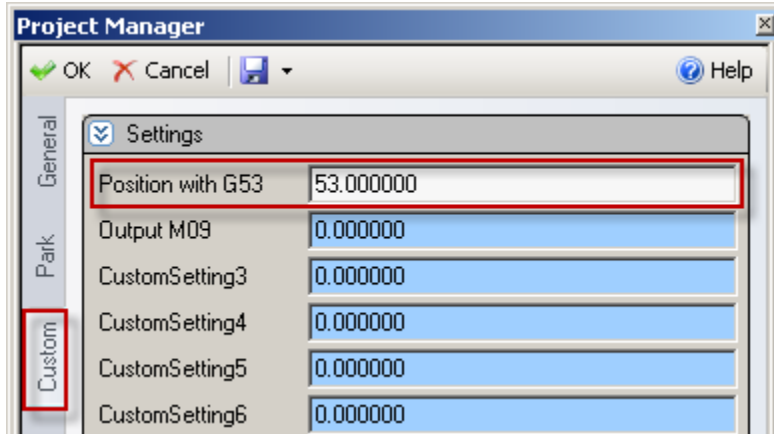
### 10.3. Park a turret to its maximal travel position (G53)

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.



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



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

			Left position	Right position	
NZX	6000	1000	Upper turret T1	G53 Z-1070.0	G53 Z230.0
			Lower turret T2	G53 Z-910.0	G53 Z90.0
		2000	Upper turret T1	G53 Z-2070.0	G53 Z80.0
			Lower turret T2	G53 Z-1910.0	G53 Z80.0
		3000	Upper turret T1		
			Lower turret T2		
	4000	1000	Upper turret T1	G53 Z-1020.0	G53 Z80.0
			Lower turret T2	G53 Z-920.0	G53 Z80.0
		1500	Upper turret T1		
			Lower turret T2		
	2000	Upper turret T1	G53 Z-2020.0	G53 Z80.0	
		Lower turret T2	G53 Z-1920.0	G53 Z80.0	
3000	Upper turret T1				
	Lower turret T2				
2500	600	Upper turret T1	G53 Z-600.0	G53 Z50.0	
		Lower turret T2	G53 Z-600.0	G53 Z50.0	
	1000	Upper turret T1	G53 Z-1000.0	G53 Z50.0	
		Lower turret T2	G53 Z-1000.0	G53 Z50.0	

Turret maximum travel position values (in the machine coordinate system G53) for NLX configuration:

Left position: -745 mm

Right position: 50 mm

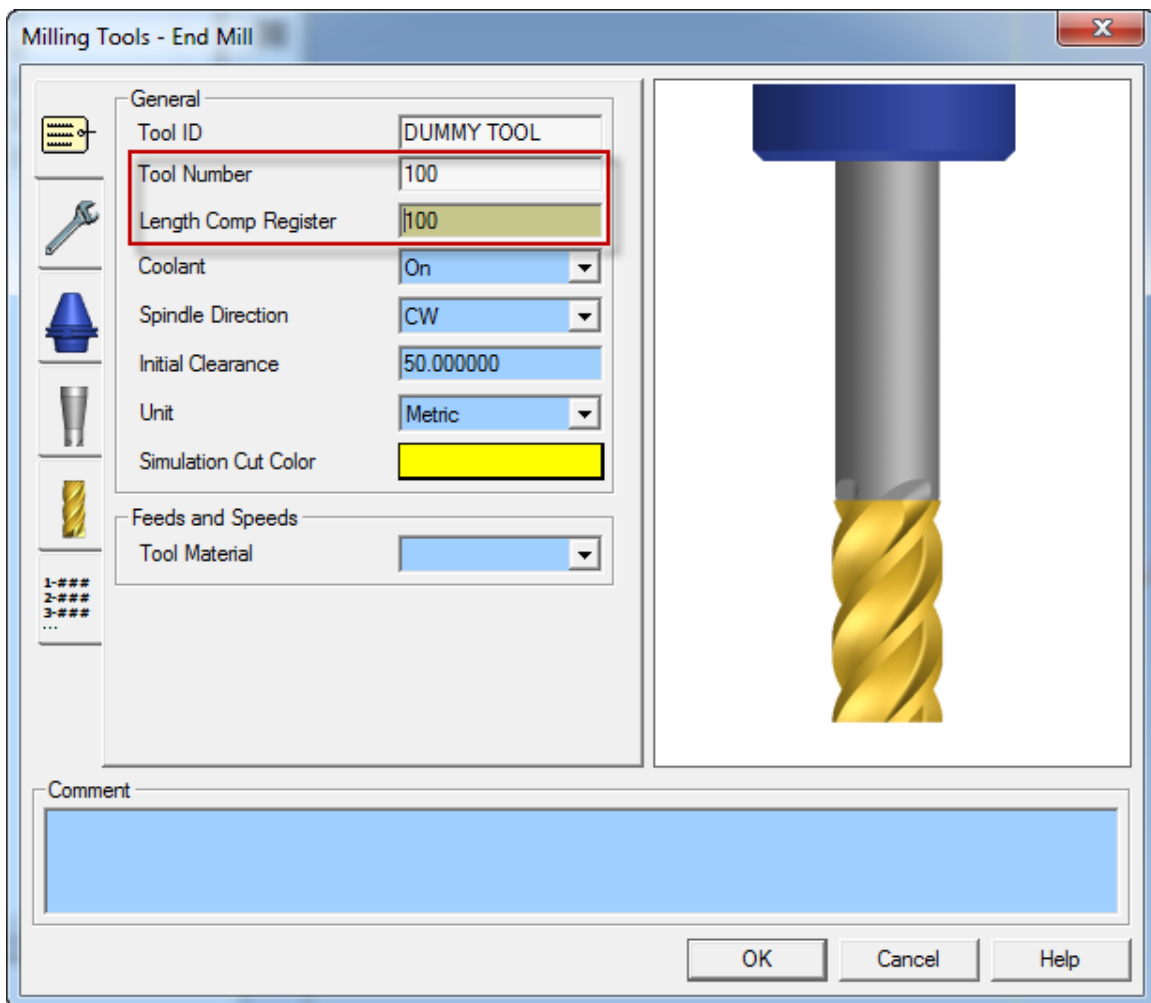
Note that if the tool you are parking is the same as the previous tool in use, the turret will be sent home in Y (Y-axis specifications only) then X before the G53 output as a safety.

#### 10.4. Park a turret prior to a transfer (for NLX configuration only)









By default, tool station 1 will be called prior to a transfer. As seen in part 4.4.5. **Tool station for part transfer and work unloader**, this tool station can be changed using the Station on Turret 1 for Transfer textbox on the POST Output Configuration.

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

You will first need to create a dummy tool on the turret, 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.



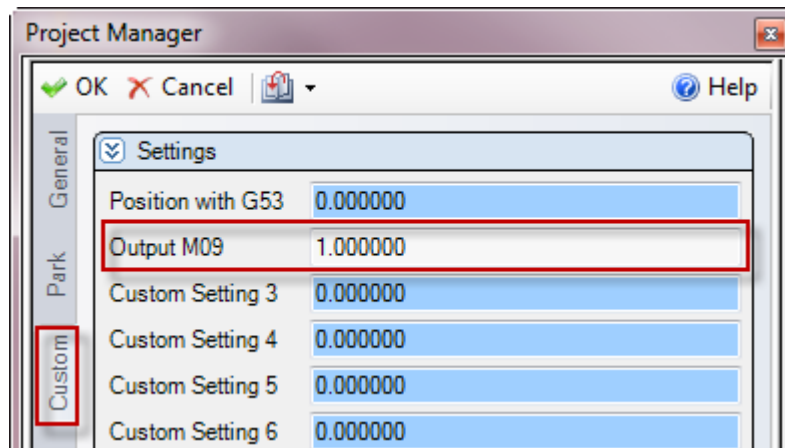
Then, simply program a Park operation using this previously defined tool prior to the transfer sequence. Note that since the tool call will be made without tool wear offset number, you will need to set Position with G53 to 53 (or -53, see part **10.3. Park a turret to its maximal travel position (G53)**) on the Park operation Custom tab. The turret will be parked using G53.

 DUMMY TOOL	
 Park turret	 Main Spindle
 Pickup	 Sub Spindle
 Cutoff Insert	
 Cutoff	 Main Spindle

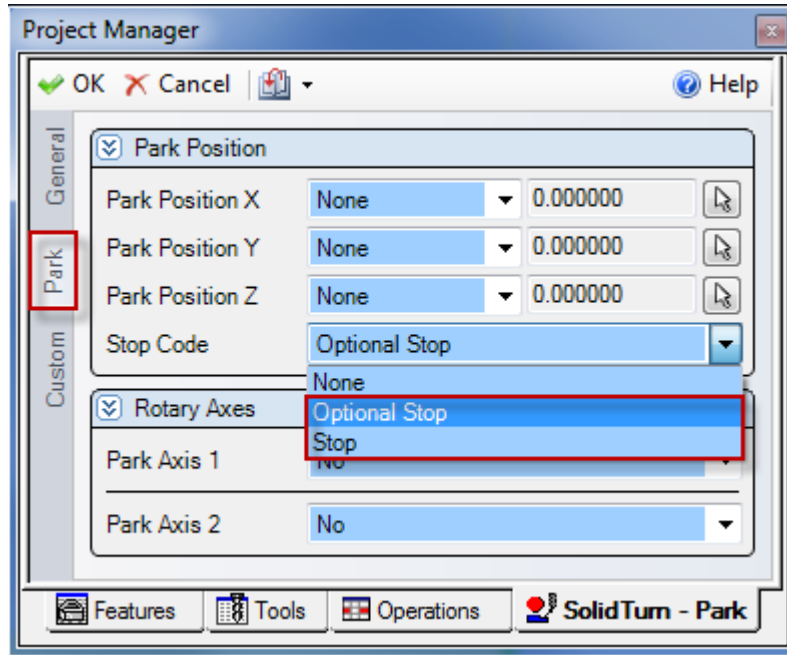
*Note:* If you omit to enter 53 in the Position with G53 field of the Park operation, you will get the following error message in your NC code: **ERROR: NO TOOL WEAR OFFSET: PARK WITH POSITION (USING G00) CANNOT BE USED.**

### 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.



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.



## 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 (for NLX configuration only) in ESPRIT. **A correct program in ESPRIT is necessary to have a correct NC code.**

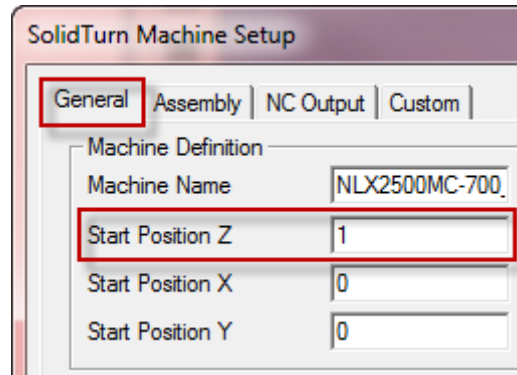
Please note that the following part catching and part transfer scenarios can be programmed automatically using the Workpiece Transfer add-in for NLX configuration.

### 11.2. Finished part catching

#### 11.2.1. Catching finished part on spindle 1 side after cut-off (for NLX configuration only)

It can be performed if your machine is equipped with a receiver installed 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



The regular steps to follow are:

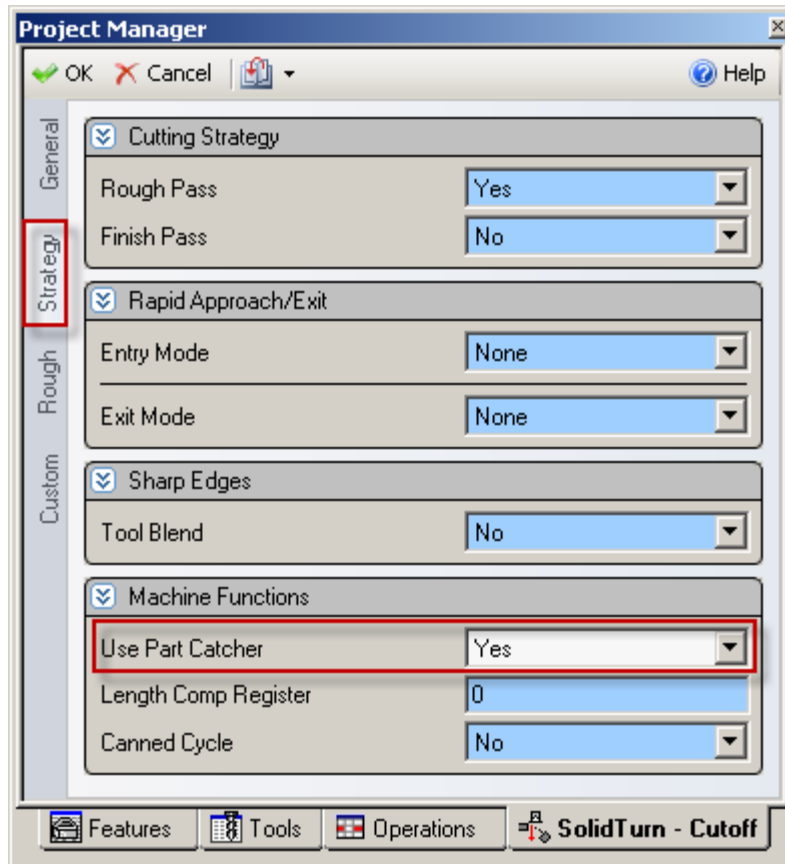
1. Machining the part on the spindle 1 side
2. Perform a cut-off operation

Steps to program this part ejection type in ESPRIT:

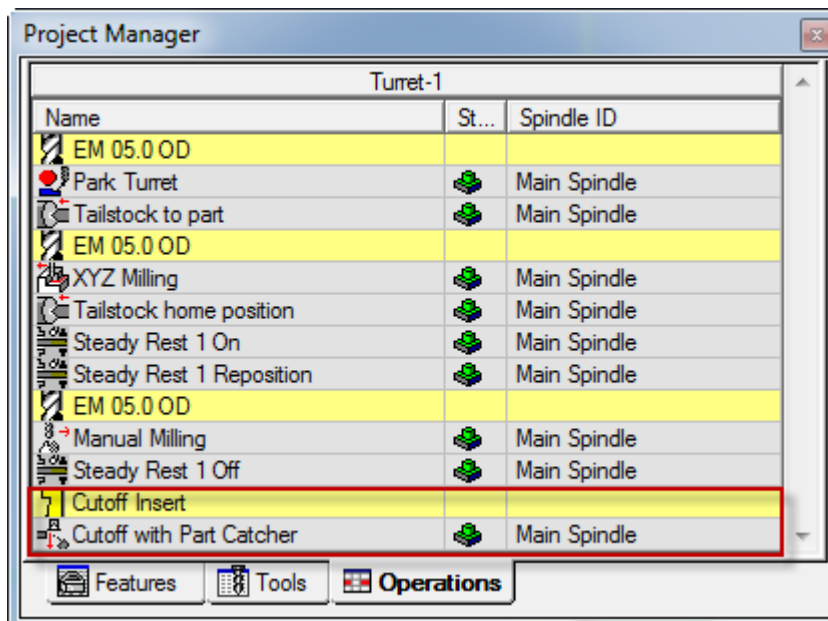
1. Machining the part on the spindle 1 side.
2. Cut-off:  
Use operation **Cutoff** using tool on **MainSpindle (Spindle Name)**.

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





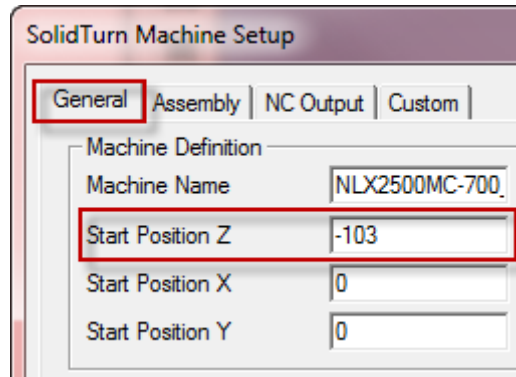
Sample operation list:



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

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:  
 $Start\ Position\ Z = - (Length\ of\ the\ finished\ part\ (Part\ Stock\ Length) + Cut-off\ tool\ width + Length\ of\ stock\ for\ facing\ the\ back\ side\ of\ the\ part\ (if\ needed))$



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:

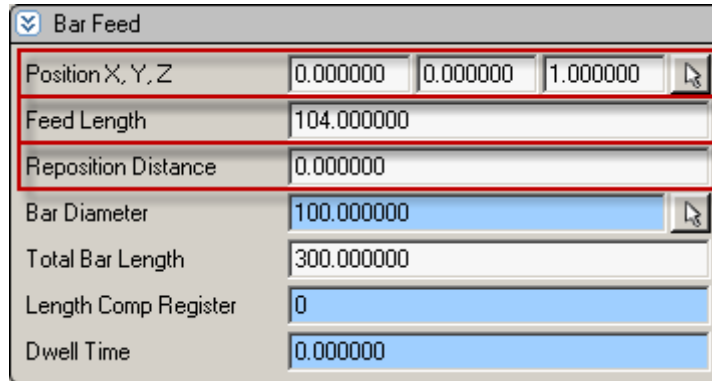
1. Perform a barfeed operation:  
Use operation **Bar Feed By Stopper (Bar Feed Type)** using tool on **MainSpindle (Spindle Name)**.

On Bar Feed tab, set the Feed Length and Reposition Distance:

$Feed\ Length\ (=Barfeed\ distance) = Length\ of\ the\ finished\ part\ (Part\ Stock\ Length) + Cut-off\ tool\ width + Length\ of\ stock\ for\ facing\ the\ front\ side\ and,\ if\ needed,\ the\ back\ side\ of\ the\ part$

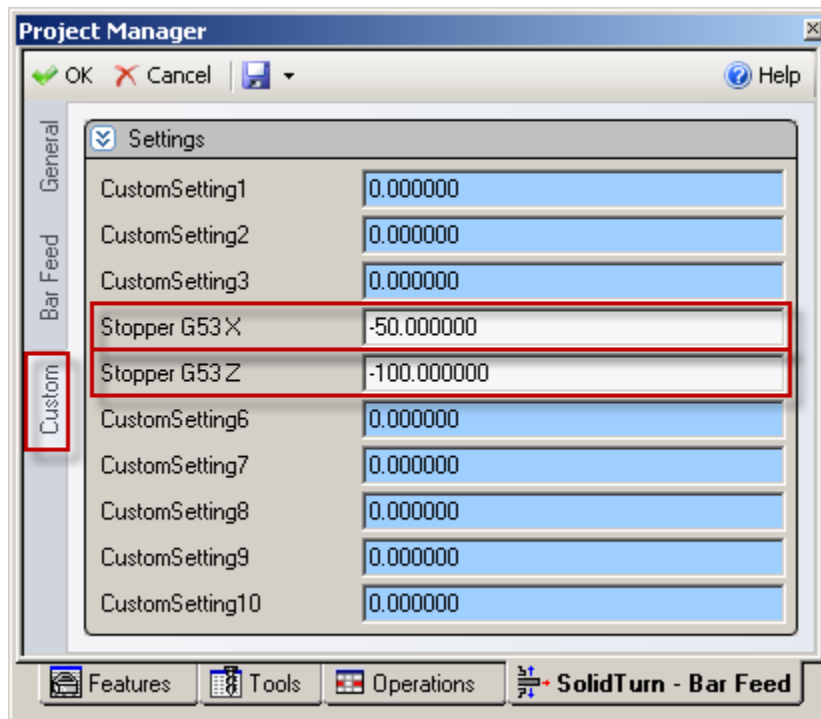
$Reposition\ Distance = 0$

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).



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 Stopper G53 Z (Custom Setting 5) field the Z position (in G53 work coordinate) of the stopper tool.

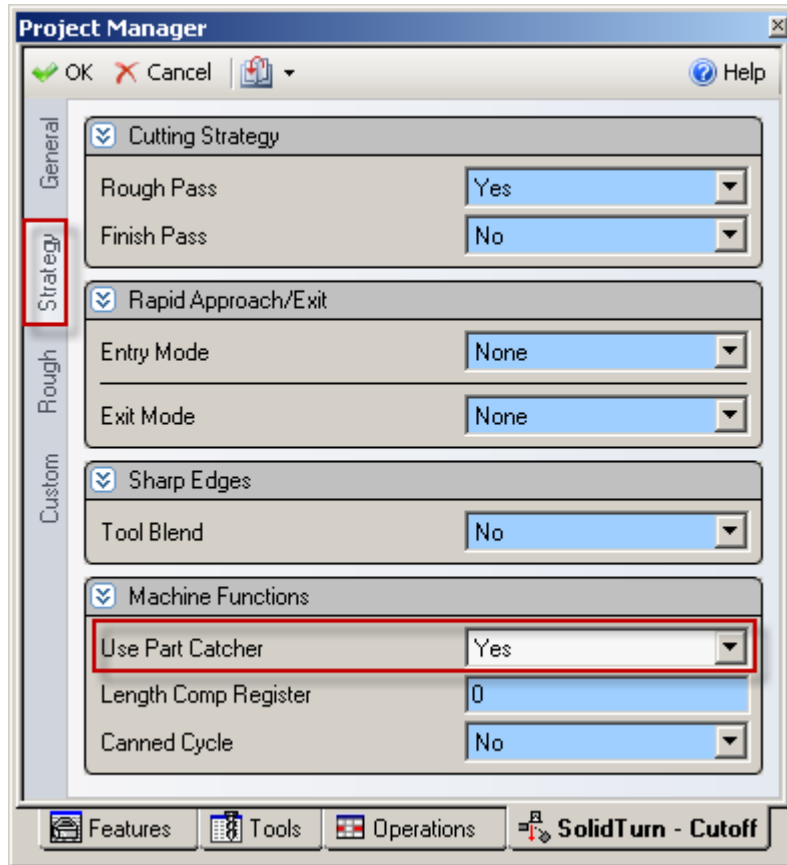


*To use current machine work coordinate defined in ESPRIT (see part 2.3. Turning Work Coordinates):* If no value is specified on the Custom tab in the Stopper G53 X and Stopper G53 Z fields (fields left equal to 0), the stopper tool will be positioned at the programmed position (Position X, Y, Z specified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

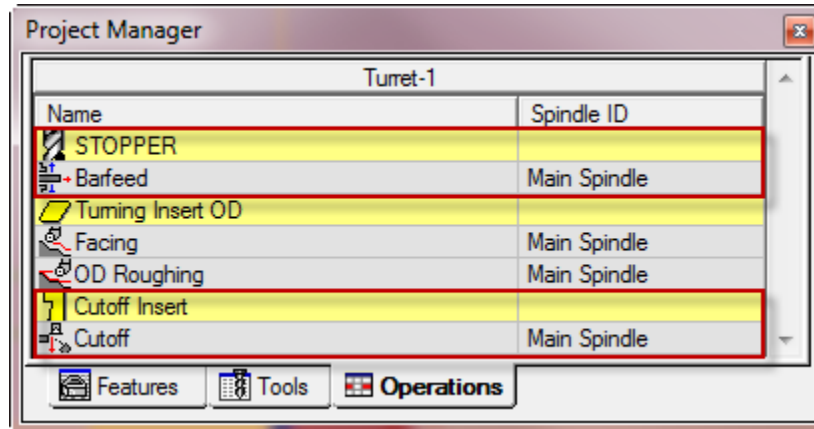
2. Machining the part on the spindle 1 side.

- Cut-off:  
Use operation **Cutoff** using tool on **MainSpindle (Spindle Name)**.

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



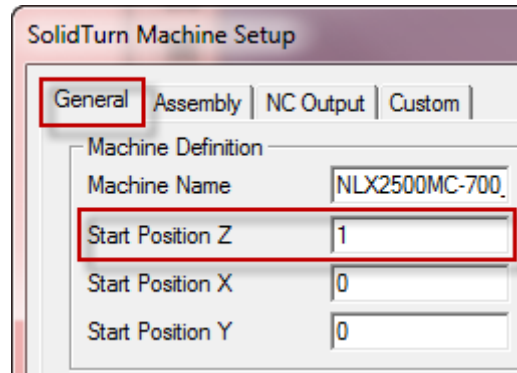
Sample operation list:



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

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*



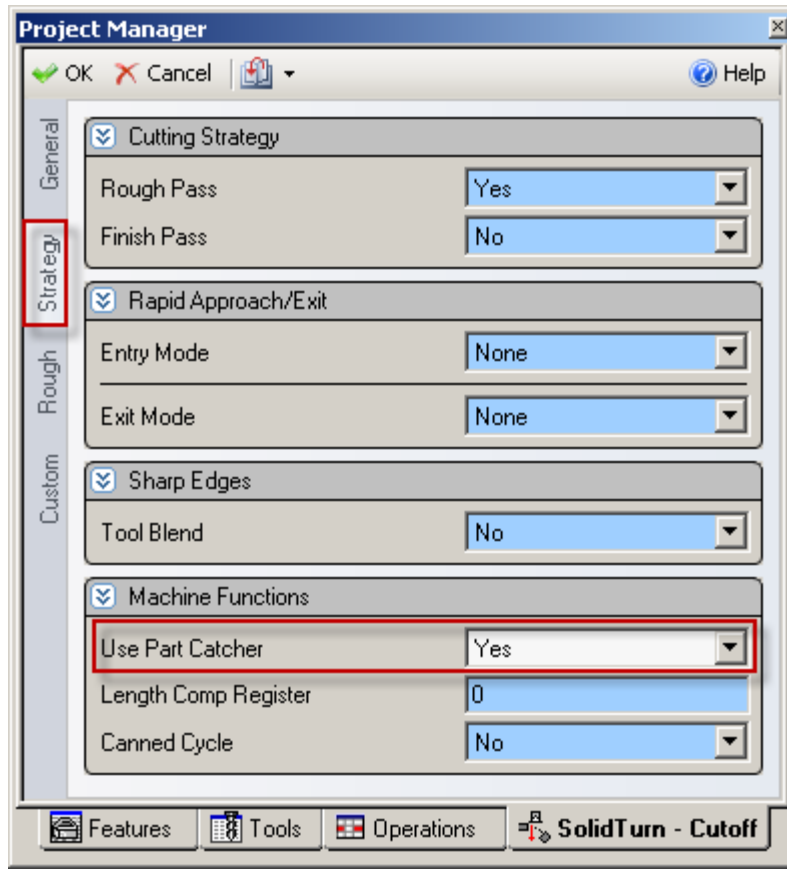
The steps to follow are the same as previous point. The only difference is that you need to program a barfeed (by stopper) operation last:

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 tool on **MainSpindle (Spindle Name)**.

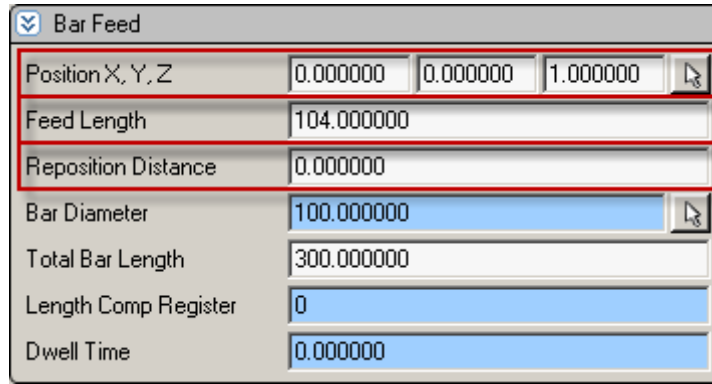
To trigger the part catching output, set Use Part Catcher to Yes on the Strategy tab.



3. Perform a barfeed operation:  
Use operation **Bar Feed By Stopper (Bar Feed Type)** using tool on **MainSpindle (Spindle Name)**.

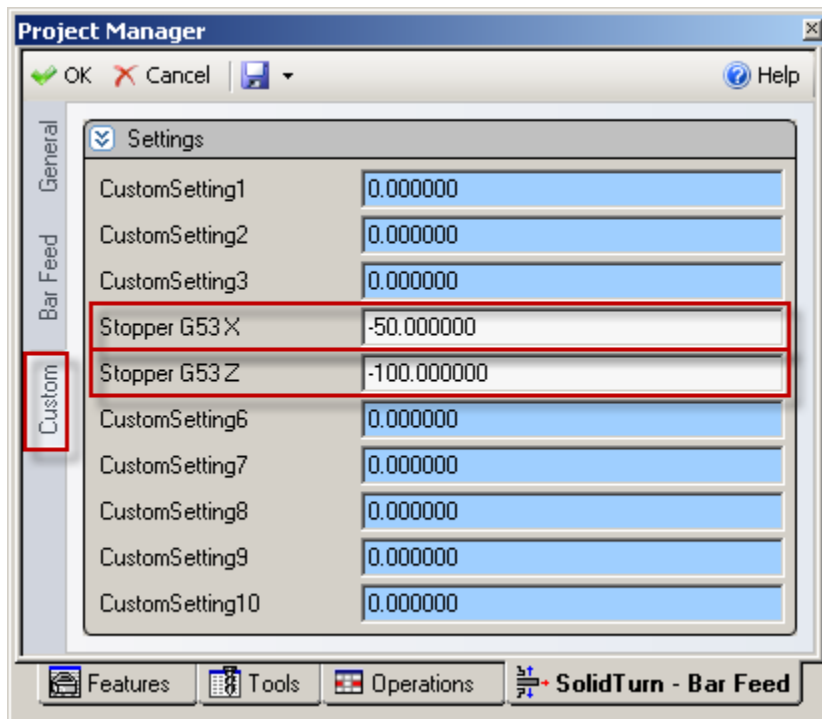
On Bar Feed tab, set the Feed Length and Reposition Distance:  
*Feed Length (=Barfeed distance) = Length of the finished part (Part Stock Length) + Cut-off tool width + Length of stock for facing the front side and, if needed, the back side of the part*  
*Reposition Distance = 0*

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).



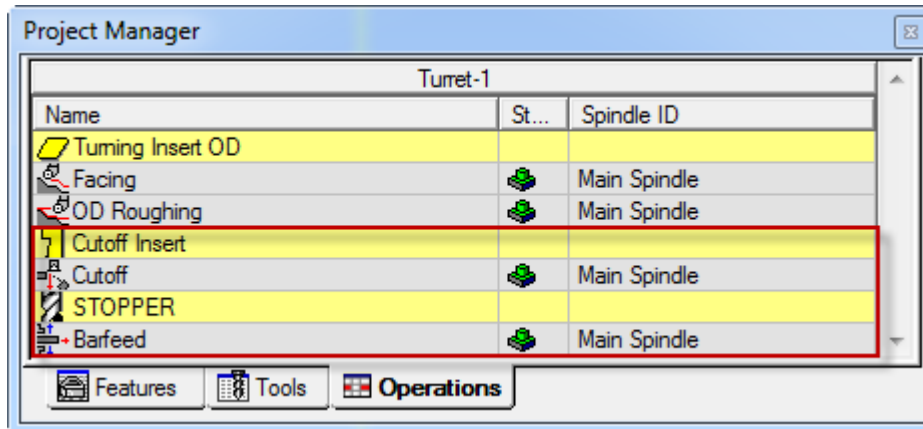
The stopper tool will either be positioned in the G53 work coordinate or the current machine work coordinate.

*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 Stopper G53 Z (Custom Setting 5) field the Z position (in G53 work coordinate) of the stopper tool.



*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.

Sample operation list:



#### 11.2.4. Manually catching finished part on spindle 1 side

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, the machine needs to be stopped with M00.

The regular steps to follow are:

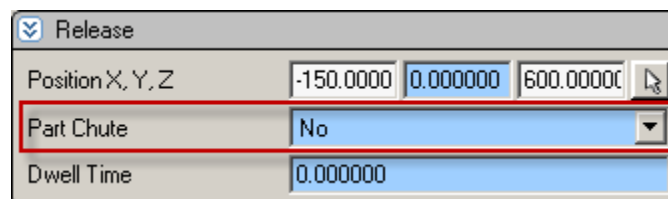
1. Machining the part on the spindle 1 side
2. Manually catching the finished part in the spindle 1

Steps to program this part ejection type in ESPRIT:

1. Machining the part on the spindle 1 side.
2. Program a wait code across the two heads for NZX configuration.
3. Manually catching the finished part in the spindle 1:  
To be able to manually catch the finished part in the spindle 1, the machine needs to be stopped. To do so, the M-code M00 (program stop) needs to be output (in both heads after the same sync code across the two heads (this is the reason why a sync code across the two heads needs to be programmed prior to the part release operation) for NZX configuration).

Use operation **Release** with **MainSpindle (Spindle Name)** on **Lower Turret (Turret Name)** (for NZX configuration).

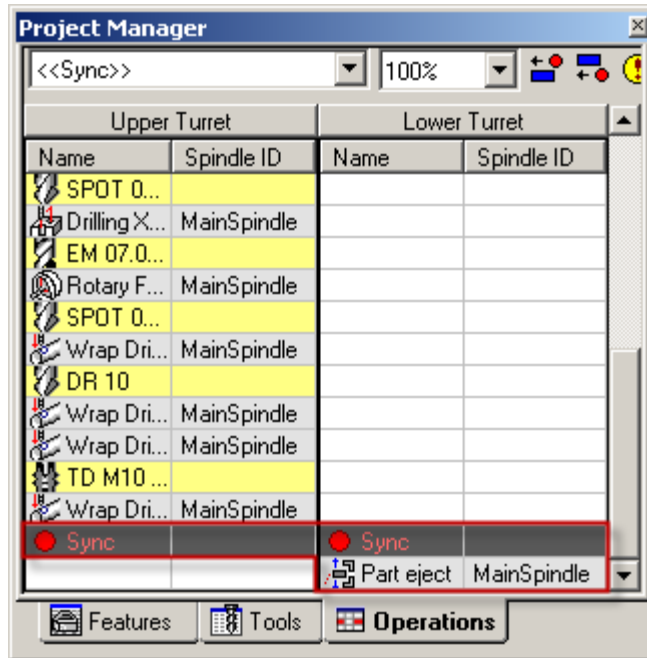
To output the stop code M00, set Part Chute to No on the Release tab.



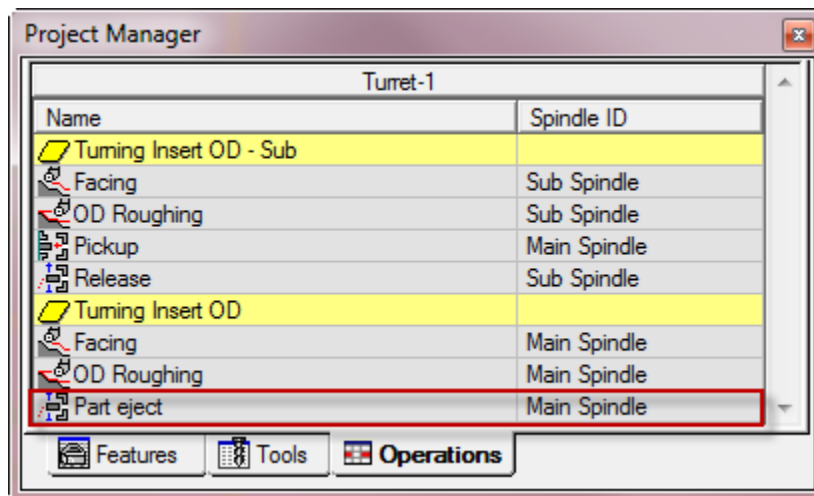


*Note for NZX configuration:* If you do not program a sync above your part release or do not set Part Chute to No, you will get the following error message in your NC code (in head 2): **ERROR: PROGRAM A SYNC ABOVE PART EJECT OPERATION & SET PART CHUTE TO NO TO OUTPUT M00.**

Sample operation list for NZX configuration:



Sample operation list for NLX configuration:



### 11.2.5. Catching finished part on spindle 2 side (for NLX configuration only)

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. Catching the finished part in the spindle 2:  
Use operation **Release** with **SubSpindle (Spindle Name)**.

Based on the setting set for Work Unloader Spindle Side on the POST Output Configuration (see part **4.4.3. Work unloader on spindle 1 or spindle 2 side**), the NC output will differ to match your machine specifications: receiver installed on spindle 1 or spindle 2 side.

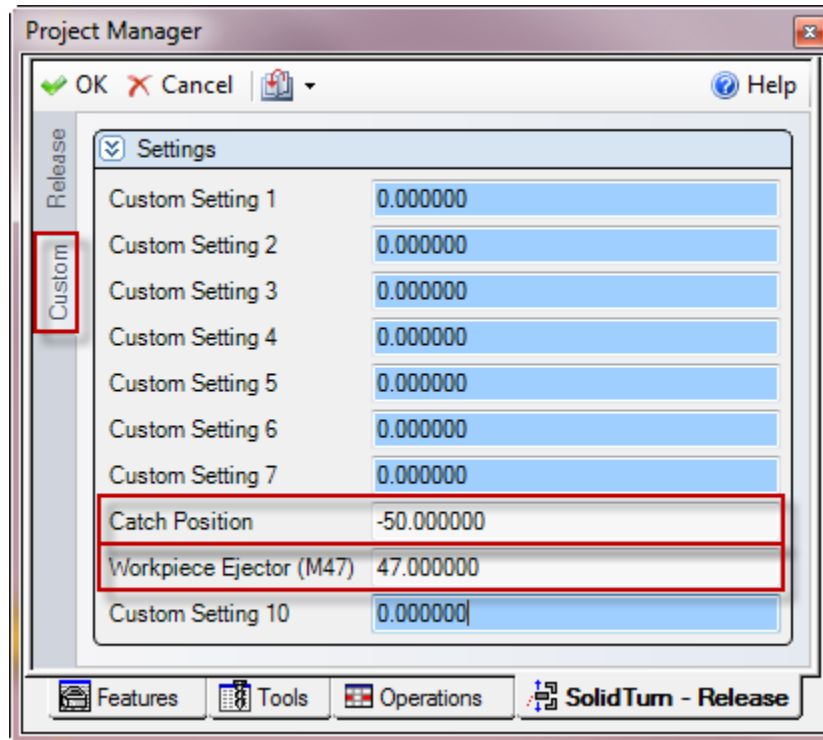
Catching the finished part in the spindle 2 on a machine with receiver installed on spindle 1 or spindle 2 side:

Once you specified on which spindle side the receiver is mounted on the machine, to trigger the part catching output, set Part Chute to Yes on the Release tab.

Release	
Position X, Y, Z	-150.0000 0.000000 600.0000
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).

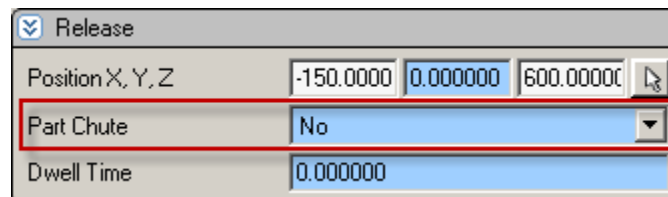
With Workpiece Ejector (M47) (Custom Setting 9), you can specify if you want to eject the finished part with the optional ejector: set Workpiece Ejector (M47) to 47 if you have the workpiece ejector OUT (M47) **option on machine**. If you omit to enter a value in the Workpiece Ejector (M47) field or enter a wrong value, no part ejection code will be output in the NC code.



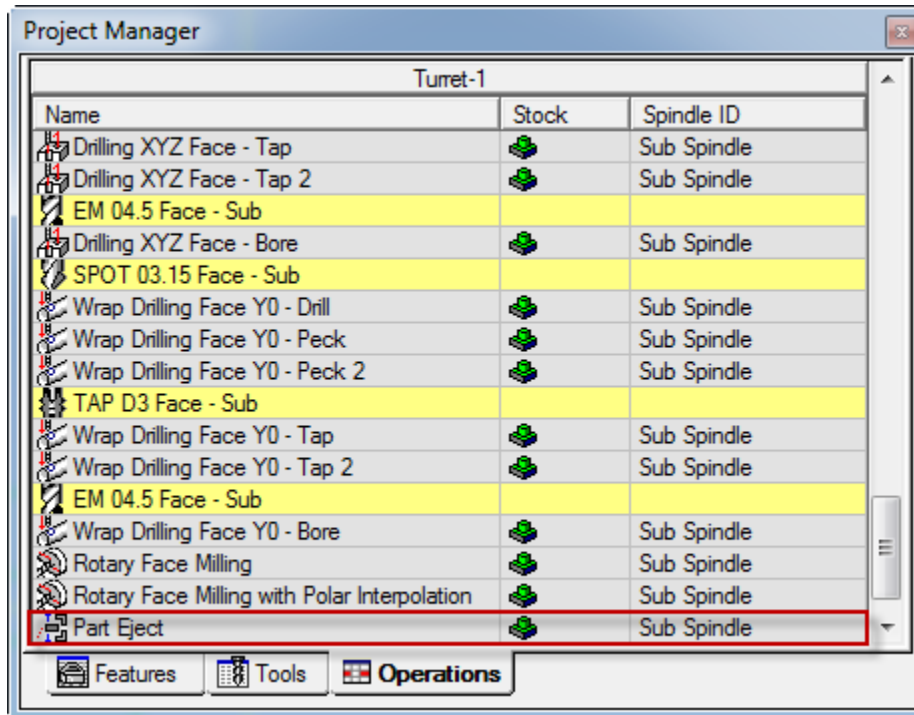
Manually catching the finished part in the spindle 2 (machine is stopped with M00):

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

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



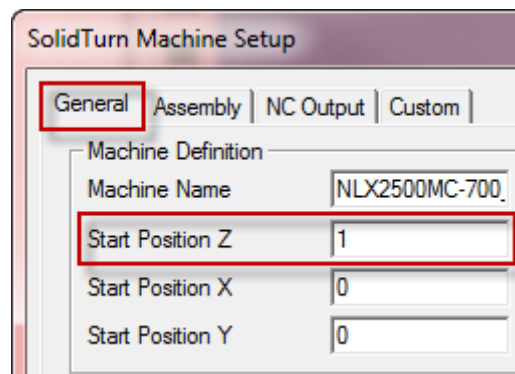
Sample operation list of finished part catching with receiver or of manual part catching:



### 11.3. Part transfer (for NLX configuration only)

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

*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*



The regular steps to follow are:

1. Machining the part on the spindle 1 side
2. Picking the part in the spindle 1 with the spindle 2
3. Pulling the bar with the spindle 2 (if needed)
4. Perform a cut-off and then the spindle 2 goes back home
5. Machining the part on the spindle 2 side
6. Catching the finished part in the spindle 2
7. Looping for next parts

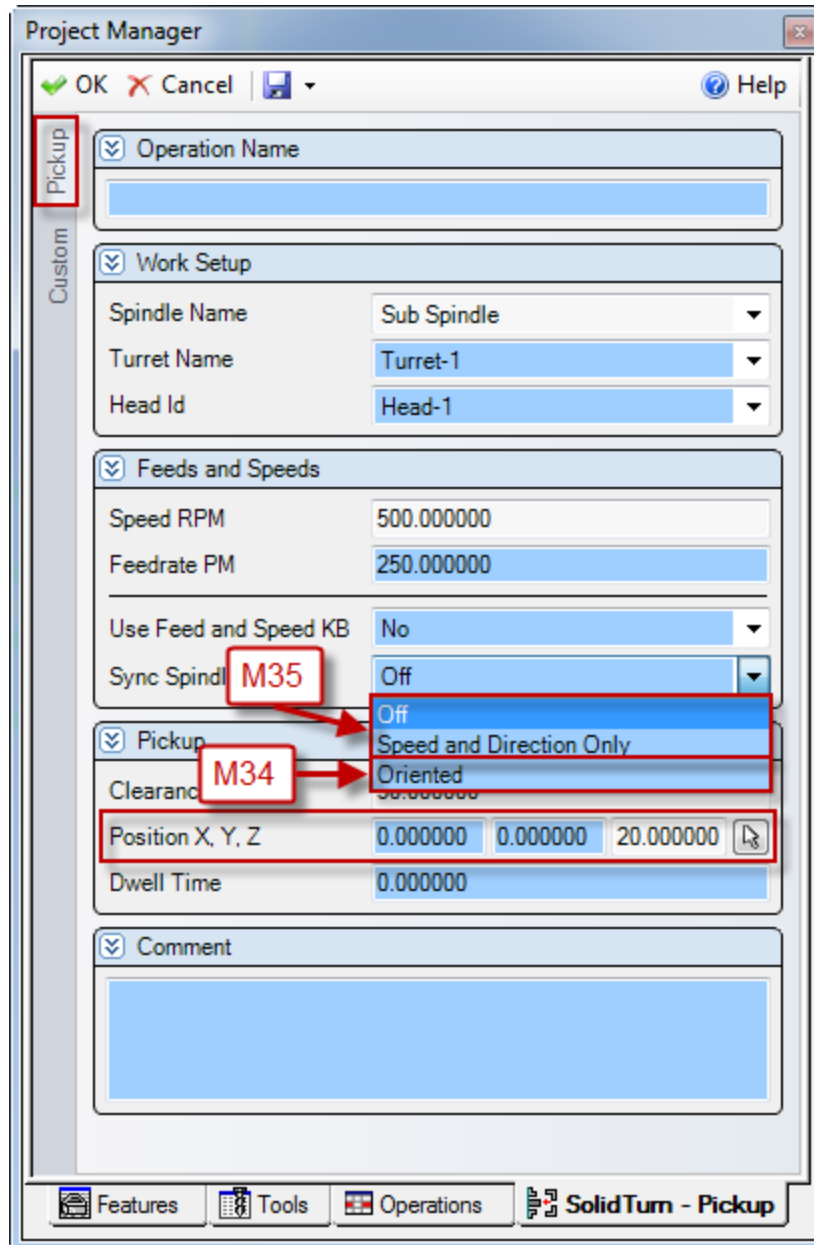
*Note:* It is also possible to perform all operations on spindle 1 and spindle 2 sides and then finish with the part transfer (after first catching finished part in the spindle 2).

Steps to program this transfer type in ESPRIT:

1. Machining the part on the spindle 1 side.
2. Picking the part in the spindle 1 with the spindle 2:  
Use operation **Pickup** with **SubSpindle (Spindle Name)**. With Position X, Y, Z, set the pickup point.

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 after synchronization of the spindles.

Enter the Feedrate PM of the spindle 2 B-axis and its clearance (from the pickup point).



3. 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)**.

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.

#### 4. Cut-off:

Use operation **Cutoff** using tool on **MainSpindle (Spindle Name)**.

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	
OK	Cancel
Help	
General	
Stock Definition	
Type	Diameter
Stock Diameter	60.000000
Rough	
Rough Passes	
Rough Stock Allowance	0.000000
Rough to Diameter	0.000000
Stock Clearance	2.000000
Dwell Time	0.000000
	G04 U
Pecking	
Peck Increment	0.000000
Features	Tools
Operations	SolidTurn - Cutoff

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

5. 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.

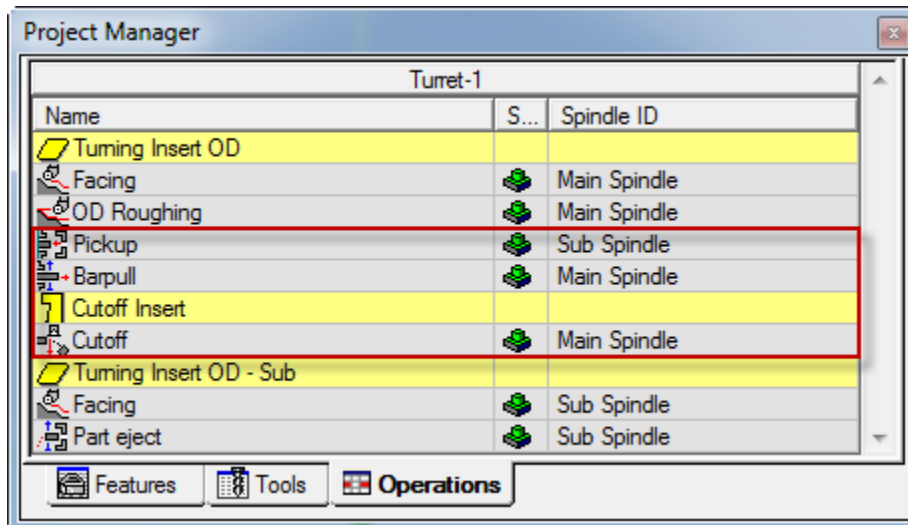
The setting “Part Stock Length” from the Machine Setup is used as default value for Turning Work Coordinates Sub Spindle Z Offset.

**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.

6. Catching the finished part in the spindle 2:

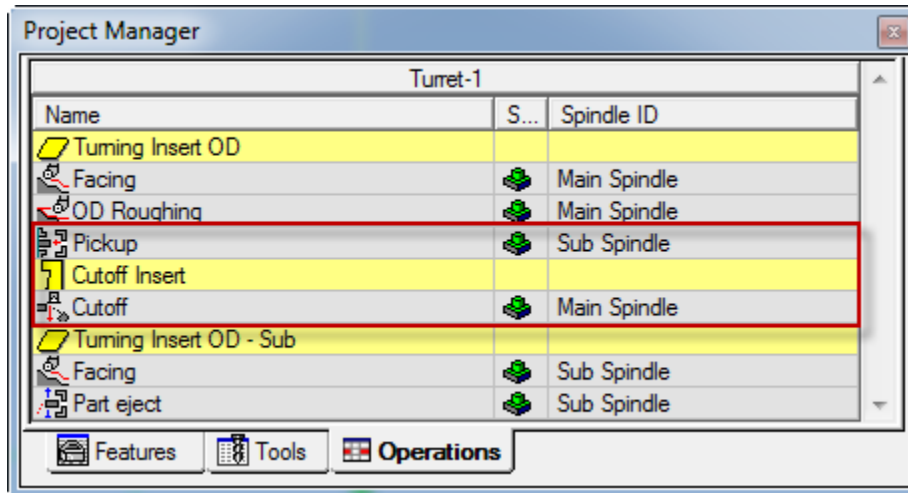
Spindle 2 needs to be empty before transferring the workpiece from spindle 1 to spindle 2 (see part **11.2.5. Catching finished part on spindle 2 side** for programming details).

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



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

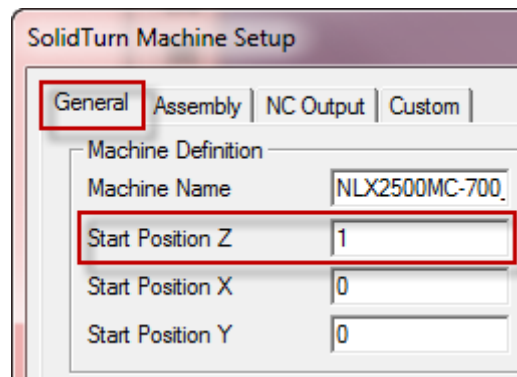




### 11.3.2. Part transfer from spindle 1 to spindle 2 with cut-off (when machine equipped with a barfeeder)

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.

*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*



The regular steps to follow are:

1. Machining the part on the spindle 1 side
2. Perform a barfeed operation
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. Machining the part on the spindle 2 side
6. Catching the finished part in the spindle 2
7. Looping for next parts

*Note:* It is also possible to perform all operations on spindle 1 and spindle 2 sides and then finish with the part transfer (after first catching finished part in the spindle 2).

Steps to program this transfer type in ESPRIT:

1. Machining the part on the spindle 1 side.
2. Perform a barfeed operation:  
Use operation **Bar Feed By Stopper (Bar Feed Type)** using tool on **MainSpindle (Spindle Name)**.

On Bar Feed tab, set the Feed Length and Reposition Distance:

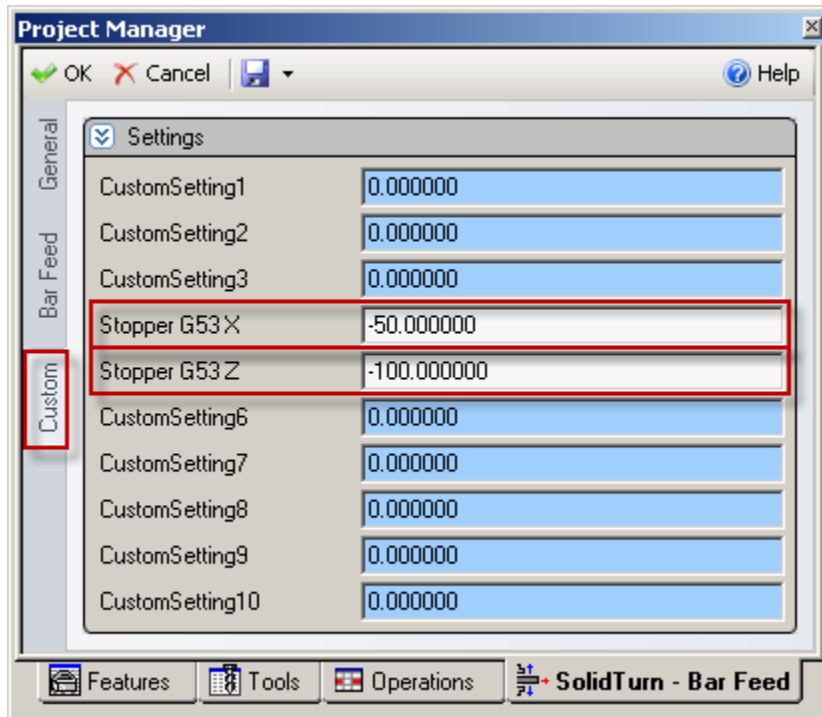
*Feed Length (=Barfeed distance = Reposition Distance) = Length of the finished part (Part Stock Length) + Cut-off tool width + Length of stock for facing the front side and the back side of the part*

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

Bar 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 Stopper G53 Z (Custom Setting 5) field the Z position (in G53 work coordinate) of the stopper tool.

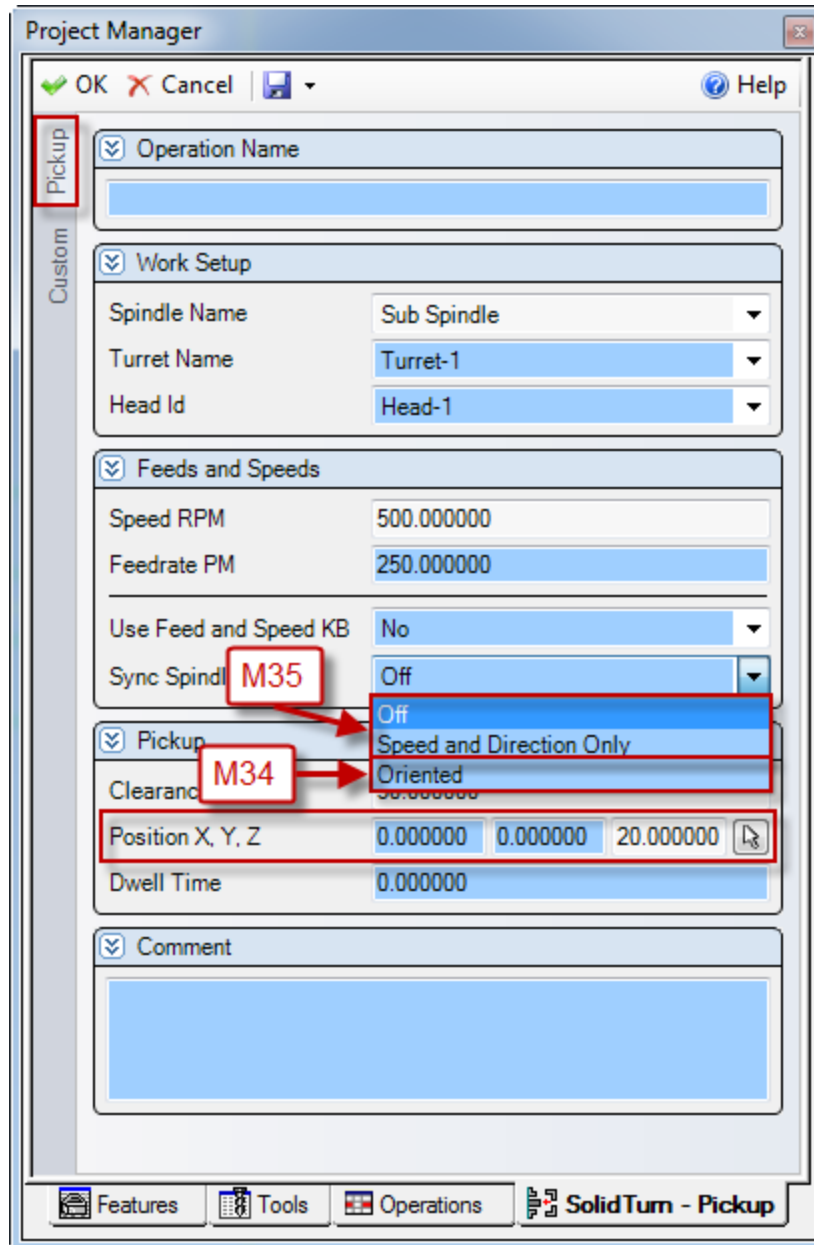


To use current machine work coordinate defined in ESPRIT (see part 2.3. **Turning Work Coordinates**): If no value is specified on the Custom tab in the Stopper G53 X and Stopper G53 Z fields (fields left equal to 0), the stopper tool will be positioned at the programmed position (Position X, Y, Z specified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

- Picking the part in the spindle 1 with the spindle 2:  
Use operation **Pickup** with **SubSpindle (Spindle 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.

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 after synchronization of the spindles.

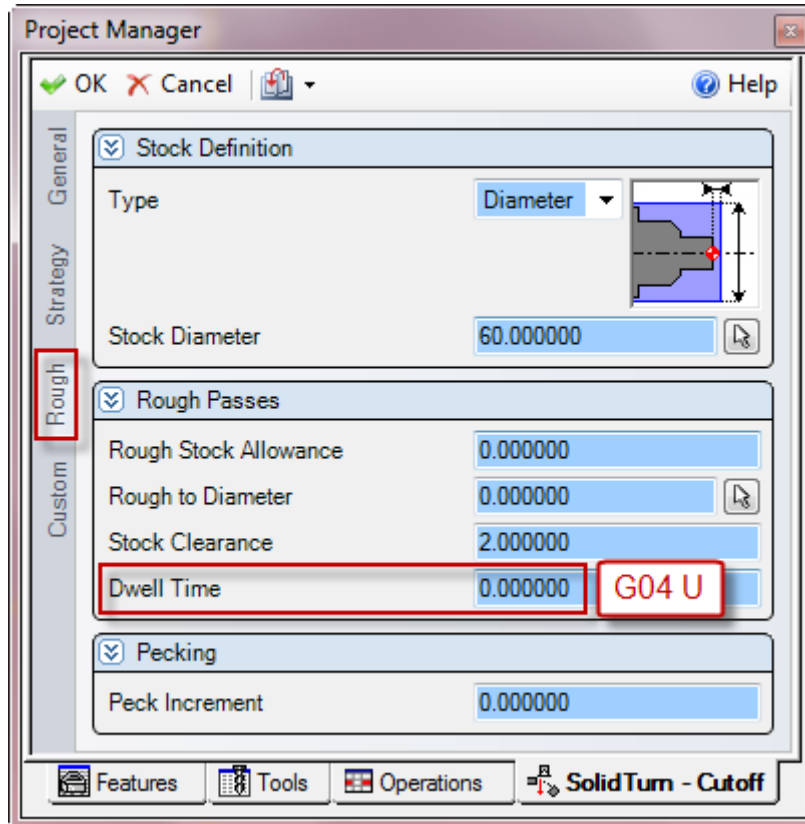
Enter the Feedrate PM of the spindle 2 B-axis and its clearance (from the pickup point).



4. Cut-off:

Use operation **Cutoff** using tool on **MainSpindle (Spindle Name)**.

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.



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

5. 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.

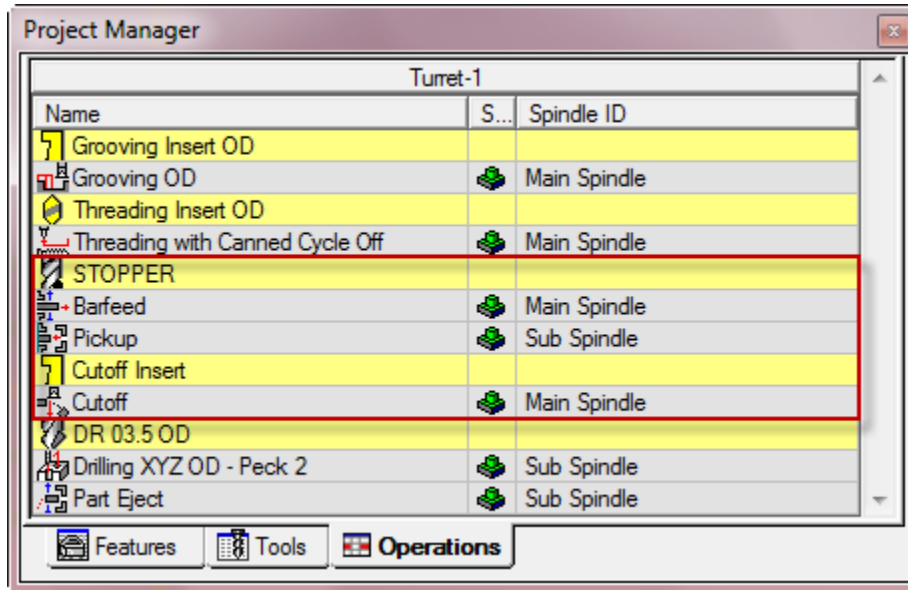
The setting “Part Stock Length” from the Machine Setup is used as default value for Turning Work Coordinates Sub Spindle Z Offset.

**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.

6. 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).

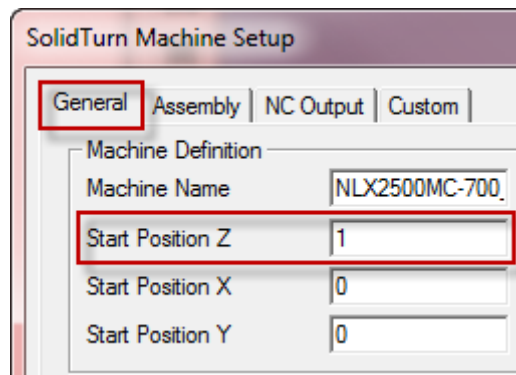
Sample operation list:



### 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.

*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*



The regular steps to follow are:

1. Machining the part on the spindle 1 side
2. Picking the part in the spindle 1 with the spindle 2
3. Perform a cut-off and then the spindle 2 goes back home
4. Perform a barfeed operation
5. Machining the part on the spindle 2 side
6. Catching the finished part in the spindle 2
7. Looping for next parts

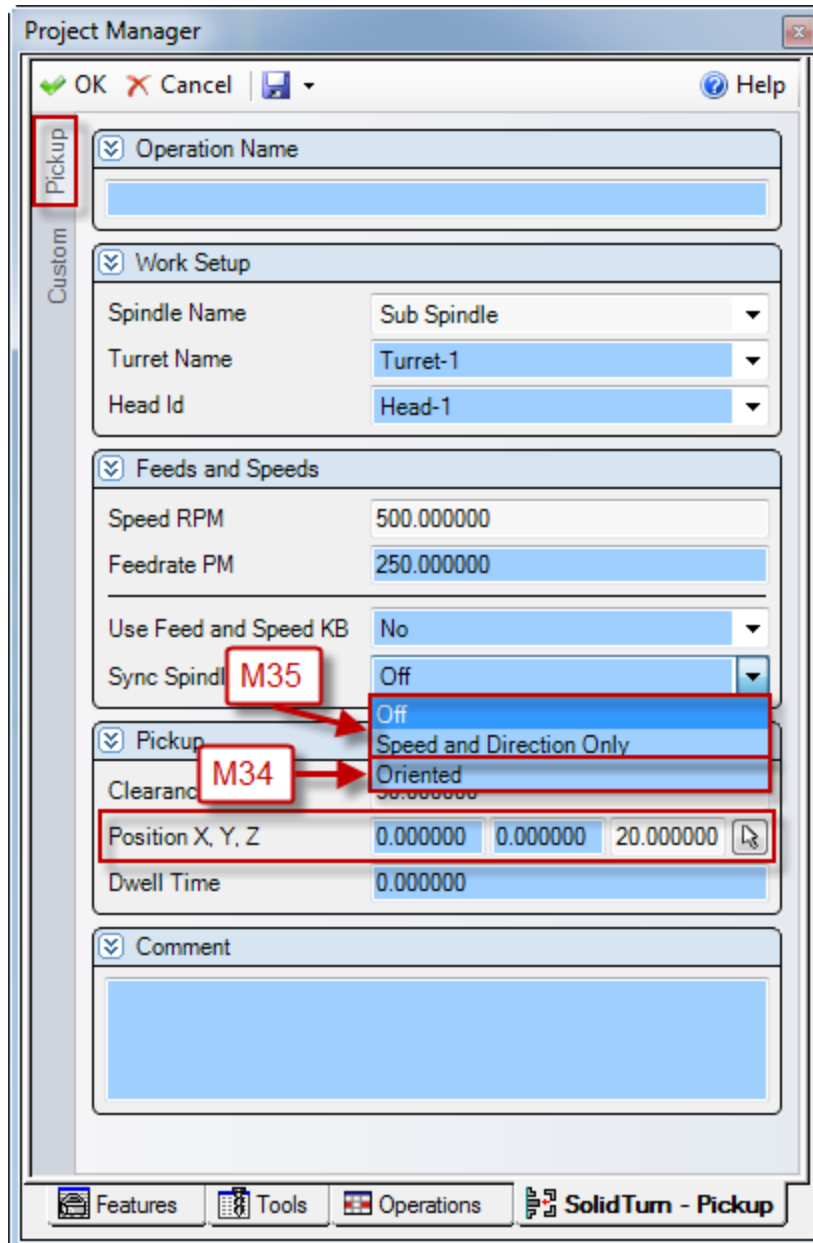
*Note:* It is also possible to perform all operations on spindle 1 and spindle 2 sides and then finish with the part transfer (after first catching finished part in the spindle 2).

Steps to program this transfer type in ESPRIT:

1. Machining the part on the spindle 1 side.
2. Picking the part in the spindle 1 with the spindle 2:  
Use operation **Pickup** with **SubSpindle (Spindle Name)**. With Position X, Y, Z, set the pickup point.

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 after synchronization of the spindles.

Enter the Feedrate PM of the spindle 2 B-axis and its clearance (from the pickup point).

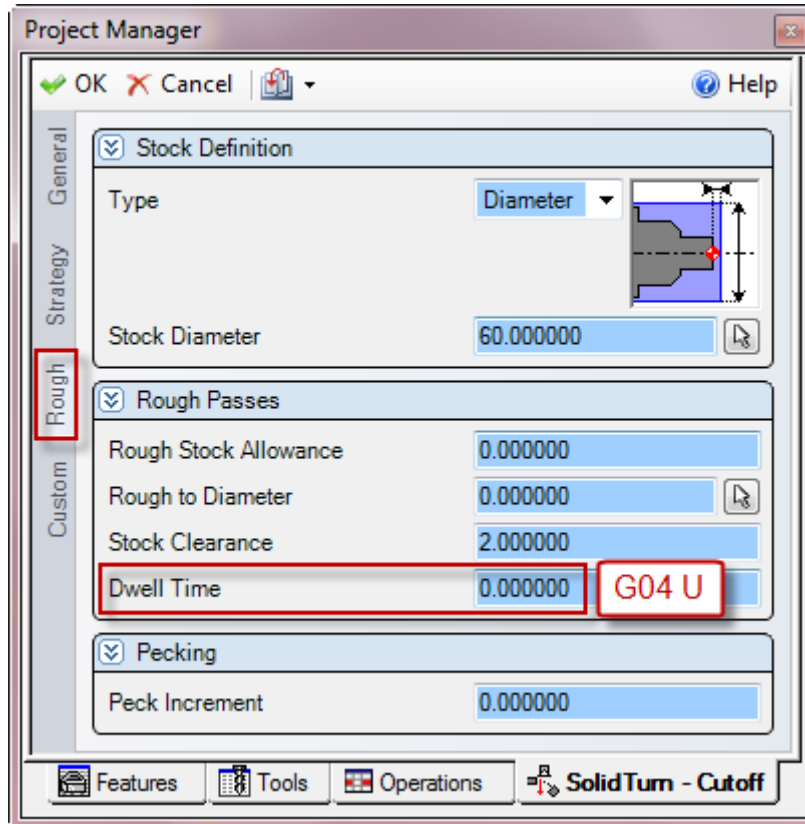


### 3. Cut-off:

Use operation **Cutoff** using tool on **MainSpindle (Spindle Name)**.

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.





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

4. Perform a barfeed operation:  
Use operation **Bar Feed By Stopper (Bar Feed Type)** using 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 Stopper G53 Z (Custom Setting 5) field the Z position (in G53 work coordinate) of the stopper tool.

Project Manager	
<input type="button" value="OK"/> <input type="button" value="Cancel"/> <input type="button" value="Help"/>	
General	Settings
Bar Feed	CustomSetting1: 0.000000
	CustomSetting2: 0.000000
	CustomSetting3: 0.000000
Custom	Stopper G53 X: -50.000000
	Stopper G53 Z: -100.000000
	CustomSetting6: 0.000000
	CustomSetting7: 0.000000
	CustomSetting8: 0.000000
	CustomSetting9: 0.000000
	CustomSetting10: 0.000000
<input type="button" value="Features"/> <input type="button" value="Tools"/> <input type="button" value="Operations"/> <input type="button" value="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.

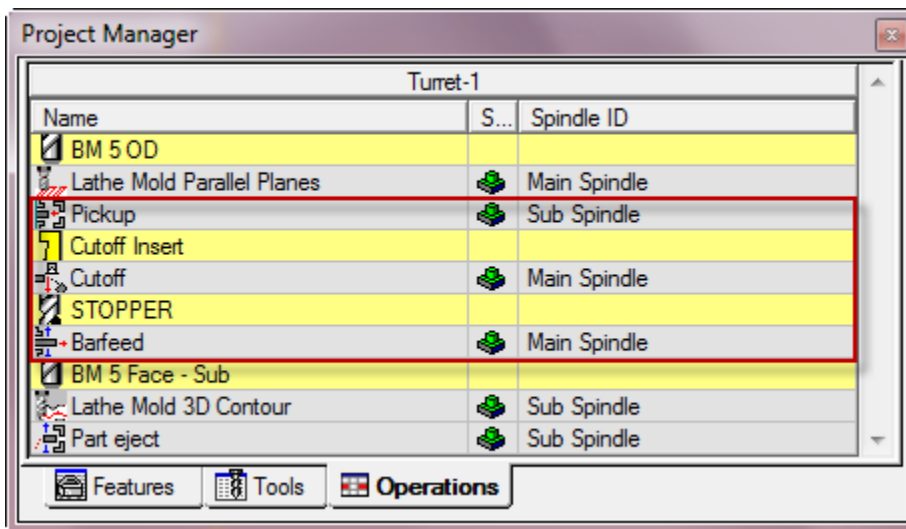
- Machining the part on the spindle 2 side:  
*Note:* The operations on the spindle 2 side are output with the origin shifted to the other face of the part (back face). During operations, only positive Z coordinates are output.

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.

- 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).

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

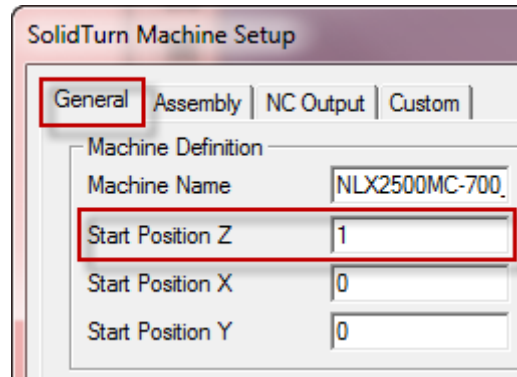


#### 11.3.4. Part transfer from spindle 1 to spindle 2 without cut-off

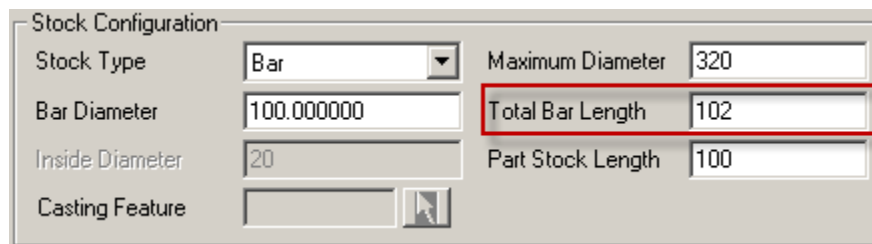
The stock is manually loaded in the spindle 1; the part is not pulled. The finished part in the spindle 2 has to be released. The entire part is transferred to the spindle 2 when the work on spindle 1 is finished.

*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*



*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*



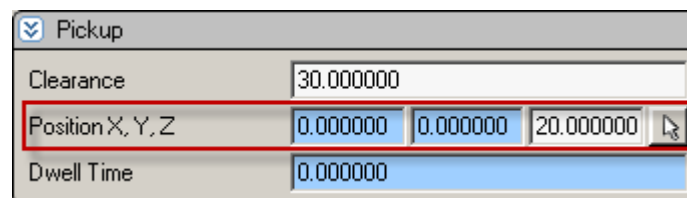
The regular steps to follow are:

1. Machining the part on the spindle 1 side
2. Picking the part in the spindle 1 with the spindle 2
3. Releasing the part from the spindle 1 and then the spindle 2 goes back home
4. Machining the part on the spindle 2 side
5. Catching the finished part in the spindle 2
6. Stop for loading a new stock in the spindle 1

*Note:* It is also possible to perform all operations on spindle 1 and spindle 2 sides and then finish with the part transfer (after first catching finished part in the spindle 2).

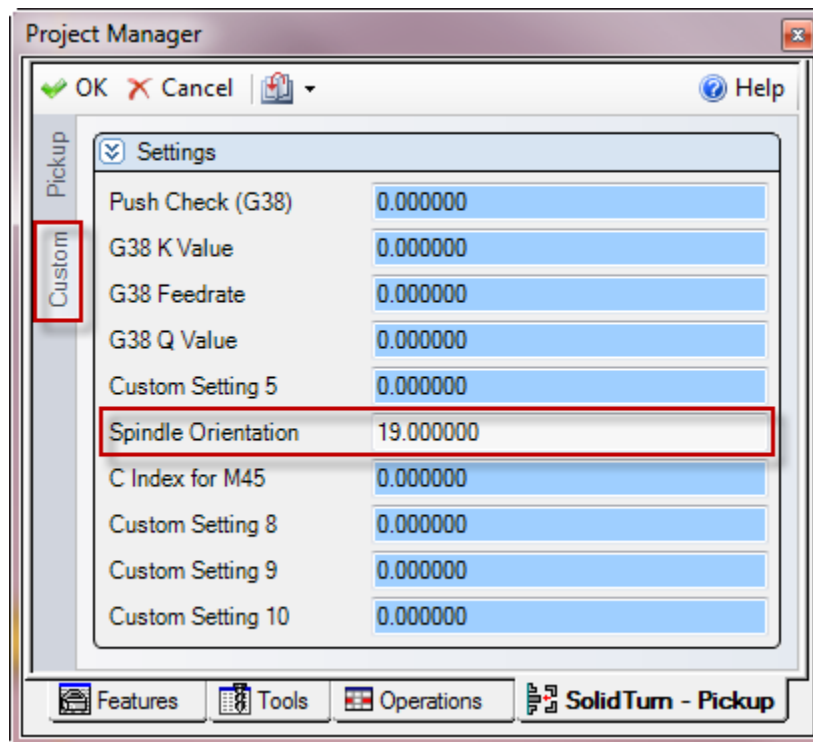
Steps to program this transfer type in ESPRIT:

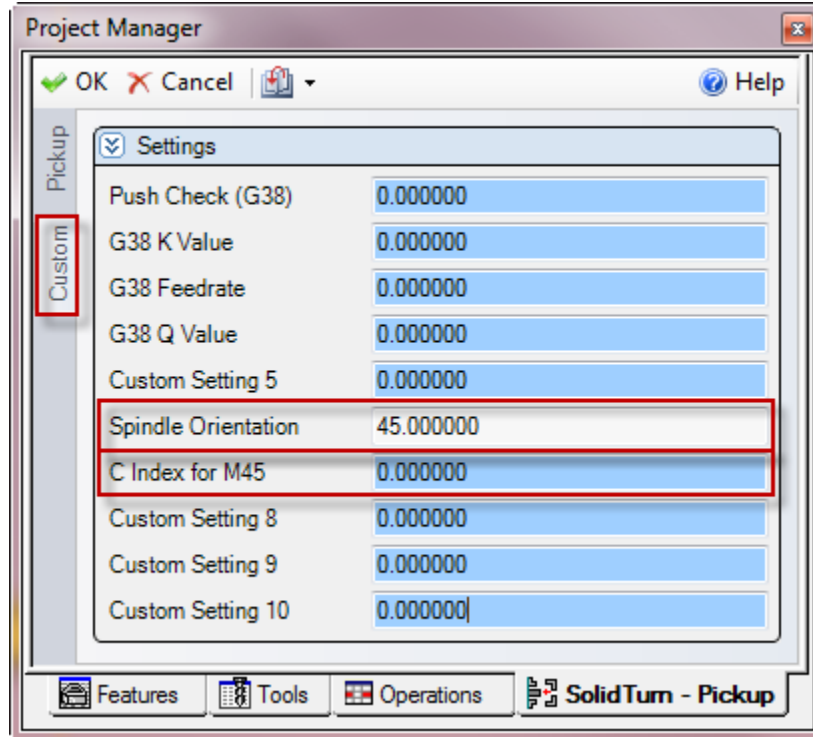
1. Machining the part on the spindle 1 side.
2. Picking the part in the spindle 1 with the spindle 2:  
Use operation **Pickup** with **SubSpindle (Spindle Name)**. With Position X, Y, Z, set the pickup point.



Enter the Feedrate PM of the spindle 2 B-axis and its clearance (from the pickup point).

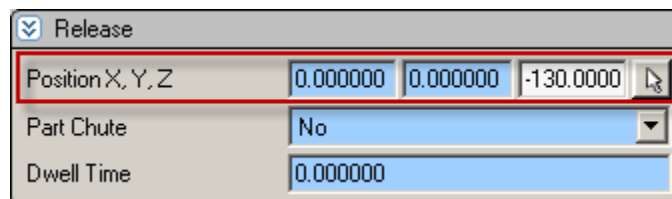
During this type of transfer, the turning spindles are stopped. Using Spindle Orientation (Custom Setting 6) on the Custom tab of the Pickup operation, you can specify how you want to orient your spindles. Set Spindle Orientation to 19 if you want to use spindle orientation (M19 / M219) or set Spindle 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. Make sure to enter a correct value for C. For example, if C-axis roll-over is on (see part **4.4.8. C-Axis roll over**), C has to be contained between  $-359.999^{\circ}$  and  $359.999^{\circ}$ .





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. Releasing the part from the spindle 1:  
Use operation **Release** with **MainSpindle (Spindle Name)**. With Position X, Y, Z, set the release point.



4. 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.

5. 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).

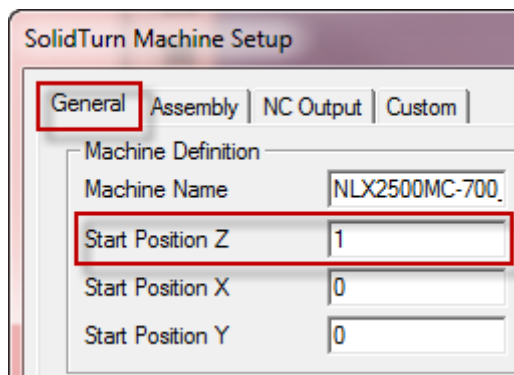
Sample operation list:

Turret-1		
Name	Stock	Spindle ID
EM 20 OD		
Spiral Milling		Main Spindle
Pickup		Sub Spindle
Release		Main Spindle
EM 20 OD		
Spiral Milling		Sub Spindle
Part eject		Sub Spindle

### 11.3.5. Part transfer from spindle 2 to spindle 1 without cut-off

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.

*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*



*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			

The regular steps to follow are:

1. Machining the part on the spindle 2 side
2. Picking the part in the spindle 2 with the spindle 1
3. Releasing the part from the spindle 2 and then the spindle 2 goes back home
4. Machining the part on the spindle 1 side
5. Stop to manually unload the finished part in the spindle 1 (and to load a new stock in the spindle 2)

*Note:* It is also possible to perform all operations on spindle 1 and spindle 2 sides and then finish with the part transfer (after first catching finished part in the spindle 1).

Steps to program this transfer 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. Picking the part in the spindle 2 with the spindle 1:  
Use operation **Pickup** with **MainSpindle (Spindle 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).

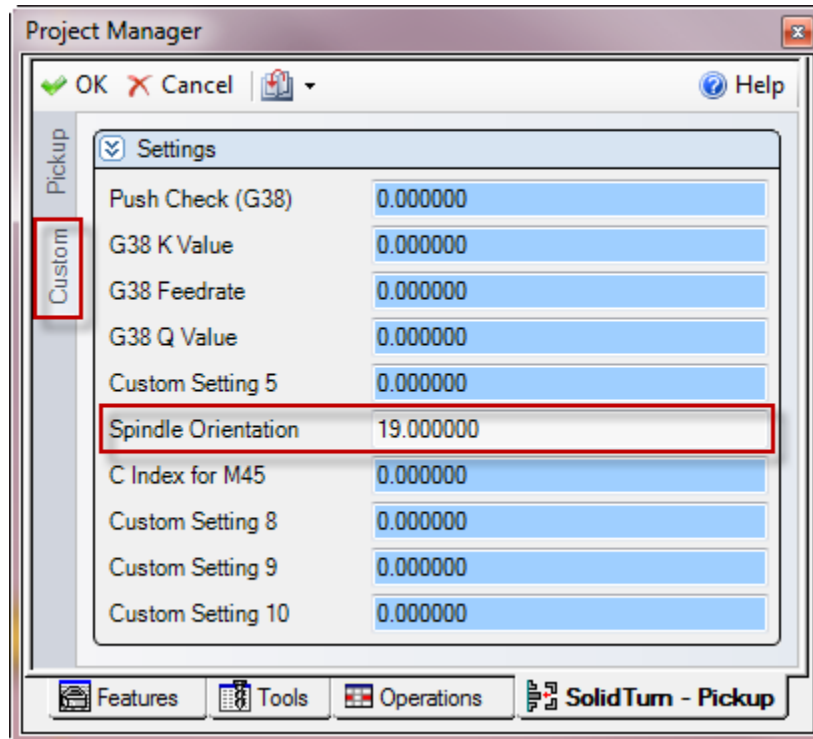
Pickup			
Clearance	45.000000		
Position X, Y, Z	0.000000	0.000000	-125.0000
Dwell Time	0.000000		

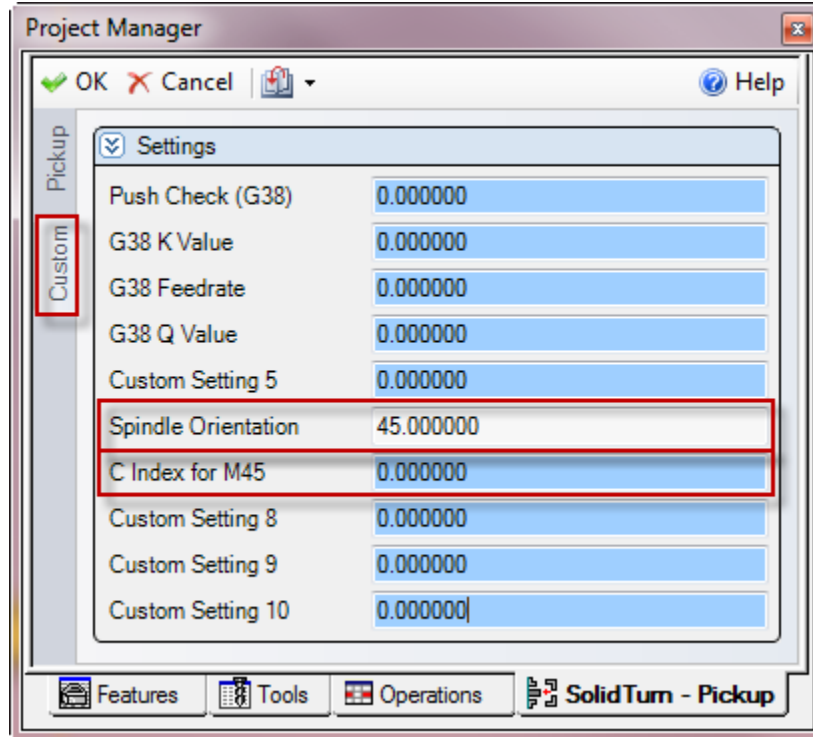
Enter the Feedrate PM of the spindle 2 B-axis and its clearance (from the pickup point).

During this type of transfer, the turning spindles are stopped. Using Spindle



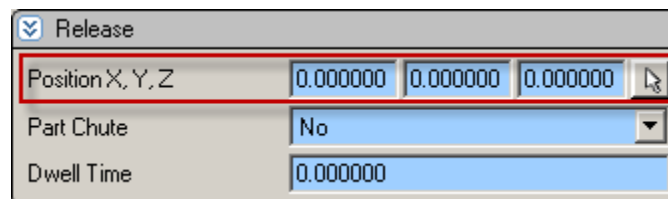
Orientation (Custom Setting 6) on the Custom tab of the Pickup operation, you can specify how you want to orient your spindles. Set Spindle Orientation to 19 if you want to use spindle orientation (M19 / M219) or set Spindle Orientation to 45 if you want to use the milling C-axis mode (M45 / M245). If 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. Make sure to enter a correct value for C. For example, if C-axis roll-over is on (see part **4.4.8. C-Axis roll over**), C has to be contained between  $-359.999^{\circ}$  and  $359.999^{\circ}$ .





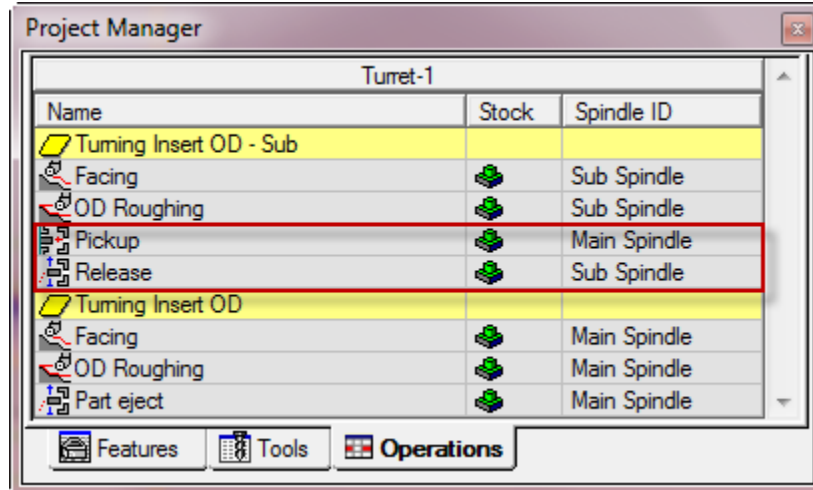
Note that the milling C-axis mode is the default. So if you omit to enter a value in the Spindle Orientation field or enter a wrong value, the spindles will be indexed using this mode.

3. Releasing the part from the spindle 2:  
Use operation **Release** with **SubSpindle (Spindle Name)**. With Position X, Y, Z, set the release point.



4. Machining the part on the spindle 1 side.
5. Stop to manually unload the finished part in the spindle 1 (and to load a new stock in the spindle 2):  
To be able to catch the finished part in the spindle 1, the machine needs to be stopped (see part **11.2.4. Manually catching finished part on spindle 1 side** for programming details).

Sample operation list:



### 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.

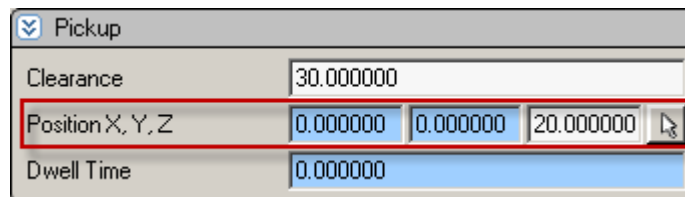
*Note:* Spindle 2 needs to be empty before repositioning since spindle 2 will pick up the part in spindle 1.

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:

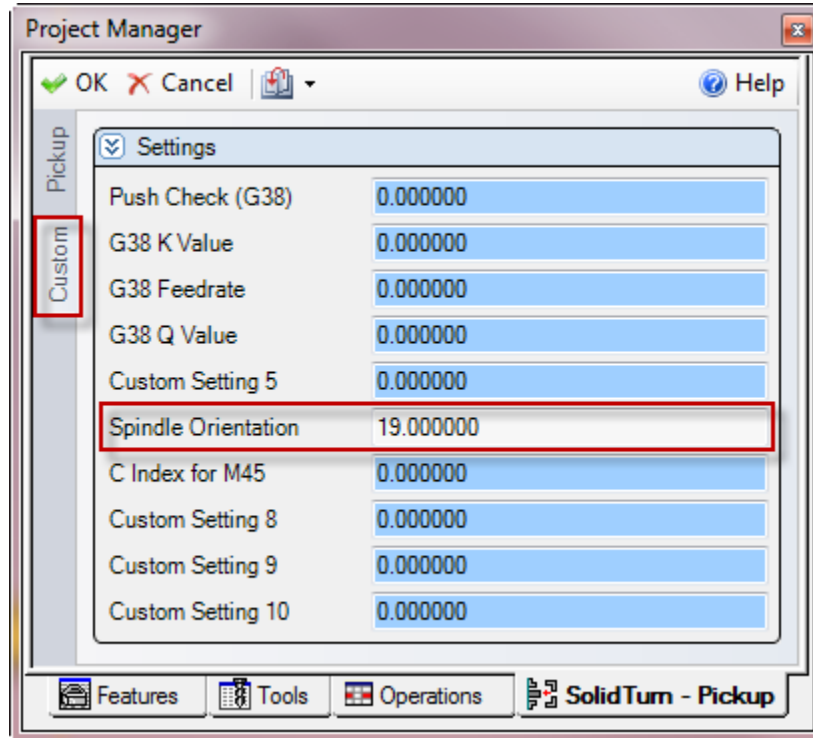
1. Picking the part in the spindle 1 with the (empty) spindle 2:  
Use operation **Pickup** with **SubSpindle (Spindle Name)**. With Position X, Y, Z, set the pickup point.

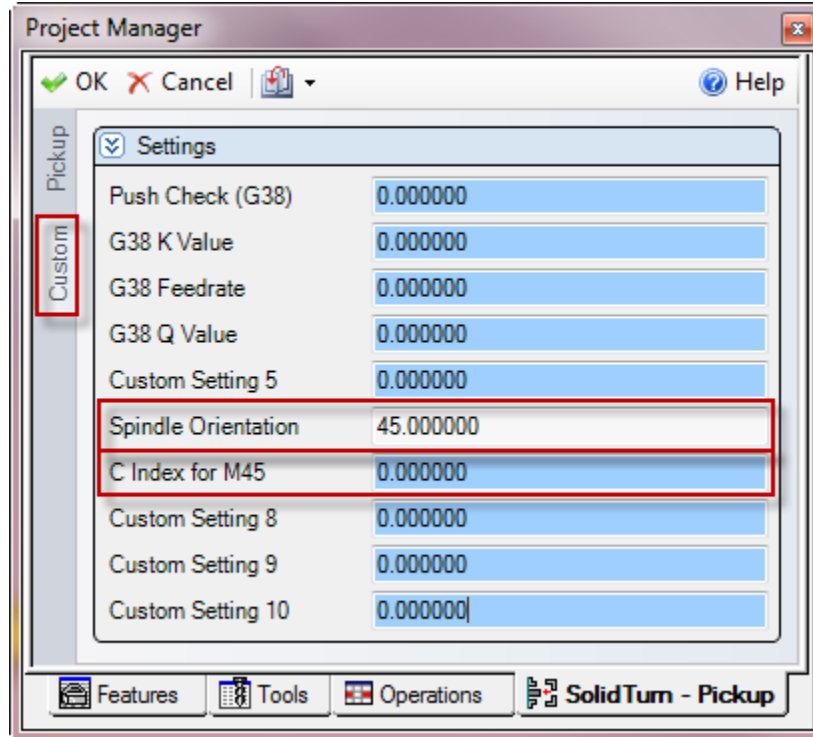


Enter the Feedrate PM of the spindle 2 B-axis and its clearance (from the pickup point).

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. Make sure to enter a correct value for C. For example, if C-axis roll-over is on (see part **4.4.8. C-Axis roll over**), C has to be contained between  $-359.999^\circ$  and  $359.999^\circ$ .





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.

2. 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)**.

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.

3. Releasing the part from the spindle 2:  
Use operation **Release** with **SubSpindle (Spindle Name)**. With Position X, Y, Z, set the release point.

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

Sample operation list:

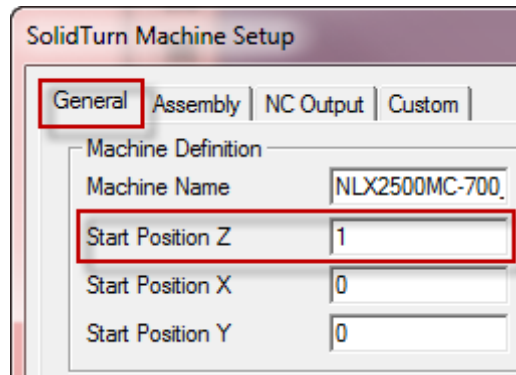
Project Manager		
Turret-1		
Name	Stock	Spindle ID
Pickup	🌱	Sub Spindle
Barpull	🌱	Main Spindle
Release	🌱	Sub Spindle
Turning Insert OD		
Facing	🌱	Main Spindle
OD Roughing	🌱	Main Spindle
Cutoff Insert		
Cutoff	🌱	Main Spindle

Features | Tools | **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 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*



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 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.

2. 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. Picking the part in the spindle 1 with the spindle 2:  
Use operation **Pickup** with **SubSpindle (Spindle Name)**. With Position X, Y, Z, set the pickup point.

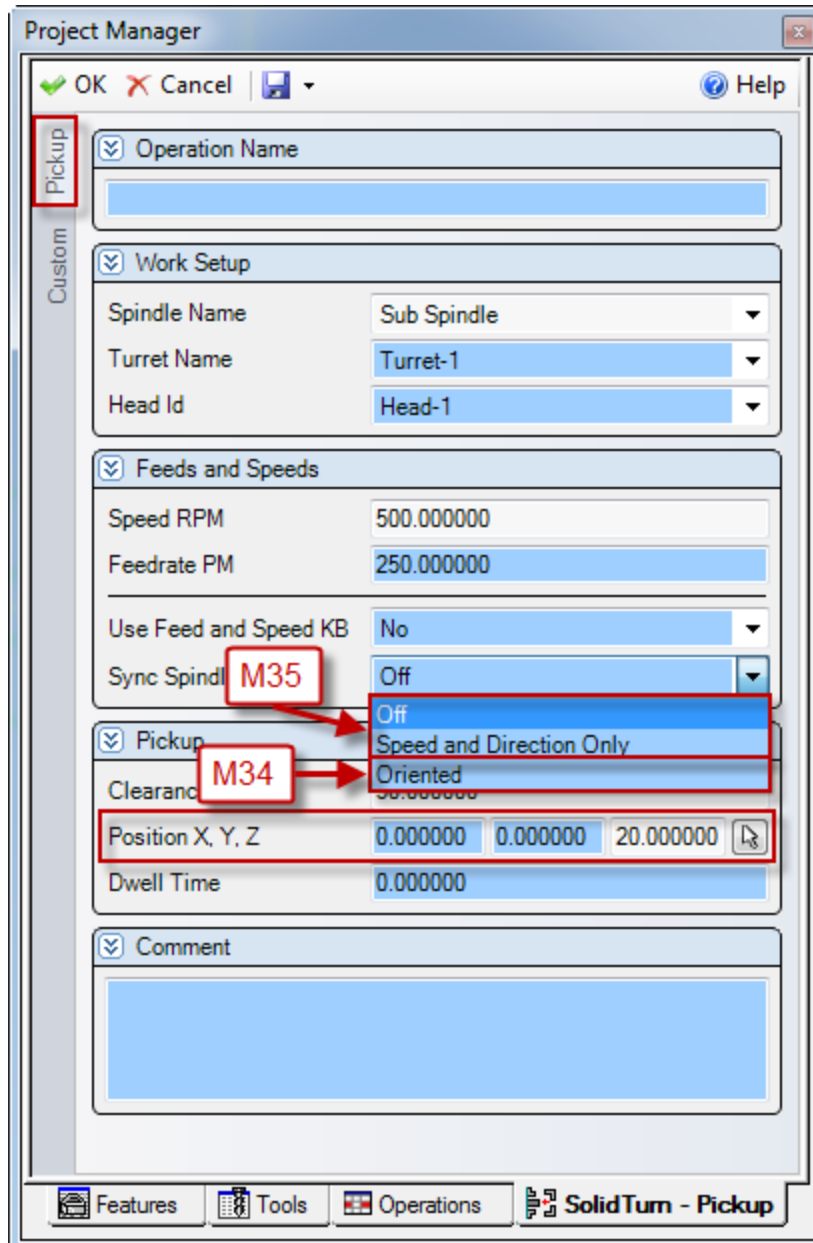
Based on the following operation, the correct synchronization code will be output in the code:

- **M34** (or **M35**): Turning operation is next; the spindle speed entered on the SolidTurn Pickup page will be output after synchronization of the spindles; the spindle direction code will be output based on the direction of the following turning operation.

- **M480**: Milling operation is next; the spindles are stopped.

Enter the Feedrate PM of the spindle 2 B-axis and its clearance (from the pickup point).





*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.

4. 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)**.

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).

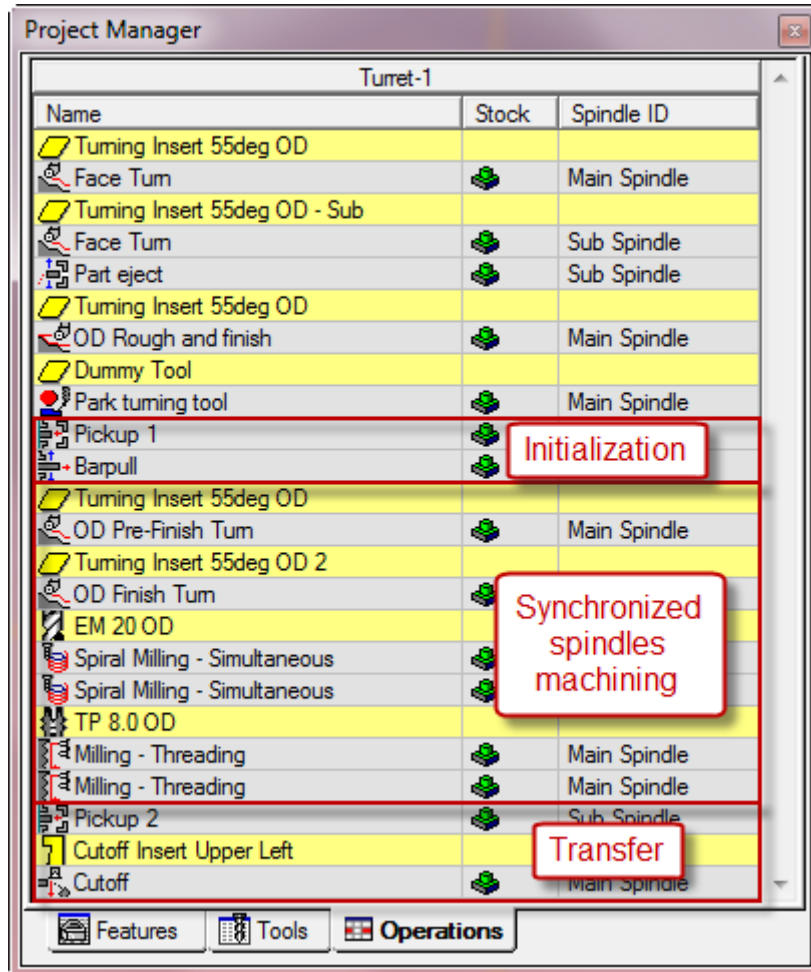
Bar Feed			
Position X, Y, Z	0.000000	0.000000	20.000000
Feed Length	41.000000		
Reposition Distance	41.000000		
Bar Diameter	60.000000		
Total Bar Length	100.000000		
Length Comp Register	0		
Dwell Time	0.000000		

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.

5. Machining the part on the spindle 1 side 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.
6. Perform one of the previously described transfer scenarios to transfer the part to spindle 2 side.

Sample operation list:



### 11.3.8. Workpiece Pushing Check (G38)

**G38 B\_ K\_ F\_ Q\_;**

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

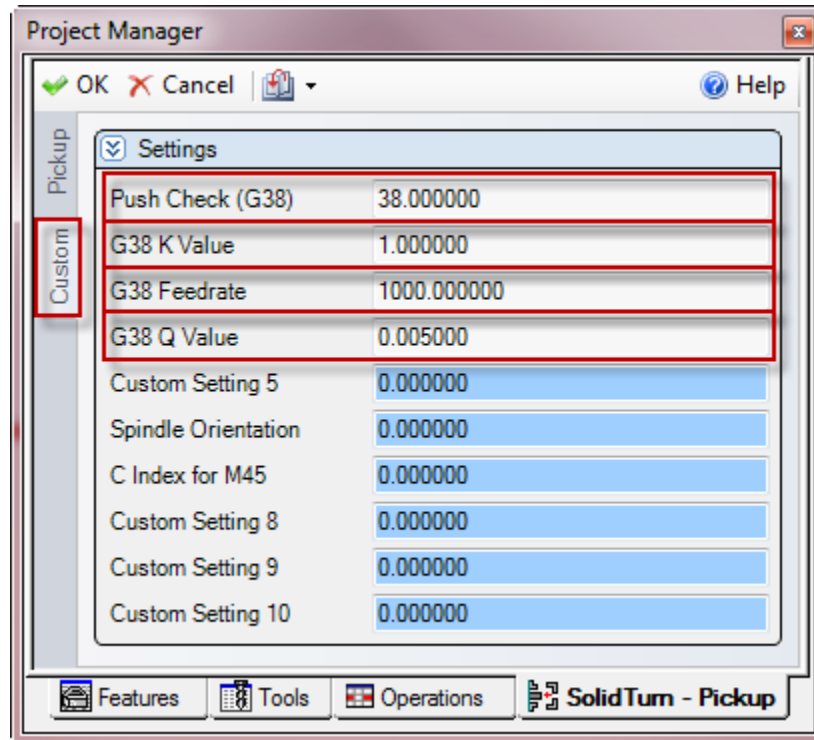
In the workpiece transfer operation, when the position error of the B-axis servomotor of headstock 2 reaches the parameter set value, the workpiece is judged to have been brought into contact with the reference face of chuck 2 or chuck 1. Then, chuck 2 or chuck 1 clamps the workpiece and the program 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)). 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.



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 the turret before transfer.

## 12. Multiple repetitive cycles

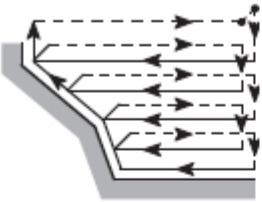
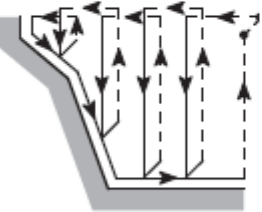
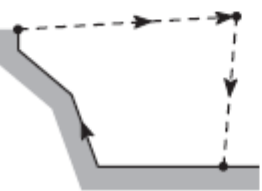
### 12.1. Introduction

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

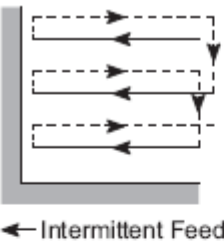
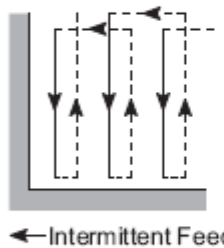
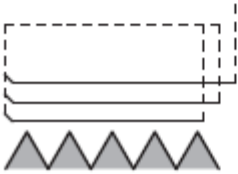
Roughing processes that require several blocks of commands can be specified by a single block of commands preceded by a G code calling a multiple repetitive cycle, and blocks that define the finished shape. The tool paths for rough cutting cycles are automatically determined.

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:**

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

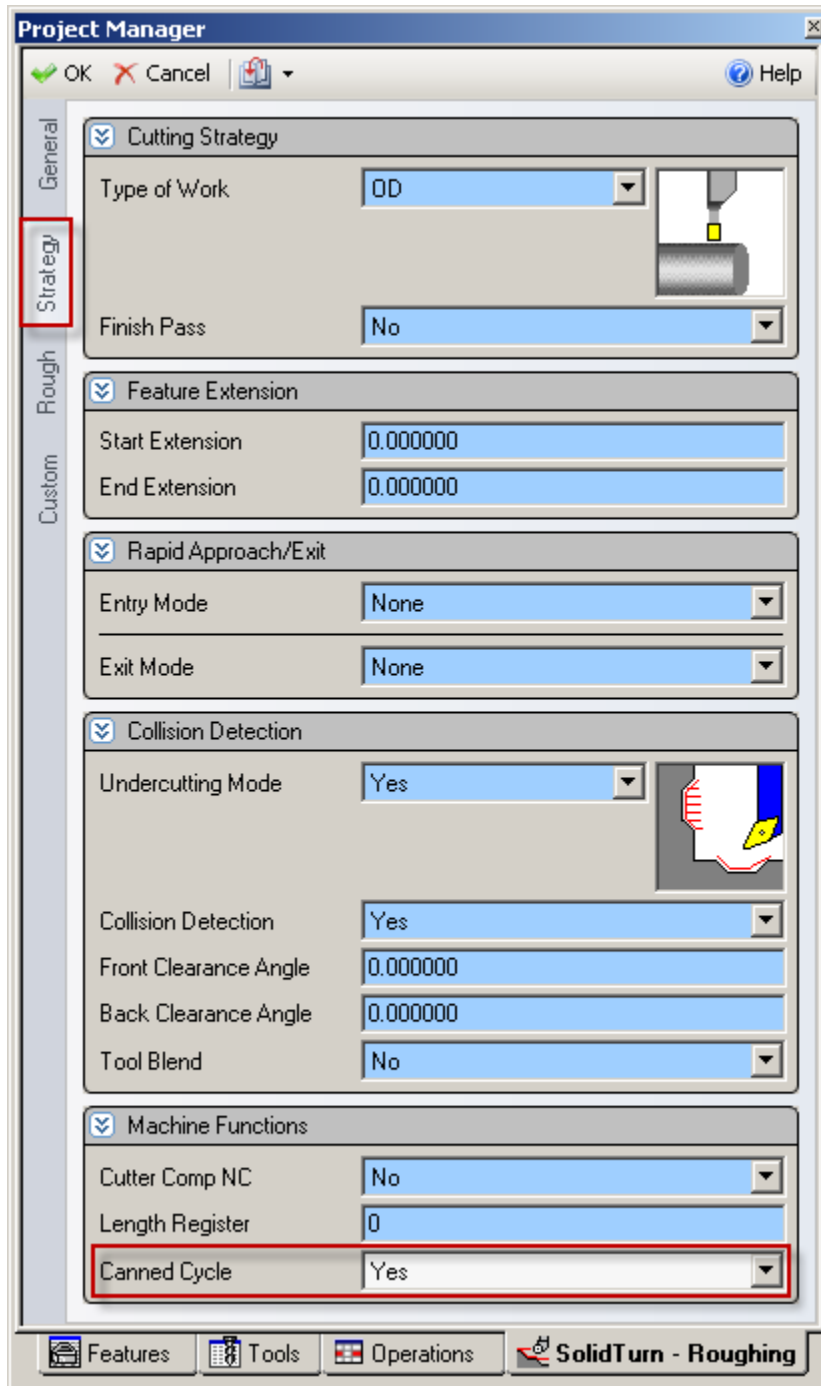
**O.D./I.D. groove, face groove and threading cycles:**

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

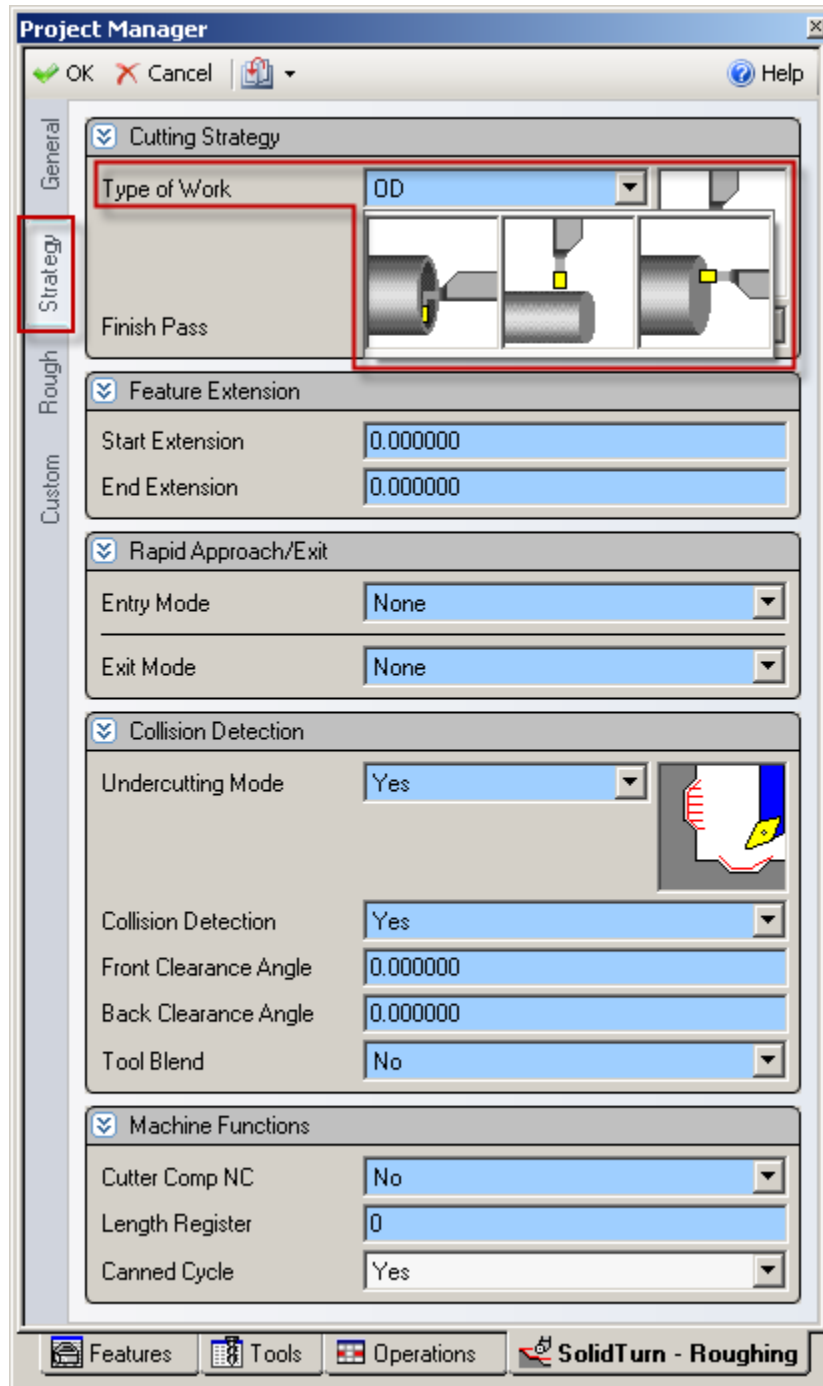
## 12.2. Roughing cycle

### 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.



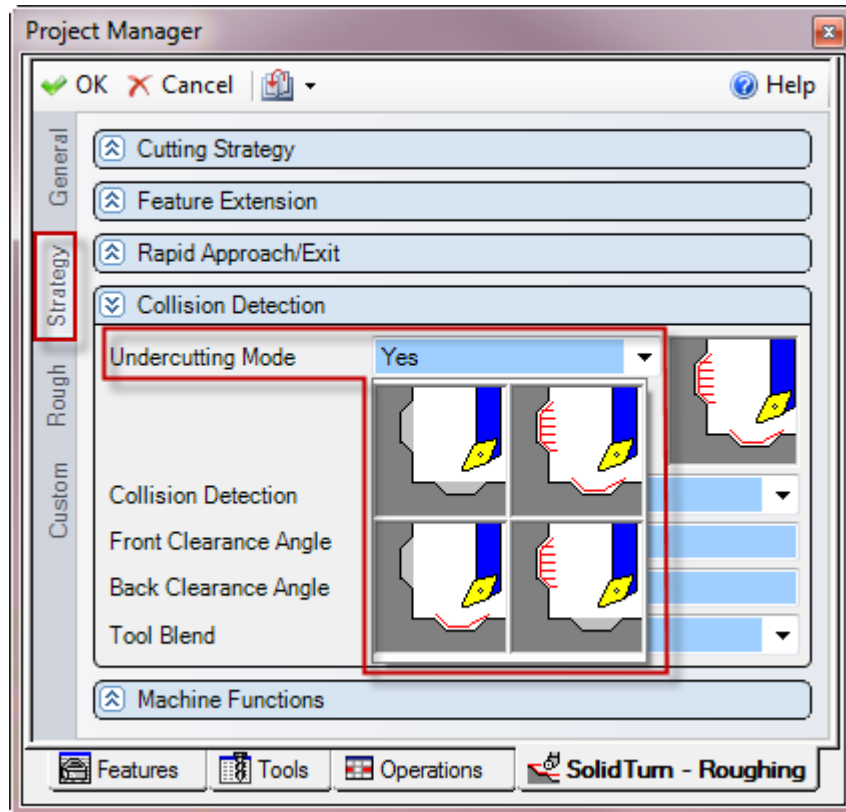
Based on the selected work strategy (Type of Work), the output G-code will be different. G71 will be output if your profile is on the O.D. or the I.D. of the part. G72 will be output if you cut on the face of the part.



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

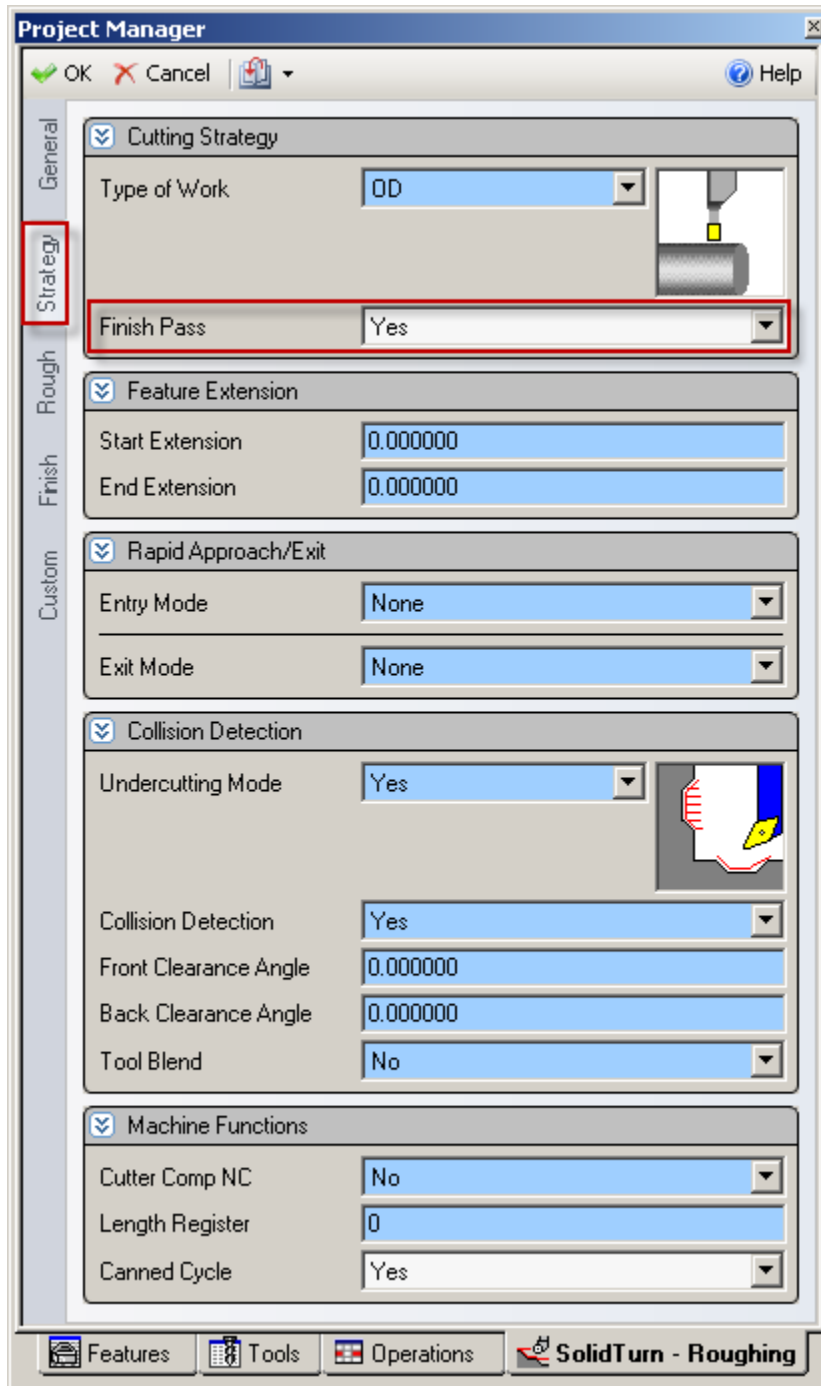
- **Yes:** All pockets on the O.D. or I.D. and the face will be cut.
- **Below:** Only pockets on the O.D. or I.D. will be cut.
- **Front:** Only pockets on the face will be cut.





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.

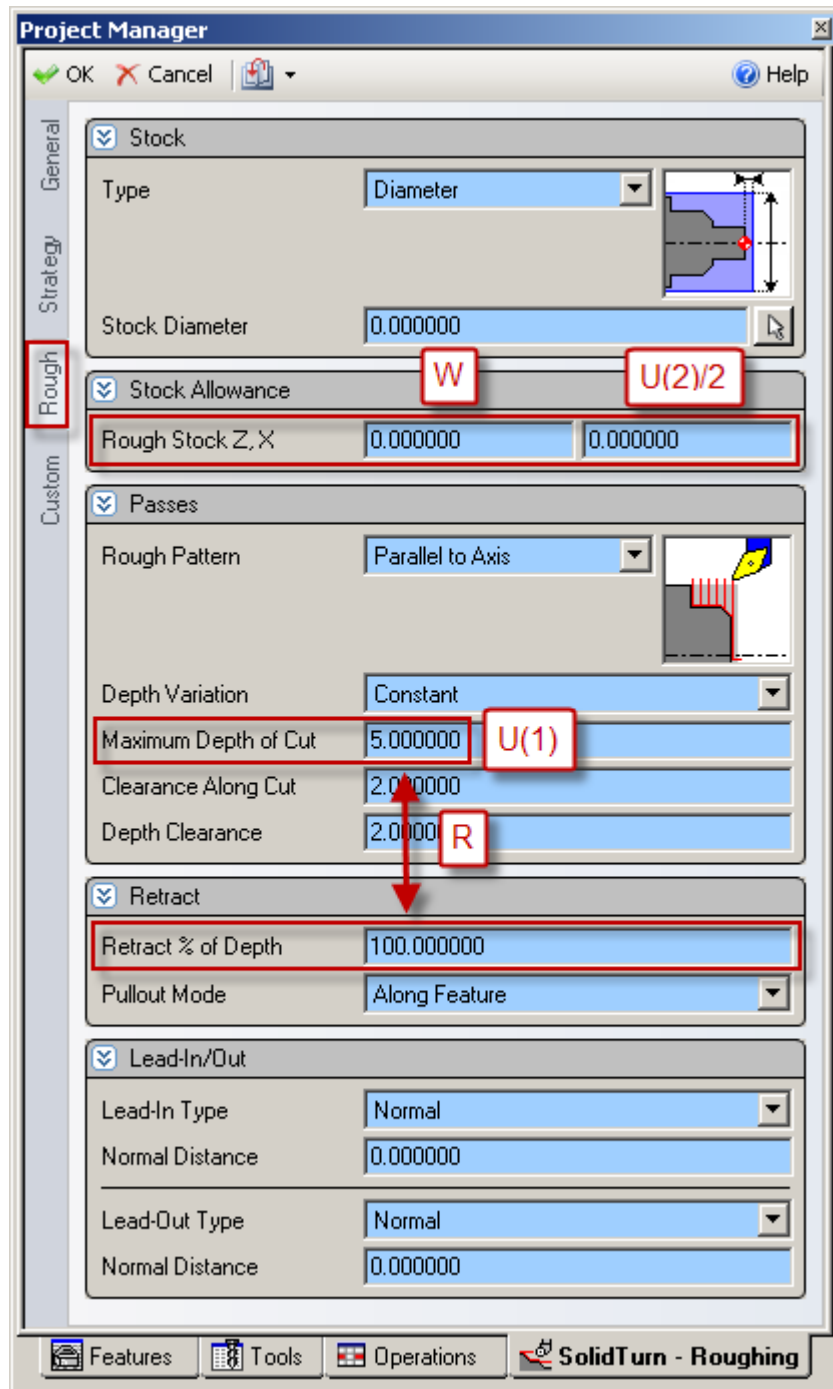


Information about some of the parameters to enter for the following cycles:

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

### 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

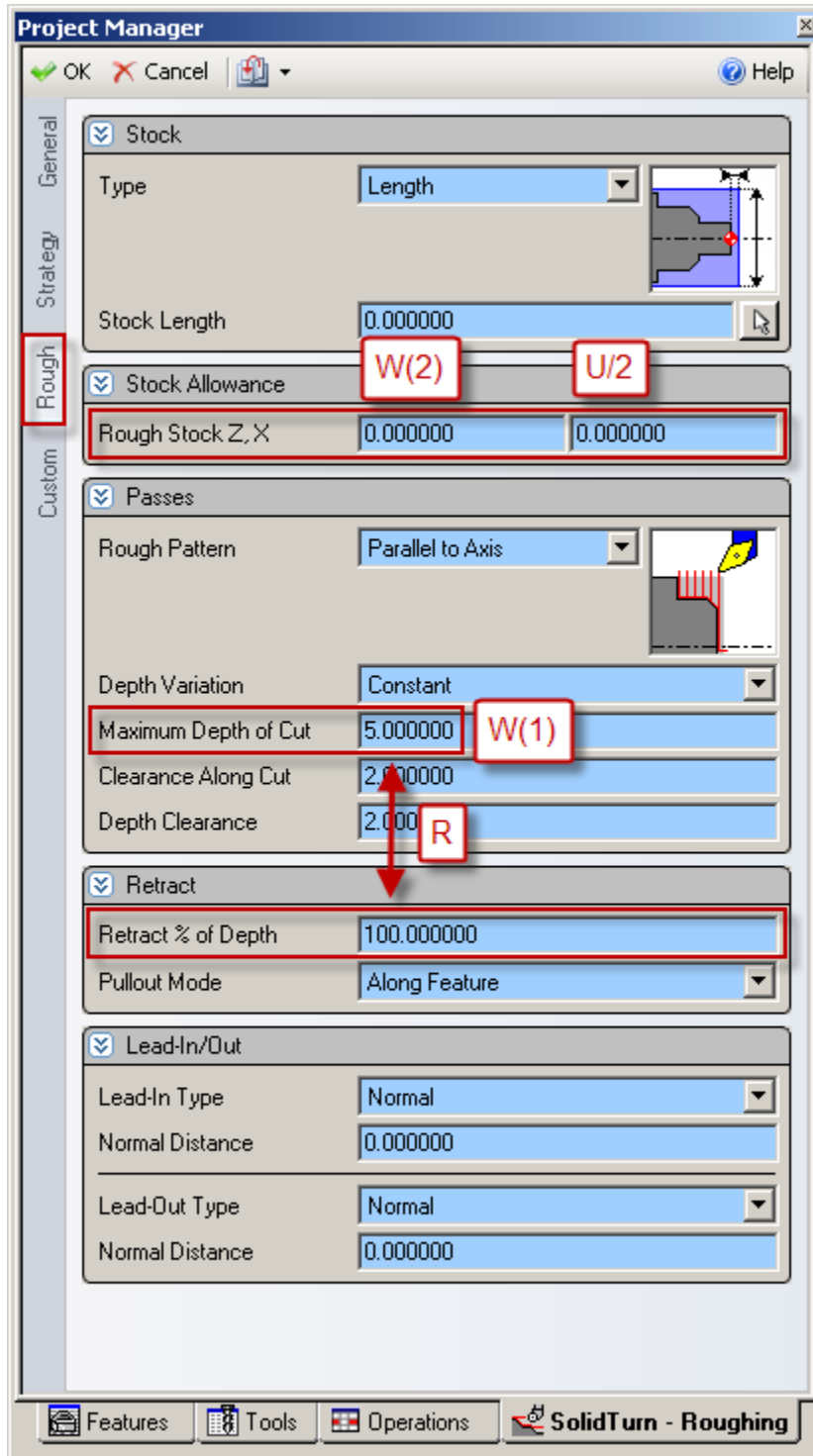
Maximum Depth of Cut and Retract % of Depth value:  $R = \text{Maximum Depth of Cut} * \text{Retract \% of Depth} / 100$ .

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\_ ;**

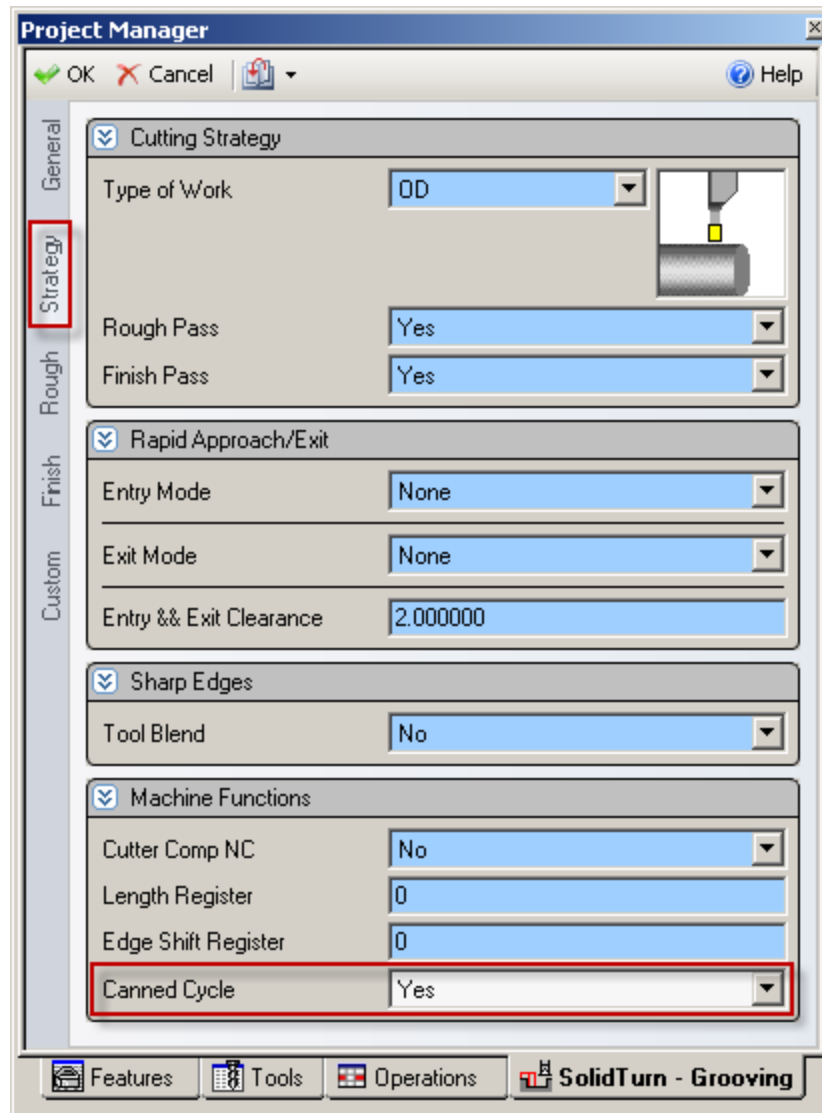


**W(1)** and **W(2)** will be output as entered in ESPRIT. **U/2** will be multiplied by 2 to have the correct value in the NC code **U**. **R** will be computed based on the entered Maximum Depth of Cut and Retract % of Depth value:  $R = \text{Maximum Depth of Cut} * \text{Retract \% of Depth} / 100$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

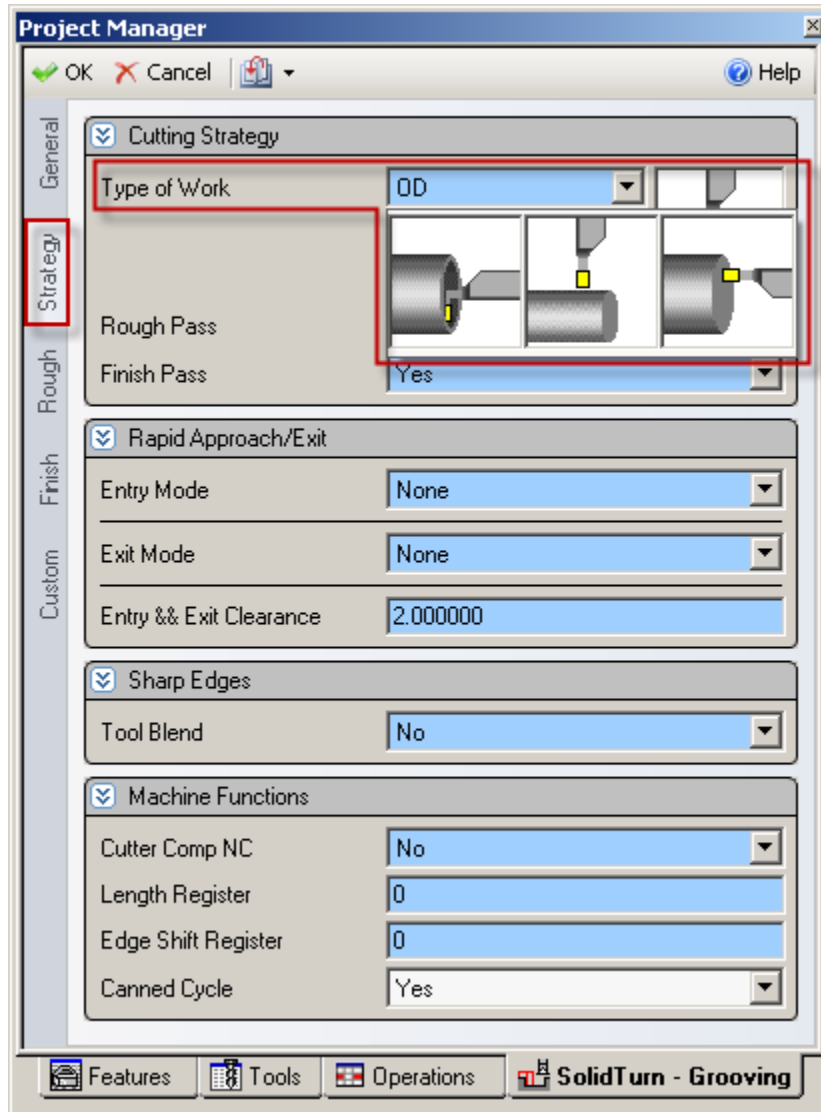
## 12.3. Grooving cycle

### 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.



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.



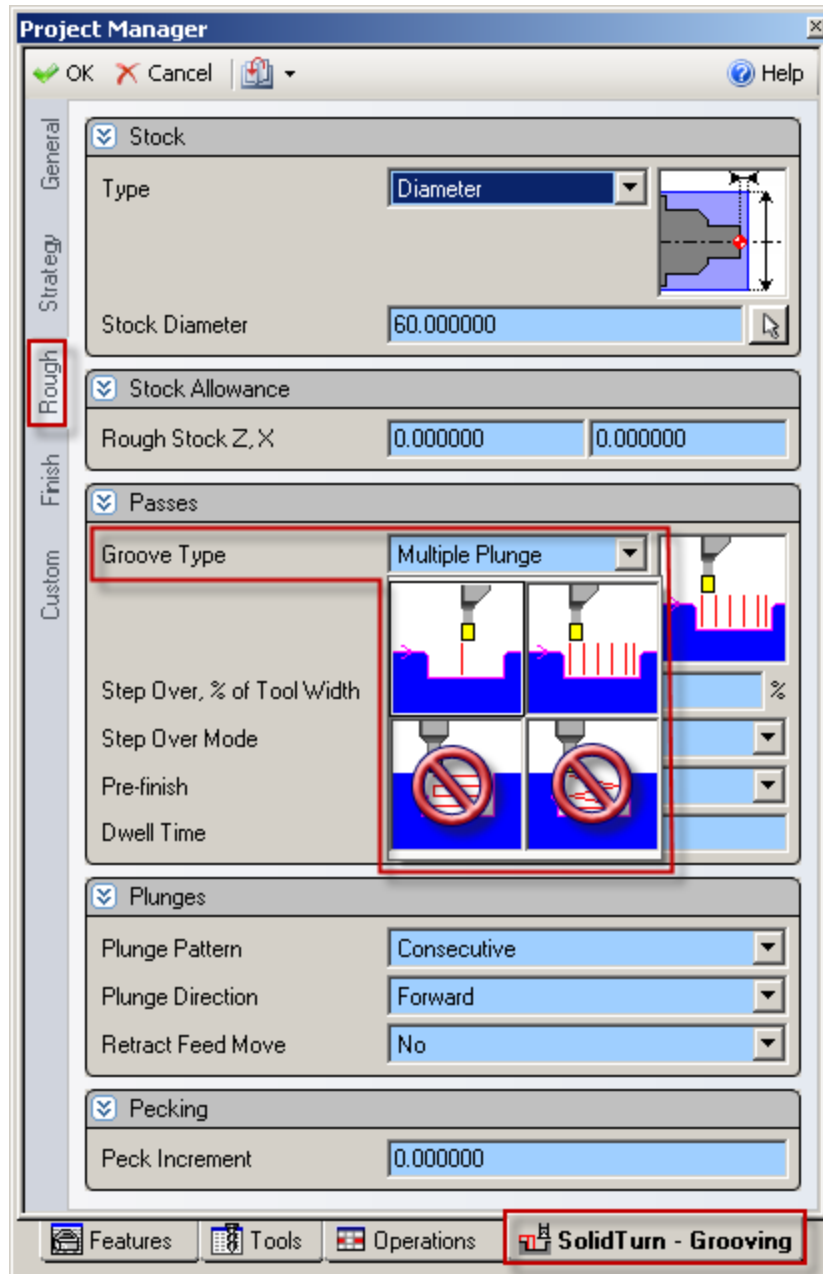
Information about some of the parameters to enter for the following cycles:

- The return amount **R(1)** will be output as entered in ESPRIT.
- **P**, in radius, and **Q** will be output as entered in ESPRIT.
- The feedrate **F** (and the cut speed **S**) can be specified on the General tab of the operation.
- 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

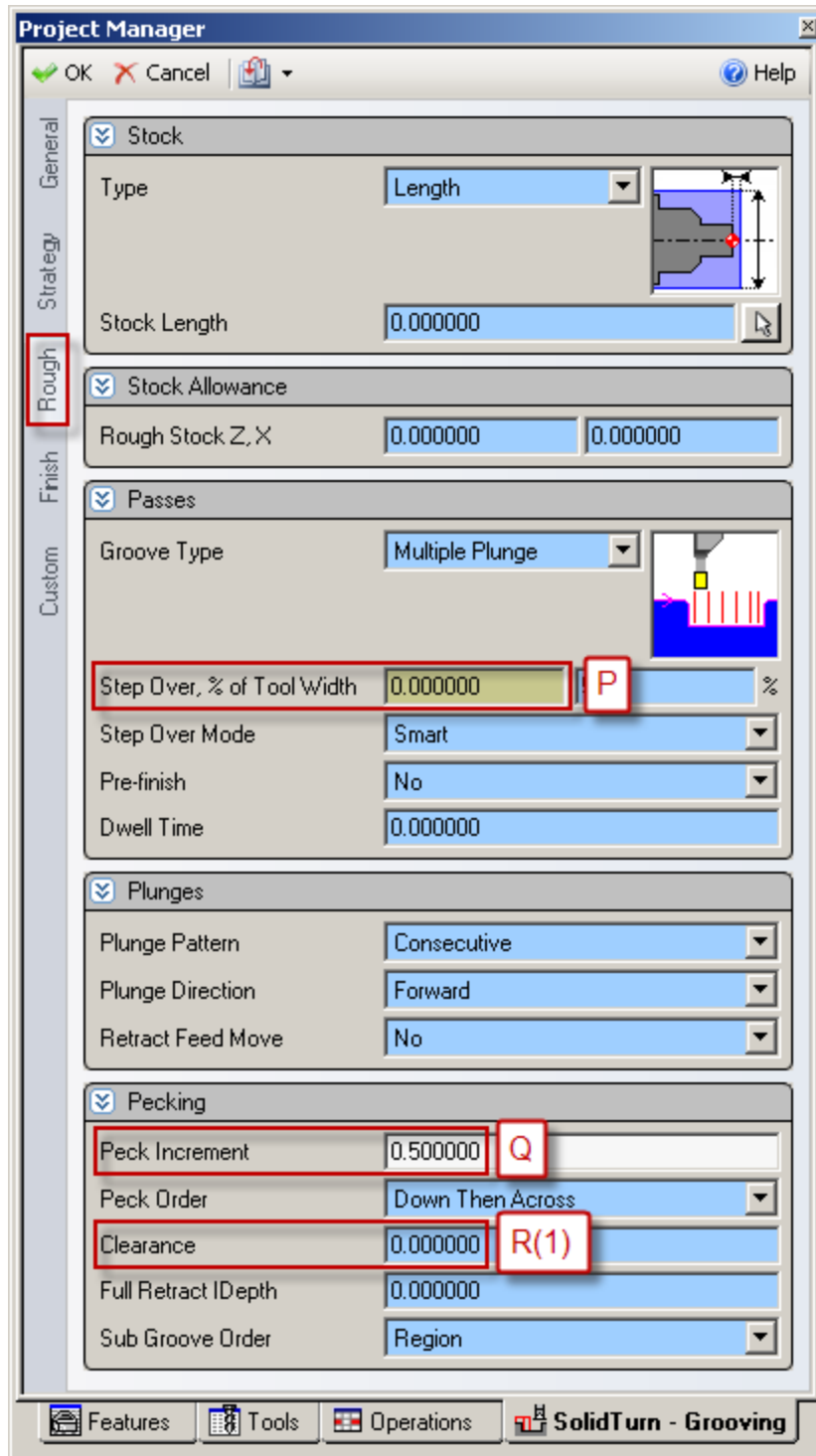
following error message in your NC code: **ERROR: SET GROOVE TYPE TO SINGLE PLUNGE OR MULTIPLE PLUNGE FOR GROOVE CANNED CYCLE.**



### 12.3.2. Face grooving

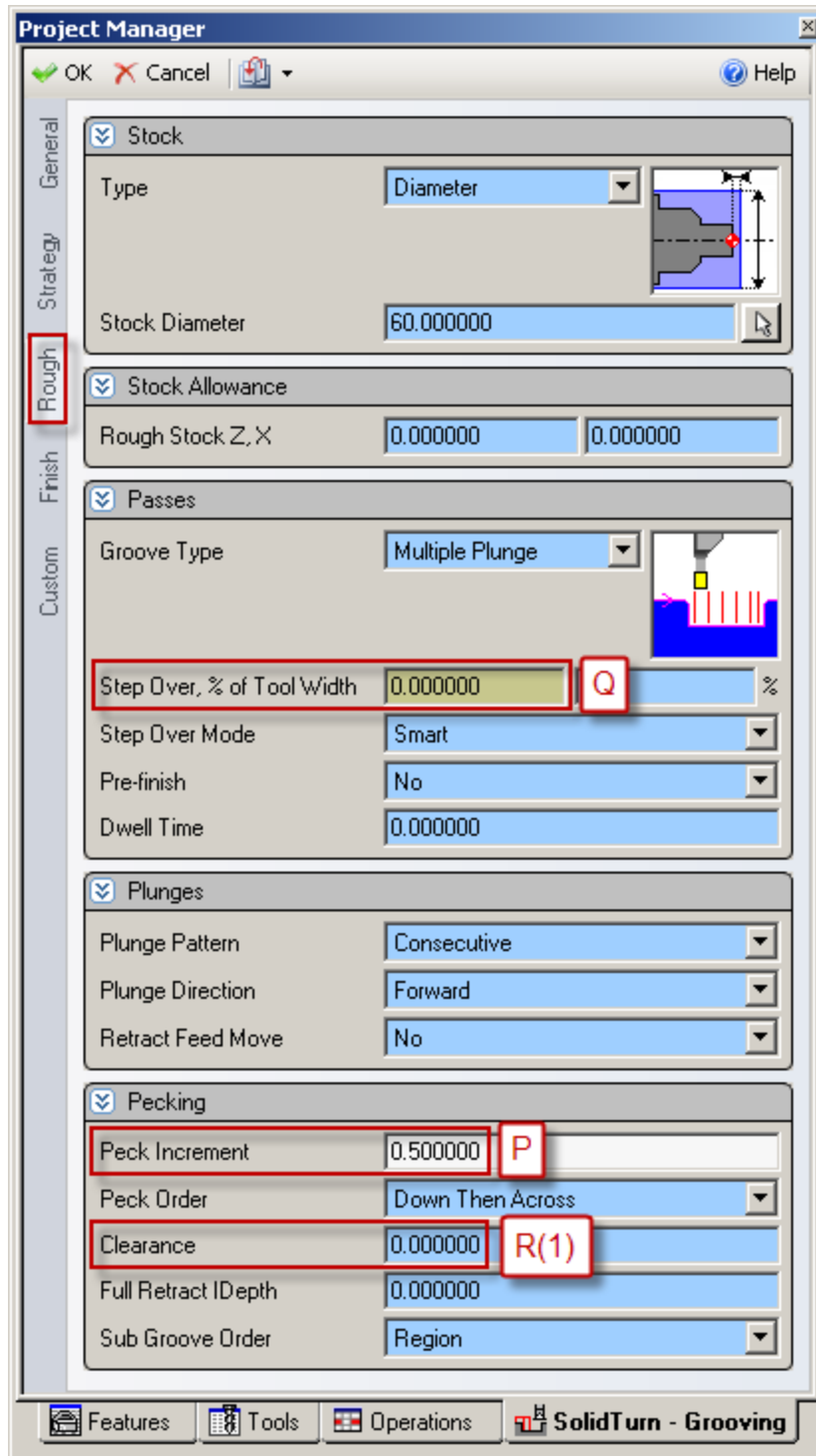
**G74 R(1);**  
**G74 X\_Z\_P\_Q\_F\_;**





### 12.3.3. O.D./I.D. grooving

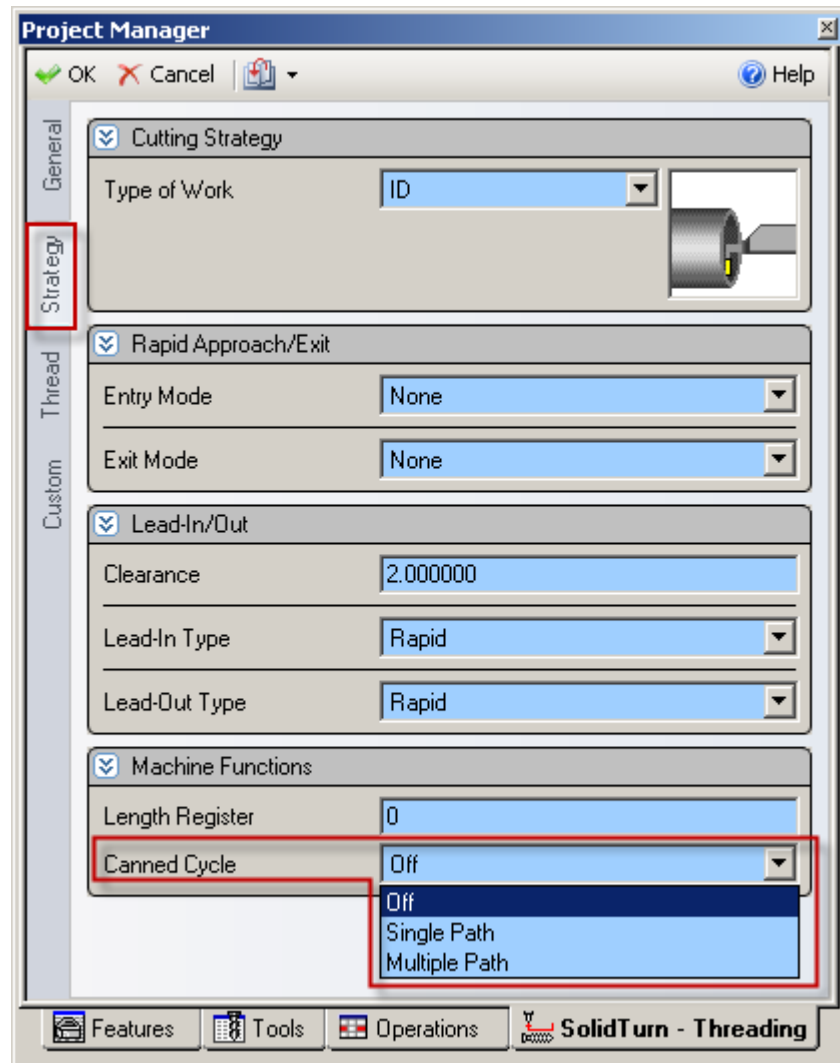
**G75 R(1);**  
**G75 X\_ Z\_ P\_ Q\_ F\_ ;**



## 12.4. Threading cycle

### 12.4.1. Introduction

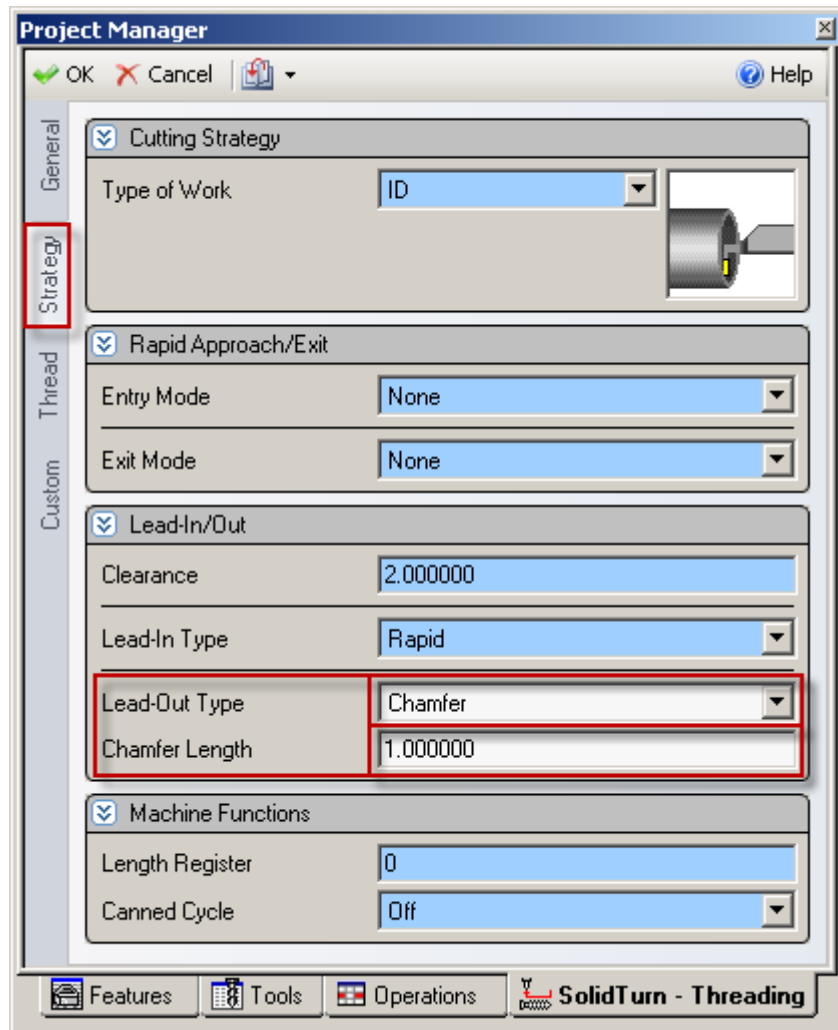
In ESPRIT, on the Strategy tab of the SolidTurn Threading operation page, set **Canned Cycle** to **Single Path** to trigger the G76 multiple repetitive cycle output. If you set this parameter to **Off**, the threading cycle will be output with G32 or G34 if Lead Variation is different from 0. When Canned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



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

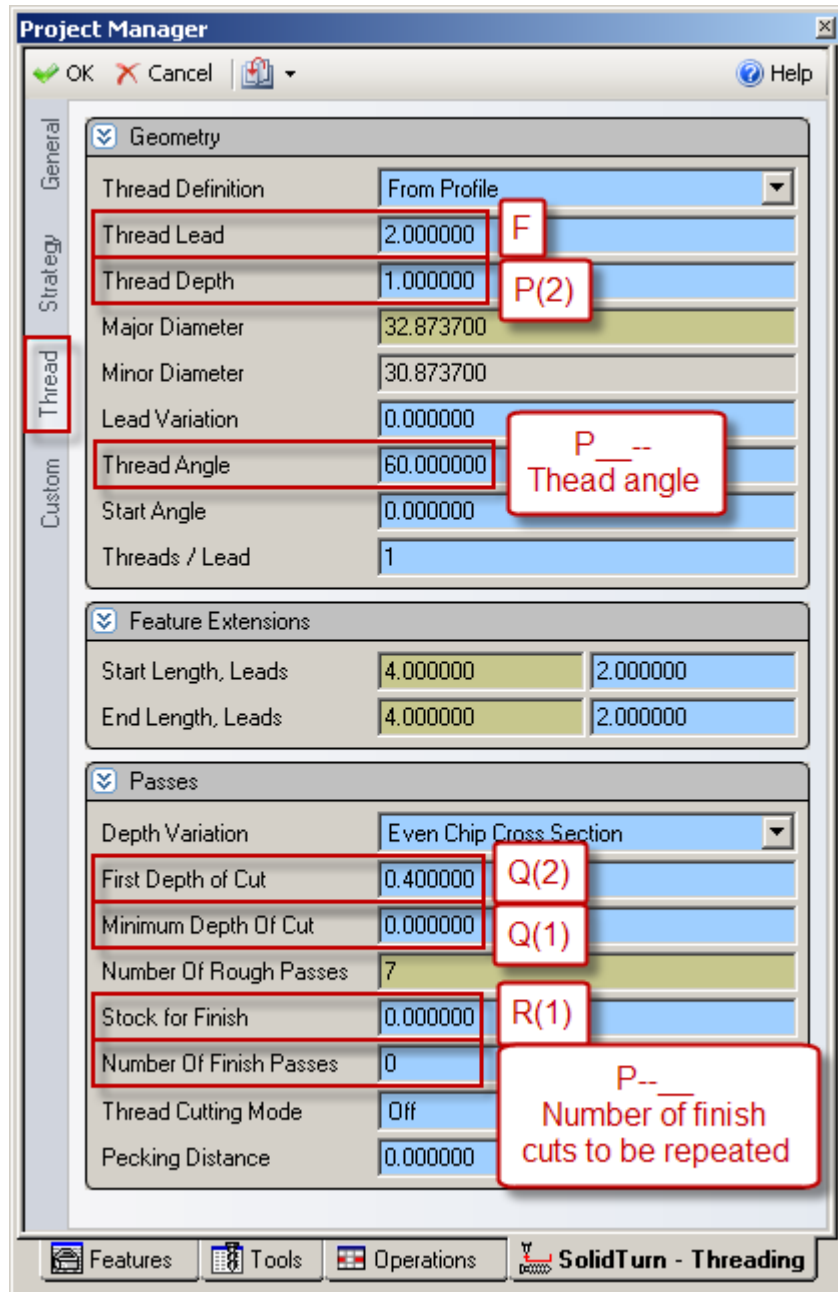
- The lead of the thread **F** will be output as entered in ESPRIT.
- The angle of shift of the thread cutting start angle **Q** will be output as entered in ESPRIT.
- The cut speed **S** can be specified on the General tab of the operation.

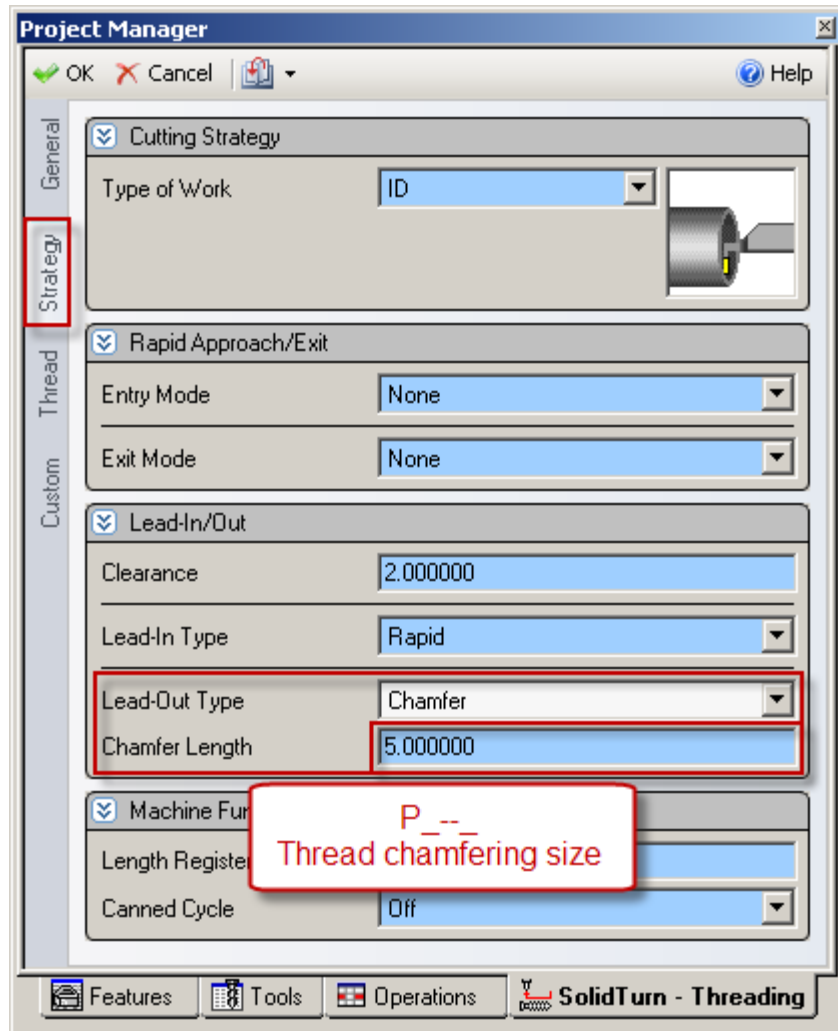
On the Strategy tab, if you set the Lead-Out Type to Chamfer and the Chamfer Length greater than 0, you will have **M23** (chamfer ON) output prior to the **G76** or **G92** commands. With a different lead-out set, the M-code output will be **M24** (chamfer OFF).



#### 12.4.2. Threading with canned cycle set to single path

**G76 P(1) Q(1) R(1) ;**  
**G76 X\_ Z\_ R(2) P(2) Q(2) F\_ ;**



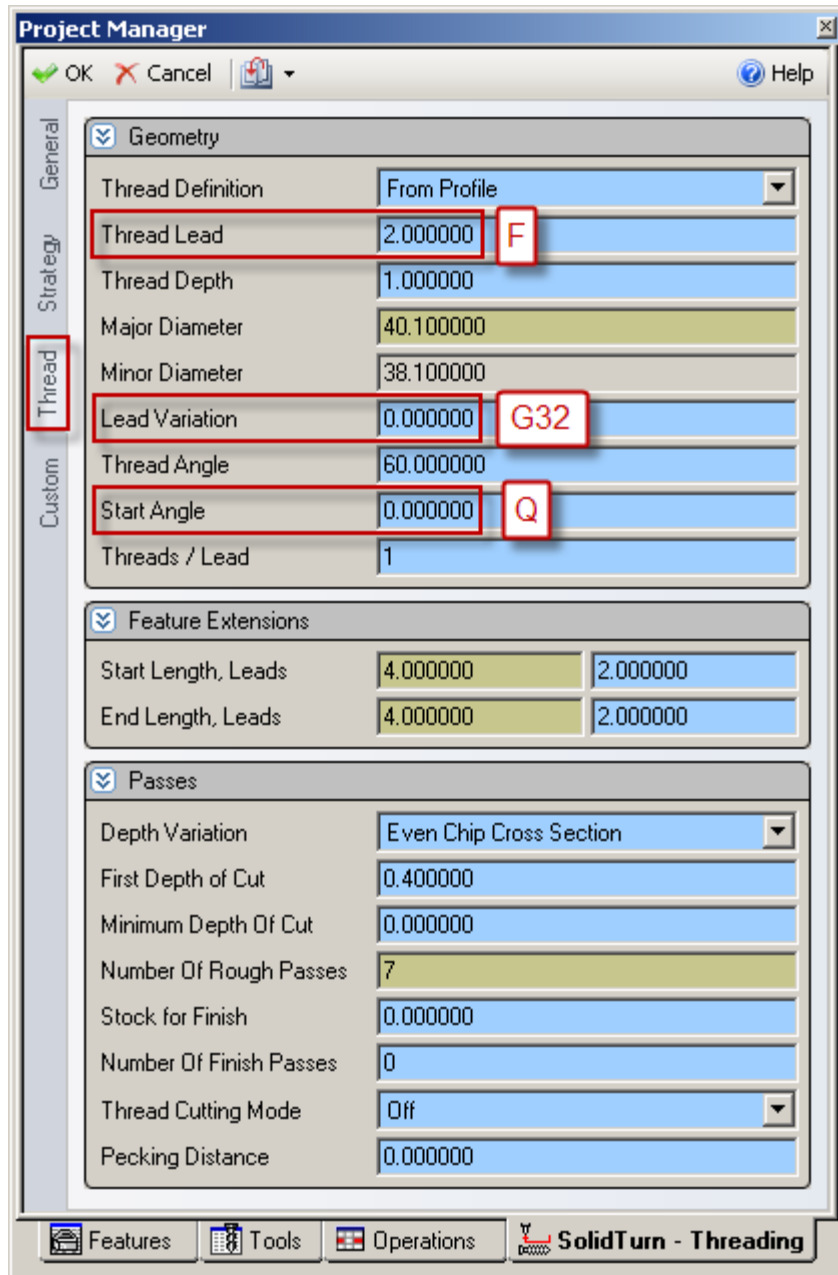


R(1), Q(1), P(2) and Q(2) will be output as entered in ESPRIT.

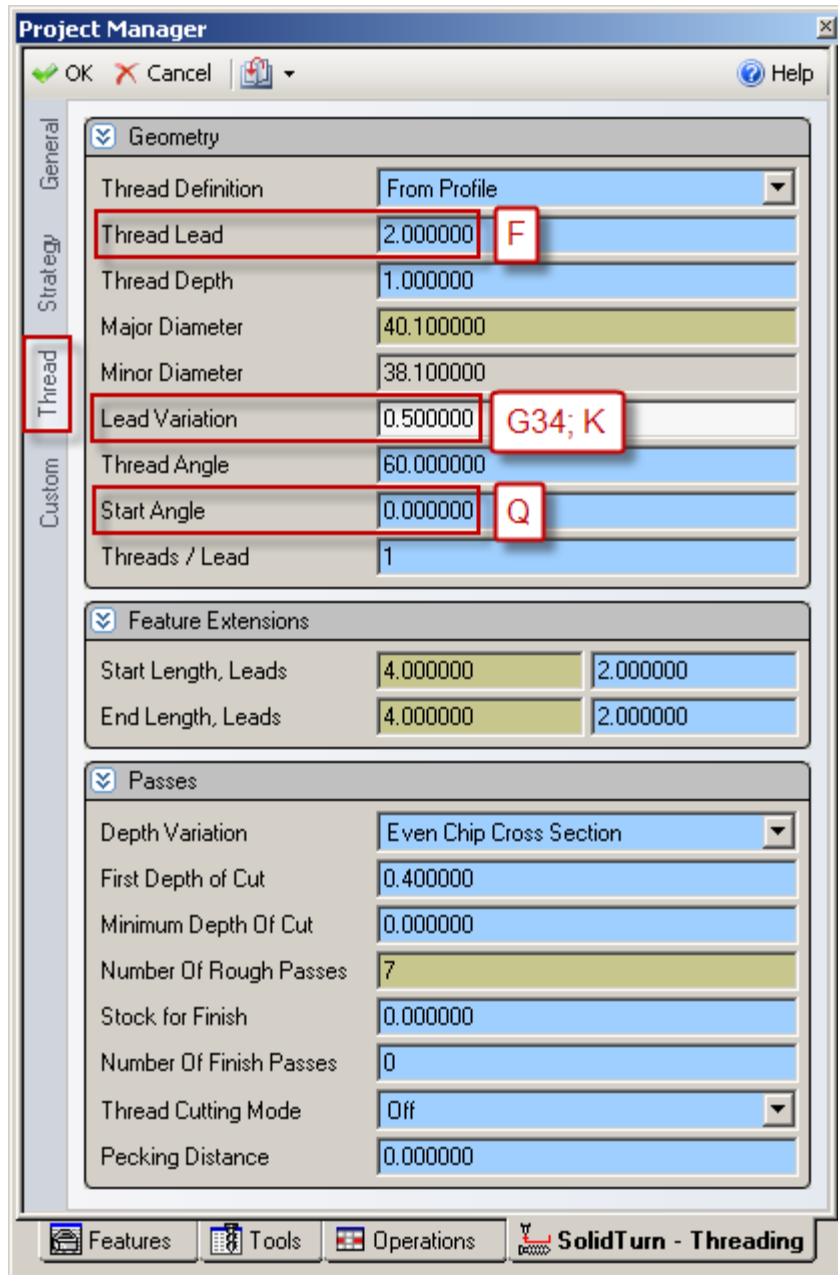
#### 12.4.3. Threading with canned cycle set to off

**Threading with canned cycle set to off, lead variation set to 0:**

**G32 X\_ Z\_ F\_ Q\_ ;**



**Threading with canned cycle set to off, lead variation different from 0:  
G34 X\_ Z\_ F\_ K\_ Q\_ ;**



**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.**

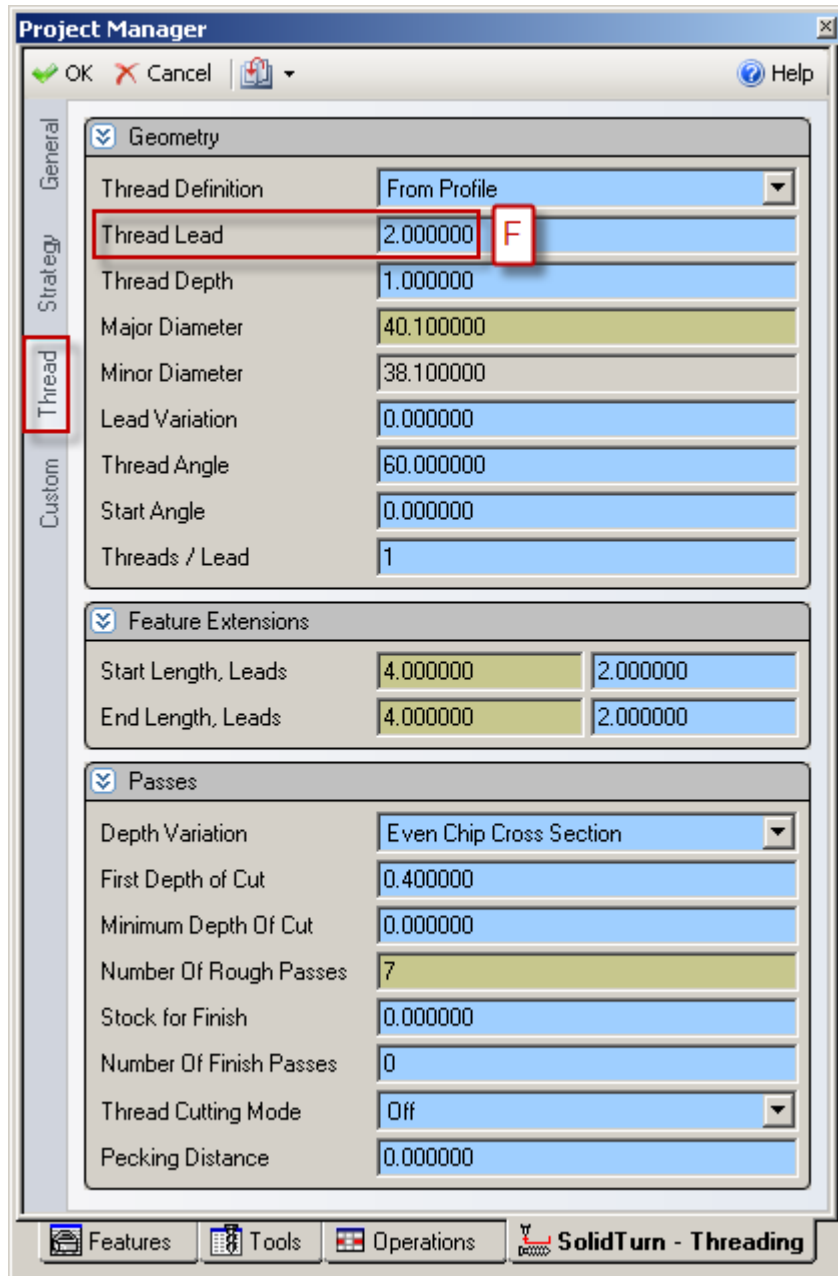


Lead-In/Out	
Clearance	2.000000
Lead-In Type	Rapid
Lead-Out Type	Rapid

**Straight thread cutting:**  
**G92 X\_Z\_F\_Q\_;**

Project Manager	
<input type="checkbox"/> OK <input type="checkbox"/> Cancel <input type="checkbox"/> Help	
<b>General</b>	
<b>Geometry</b>	
Thread Definition	From Profile
Thread Lead	2.000000 <b>F</b>
Thread Depth	1.000000
Major Diameter	40.100000
Minor Diameter	38.100000
Lead Variation	0.000000
Thread Angle	60.000000
Start Angle	0.000000 <b>Q</b>
Threads / Lead	1
<b>Strategy</b>	
<b>Thread</b>	
<b>Custom</b>	
<b>Feature Extensions</b>	
Start Length, Leads	4.000000 2.000000
End Length, Leads	4.000000 2.000000
<b>Passes</b>	
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 <b>SolidTurn - Threading</b>	

**Tapered thread cutting:**  
**G92 X\_ Z\_ R\_ F\_ ;**



## 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:

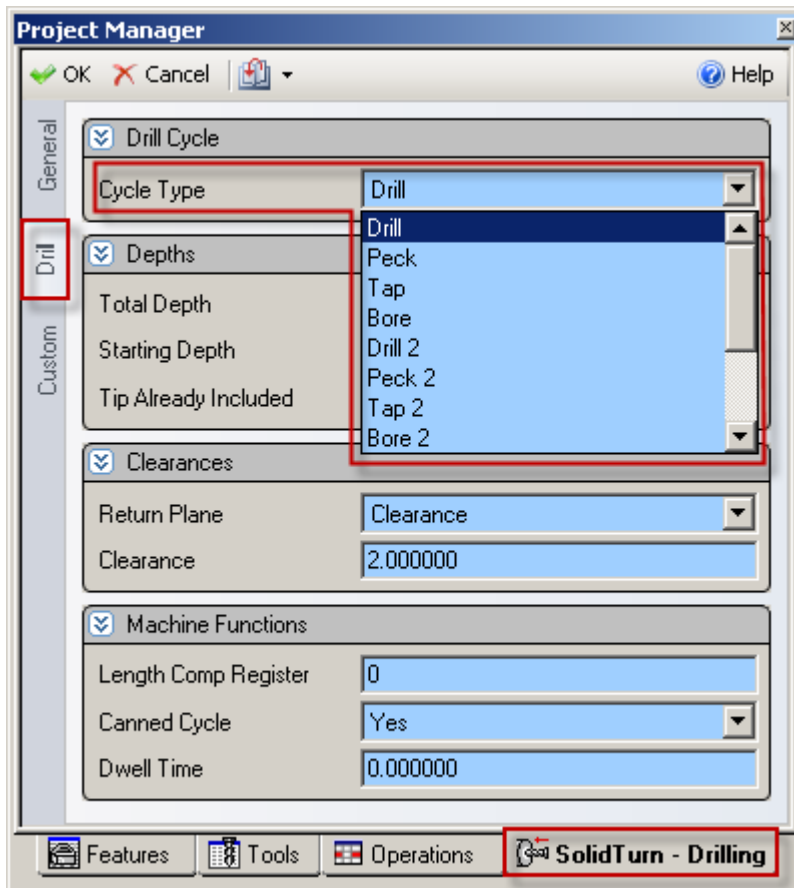
The Face Hole Machining Canned Cycle:

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

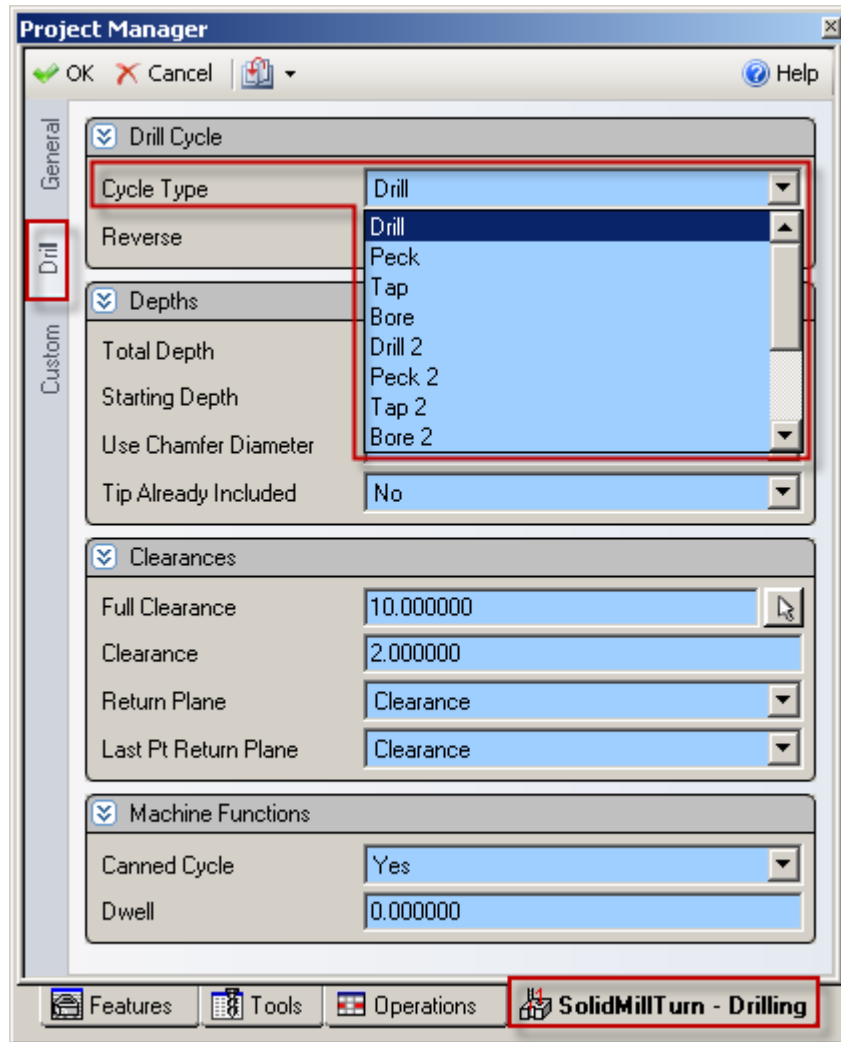
The Side Hole Machining Canned Cycle:

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

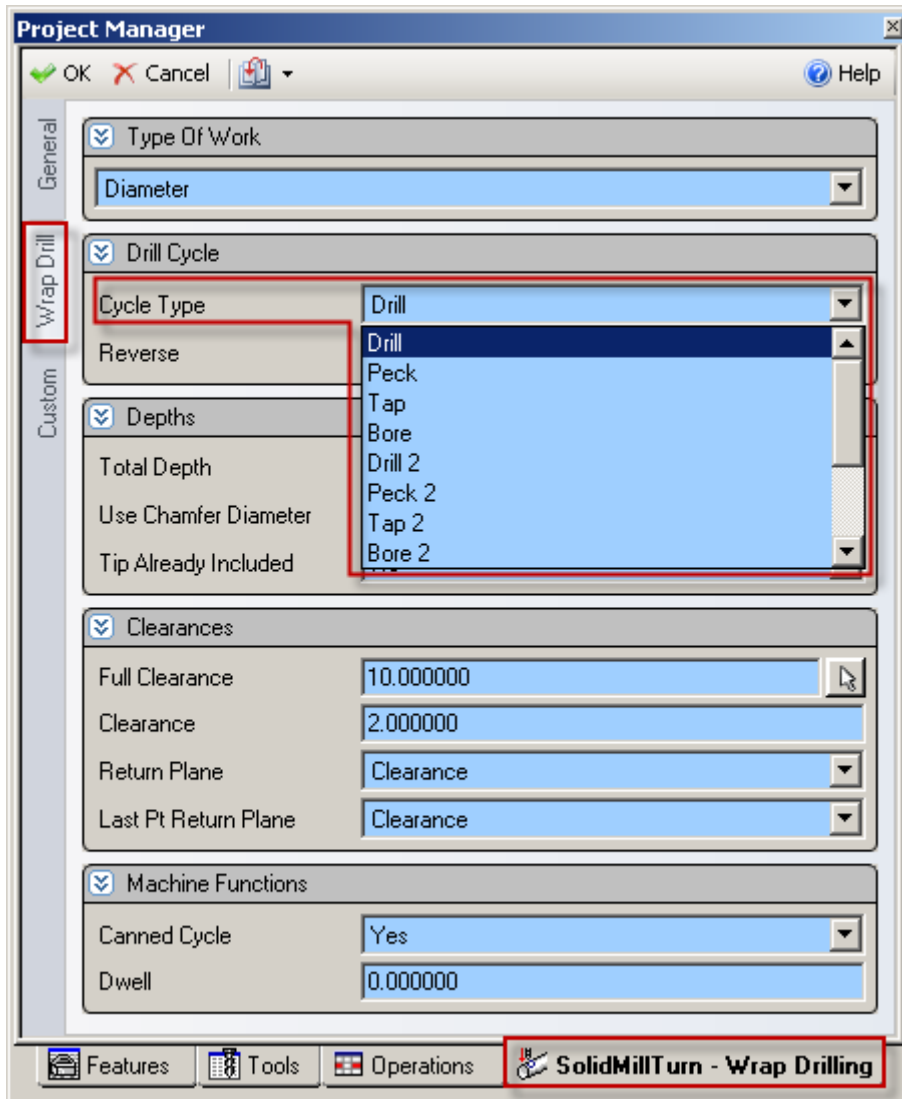
In ESPRIT, you can choose between the different drilling canned cycles using the Cycle Type pull-down on the Drill tab for the SolidTurn Drilling cycle:



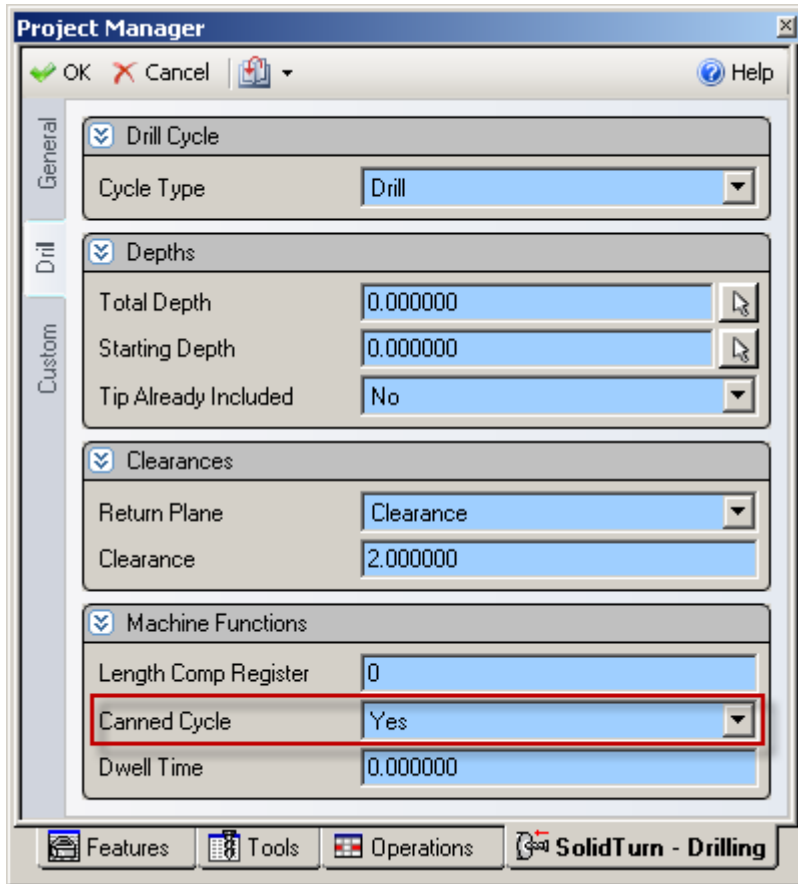
On the Drill tab for the SolidMillTurn Drilling cycle:



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



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



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:

- For G83 / G87, the depth of cut per one infeed motion **Q** is an unsigned value, in radius for side drilling. Specify **Q** in mm, it will then be output in units of 0.001 mm without decimal point. For example, if you enter 2 for **Q** in ESPRIT, the output will be in the NC code Q2000.
- For G74, the depth of cut per one infeed motion **Q** will be output as entered in ESPRIT.
- Enter the dwell **P** in s, it will then be output in units of 0.001s without decimal point. For example, if you enter 1.5 for **P** in ESPRIT, the output will be in the NC code P1500.
- The feedrate **F** (and the cut speed **S**) can be specified on the General tab of the operation.

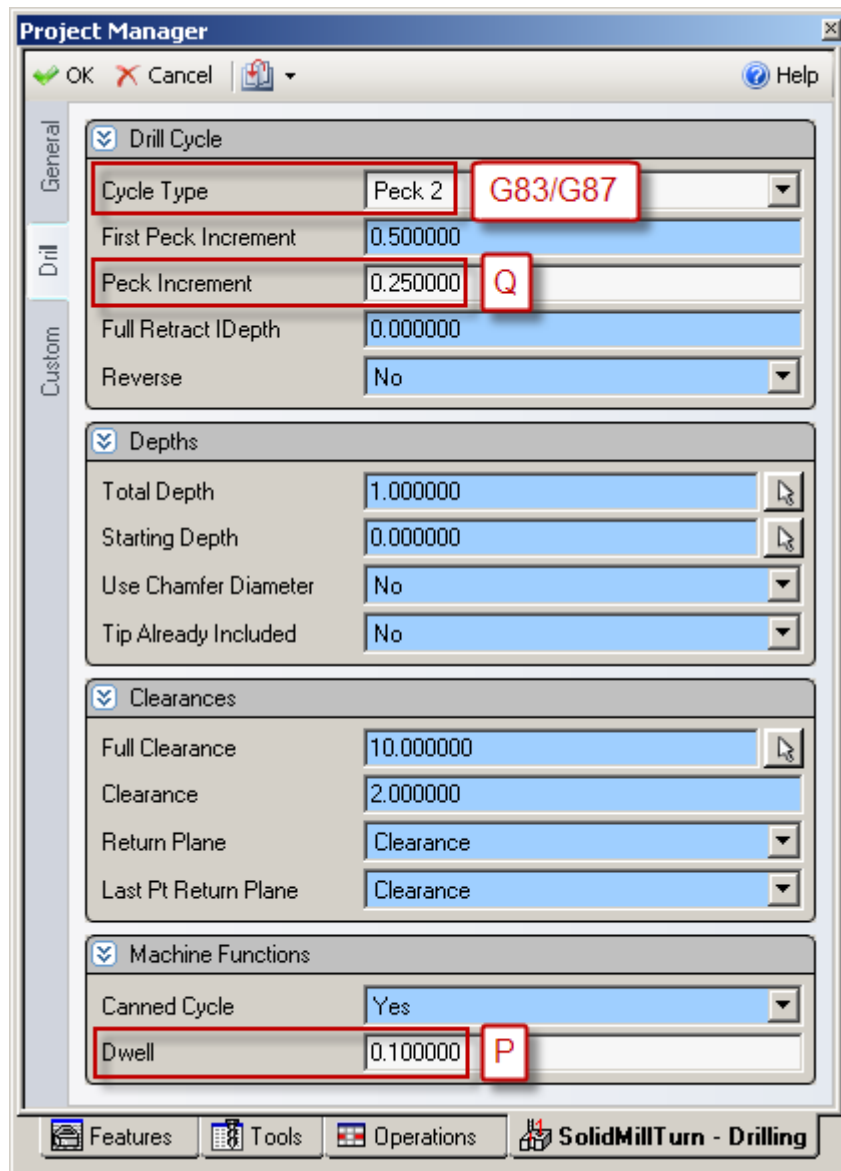
## 13.2. Face and side high-speed deep hole drilling / Face and side deep hole drilling / Deep hole drilling in turning mode / Deep hole drilling with G74

### 13.2.1. Face and side high-speed deep hole drilling (G83/G87) (Parameter #8115 = 1)

**CAUTION !!!** Use **SolidMillTurn Drilling** or **SolidMillTurn Wrap Drilling**

G83 X\_ (Y\_) C\_ Z\_ R\_ Q\_ P\_ F\_ ;

G87 Z\_ (Y\_) C\_ X\_ R\_ Q\_ P\_ F\_ ;

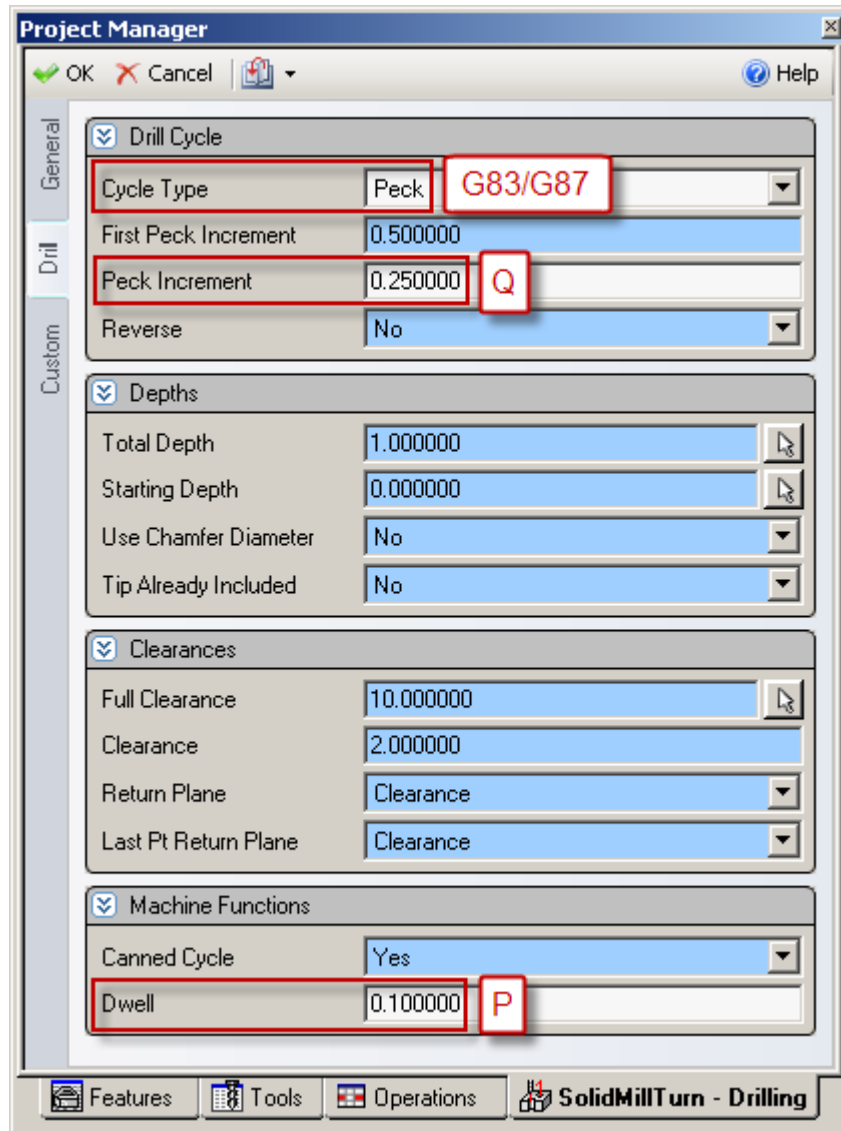


### 13.2.2. Face and side deep hole drilling (G83/G87) (Parameter #8115 = 0)

**CAUTION !!!** Use **SolidMillTurn Drilling** or **SolidMillTurn Wrap Drilling**

**G83 X\_ (Y\_) C\_ Z\_ R\_ Q\_ P\_ F\_ ;**

**G87 Z\_ (Y\_) C\_ X\_ R\_ Q\_ P\_ F\_ ;**

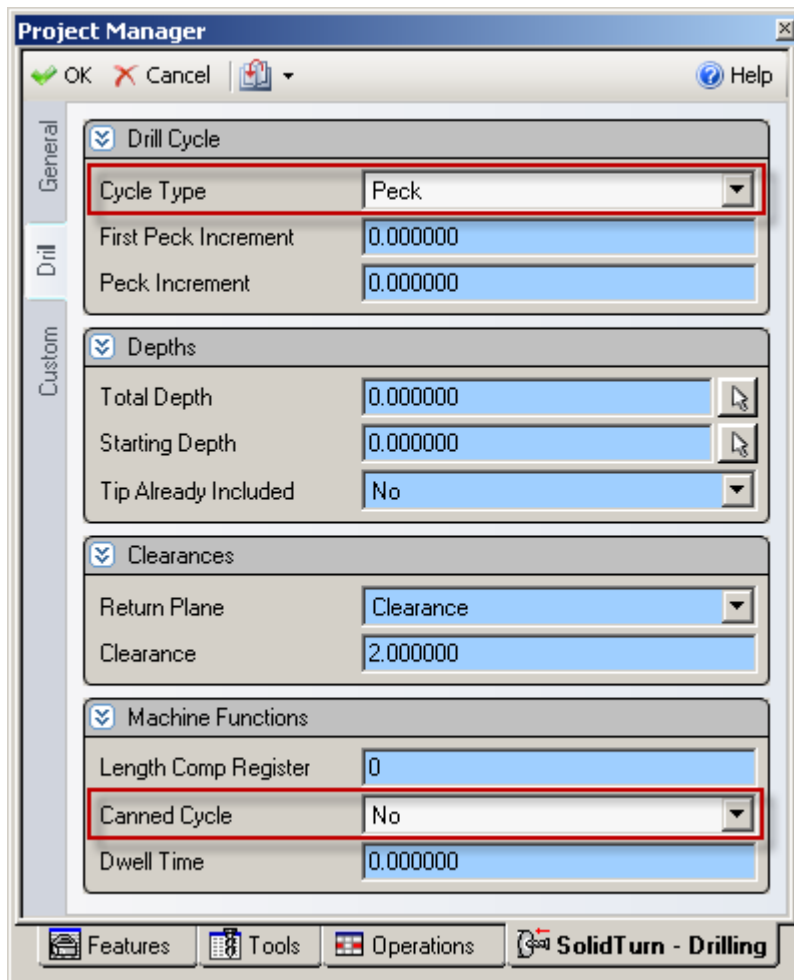


### 13.2.3. Deep hole drilling in turning mode

**CAUTION !!!** Use **SolidTurn Drilling**

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





*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 CYCLE.**

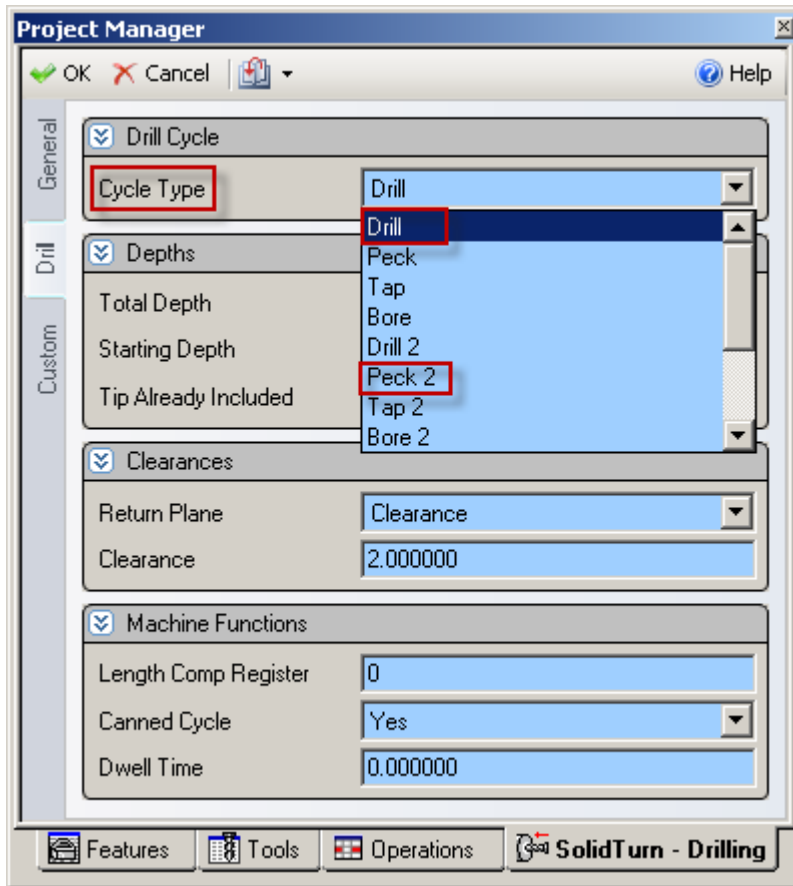
#### 13.2.4. Deep hole drilling with G74

**CAUTION !!!** Use **SolidTurn Drilling**

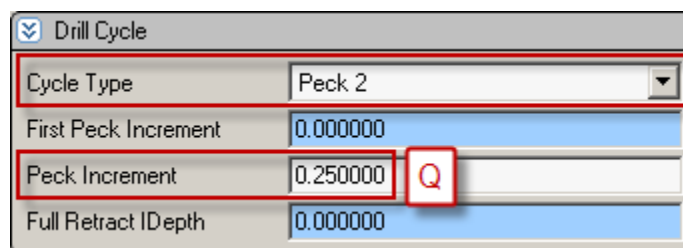
**G74 R(1)**

**G74 Z\_ Q\_ F\_ ;**

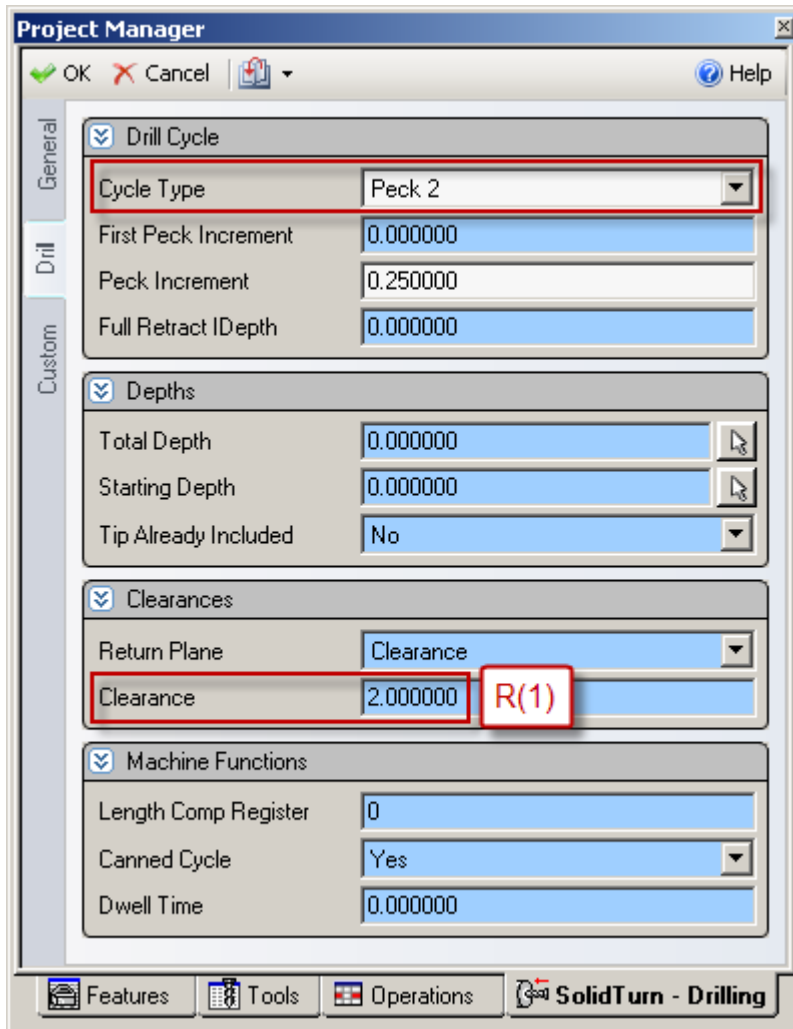
On the SolidTurn Drilling cycle, set the Cycle Type to Drill or Peck 2 to output your drilling cycle with G74.



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



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.

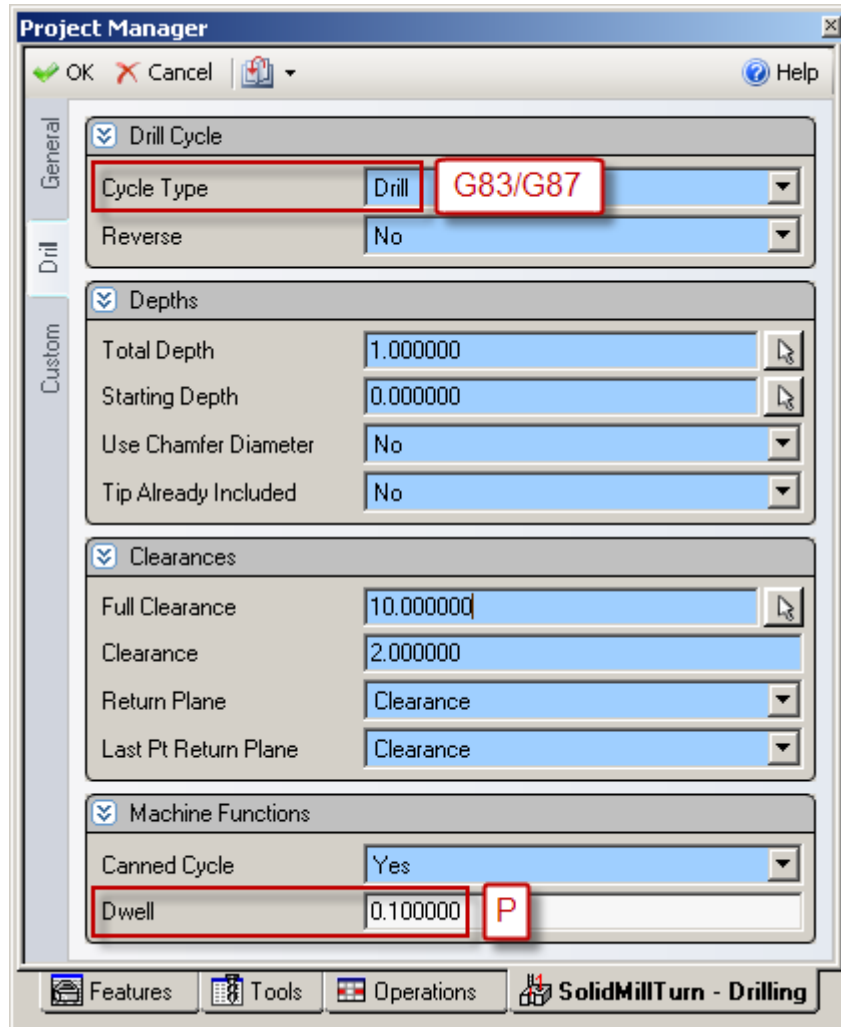


### 13.3. Face and side spot drilling (G83/G87)

**CAUTION !!!** Use SolidMillTurn Drilling or SolidMillTurn Wrap Drilling

G83 X\_ (Y\_) C\_ Z\_ R\_ P\_ F\_ ;

G87 Z\_ (Y\_) C\_ X\_ R\_ P\_ F\_ ;

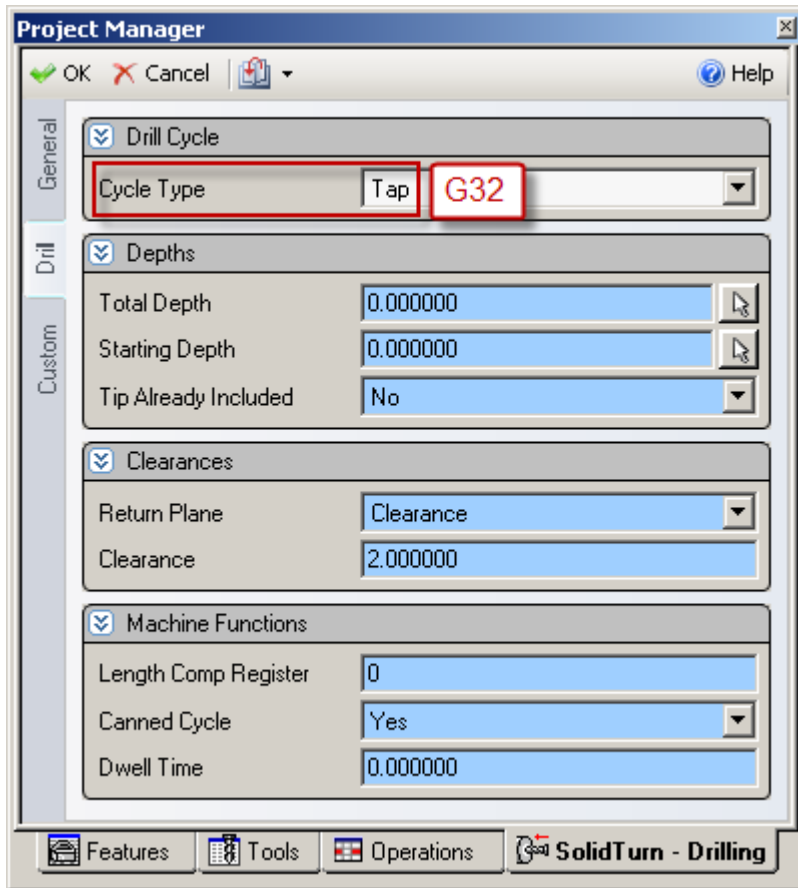


### 13.4. Tapping at center of spindle (G32)

**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.



*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 synchronized reverse tapping***

### ***13.5.1. Restriction of synchronized tapping***

**CAUTION !!!** During the synchronized tapping cycle, the maximum speeds for the rotary tool is  $3000 \text{ min}^{-1}$ .

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.

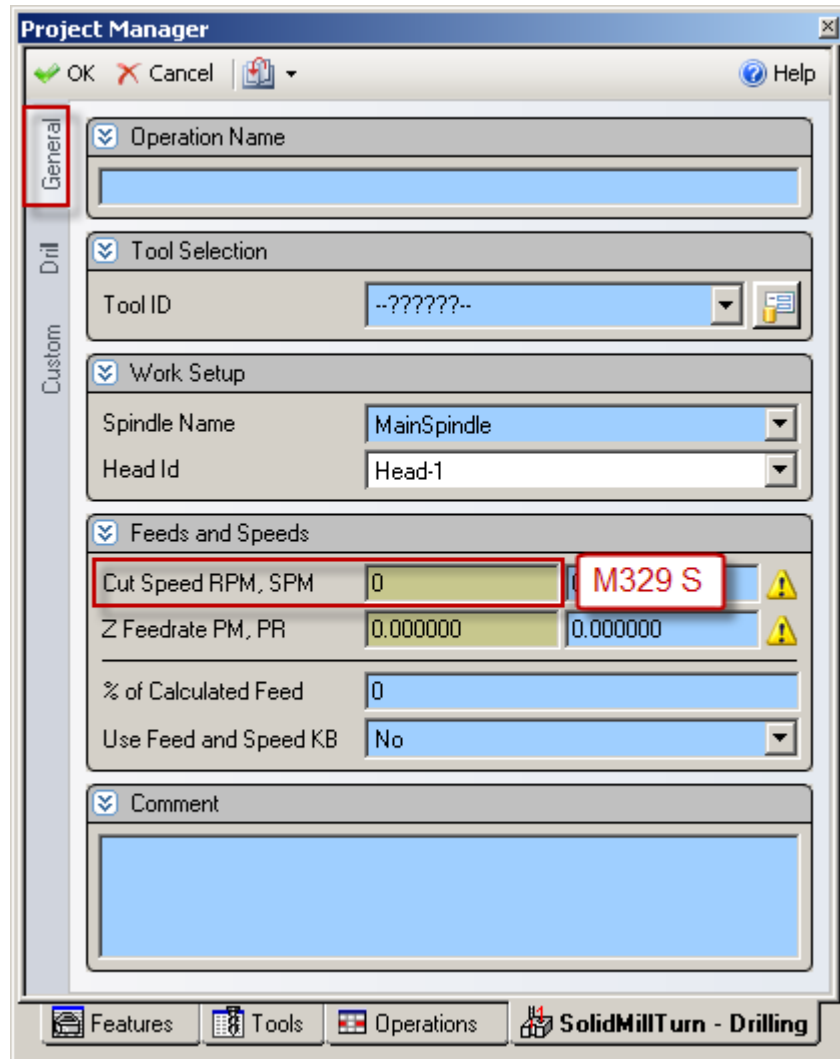
*Note:* Synchronized tapping can only be used with milling drilling operations. On the SolidTurn Drilling operation, if you set Cycle Type to:

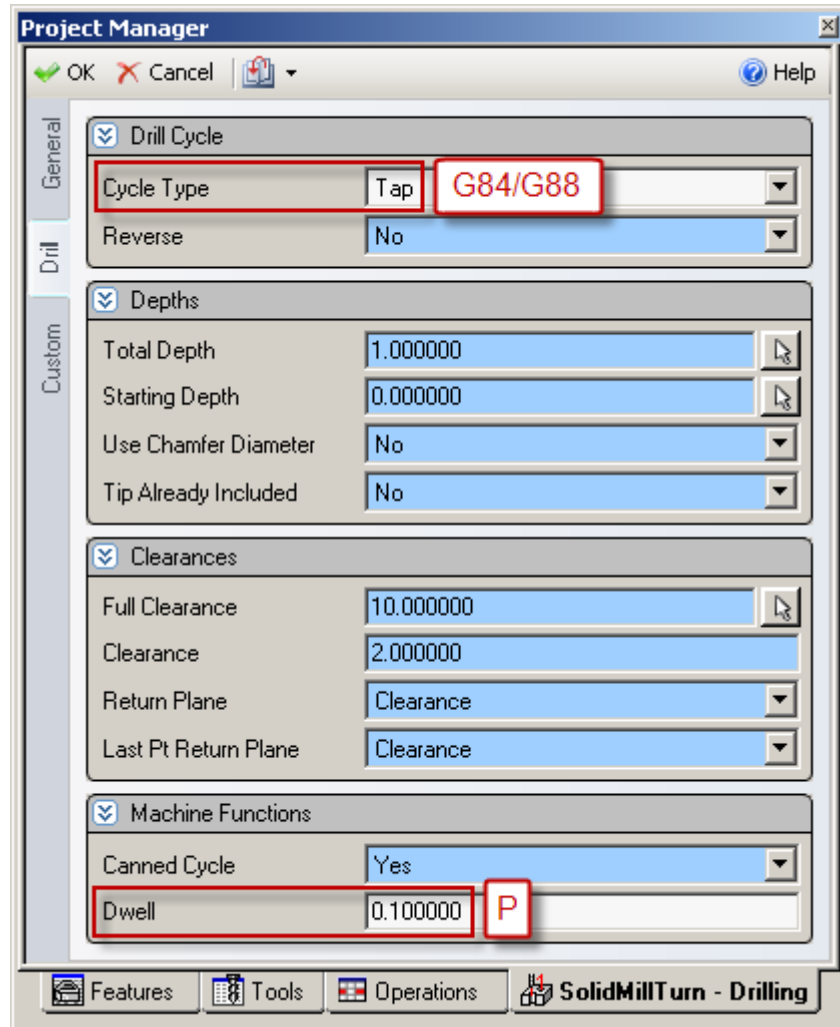
- Tap, the code will be output with G32 (tapping at center of spindle).
- Tap 2, you will get the following error message in your NC code: **ERROR: CANNED CYCLE NOT SUPPORTED FOR LATHE CYCLE.**

### 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\_ ;

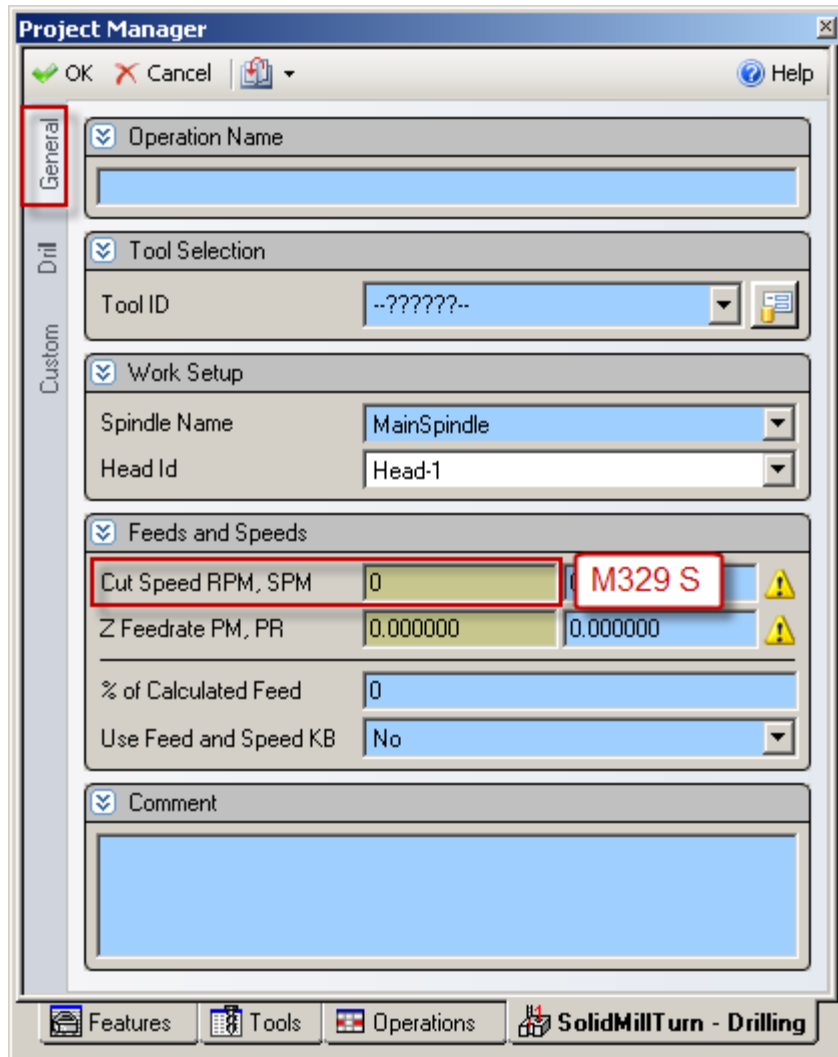




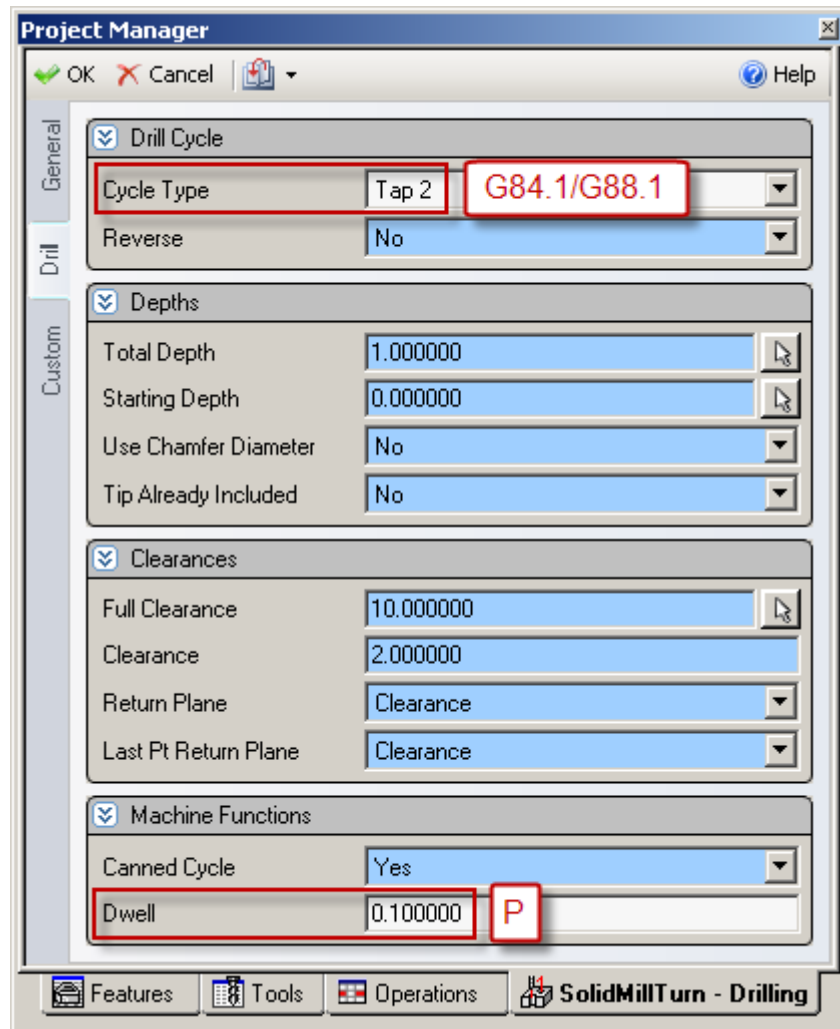
### 13.5.3. Face and side synchronized reverse tapping (M329 G84.1/M329 G88.1)

**CAUTION !!!** Use **SolidMillTurn Drilling** or **SolidMillTurn Wrap Drilling**

M329 S\_ ;  
 G84.1 X\_ (Y\_) C\_ Z\_ R\_ P\_ F\_ ;  
 M329 S\_ ;  
 G88.1 Z\_ (Y\_) C\_ X\_ R\_ P\_ F\_ ;







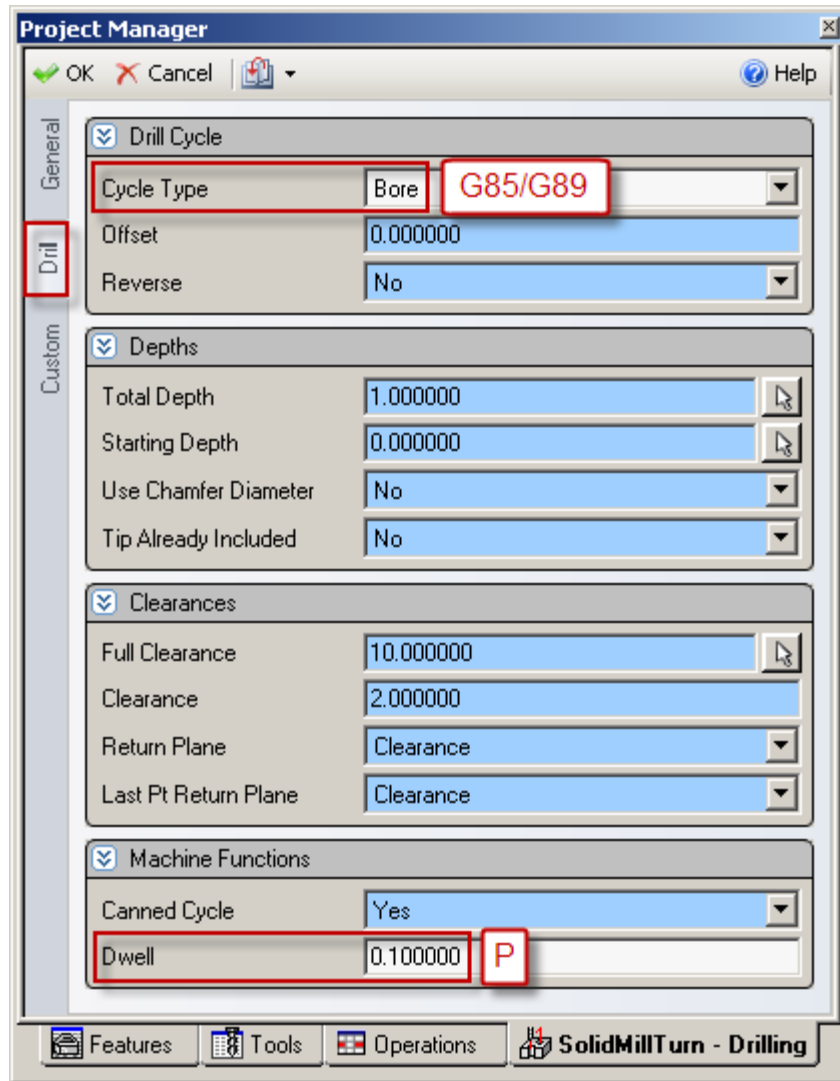
## 13.6. Face and side boring / Boring in turning mode

### 13.6.1. Face and side boring (G85/G89)

**CAUTION !!!** Use **SolidMillTurn Drilling** or **SolidMillTurn Wrap Drilling**

**G85 X\_ (Y\_) C\_ Z\_ R\_ P\_ F\_ ;**

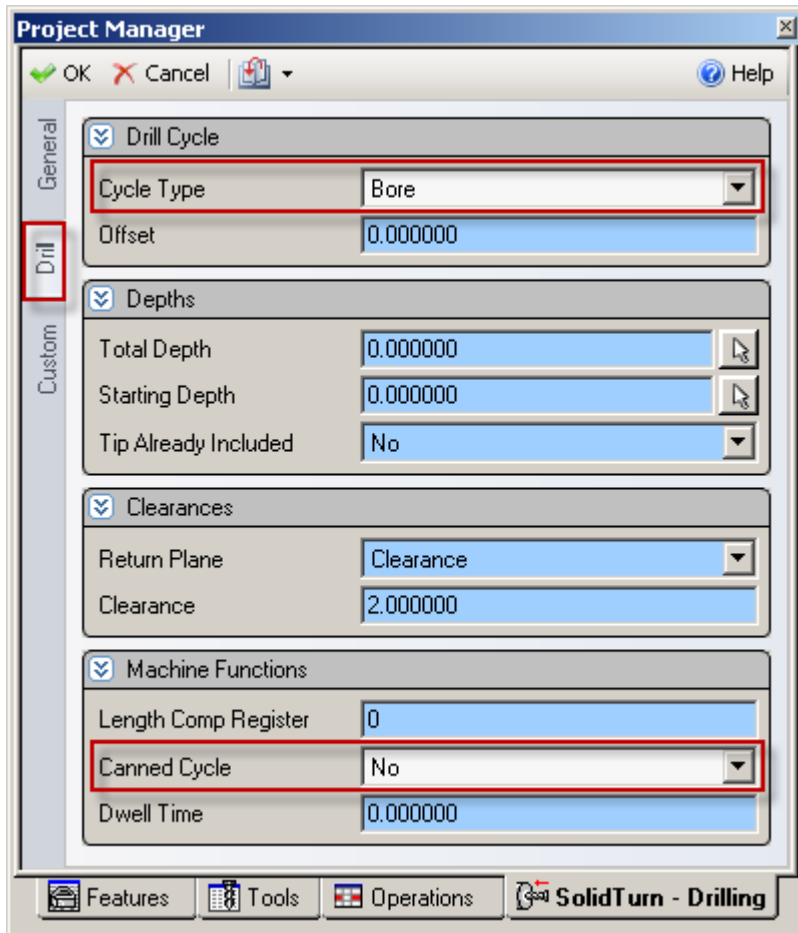
**G89 Z\_ (Y\_) C\_ X\_ R\_ P\_ F\_ ;**



### 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.



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 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 NLX configuration) commands is 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.

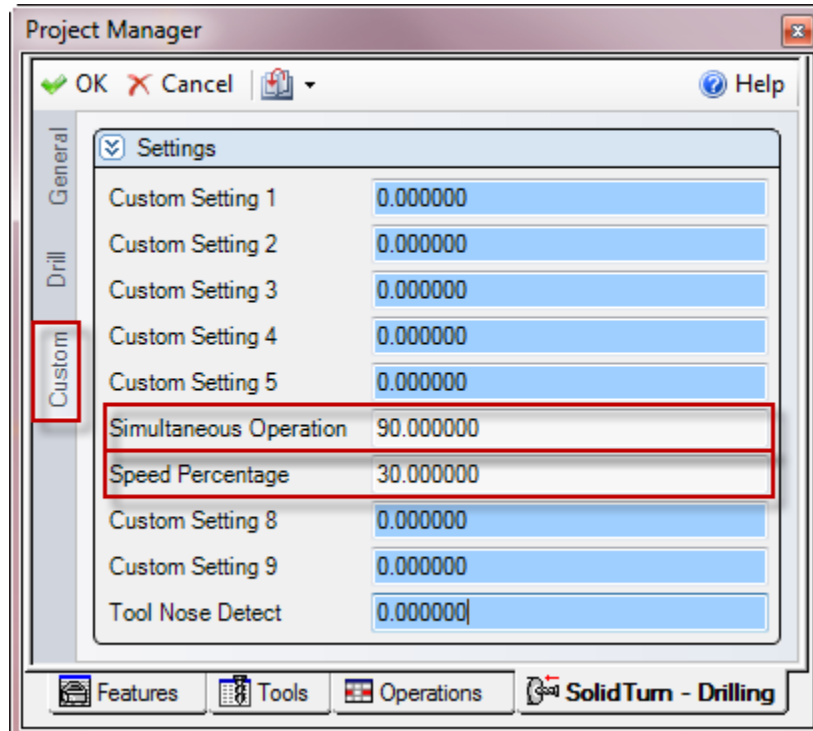
On the Custom tab of the SolidTurn Drilling operation, set Simultaneous Operation (Custom Setting 6) to 90 to trigger the output of the M90 command.

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.

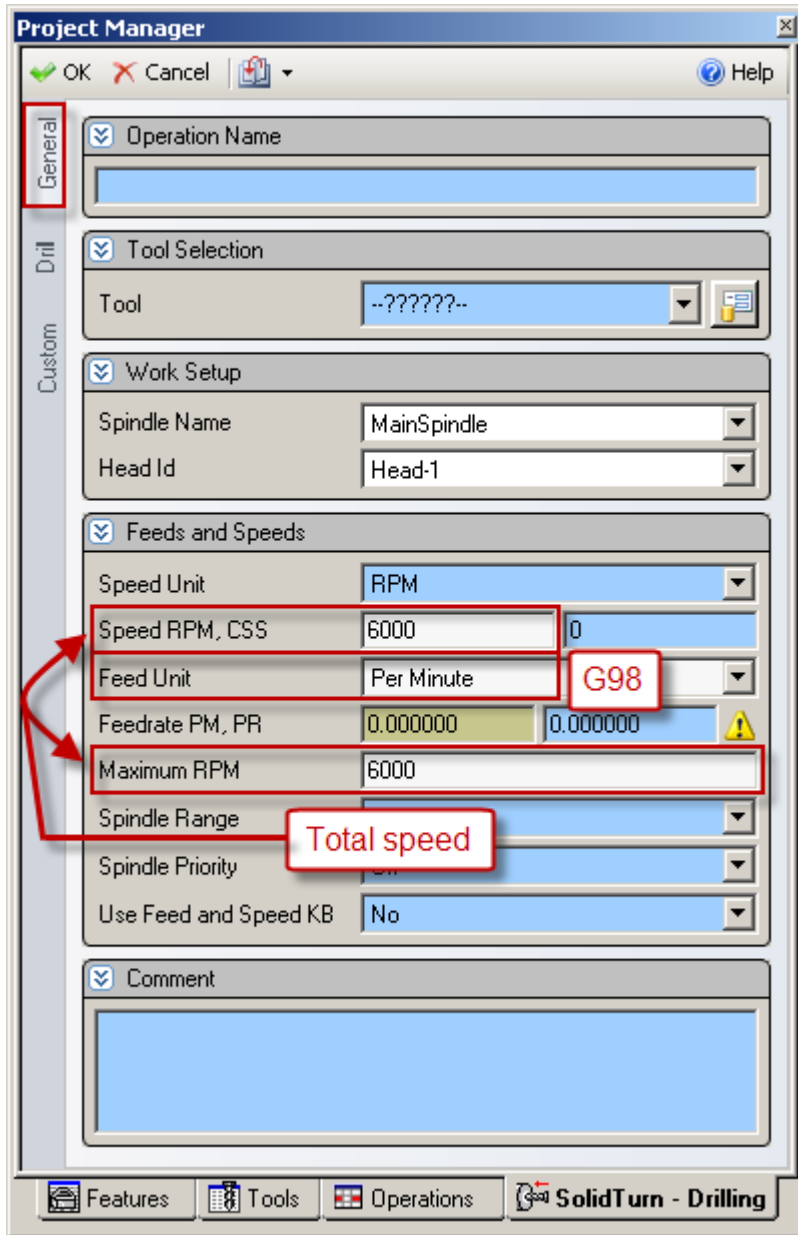
*Example:*

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.



*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  $6000 \text{ min}^{-1}$ , which is the sum of the turning spindle speed ( $4200 \text{ min}^{-1}$ ) and the rotary tool speed ( $1800 \text{ min}^{-1}$ ).
- 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.



## 14. 4-axis wrap milling cycles with interpolation

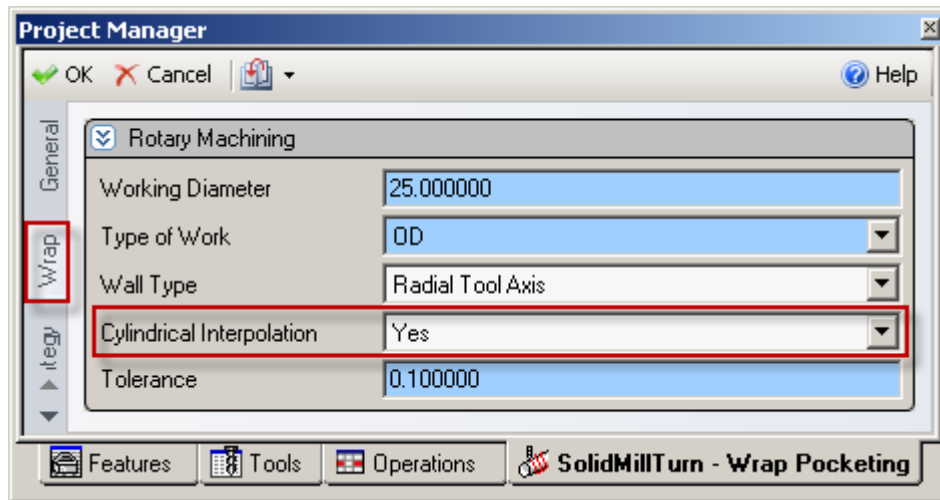
### 14.1. Introduction

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

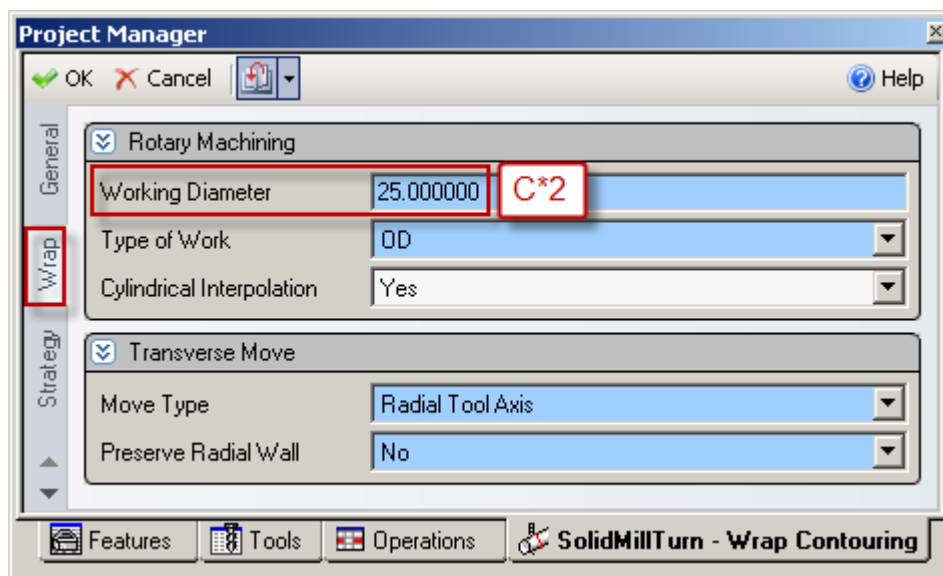
### 14.2. Cylindrical interpolation

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

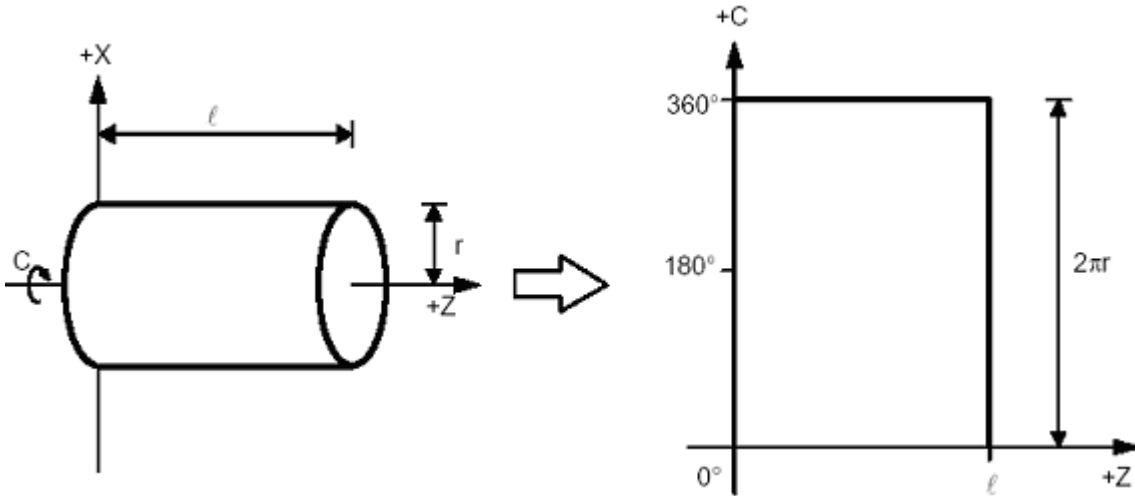
On the Wrap tab of the operation pages, set Cylindrical Interpolation to Yes to output 4-axis wrap milling operations with cylindrical interpolation.



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:



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



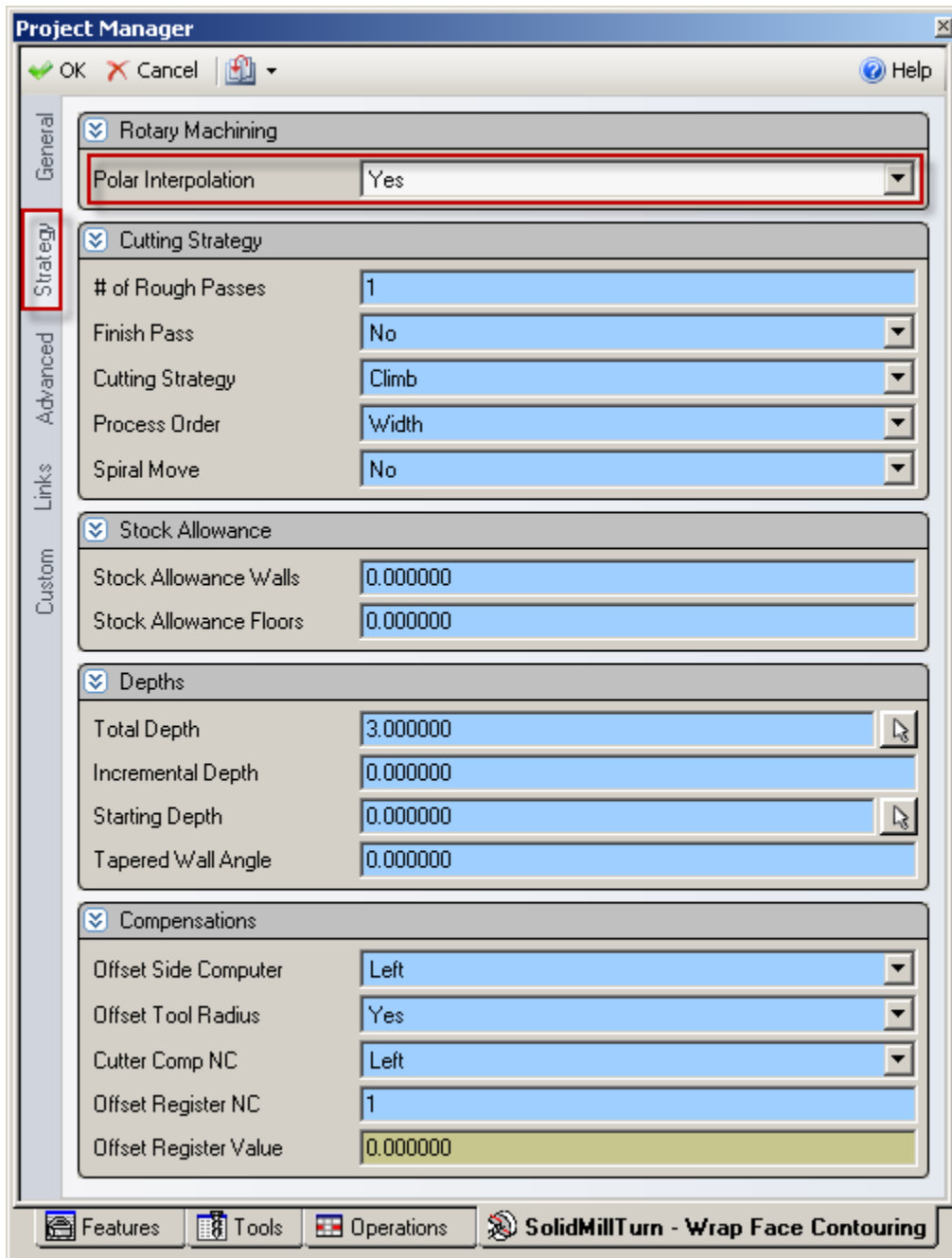
Using cylindrical interpolation can shorten the program length because ESPRIT outputs the circular motions in the cylindrical plane.

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

### ***14.3. Polar coordinate interpolation (Notching)***

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

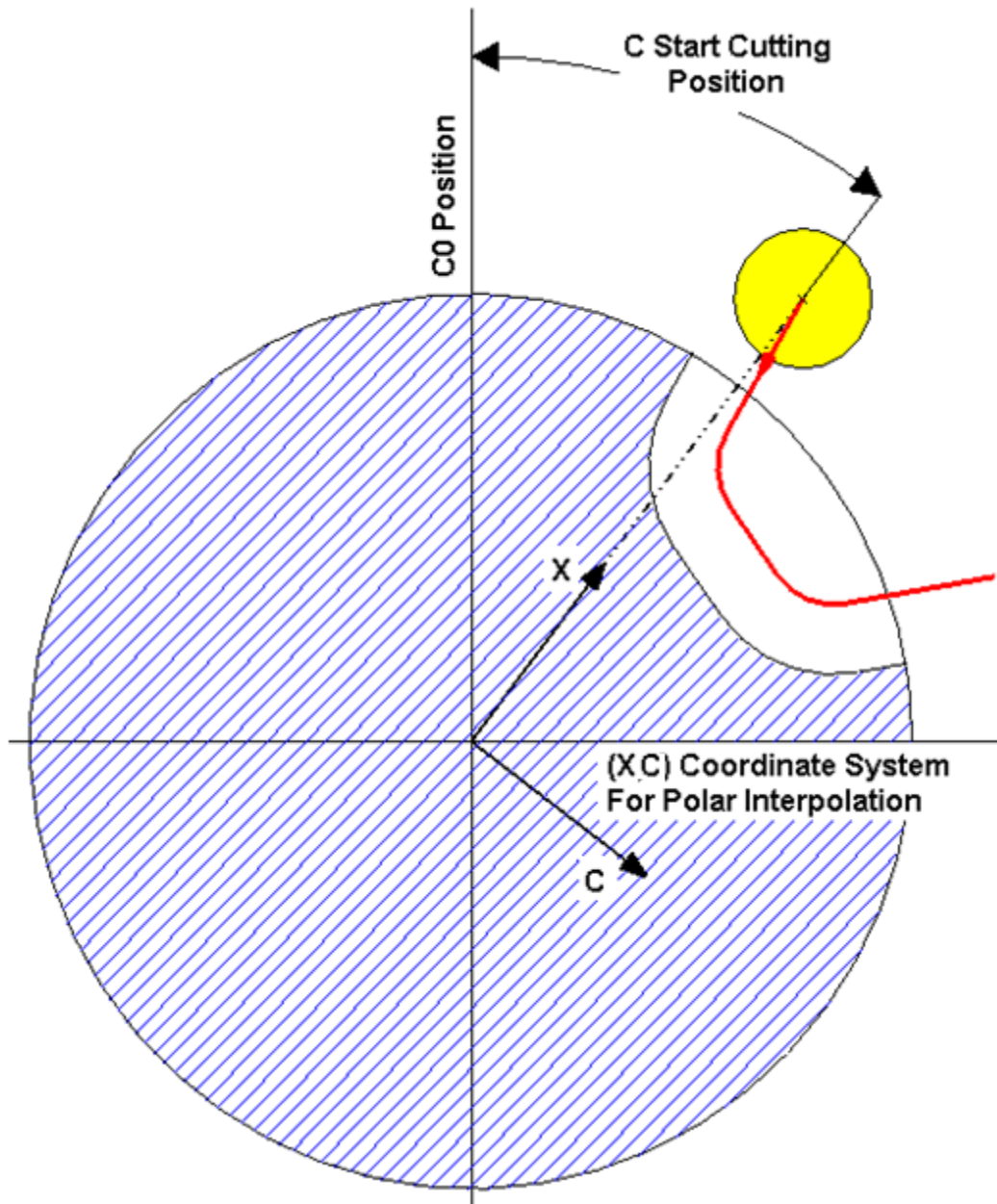
On the Strategy tab of the operation pages, set Polar Interpolation to Yes to output rotary face milling operations with polar interpolation.



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.





## 15. Other Functionalities

### 15.1. Tailstock

#### 15.1.1. Introduction

When machining long work pieces with the Mori Seiki NZX & NLX machines, it is sometimes beneficial to use a tailstock to support the workpiece while you are machining.

*Note:*

For NZX configuration, based on the type of tailstock and machine, you will have to program your tailstock on head 1 or on head 2 (see part **4.4.4. Type of tailstock installed on the machine**):

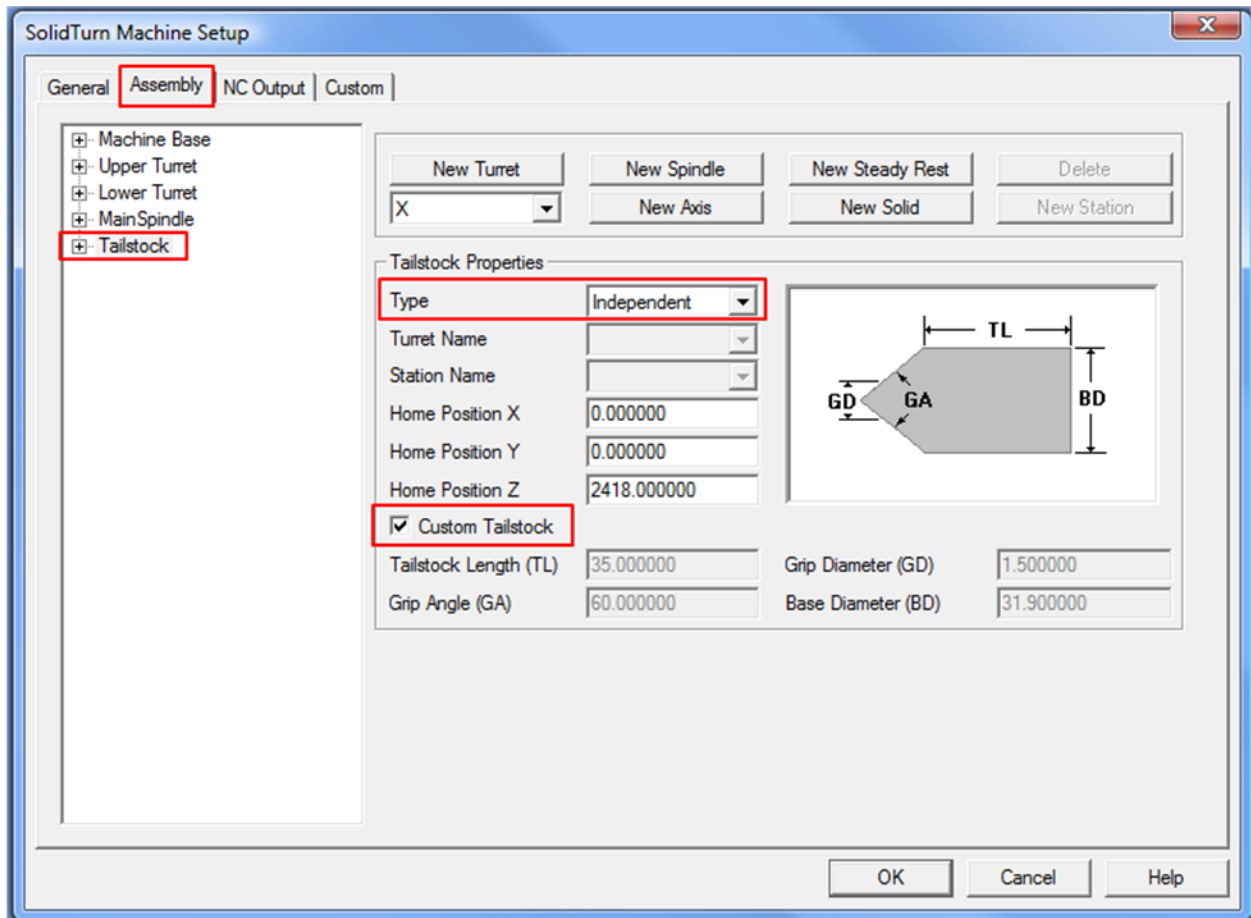
	NZX2500	NZX4000	NZX6000
Servo driven tailstock	HEAD1	-	HEAD1
Carriage direct-coupled tailstock	HEAD1	HEAD2	HEAD2

For NLX configuration, based on the type of tailstock installed on the machine, the NC output will differ. The type of tailstock is defined in the Machine Setup in ESPRIT (see part **4.4.4. Type of tailstock installed on the machine**).

### *15.1.2. Defining the tailstock in the Machine Setup*

Open the Machine Setup page and navigate to the Assembly tab. Click on Tailstock to view the Tailstock Properties window. Under Type select Independent. It is also possible to define specific geometric properties of the tailstock in this section. Check Custom Tailstock to use the geometry from the Machine Setup as the tailstock.

**Machine Setup Assembly Page:**



### 15.1.3. Creating the tailstock operation

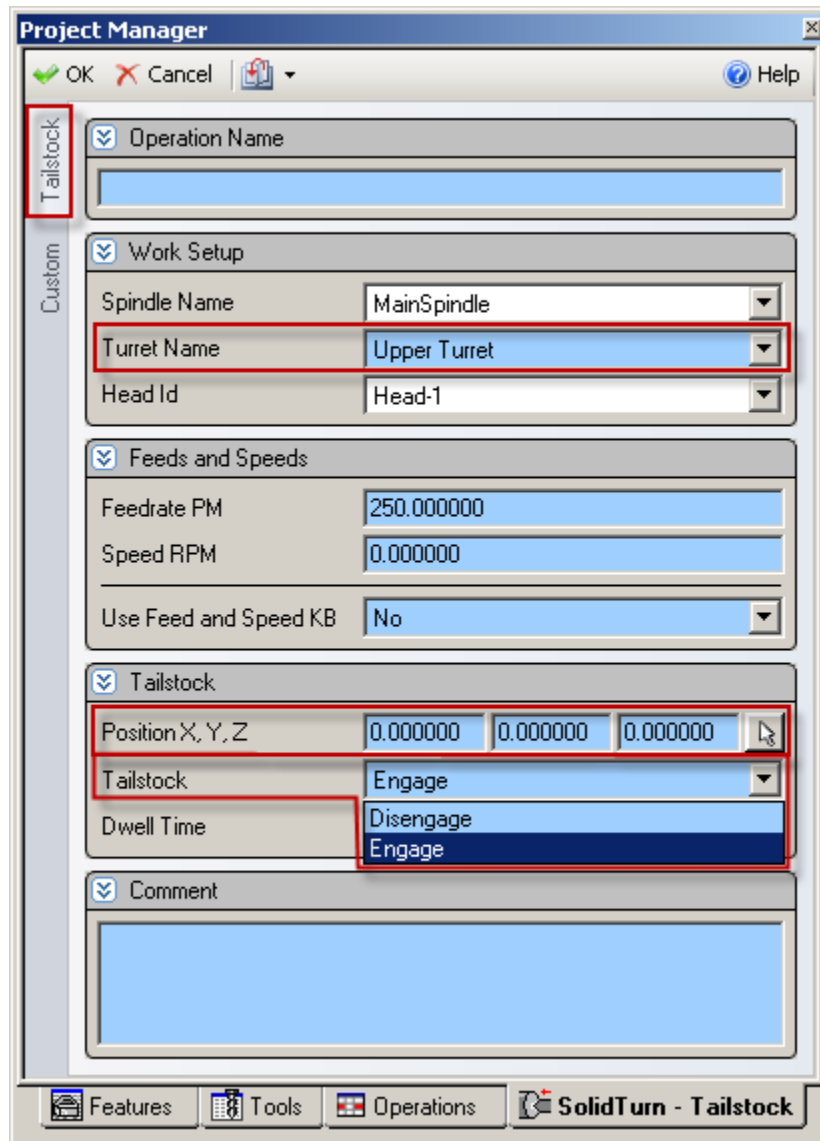
To create a Tailstock operation, click on the Tailstock operation on the SolidTurn toolbar.



On the Tailstock tab, select **Upper Turret** or **Lower Turret** (based on type of tailstock and machine) for **Turret Name** for NZX configuration.

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

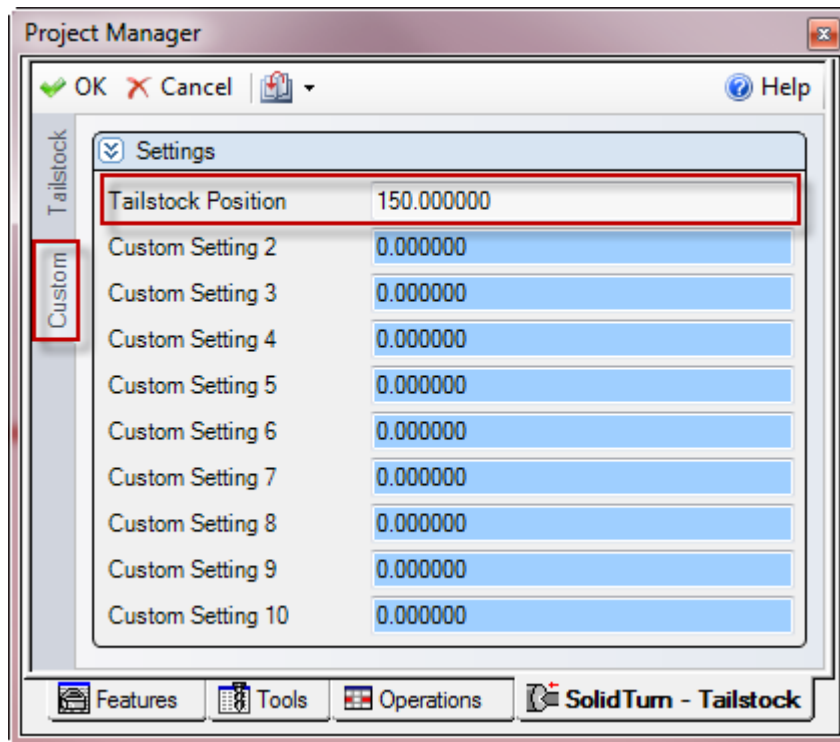
With Position X, Y, Z, set the reference tailstock engage point. This is the point that the tailstock will engage the part at during simulation.



In the Dwell Time field, you can specify a dwell that will be output at the end of the tailstock engagement.



In the Tailstock Position (Custom Setting 1) field of the operation page, enter the position of the tailstock (machine coordinate value) **V** for NZX & NLX with carriage direct-coupled tailstock configurations. The tailstock will travel 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.



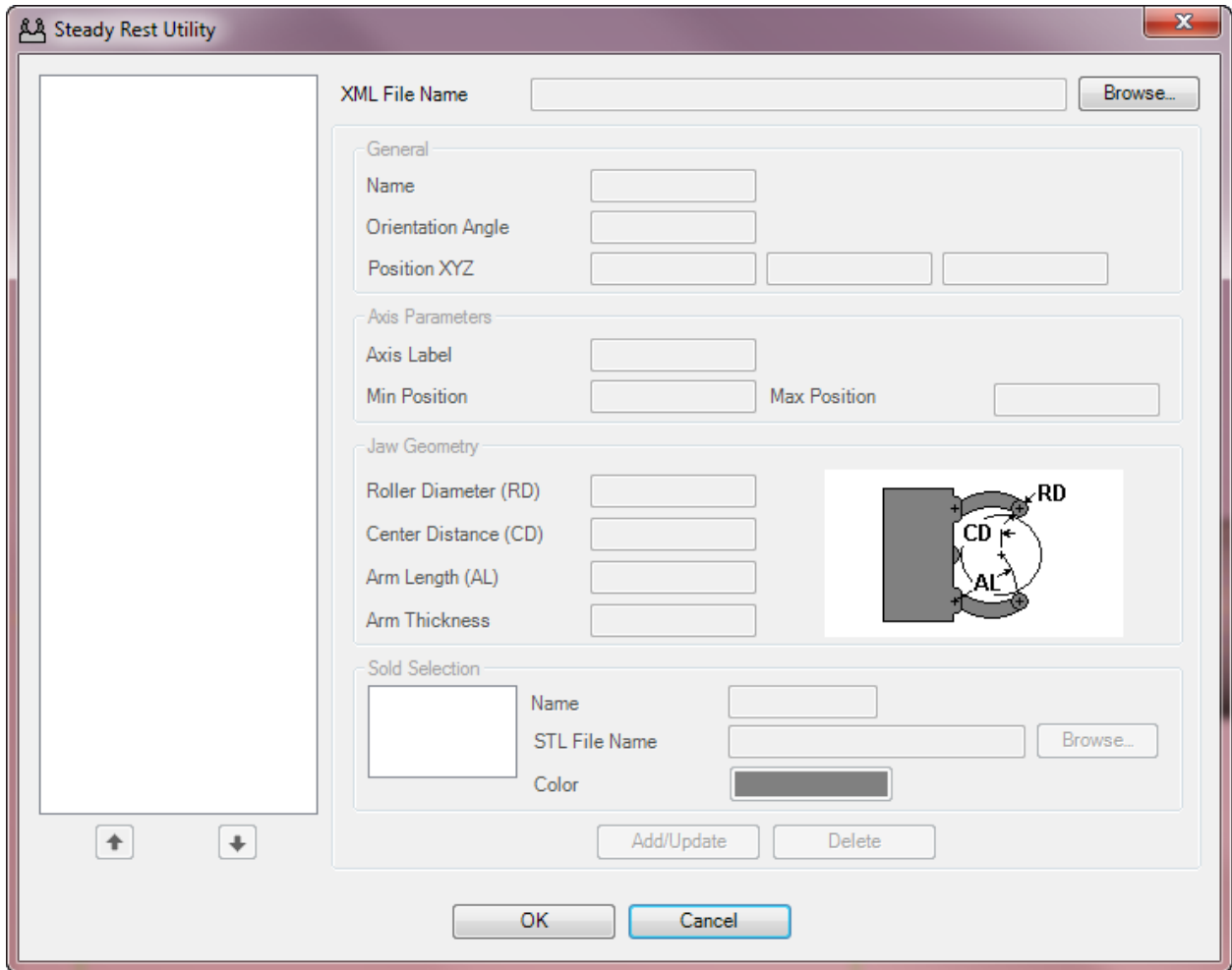
## **15.2. Independent steady rest**

### *15.2.1. Introduction*

When machining long work pieces with the Mori Seiki NZX & NLX machines, it is sometimes beneficial to use an independent steady rest to support the workpiece while you are machining.

### *15.2.2. Defining the steady rest*

On the Steady Rest Utility of the MoriSeiki AddIn, you will be able to create your independent steady rest.



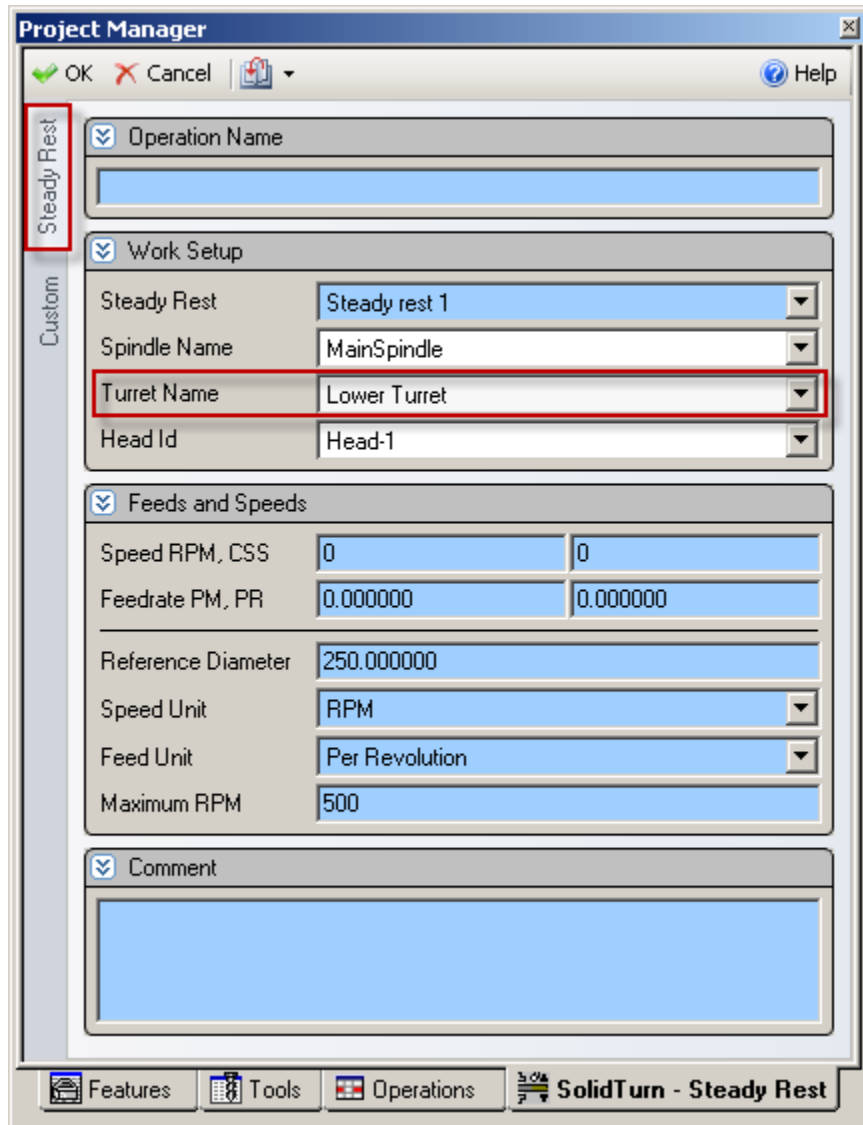
*Note:* Please refer to your MoriSeiki AddIn Manual to have detailed instructions on how to create the steady rest with this utility.

### 15.2.3. Creating the steady rest operation

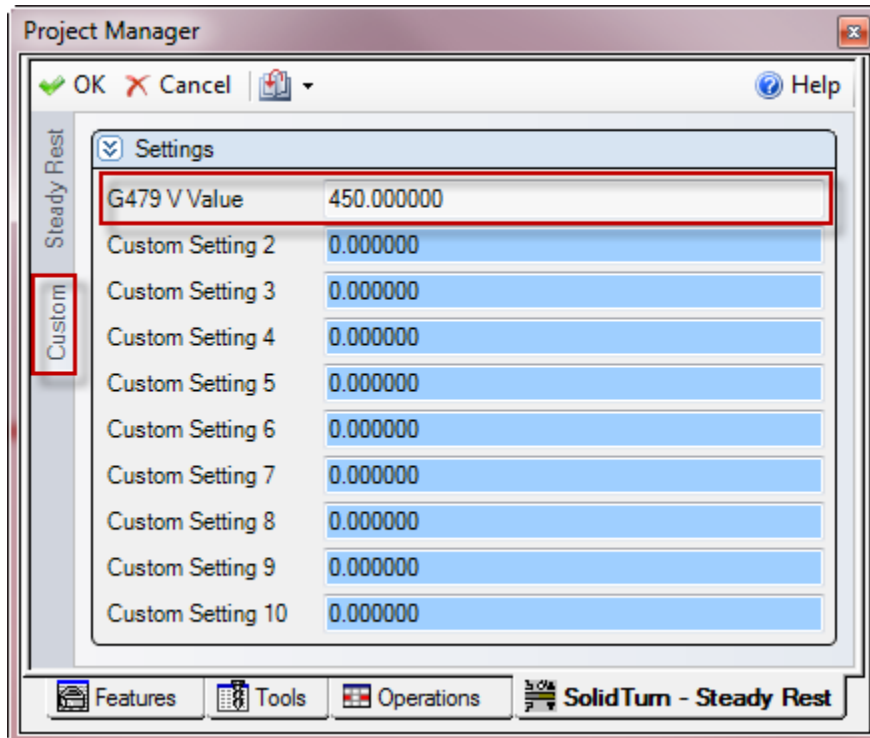
To create a Steady Rest operation, click on the Steady Rest operation on the SolidTurn toolbar.



On the Steady Rest tab, select **Lower Turret** for **Turret Name** since the steady rest is controlled by the lower turret for NZX configuration.



In the G479V Value (Custom Setting 1) field of the operation page, enter the position of the steady rest (machine coordinate value) **V**. The steady rest will travel to this position.



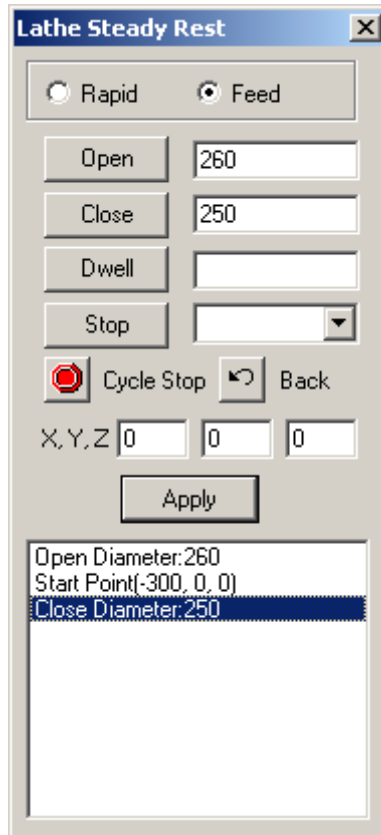
When you click OK, a new window will pop up that allows you to create the path for your steady rest. In this window you will have several options to create your steady rest operation:

- Rapid or Feed moves: Use your cursor to select a point to move to and then click the “Apply” button to record that move; Note that Rapid and Feed modes will produce the same NC code since the steady rest is only moved using a macro (**G479 V**)
- Open (**M340**) and Close (**M341**) at specified diameters: To record an open or close move, enter the diameter in the field next to the button (Open or Close) you want to use and simply click on the respective button
- Dwell times and Stop codes: Not used

Note that while programming the steady rest movements, you can use Back if you did a mistake and need to undo the last created entry.

When you are finished programming your steady rest movements, click on Cycle Stop to close the window and create your Steady Rest operation.





In the NC code (of upper and lower turrets for NZX configuration), the steady rest coolant code (ON/OFF) will automatically be output along with regular coolant codes while the steady rest is closed.

### ***15.3. Long Boring Bar (for NZX6000 configuration only)***

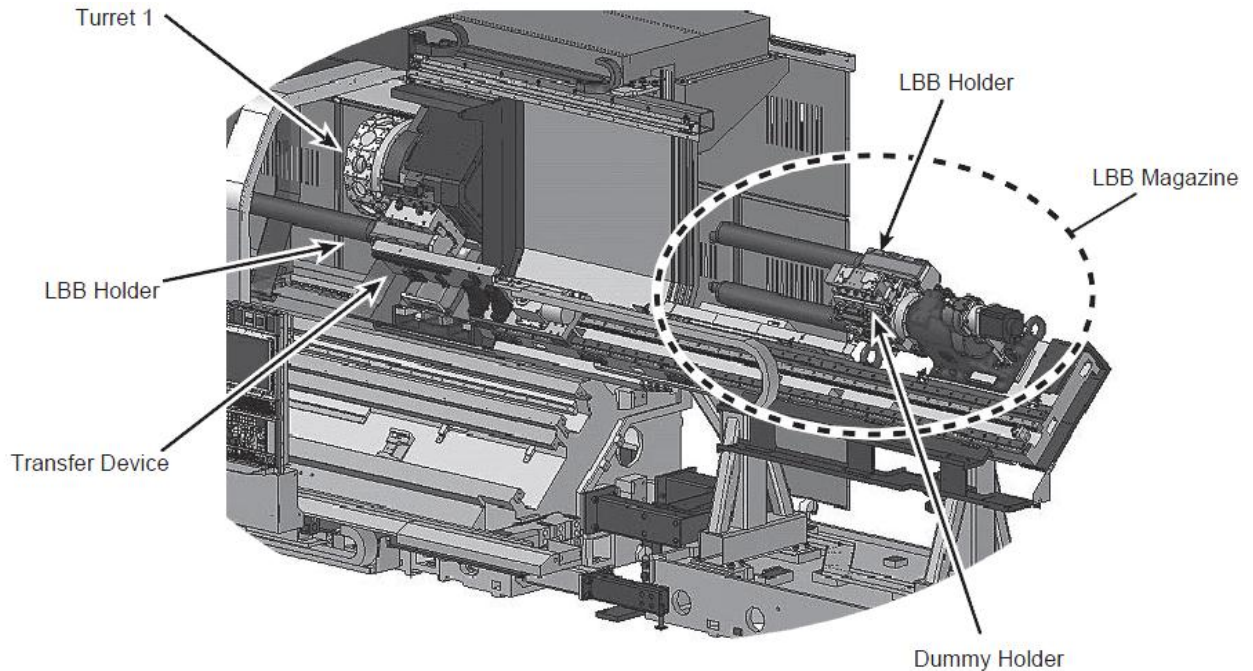
**G361 M\_ ;**

NZX6000 machines support Long Boring Bar (LBB) specification. LBB cannot be mounted on standard tool holders on the turret due to their size.

LBB holders are directly mounted on the X-axis of the upper turret. They have an automatic tool changer. Three LBB can be placed in the tool changer.

For LBB specification, tool station 12 on the upper turret is dedicated for LBB mounting.

Once a LBB is mounted in station 12, the upper turret cannot be indexed. Upper turret can only be indexed when dummy holder is mounted in station 12.



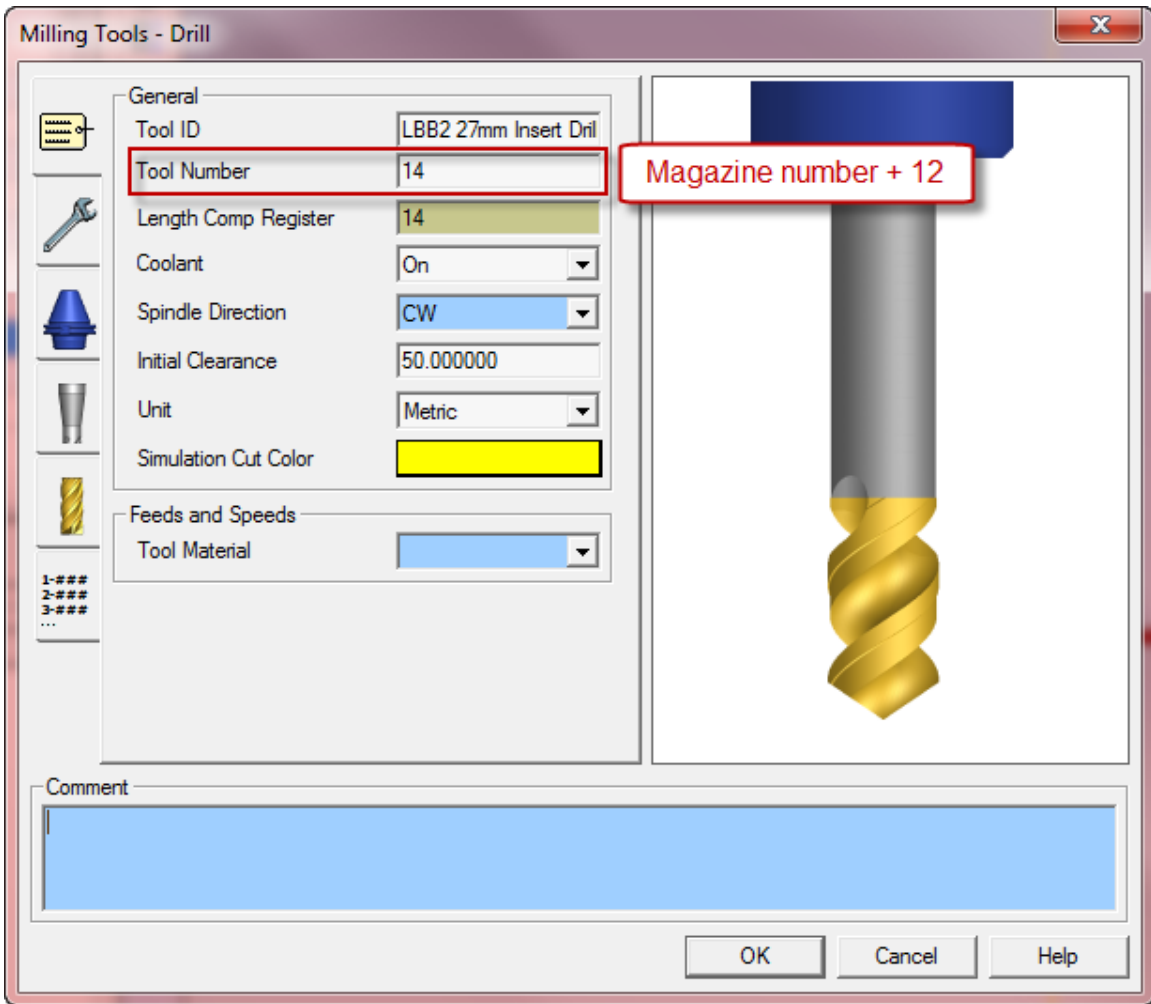
The LBB tool change command G-code (G361) is used to call the holder in the LBB magazine. The different options for M\_ are:

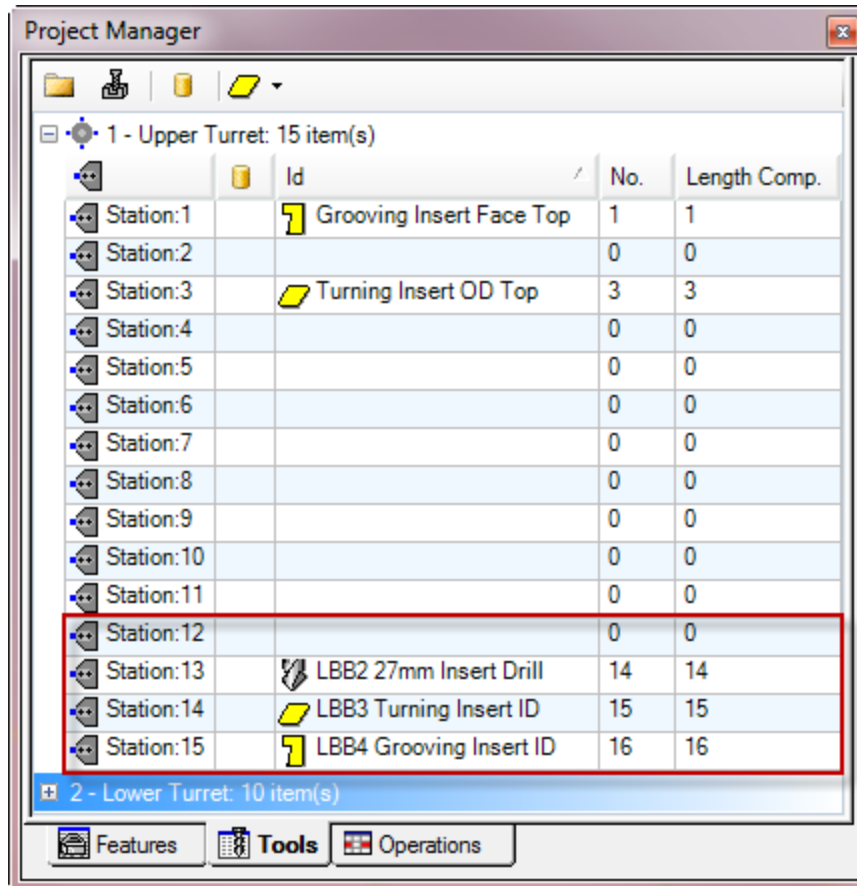
- **M1:** Dummy holder on LBB magazine 1
- **M2:** LBB holder 1 on LBB magazine 2
- **M3:** LBB holder 2 on LBB magazine 3
- **M4:** LBB holder 3 on LBB magazine 4

In ESPRIT, to program an operation using a LBB tool, simply create your LBB in station 13, 14 or 15. Station 13 will contain LBB holder 1 (located on LBB magazine 2), station 14 will contain LBB holder 2 (located on LBB magazine 3) and station 15 will contain LBB holder 3 (located on LBB magazine 4).

The corresponding G361 M\_ codes will be output in the NC code.

Note that in ESPRIT, you need to set your tool number to the magazine number + 12. For example for magazine 2, you would set your tool number to 14 (12+2).





When switching to a regular tool holder (located in station 1 to 11), G361 M1 will automatically be output in the NC code to load the dummy holder prior to index the upper turret to the regular tool.

## 16. Custom Settings index

### 16.1. On operation pages

	See page	Use	Comments
Custom Setting 1	158	Tailstock position (G479)	NZX Head 1 & 2 / NLX: Used for Tailstock cycle
	161	Steady Rest position (G479)	NZX Head 2 / NLX: Used for Steady Rest cycle
	62	Park turret with G53	NZX Head 1 & 2 / NLX: Used for Tailstock cycle / Used for Park cycle
	110	Workpiece pushing check (G38)	NLX: Used for Part pickup cycle
	48	Output of rapid moves (G00) with linear interpolation moves (G01 F)	NZX Head 1 / NLX: Used for all milling cycles
Custom Setting 2	64	Output stop coolant code (M09)	NZX Head 1 & 2 / NLX: Used for Park cycle
	110	B-axis retraction stroke	NLX: Used for Part pickup cycle / Goes with Custom Setting 1
Custom Setting 3	110	B-axis feedrate	NLX: Used for Part pickup cycle / Goes with Custom Setting 1

Custom Setting 4	69	X position of bar stopper tool (G53)	NLX: Used for Barfeed (By stopper) cycle
	110	Tolerance for workpiece transfer position	NLX: Used for Part pickup cycle / Goes with Custom Setting 1
Custom Setting 5	69	Z position of bar stopper tool (G53)	NLX: Used for Barfeed (By stopper) cycle
Custom Setting 6	150	Spindle/Rotary Tool Spindle Simultaneous Operation Mode	NZX Head 1 / NLX: Used for SolidTurn Drilling cycle
	94	C-Axis mode for spindle orientation	NLX: Used for Part pickup cycle
Custom Setting 7	150	Percentage of the total speed you want to apply to the rotary tool	NZX Head 1 / NLX: Used for SolidTurn Drilling cycle / Goes with Custom Setting 6
	94	C-Axis angle value	NLX: Used for Part pickup cycle / Goes with Custom Setting 6
Custom Setting 8	77	Spindle 2 catching position	NLX: Used for Part release cycle
Custom Setting 9	77	Workpiece ejector (M47)	NLX: Used for Part release cycle
Custom Setting 10	41	Automatic determination of tool nose radius	NZX Head 1 & 2 / NLX: Used on all operation pages

## 16.2. On tool pages

	See page	Use	Comments
Custom Setting 1	51	Second coolant type	Used to set a second coolant (Optional)
Custom Setting 2		Not used	
Custom Setting 3	29	Second home position	Used to use second home position (G30)
Custom Setting 4		Not used	
Custom Setting 5		Not used	
Custom Setting 6	52	Special coolant ON	Used to turn on special coolant (Optional)
Custom Setting 7	52	Special coolant OFF	Used to turn off special coolant (Optional) / Goes with Custom Setting 6
Custom Setting 8	25	Tool life management	Used to turn on/off tool life management
Custom Setting 9	25	Tool life management - Counting by times used (M89)	Used to output M89 (Counting by times used) / Goes with Custom Setting 8
Custom Setting 10		Not used	

## 16.3. In Machine Setup

	See page	Use	Comments
Custom Setting 1	15	Output of Program End (M02), Program End and Rewind (M30) or Sub-Program End (M99)	
Custom Setting 2	15	Work unloader on spindle 1 or 2 side	For NLX configuration only
Custom Setting 3	16	Type of tailstock installed on the machine	
Custom Setting 4	16	Tool station for part transfer and work unloader	For NLX configuration only
Custom Setting 5		Not used	
Custom Setting 6		Not used	
Custom Setting 7	17	C-axis brake clamp/unclamp	
Custom Setting 8	17	Set how to skip spindle control using comment form, Block Skip or Block Skip 2	For NZX configuration only
Custom Setting 9	17	C-axis roll over	
Custom Setting 10	18	Set position of optional stop code (M01) in the NC code	
Custom String 10	18	Set position of sequence numbers (N)	